Created in COMSOL Multiphysics 6.1

Electrodynamics of a Power Switch

Introduction

Events such as overcurrent or overload can seriously damage electrical circuits or power lines. A possible solution to this problem is the implementation of circuit breakers in the form of automatic electrical switches, which mechanically interrupt the current flow by moving a plunger as soon as a defect is detected. In contrast to a fuse, which has to be replaced after its activation, a circuit breaker can be reset.

Circuit breakers can be classified according to features like voltage rating, construction type, structural features, and interruption technique. The main purpose of this model is to illustrate the working principle and some possible solutions for modeling a particular class of circuit breakers, namely *magnetic power switches*. These are electromechanical devices in which iron plungers are moved by means of the magnetic attraction exerted by current flowing in coils. Turning off the driving current resets the switch to the initial state. Similar technology is present also in electrovalves and many other magnetic actuators.

The model includes rigid-body dynamics under the influence of magnetic forces, induced currents, and spring/constraint arrangements that keep the plunger in its equilibrium position.

Model Definition

The geometry of the electromechanical device is shown in Figure 1. Two bulk E-shaped iron cores are separated by an air gap. A copper coil is placed on the central leg of the lower E-core, which is kept fixed. As current flows in the coil, an attractive force is exerted on the upper E-core (the moving plunger), which is held in place by a prestressed spring. When the force reaches a threshold value, the plunger moves toward the lower E-core, closing the air gap. The model illustrates how to properly simulate the movement and the closing time, which depends on the spring stiffness.

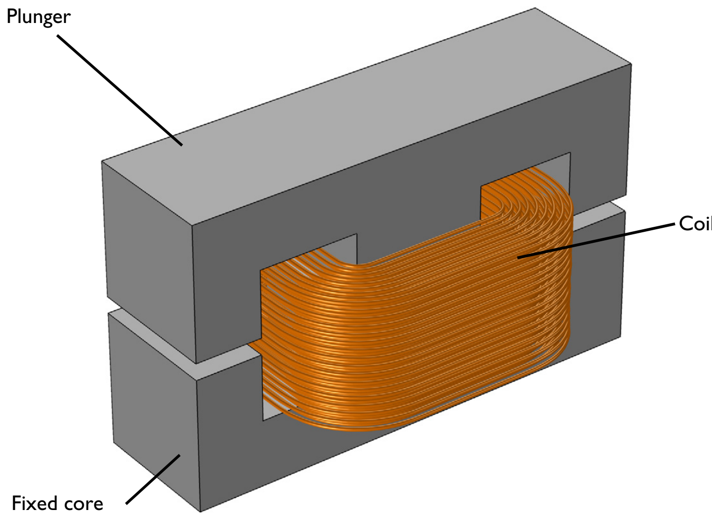


Figure 1: Geometry of the switch.

The geometry is built using COMSOL Multiphysics' CAD tools and taking advantage of parameterized Geometry Parts, allowing a finer control on the geometry. Due to the symmetry of the device, it is possible to model only a quarter of the geometry. Figure 2 shows the simulation geometry, complete with size specifications, which are added as model parameters in COMSOL Multiphysics. In order to compute the electromagnetic fields, the power switch is embedded in an air domain.

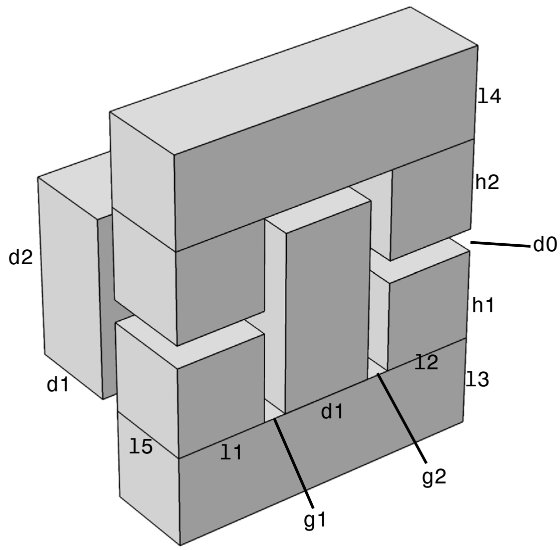


Figure 2: Switch specifications.

The model solves a preprocessing study to compute the parameterization of the air gap and the coil direction, as detailed in the following sections, then a main Time Dependent study step from $t = 0$ s to $t = 1$ s.

PHYSICS INTERFACES

This model uses the Magnetic Fields interface together with the Moving Mesh features, which allows computing magnetic fields in a geometry that changes with time due to the closure of the air gap. The Magnetic Fields interface computes the time-dependent magnetic field generated by the coil, the current densities induced in the coils, and their magnetic effects. A Coil feature using a homogenized multiturn model is used for the excitation. The direction of the current flow in the coil is computed automatically in a Coil Geometry Analysis study step. The attractive force acting on the moving domain is computed using a Force Calculation feature, and the value is used in the ordinary differential equation (in a Global ODEs and DAEs interface) that describes the plunger dynamics according to Newtonian mechanics.

To improve the stability of the solution in presence of nonlinear magnetic materials, linear elements are used for the discretization of the magnetic vector potential.

MESH DEFORMATION

During the switching process, the plunger moves rigidly in the vertical direction, while the lower core remains fixed. The mesh in the air gap must then be deformed consistently to accommodate this movement. This is taken care of by the Yeoh smoother.

In order to avoid the complete collapse of the mesh, displacement is limited to be at most 95% of the initial air gap, with a marginal impact on the results. The model setup considers a completely inelastic impact. A possible alternative solution to model the complete collapse of gap is to use a Stop Condition in the solver sequence, continuing the simulation with a geometry without the gap, or by means of the Events interface. With this approach it is also possible to treat an elastic or partially elastic collision between the plunger and fixed core.

Results and Discussion

Different stages can be identified in the transient analysis. During the first 45 ms of the simulation the current grows but the electromagnetic attractive force is not enough to overcome the opposing spring force. In the interval between 45 ms and 85 ms the electromagnetic force increases further and the plunger starts to translate downward, toward the iron core; once it reaches this new position it stops its movement. During this stage the current decreases due to the inductance changing, reaching its minimum value with the closing of the air gap. When the plunger and core make contact, a new stationary RL circuit is created. The current starts to increase again with a slope depending on the new characteristic time of the device.

Figure 3 and Figure 4 show the magnetic fields on symmetry planes when the gap is still open (Figure 3) and at a time where the gap is closed (Figure 4). In both cases, it is evident that induced eddy currents are screening the interior of the core from the field. These currents are limited to a region as large as the skin depth, resolved by the boundary layer mesh.

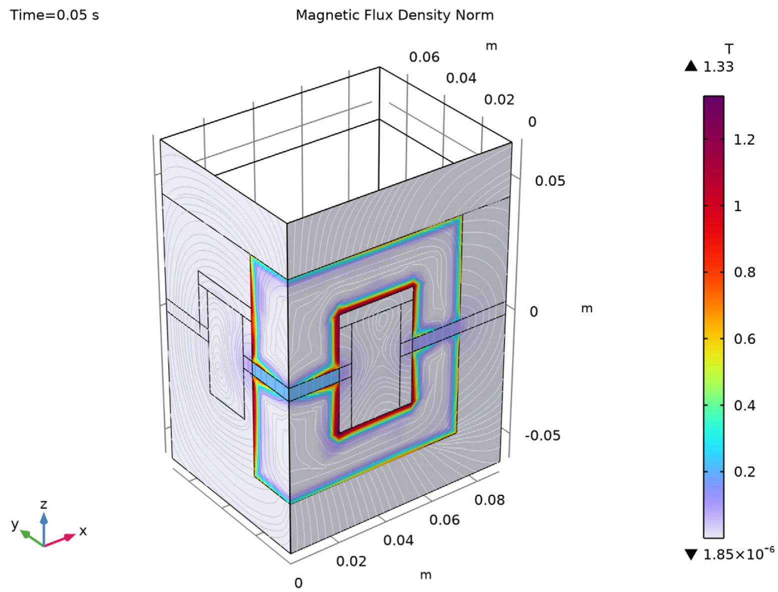


Figure 3: Magnetic flux density norm at t = 0.05 s, when the gap is open.

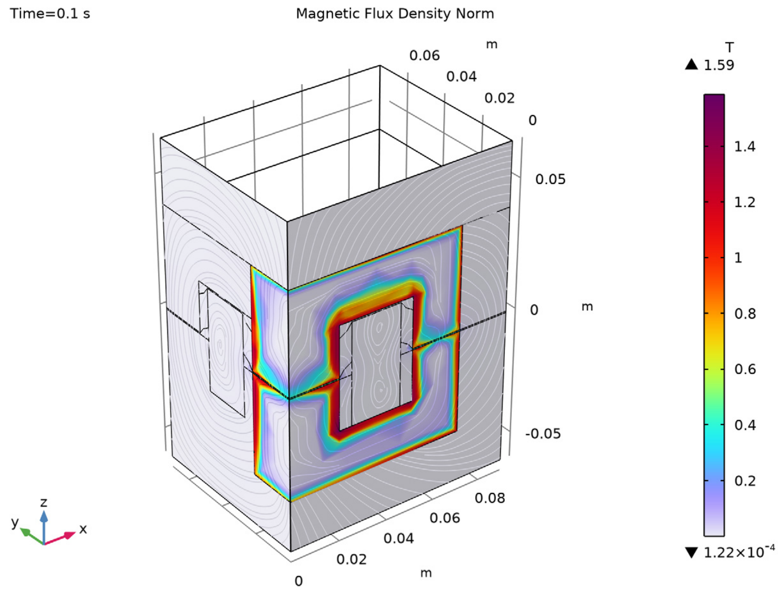


Figure 4: Magnetic flux density norm at t = 0.1 s, when the gap is closed.

Figure 5 shows the evolution of magnetic field streamlines at different instants of the simulation. The upper-left image refers to a time instant in which the spring is still prestressed. As the magnetic force increases, it starts to compress the spring (upper-right) until it reaches its maximum compression (lower-left). Well before the final time of the simulation, the spring is completely compressed and the induced currents in the core have decayed (lower-right).

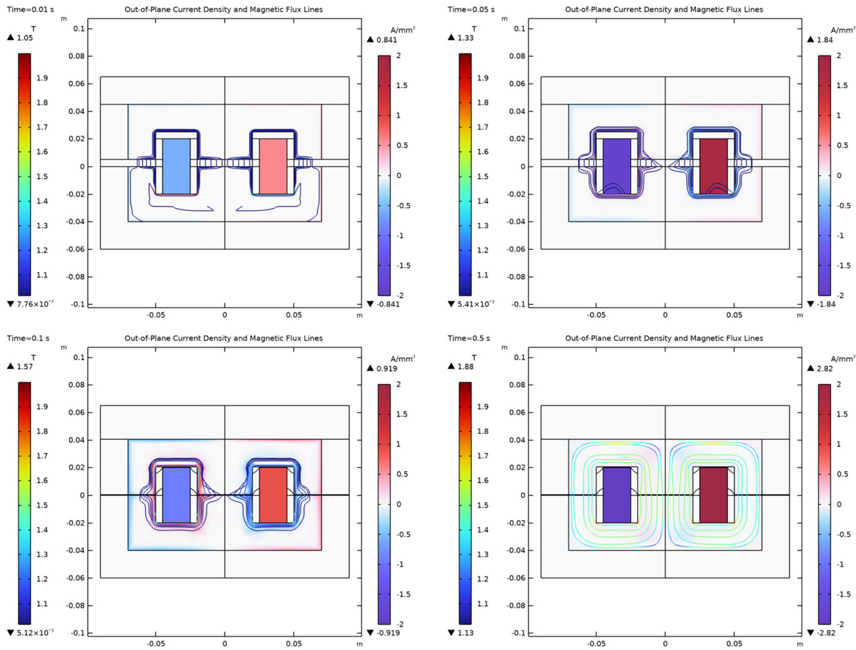


Figure 5: Evolution of current density (surface) and magnetic flux density (streamlines) at different times.

Figure 6 shows the core losses in the device due to induced current density. This information may be relevant for predicting possible overheating of the device.

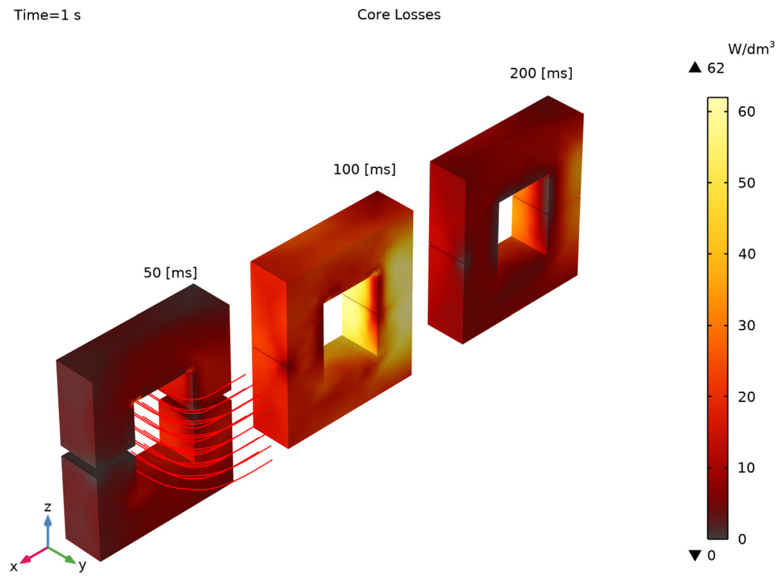


Figure 6: Core losses due to induced currents at different time instants.

A series of 1D plots are also created to highlight the dynamics of the magnetic switch, before, during, and after plunger motion:

- *Before plunger motion*: Figure 7 shows the first stage of the simulation, when the spring is not yet compressed. The blue and green lines represent normalized currents and gap size respectively. The red line is an exponential fit for the RL current dynamics of the initially nonmoving inductor — the response of an ideal system.
- *During plunger motion*: the compression of the spring and the resulting closure of the gap are visualized in Figure 8. The normalized currents and gap size are represented by blue and green lines, respectively, while the red line shows the mechanical power (which is nonzero only during the motion of the plunger).
- *After plunger motion*: Figure 9 refers to the last stage of the simulation, when the spring is completely compressed. The red line shows the induction losses in the core, which are significant during the movement of the plunger. Depending on the details of the device and the desired performance, this aspect may need to be taken into account

during the design process. After the movement has been completed, the current starts increasing again, as expected in a (nonlinear) RL circuit.

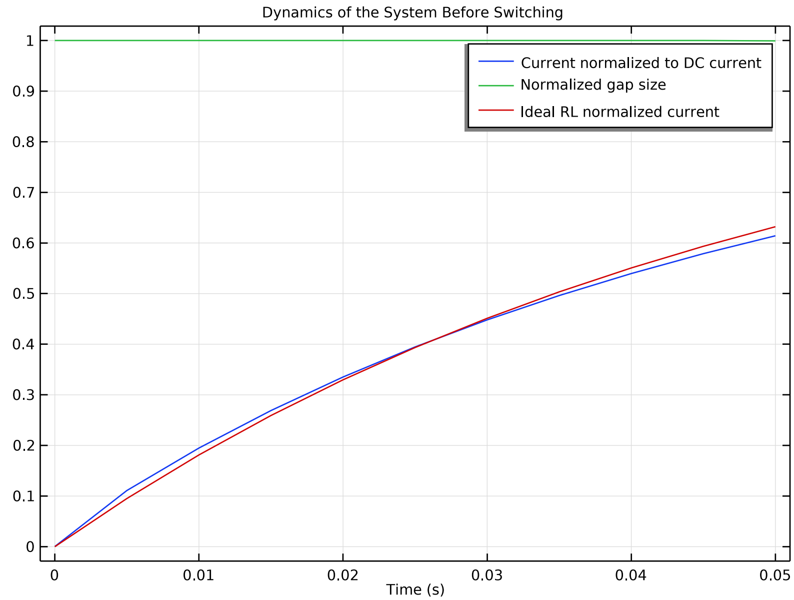


Figure 7: Coil current and gap size plotted versus time, and normalized to the DC current and initial gap size, respectively. Also plotted is a normalized ideal RL current for reference.

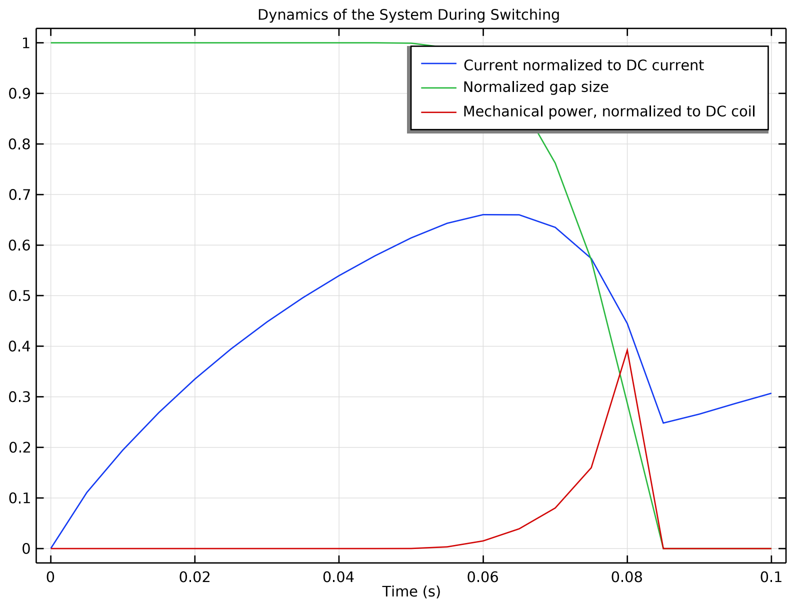


Figure 8: Coil current, gap size, and mechanical power exerted on the plunger, plotted versus time and normalized to the DC current, initial gap size, and DC coil power, respectively.

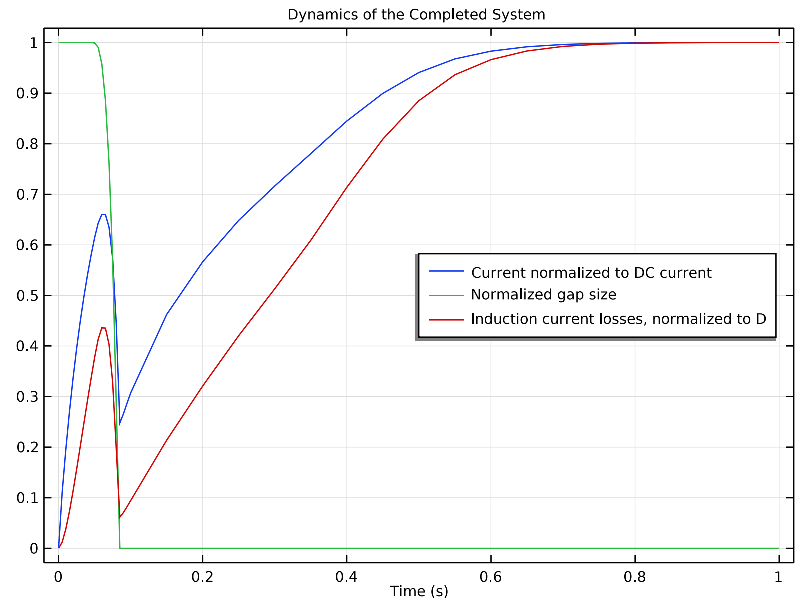


Figure 9: Coil current, gap size, and induction current losses, plotted versus time and normalized to the DC current, initial gap size, and DC coil power, respectively.

It is worth recalling that the blue and green curves in Figure 7, Figure 8, and Figure 9 represent the same physical quantities: the normalized current and the normalized gap size, respectively. The reason why they look different is the limit of the x-axis (time scale).

Application Library path: `ACDC_Module/Devices,_Transducers_and_Actuators/power_switch`

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 **3D**.
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 **Preset Studies for Selected Physics Interfaces> Coil Geometry Analysis**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Import the geometric and physical parameters from an external file.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `power_switch_parameters.txt`.

   In order to draw the parameterized geometry of the solid parts, create two 2D **Geometry Parts**, one for the core projection and one for the coil. These parts will be successively combined in a third 3D part.

   Note that the **Geometry representation** must be set to **COMSOL kernel** to fit with the numbering of the domains used in these instructions.

CORE SECTION

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts> 2D Part**.
2 In the **Settings** window for **Part**, type `Core Section` in the **Label** text field.

Make use of the Difference operator to create a horseshoe shape. Then mirror the shape to create the core section.

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 **Width** text field, type `l1+g1+d1+g2+l2`.

4 In the **Height** text field, type `h1+l3`.

5 Locate the **Position** section. In the **y** text field, type `-(h1+l3)`.

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 **Width** text field, type `g1+d1+g2`.

4 In the **Height** text field, type `h1`.

5 Locate the **Position** section. In the **x** text field, type `l1`.

6 In the **y** text field, type `-h1`.

Difference 1 (dif1)

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

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **r2** only.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Point on Line of Reflection** section.

3 In the **y** text field, type `d0/2`.

4 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

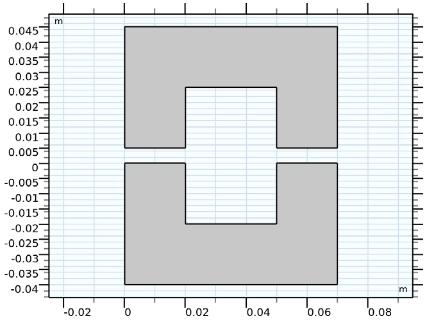
5 In the **y** text field, type `1`.

6 Select the object **dif1** only.

7 Locate the **Input** section. Select the **Keep input objects** check box.

8 Click **Build Selected**.

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



GLOBAL DEFINITIONS

Proceed to create the geometry part for the coil.

COIL SECTION

1 In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **2D Part**.

2 In the **Settings** window for **Part**, type `Coil Section` in the **Label** text field.

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 **Width** text field, type `d1`.

4 In the **Height** text field, type `15`.

5 Locate the **Position** section. In the **x** text field, type `l1+g1`.

Circle 1 (c1)

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

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

3 In the **Radius** text field, type `g1+d1`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **x** text field, type `l1`.

6 In the **y** text field, type `15`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`d1`

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 **Width** text field, type `l1`.

4 In the **Height** text field, type `d1`.

5 Locate the **Position** section. In the **y** text field, type `l5+g1`.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Coil Section** and choose **Delete Entities**.

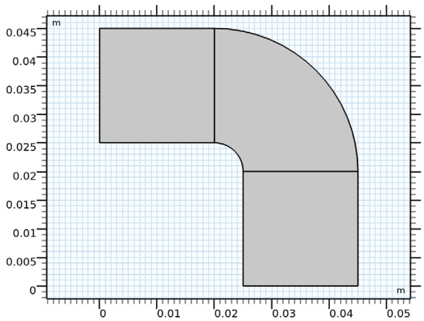
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **c1**, select Domain 1 only.

5 Click **Build Selected**.

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



GLOBAL DEFINITIONS

Create a three-dimensional geometry part for the solid object, combining the two previously created parts.

SOLID PARTS

1 In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **3D Part**.

2 In the **Settings** window for **Part**, type `Solid Parts` in the **Label** text field.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **xz-plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Core Section 1 (pi1)

In the **Work Plane** toolbar, click **Parts** and choose **Core Section**.

Nonlinear Core

1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Solid Parts** right-click **Work Plane 1 (wp1)** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, type `Nonlinear Core` in the **Label** text field.
3 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
4 From the **Show in instances** list, choose **All levels**.
5 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
15

6 Select the **Reverse direction** check box.

Upper Core

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, type `Upper Core` in the **Label** text field.
3 On the object **ext1**, select Domain 2 only.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 In the **z-coordinate** text field, type `-h1`.

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2)>Coil Section 1 (pi1)

In the **Work Plane** toolbar, click **Parts** and choose **Coil Section**.

Coil

1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Solid Parts** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, type `Coil` in the **Label** text field.

3 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

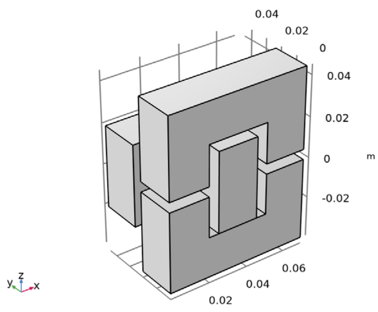
4 From the **Show in instances** list, choose **All levels**.

5 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
d2

6 In the **Geometry** toolbar, click **Build All**.

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



Now create the actual simulation geometry. Start by creating an instance of the three-dimensional geometry part just added. After that, a few additional solid parts have to be drawn in order to partition properly the air domain surrounding the coil and the core. The purpose of partitioning the air gap is to minimize the distortion of the mesh during deformation.

GEOMETRY 1

Solid Parts 1 (pi1)

1 In the **Geometry** toolbar, click **Parts** and choose **Solid Parts**.

2 In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.

3 In the table, enter the following settings:

Name	**Keep**	**Physics**	**Contribute to**
Nonlinear Core	√	√	None
Upper Core	√	√	None
Coil	√	√	None

4 Click **Build All Objects**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `l1+g1+d1+g2+l2+2[cm]`.

4 In the **Depth** text field, type `l5+g1+d1+g3+2[cm]`.

5 In the **Height** text field, type `h1+l3+d0+h2+l4+4[cm]`.

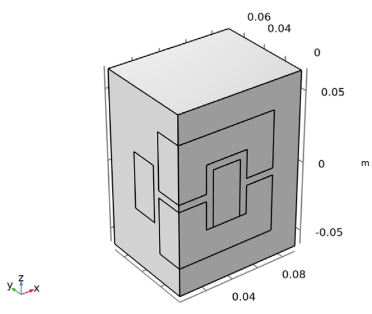
6 Locate the **Position** section. In the **z** text field, type `-h1-l3-2[cm]`.

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

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



Partition Domains 1 (pard1)

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

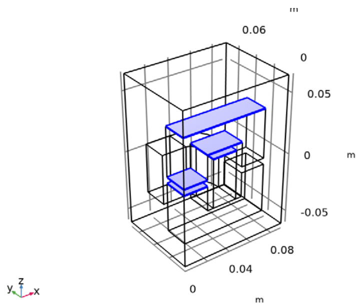
2 On the object **uni1**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended faces**.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

6 On the object **uni1**, select Boundaries 7, 10, 11, 24, and 33 only.



Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Offset type** list, choose **Through vertex**.

4 Find the **Offset vertex** subsection. Click to select the **Activate Selection** toggle button.

5 On the object **pard1**, select Point 34 only.

6 Click to expand the **Local Coordinate System** section. From the **Origin** list, choose **Vertex projection**.

7 Find the **Vertex for origin** subsection. Click to select the **Activate Selection** toggle button.

8 On the object **pard1**, select Point 34 only.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Cross Section 1 (cro1)

In the **Work Plane** toolbar, click **Cross Section**.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 Right-click **Plane Geometry** and choose **Delete Entities**.

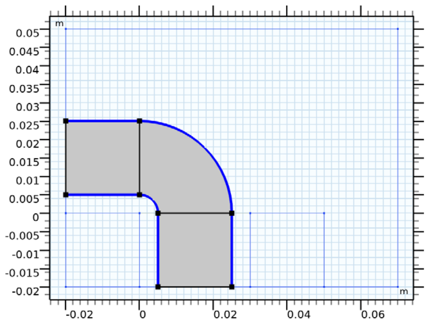
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **cro1**, select Domains 1, 2, 4, and 7 only.

Work Plane 1 (wp1)>Offset 1 (off1)

1 In the **Work Plane** toolbar, click **Offset**.
2 In the **Settings** window for **Offset**, locate the **Input** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 On the object **del1**, select Boundaries 2, 3, 5, and 8–10 only.



5 Clear the **Keep input objects** check box.
6 Locate the **Options** section. In the **Distance** text field, type `d0`.

Work Plane 1 (wp1)>Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Coordinates** section. In the table, enter the following settings:

xw (m)	yw (m)
0	d1+2*d0
0	0
d1+2*d0	0

5 Click **Build Selected**.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 From the **Input object handling** list, choose **Keep**.

4 Locate the **Distances** section. From the **Specify** list, choose **Vertices to extrude to**.

5 On the object **pard1**, select Point 11 only.

6 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Cylinder Selection 1 (cylsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Cylinder Selection**.

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

3 In the **Top distance** text field, type `d0+h2+l4`.

4 In the **Bottom distance** text field, type `d0`.

5 Locate the **Position** section. In the **y** text field, type `l5+g1+d1+g3+2[cm]`.

Form Composite Domains 1 (cmd1)

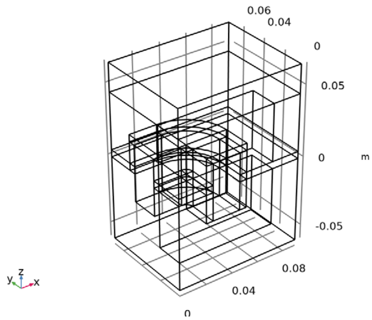
1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 In the **Settings** window for **Form Composite Domains**, locate the **Input** section.

3 From the **Domains to composite** list, choose **Cylinder Selection 1**.

4 Click **Build Selected**.

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



Proceed to create a number of selections in order to have a robust geometric parameterization.

DEFINITIONS

Plunger

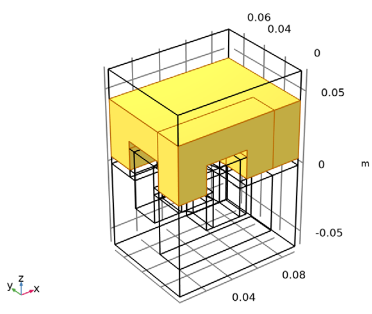
1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Plunger` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Upper Core (Solid Parts 1)** and **Cylinder Selection 1**.

5 Click **OK**.

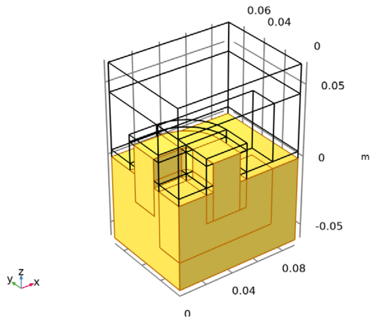


Fixed Domains

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

2 In the **Settings** window for **Box**, type `Fixed Domains` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z maximum** text field, type `-d0`.



Deformed Domains

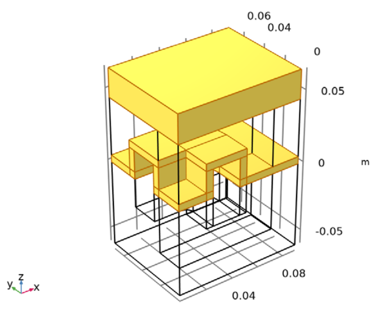
1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Deformed Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, in the **Selections to invert** list, choose **Plunger** and **Fixed Domains**.

5 Click **OK**.

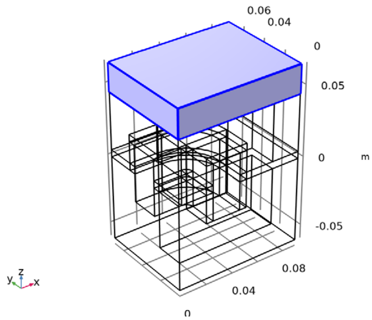


Top Domain

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

2 In the **Settings** window for **Explicit**, type `Top Domain` in the **Label** text field.

3 Select Domain 5 only.



Domain in between

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

2 In the **Settings** window for **Difference**, type `Domain in between` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **Deformed Domains** in the **Selections to add** list.

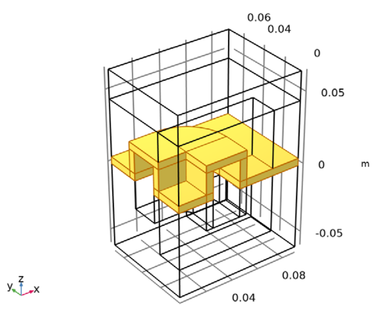
5 Click **OK**.

6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog box, select **Top Domain** in the **Selections to subtract** list.

9 Click **OK**.



Ext. boundaries to Deformed Domains

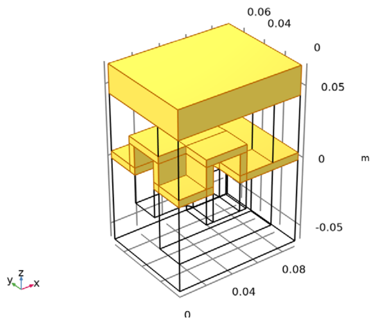
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Ext. boundaries to Deformed Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Deformed Domains** in the **Input selections** list.

5 Click **OK**.



Ext. Boundaries to Plunger

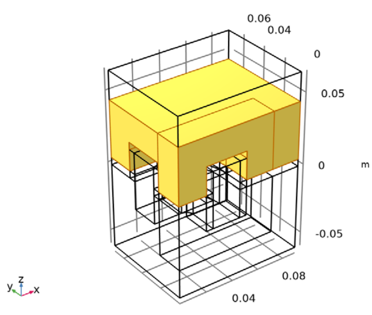
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Ext. Boundaries to Plunger` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Plunger** in the **Input selections** list.

5 Click **OK**.



Ext. Boundaries to Fixed Domains

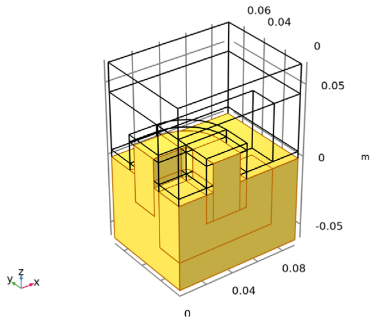
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Ext. Boundaries to Fixed Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Fixed Domains** in the **Input selections** list.

5 Click **OK**.



Fixed Boundaries at Plunger

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

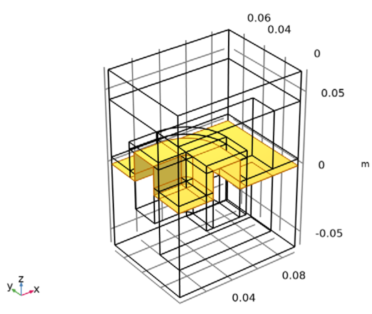
2 In the **Settings** window for **Intersection**, type `Fixed Boundaries at Plunger` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Ext. boundaries to Deformed Domains** and **Ext. Boundaries to Fixed Domains**.

6 Click **OK**.



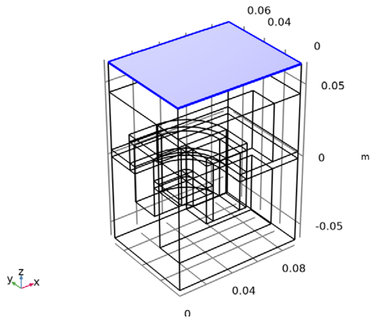
Top boundary

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

2 In the **Settings** window for **Explicit**, type `Top boundary` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 16 only.



Fixed Boundaries

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

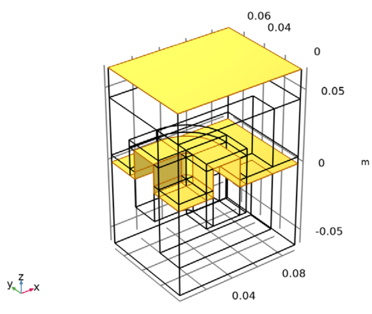
2 In the **Settings** window for **Union**, type `Fixed Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Fixed Boundaries at Plunger** and **Top boundary**.

6 Click **OK**.



Moving Boundaries

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

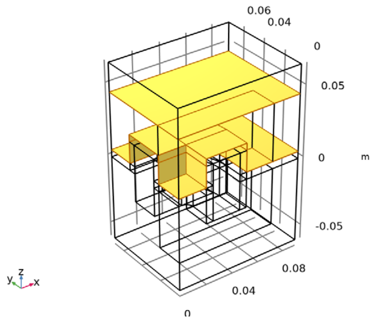
2 In the **Settings** window for **Intersection**, type `Moving Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Ext. boundaries to Deformed Domains** and **Ext. Boundaries to Plunger**.

6 Click **OK**.



Side boundaries

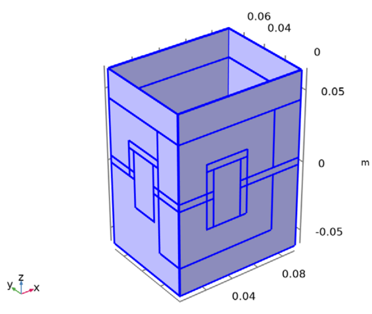
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Side boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

5 Select Boundaries 1, 2, 4, 5, 7, 8, 10, 11, 13, 14, 18, 21, 24, 27, 31, 38, 41, 45, 51–54, 56, 59, 62, 65, 97, 105, 108, 111, 118, and 125–129 only.



Ext. Boundaries to cores

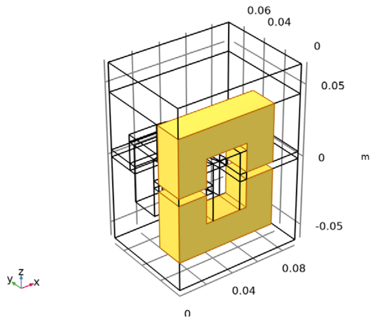
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Ext. Boundaries to cores` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Nonlinear Core (Solid Parts 1)** in the **Input selections** list.

5 Click **OK**.



Boundary layer mesh

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

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Boundary layer mesh`.

5 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

6 In the **Add** dialog box, select **Ext. Boundaries to cores** in the **Selections to add** list.

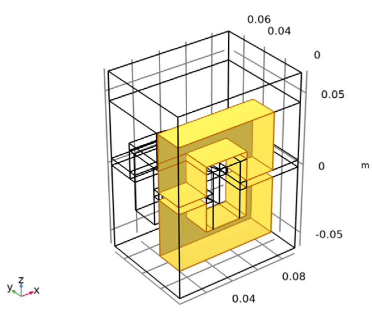
7 Click **OK**.

8 In the **Settings** window for **Difference**, locate the **Input Entities** section.

9 Under **Selections to subtract**, click **Add**.

10 In the **Add** dialog box, select **Side boundaries** in the **Selections to subtract** list.

11 Click **OK**.



MATERIALS

Next, add the materials — air, a coil material, and soft iron for the coil. For air, use a small and finite conductivity in order to be able to solve using the same formulation used in the conductors.

Air

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Air` in the **Label** text field.
3. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Coil Material

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Coil Material` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Coil (Solid Parts 1)**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0		S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

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 **AC/DC>Soft Iron (With Losses)**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

MATERIALS

Soft Iron (With Losses) (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Nonlinear Core (Solid Parts 1)**.

MESH 1

Move now to the mesh setup, using a **Coarser** setting for all domains that will later be overwritten in specific parts of the geometry.

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 **Coarser**.

Swept 1

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

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

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

4 From the **Selection** list, choose **Domain in between**.

Size 1

1 Right-click **Swept 1** and choose **Size**.

2 Select Boundaries 9, 20, 40, 60, 70, 88, 109, and 119 only.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `d0*0.5`.

8 Select the **Maximum element growth rate** check box. In the associated text field, type `1.2`.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 Select Domains 3, 6, 8, 11, 13, 14, 16, 17, 19, 21, and 24 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `2`.

Distribution 2

1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domains 7, 12, 15, 18, 22, and 25 only.

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Nonlinear Core (Solid Parts 1)**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Boundary layer mesh**.

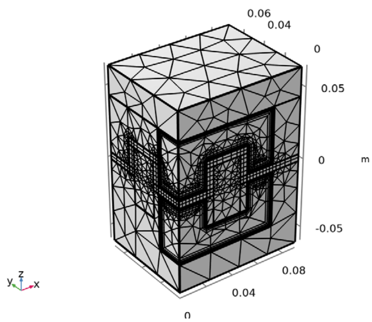
4 Locate the **Layers** section. In the **Number of layers** text field, type `7`.

5 In the **Stretching factor** text field, type `1.4`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type `0.2[mm]`.

8 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



Move on to set up the moving mesh to describe the plunger movement.

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains> Deforming Domain**.

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

3 From the **Selection** list, choose **Deformed Domains**.

4 Locate the **Smoothing** section. In the C_2 text field, type `100`.

Prescribed Deformation 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains> Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Geometric Entity Selection** section.

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

4 Locate the **Prescribed Deformation** section. Specify the *dx* vector as

0	X
0	Y
disp	Z

Prescribed Normal Mesh Displacement 1

1. In the **Definitions** toolbar, click **Moving Mesh** and choose **Boundaries>Prescribed Normal Mesh Displacement**.
2. In the **Settings** window for **Prescribed Normal Mesh Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Side boundaries**.

Prescribed Mesh Displacement 1

1. In the **Definitions** toolbar, click **Moving Mesh** and choose **Boundaries>Prescribed Mesh Displacement**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Fixed Boundaries**.

Prescribed Mesh Displacement 2

1. In the **Definitions** toolbar, click **Moving Mesh** and choose **Boundaries>Prescribed Mesh Displacement**.
2. In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Moving Boundaries**.
4. Locate the **Prescribed Mesh Displacement** section. Specify the *dx* vector as

0	X
0	Y
disp	Z

Continue to set up the physics of the model.

MAGNETIC FIELDS (MF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.
2. In the **Settings** window for **Magnetic Fields**, click to expand the **Discretization** section.
3. From the **Magnetic vector potential** list, choose **Linear**.

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 **Coil (Solid Parts 1)**.

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

5 From the **Coil type** list, choose **Numeric**.

6 From the **Coil excitation** list, choose **Voltage**.

7 In the V_{coil} text field, type `10[V]`.

8 Locate the **Homogenized Multiturn Conductor** section. In the N text field, type `filling*d1*d2/a_coil`.

9 In the a_{wire} text field, type `a_coil`.

Geometry Analysis 1

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

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

3 Find the **Symmetry specification** subsection. In the F_L text field, type `4`.

Input 1

1 In the **Model Builder** window, expand the **Geometry Analysis 1** node, then click **Input 1**.

2 Select Boundary 97 only.

Geometry Analysis 1

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

Output 1

1 In the **Physics** toolbar, click **Attributes** and choose **Output**.

2 Select Boundary 31 only.

Ampère's Law 2

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 **Nonlinear Core (Solid Parts 1)**.

Force Calculation 1

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

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

3 From the **Selection** list, choose **Upper Core (Solid Parts 1)**.

Gauge Fixing for A-field 1

1 In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-field**.

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

3 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

4 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	**f(u,ut,utt,t) (1)**	**Initial value (u_0) (1)**	**Initial value (u_t0) (1/s)**	**Description**
`vv`	`vvt-if(at_bottom,0,min(mf.Forcez_0+k0*(x0-disp),0)/mass)`	`0`	`0`	`Plunger vertical velocity`

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `velocity` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Velocity (m/s)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `acceleration` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Acceleration (m/s^2)**.

14 Click **OK**.

Global Equations 2

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt, t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
dd	vv-ddt	0	0	Plunger vertical displacement

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `displacement` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Displacement (m)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `velocity` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Velocity (m/s)**.

14 Click **OK**.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
at_bottom	dd<(-maxdisp)		Logical variable whose value is one when the plunger is at the bottom position

Name	Expression	Unit	Description
`disp`	`if(at_bottom, -maxdisp, dd)`	m	`Plunger displacement`
`vel`	`if(at_bottom, 0, vv)`	m/s	`Velocity of plunger`

The first study consists of a preprocessing step to solve **Coil Geometry Analysis** which is needed in order to compute the direction of the coil.

STUDY 1 (PREPROCESSING)

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

2 In the **Settings** window for **Study**, type `Study 1 (Preprocessing)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Coil Geometry Analysis

1 In the **Model Builder** window, under **Study 1 (Preprocessing)** click **Step 1: Coil Geometry Analysis**.

2 In the **Settings** window for **Coil Geometry Analysis**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Moving mesh (Component 1)**.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Preprocessing: Normalized Air Gap Parameterization and Coil Direction

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

2 Right-click **Results** and choose **3D Plot Group**.

3 In the **Settings** window for **3D Plot Group**, type `Preprocessing: Normalized Air Gap Parameterization and Coil Direction` in the **Label** text field.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Volume 1

1 Right-click **Preprocessing: Normalized Air Gap Parameterization and Coil Direction** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

Filter 1

1 Right-click **Volume 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type y<x.

4 In the **Preprocessing: Normalized Air Gap Parameterization and Coil Direction** toolbar, click **Plot**.

Streamline 1

1 In the **Model Builder** window, right-click **Preprocessing: Normalized Air Gap Parameterization and Coil Direction** and choose **Streamline**.

2 Select Boundary 97 only.

3 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Fields>Coil parameters>mf.coil1.eCoilx,...,mf.coil1.eCoilz - Coil direction (spatial frame)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

5 Find the **Point style** subsection. From the **Color** list, choose **Yellow**.

6 In the **Preprocessing: Normalized Air Gap Parameterization and Coil Direction** toolbar, click **Plot**.

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

ROOT

Now create the main study containing the **Time Dependent** step. Specify that the step must use the values computed in the preprocessing study.

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 **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 (TIME DEPENDENT)

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study**, type `Study 2 (Time Dependent)` in the **Label** text field.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 2 (Time Dependent)** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0,0.005,0.1) range(0.15,0.05,1)`.
4. Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.
6. From the **Study** list, choose **Study 1 (Preprocessing), Coil Geometry Analysis**.

Solution 2 (sol2)

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

   Apply typical solver settings for strongly nonlinear problems.
2. In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
3. In the **Model Builder** window, expand the **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Magnetic vector potential (spatial frame) (comp1.A)**.
4. In the **Settings** window for **Field**, locate the **Scaling** section.
5. From the **Method** list, choose **Manual**.
6. In the **Scale** text field, type `1e-3`.
7. In the **Model Builder** window, under **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Divergence condition variable (comp1.mf.psi)**.
8. In the **Settings** window for **Field**, locate the **Scaling** section.
9. From the **Method** list, choose **Manual**.
10. In the **Model Builder** window, under **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Coil current (comp1.mf.coil1.ICoil_ode)**.
11. In the **Settings** window for **State**, locate the **Scaling** section.
12. From the **Method** list, choose **Manual**.

13 In the **Model Builder** window, under **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Plunger vertical velocity (comp1.ODE1)**.

14 In the **Settings** window for **State**, locate the **Scaling** section.

15 From the **Method** list, choose **Manual**.

16 In the **Model Builder** window, expand the **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** node.

17 Right-click **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1>Direct** and choose **Enable**.

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

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

20 In the **Model Builder** window, under **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.

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

22 From the **Maximum BDF order** list, choose **2**.

23 Click to expand the **Absolute Tolerance** section. From the **Tolerance method** list, choose **Manual**.

24 In the **Absolute tolerance** text field, type `0.01`.

25 Right-click **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** and choose **Fully Coupled**.

26 In the **Settings** window for **Fully Coupled**, locate the **General** section.

27 From the **Linear solver** list, choose **Direct**.

28 Click to expand the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.

29 In the **Study** toolbar, click **= Compute**. The solution process will need about 50 minutes on a typical workstation.

RESULTS

Modify the previously generated plot to visualize the normalized displacement in the moving mesh regions. It is possible to verify that the material frame representation of this quantity does not depend significantly on the time after the moving anchor has moved away from its rest position.

Volume 1

1 In the **Model Builder** window, under **Results> Preprocessing: Normalized Air Gap Parameterization and Coil Direction** click **Volume 1**.

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

3 From the **Dataset** list, choose **Study 2 (Time Dependent)/Solution 2 (sol2)**.

4 From the **Time (s)** list, choose **0.05**.

5 Locate the **Expression** section. In the **Expression** text field, type `(z-Z)/disp`.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Deformed Domains**.

Multislice 1

1 In the **Model Builder** window, expand the **Results>Magnetic Flux Density Norm (mf)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Coordinates** text field, type `0`.

4 Find the **y-planes** subsection. In the **Coordinates** text field, type `0`.

Streamline Multislice 1

1 In the **Model Builder** window, click **Streamline Multislice 1**.

2 In the **Settings** window for **Streamline Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Coordinates** text field, type `0`.

4 Find the **y-planes** subsection. In the **Coordinates** text field, type `0`.

Magnetic Flux Density Norm

1 In the **Model Builder** window, click **Magnetic Flux Density Norm (mf)**.

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

The generated plot shows the magnetic fields on the symmetry planes at the last time step, when the gap is closed and the induction currents have decayed.

3 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

4 From the **Time (s)** list, choose **0.05**.

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

The generated plot shows the magnetic fields on the symmetry planes at a time when the gap was still open.

6 In the **Label** text field, type `Magnetic Flux Density Norm`.

7 Locate the **Title** section. From the **Title type** list, choose **Label**.

8 Locate the **Color Legend** section. Select the **Show units** check box.

9 In the **Model Builder** window, click **Magnetic Flux Density Norm**.

10 Locate the **Data** section. From the **Time (s)** list, choose **0.1**.

11 In the **Magnetic Flux Density Norm** toolbar, click **Plot**.

The generated plot shows the magnetic fields on the symmetry planes at a time step in which the gap is closed and induction currents are screening the interior of the core to the field.

A 2D section of the complete geometry can be produced with the following instructions.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2 (Time Dependent)/Solution 2 (sol2)**.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Out-of-Plane Current Density and Magnetic Flux Lines

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Out-of-Plane Current Density and Magnetic Flux Lines` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

6 Select the **Show units** check box.

Surface 1

1 Right-click **Out-of-Plane Current Density and Magnetic Flux Lines** and choose **Surface**.

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

3 In the **Expression** text field, type `mf.Jy`.

4 In the **Unit** field, type `A/mm^2`.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Minimum** text field, type `-2`.

7 In the **Maximum** text field, type `2`.

8 Locate the **Coloring and Style** section. Click **Change Color Table**.

9 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

10 Click **OK**.

11 In the **Out-of-Plane Current Density and Magnetic Flux Lines** toolbar, click **Plot**.

The plot shows the currents perpendicular to the *xz* -plane, simply mirrored. Use a side indicator variable to provide the correct sign for the currents.

Mirror 3D 1

1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D 1**.

2 In the **Settings** window for **Mirror 3D**, click to expand the **Advanced** section.

3 Select the **Define variables** check box.

Surface 1

1 In the **Model Builder** window, under **Results>Out-of-Plane Current Density and Magnetic Flux Lines** click **Surface 1**.

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

3 In the **Expression** text field, type `mf.Jy*sign(mir1x)`.

4 In the **Out-of-Plane Current Density and Magnetic Flux Lines** toolbar, click **Plot**.

Out-of-Plane Current Density and Magnetic Flux Lines

Vector quantities are automatically correct when inherited from a mirror dataset.

Streamline 1

1 In the **Model Builder** window, right-click **Out-of-Plane Current Density and Magnetic Flux Lines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Expression** section.

3 In the **y-component** text field, type `mf.Bz`.

4 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `range(-0.018,0.004,0.018)`.

6 In the **y** text field, type `0`.

7 Click to expand the **Advanced** section. In the **Loop tolerance** text field, type `0.1`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.
2 In the **Settings** window for **Color Expression**, click to expand the **Range** section.
3 Select the **Manual color range** check box.
4 In the **Maximum** text field, type `2`.

Out-of-Plane Current Density and Magnetic Flux Lines

1 In the **Model Builder** window, under **Results** click **Out-of-Plane Current Density and Magnetic Flux Lines**.
2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.
3 From the **Position** list, choose **Alternating**.
4 Locate the **Data** section. From the **Time (s)** list, choose **0.01**.
5 In the **Out-of-Plane Current Density and Magnetic Flux Lines** toolbar, click **Plot**.

   The plot shows the field and geometry configuration at a time instant in which the gap is still open. Use the **Time** list box to visualize the solution at different time instants.

Create a plot showing the losses in the core at different times, in the same 3D visualization.

Core Losses

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Core Losses` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 (Time Dependent)/ Solution 2 (sol2)**.
4 Locate the **Title** section. From the **Title type** list, choose **Label**.
5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.
6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.
7 Select the **Show units** check box.

Streamline 1

1 Right-click **Core Losses** and choose **Streamline**.
2 In the **Settings** window for **Streamline**, locate the **Expression** section.
3 In the **x-component** text field, type `mf.Jx`.
4 In the **y-component** text field, type `mf.Jy`.
5 In the **z-component** text field, type `mf.Jz`.
6 Select Boundary 97 only.

Volume 1

1 In the **Model Builder** window, right-click **Core Losses** and choose **Volume**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2 (Time Dependent)/Solution 2 (sol2)**.
4 From the **Time (s)** list, choose **0.05**.
5 Locate the **Expression** section. In the **Expression** text field, type `mf.Qrh`.
6 In the **Unit** field, type `W/dm^3`.
7 Locate the **Coloring and Style** section. Click **Change Color Table**.
8 In the **Color Table** dialog box, select **Thermal>GrayBody** in the tree.
9 Click **OK**.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Nonlinear Core (Solid Parts 1)**.

Volume 2

1 In the **Model Builder** window, under **Results>Core Losses** right-click **Volume 1** and choose **Duplicate**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Time (s)** list, choose **0.1**.

Translation 1

1 Right-click **Volume 2** and choose **Translation**.
2 In the **Settings** window for **Translation**, locate the **Translation** section.
3 In the **x** text field, type `-0.1`.

Volume 2

1 In the **Model Builder** window, click **Volume 2**.
2 In the **Settings** window for **Volume**, click to expand the **Inherit Style** section.
3 From the **Plot** list, choose **Volume 1**.

Volume 3

1 Right-click **Results>Core Losses>Volume 2** and choose **Duplicate**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Time (s)** list, choose **0.2**.

Translation 1

1 In the **Model Builder** window, expand the **Volume 3** node, then click **Translation 1**.
2 In the **Settings** window for **Translation**, locate the **Translation** section.
3 In the **x** text field, type `-0.2`.

Annotation 1

1 In the **Model Builder** window, right-click **Core Losses** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type `50 [ms]`.
4 Locate the **Position** section. In the **x** text field, type `0.03`.
5 In the **z** text field, type `0.07`.
6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Annotation 2

1 Right-click **Core Losses** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type `100 [ms]`.
4 Locate the **Position** section. In the **x** text field, type `-0.07`.
5 In the **z** text field, type `0.07`.
6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Annotation 3

1 Right-click **Core Losses** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type `200 [ms]`.
4 Locate the **Position** section. In the **x** text field, type `-0.17`.
5 In the **z** text field, type `0.07`.
6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Then generate the plot and rotate the view with the mouse to visualize the solution.

Create a first 1D plot to visualize the dynamics at the beginning of the process, when the spring is not yet compressed. Plot the normalized currents, gap size, and an exponential fit for the RL current dynamics in a charging inductor.

Dynamics of the System Before Switching

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 **Label**.

4 In the **Label** text field, type `Dynamics of the System Before Switching`.

Global 1

1 Right-click **Dynamics of the System Before Switching** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 2 (Time Dependent)/Solution 2 (sol2)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(0,0.005,0.05)`.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
mf.ICoil_1*mf.RCoil_1/ mf.VCoil_1	1	Current normalized to DC current
1+disp/maxdisp	1	Normalized gap size
1-exp(-t/50[ms])		Ideal RL normalized current

7 Click to expand the **Coloring and Style** section. In the **Dynamics of the System Before Switching** toolbar, click **Plot**.

Dynamics of the System Before Switching

From the first 1D plot, create a second one is to visualize the dynamics of the spring compression. Plot normalized current, gap size, and mechanical power of the moving plunger. Mechanical power is directly linked to the change in inductance, in turn forcing the current to decrease as the gap closes.

Dynamics of the System During Switching

1 In the **Model Builder** window, right-click **Dynamics of the System Before Switching** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Dynamics of the System During Switching` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Dynamics of the System During Switching** node, then click **Global 1**.

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

3 In the **Times (s)** text field, type `range(0,0.005,0.1)`.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.ICoil_1*mf.RCoil_1/ mf.VCoil_1`	1	Current normalized to DC current
`1+disp/maxdisp`	1	Normalized gap size
`4*mass*d(vel,t)*vel/at(1, mf.ICoil_1*mf.VCoil_1)`	1	Mechanical power, normalized to DC coil power

5 In the **Dynamics of the System During Switching** toolbar, click **Plot**.

Dynamics of the System During Switching

Create a third 1D plot to visualize the dynamics of the current after the spring has been completely compressed. Plot the normalized current, gap size, and the induction losses in the core. After the gap is closed, the current will start increasing again as expected in a RL circuit. The curve deviates slightly from the expected exponential behavior because of the induction currents and the nonlinearity of the iron core.

Dynamics of the Completed System

1 In the **Model Builder** window, right-click **Dynamics of the System During Switching** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Dynamics of the Completed System` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type .

Global 1

1 In the **Model Builder** window, expand the **Dynamics of the Completed System** node, then click **Global 1**.

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

3 From the **Time selection** list, choose **All**.

4 In the table, select the third row then click the **Delete** button below the table.

Integral 1

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Integral**.

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

3 From the **Dataset** list, choose **Study 2 (Time Dependent)/Solution 2 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Coil (Solid Parts 1)**.

Global 2

1 In the **Model Builder** window, right-click **Dynamics of the Completed System** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
4*mf.Qrh/at(1,mf.ICoil_1*mf.VCoil_1)	1	

4 Locate the **Data** section. From the **Dataset** list, choose **Integral 1**.
5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
4*mf.Qrh/at(1,mf.ICoil_1* mf.VCoil_1)	1	Induction current losses, normalized to DC coil power

6 In the **Dynamics of the Completed System** toolbar, click **Plot**.