



# Electrodynamics of a Power Switch — Multibody Version

## Introduction

---

Events such as overcurrents or overloads 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 one class of circuit breakers: *magnetic power switches*, 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 the rigid body dynamics under the influence of magnetic forces, induced currents, and spring/constraint arrangements that keep the plunger in its equilibrium position.

---

**Note:** This model requires the AC/DC Module and the Multibody Dynamics Module.

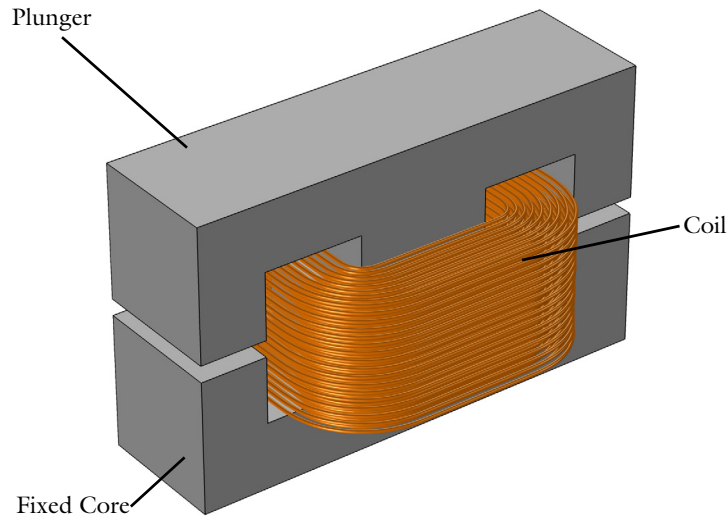
---

## 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,

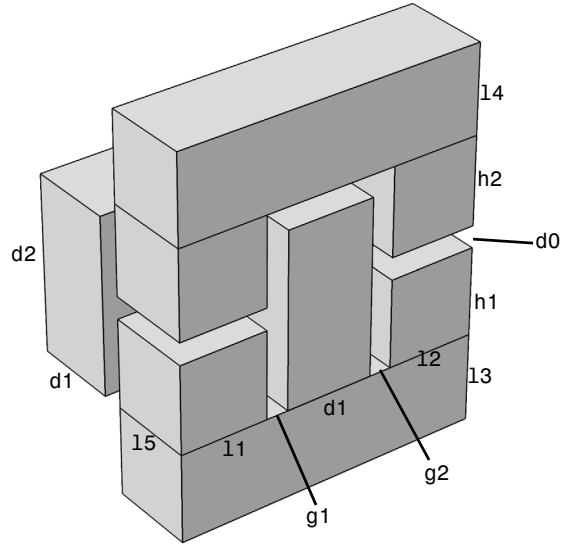
closing the air gap. The model illustrates how to properly simulate the movement and the closing time, which depends on the spring stiffness.



*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 represent 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 and Multibody Dynamics interfaces coupled to the Moving Mesh, 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. Plunger dynamics is described using Rigid Domain node in the Multibody Dynamics interface. The attractive force acting on the moving domain is modeled as an applied force on the rigid plunger with a given mass.

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 complete collapse of the mesh, displacement is limited to be at most 90% of the initial air gap, with a marginal impact on the results. This model is set up in order to consider an elastic but highly damped collision between the plunger and fixed core. It is achieved using the locking functionality available in the **Joint** node in the Multibody Dynamics interface. An 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.

### *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.

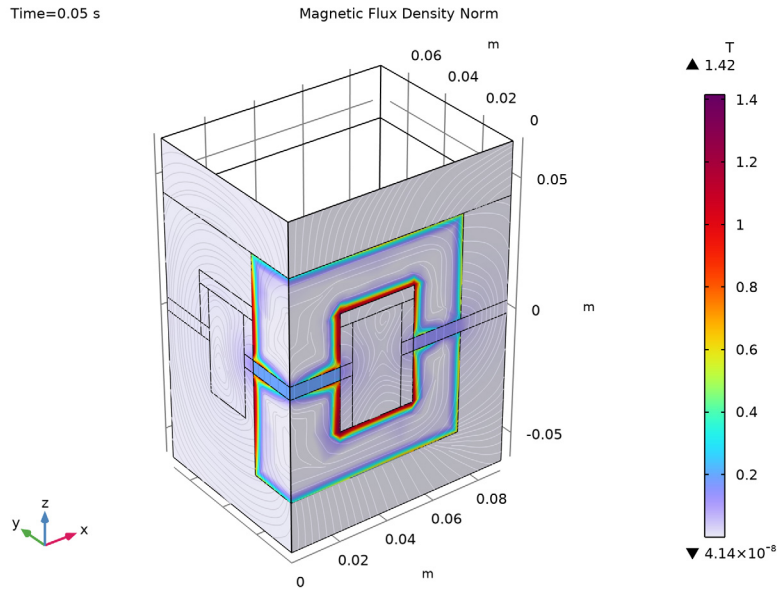


Figure 3: Magnetic flux density norm at  $t = 0.05$  s, when the gap is open.

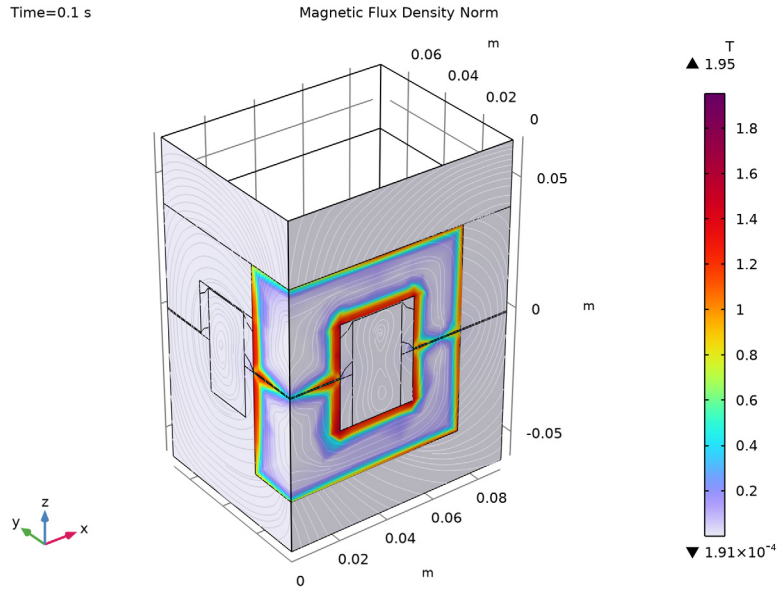


Figure 4: Magnetic flux density norm at  $t = 0.1$  s, when the gap is closed.

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 (upper 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).

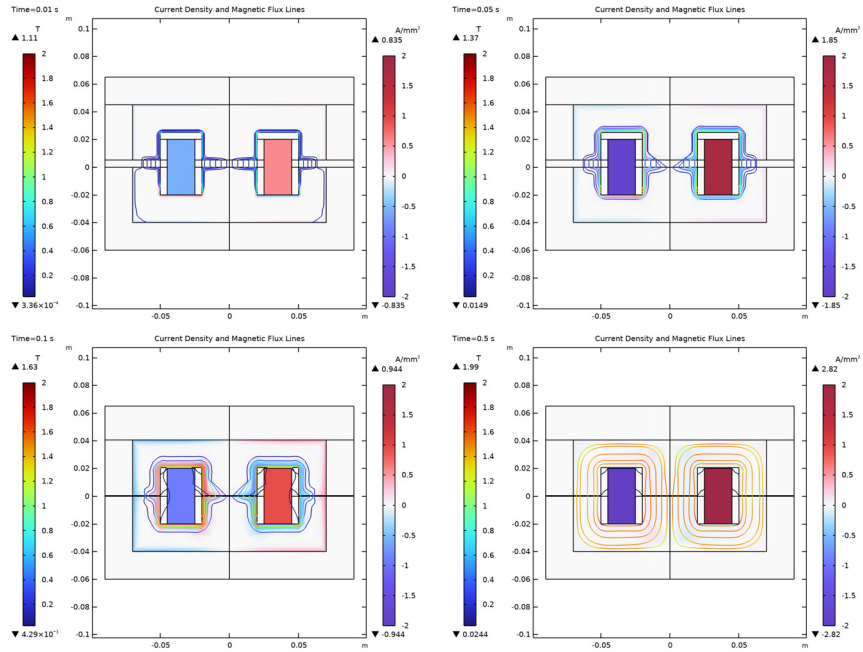


Figure 5: Evolution of current density (surface) and magnetic flux density (streamlines) at different times.



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

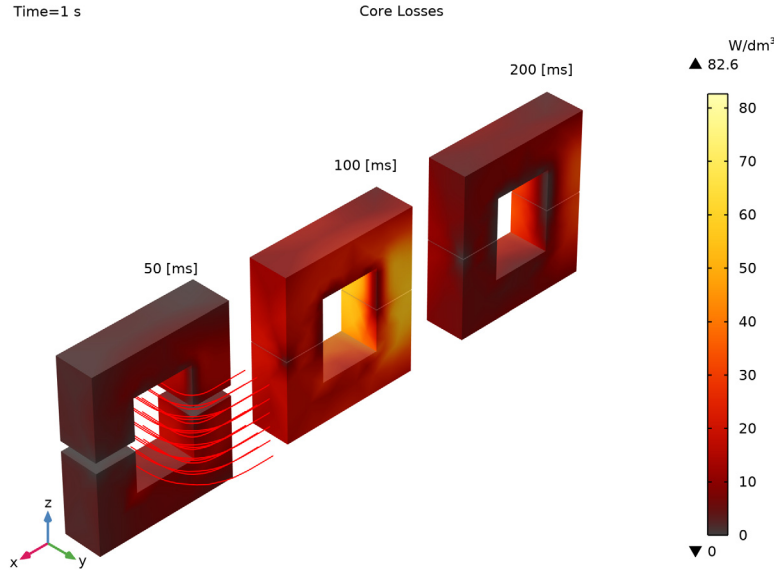
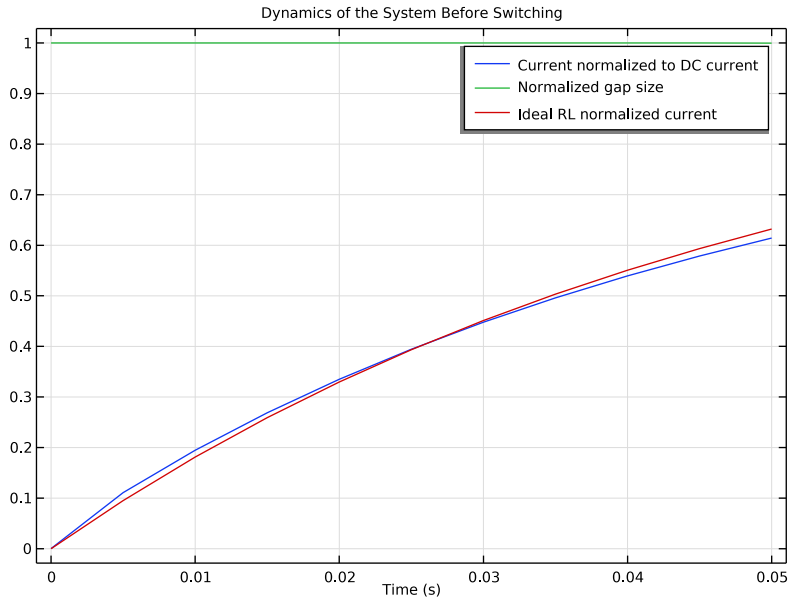


Figure 6: Core losses due to induced currents at different time instants.

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. Blue and green lines represent normalized currents and gap size respectively. 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. Normalized currents and gap size are represented by blue and green lines respectively, the red line showing instead the mechanical power (which is nonzero only during the motion of the plunger).
- *After plunger motion:* Figure 9 refers to the last stage of 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 is completed, the current starts increasing again as expected in a (nonlinear) RL circuit.



*Figure 7: Magnetic flux density norm at  $t = 0.1$  s, when the gap is closed.*

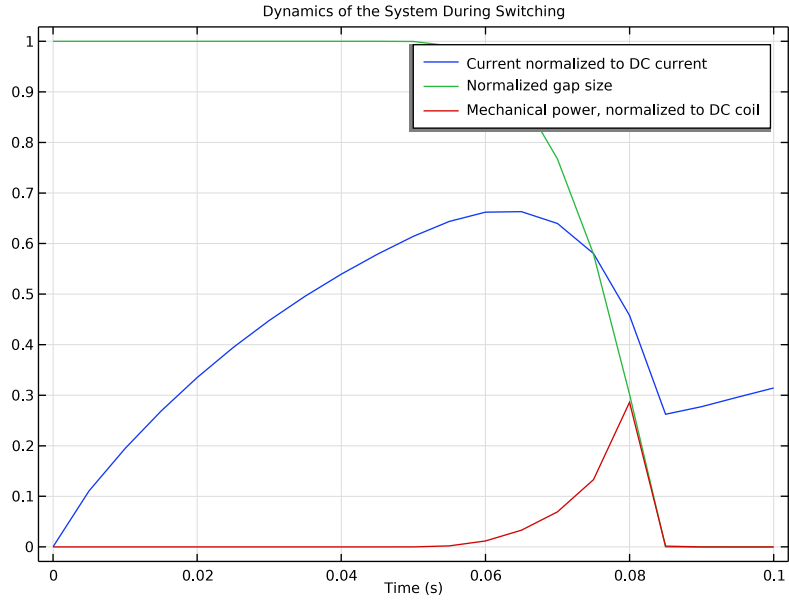
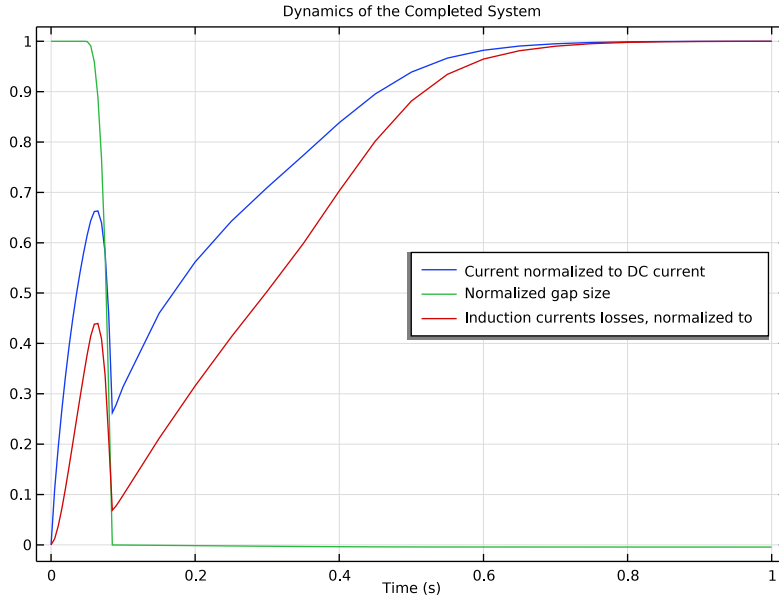


Figure 8: Magnetic flux density norm at  $t = 0.1$  s, when the gap is closed.



*Figure 9: Magnetic flux density norm at  $t = 0.1$  s, when the gap is closed.*

It is worth recalling that blue and green curves in [Figure 7](#), [Figure 8](#), and [Figure 9](#) are representing 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). [Figure 10](#) shows the variation of locking force with time.

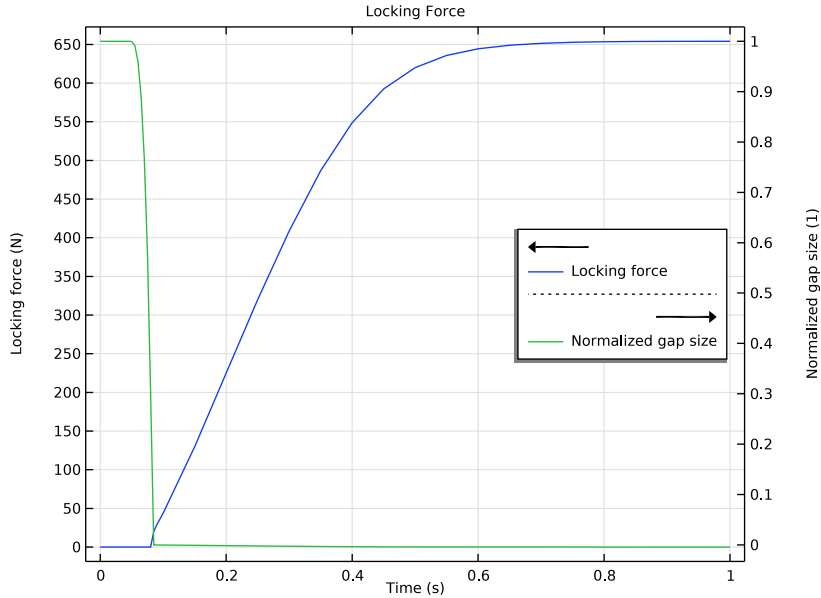


Figure 10: Variation of locking force and gap with time.

---

**Application Library path:** Multibody\_Dynamics\_Module/Electrical\_Machinery/power\_switch\_multibody


---

### 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 In the **Select Physics** tree, select **Structural Mechanics>Multibody Dynamics (mbd)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Coil Geometry Analysis**.
- 8 Click  **Done**.

## GLOBAL DEFINITIONS

### *Parameters* /

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_multibody_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 into 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  $11+g1+d1+g2+12$ .
- 4 In the **Height** text field, type  $h1+13$ .
- 5 Locate the **Position** section. In the **y** text field, type  $-(h1+13)$ .




### Rectangle 2 (r2)

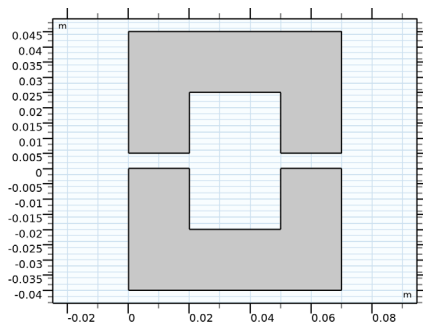
- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type  $g1+d1+g2$ .
- 4 In the **Height** text field, type  $h1$ .
- 5 Locate the **Position** section. In the **x** text field, 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 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 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  $11+g1$ .

#### *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  $11$ .
- 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)*

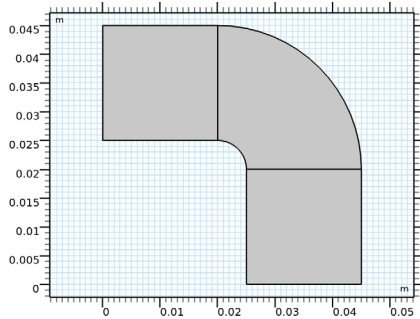
- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type  $11$ .
- 4 In the **Height** text field, type  $d1$ .
- 5 Locate the **Position** section. In the **y** text field, type  $15+g1$ .

#### *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  **Part Instance** 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  **Part Instance** and choose **Coil Section**.

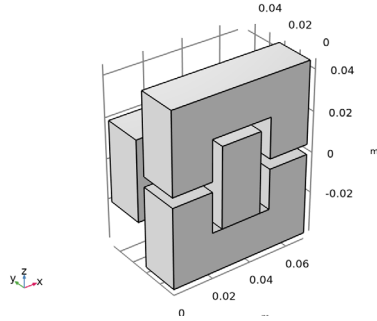
#### *Coil*

- 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 I


### Solid Parts I (piI)

- 1 In the **Geometry** toolbar, click  **Part Instance** 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

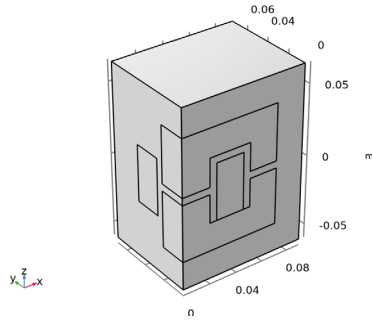
- 4 Click  **Build All Objects**.

### Block I (blkI)



- 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].

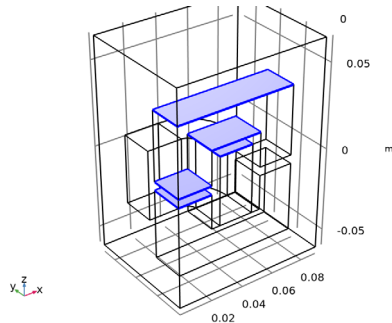
### Union 1 (un1)

- 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 **un1**, 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 **un1**, select Boundaries 7, 10, 11, 24, and 33 only.



### Work Plane 1 (wpl1)


- 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 Click to select the  **Activate Selection** toggle button for **Offset vertex**.
- 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 Click to select the  **Activate Selection** toggle button for **Vertex for origin**.
- 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)*

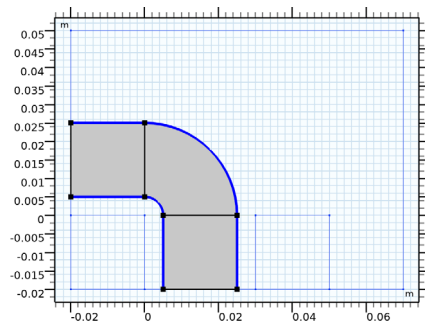
In the **Work Plane** toolbar, click  **Cross Section**.

*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.
- 5 Click  **Build Selected**.


*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.



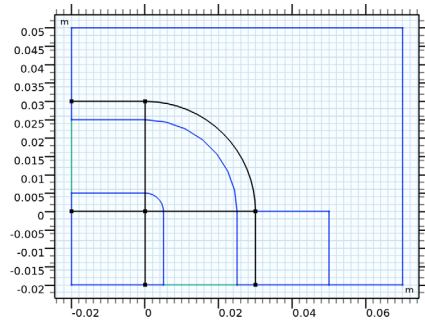
- 5 Clear the **Keep input objects** check box.
- 6 Locate the **Options** section. In the **Distance** text field, type d0.

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 **part1**, 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**.

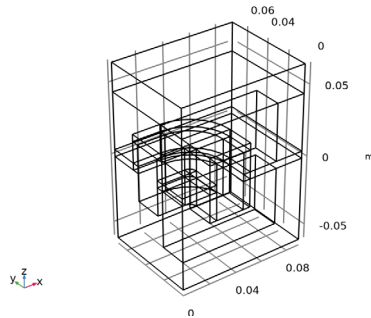
Cylinder Selection 1 (cylsel1)

- 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  $d0+h2+14$ .
- 4 In the **Bottom distance** text field, type  $d0$ .
- 5 Locate the **Position** section. In the **y** text field, type  $15+g1+d1+g3+2$  [cm].

#### *Form Composite Domains I (cmd I)*



- 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 I**.
- 4 Click  **Build Selected**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.



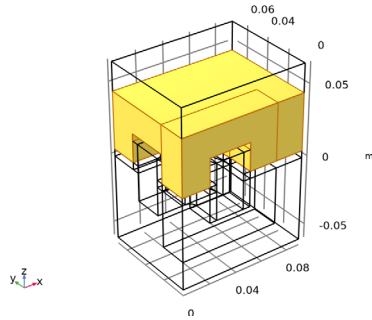
Proceed to create a number of selections in order to have a robust geometric parameterization.

## **DEFINITIONS**


### *Plunger*

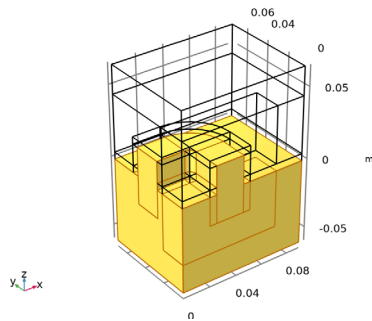
- 1 In the **Definitions** toolbar, click  **Union**.
- 2 In the **Settings** window for **Union**, type **Plunger** in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Selections to add**, click  **Add**.
- 4 In the **Add** dialog box, in the **Selections to add** list, choose **Upper Core (Solid Parts I)** and **Cylinder Selection I**.

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.

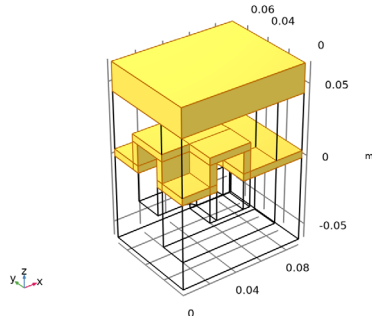


### Deformed Domains


- 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**.

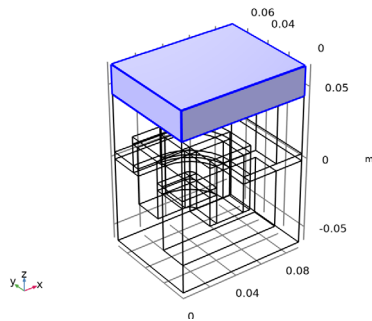


5 Click **OK**.






### *Top Domain*

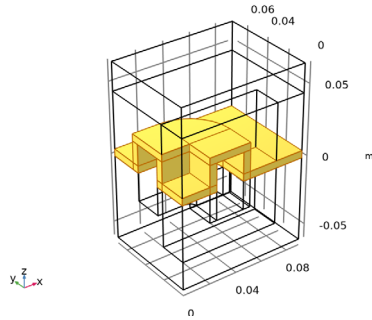
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Top Domain** in the **Label** text field.
- 3 Select Domain 5 only.





### *Domain in between*

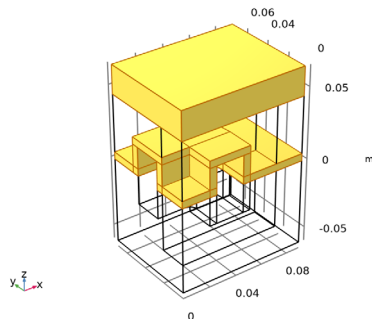
- 1 In the **Definitions** toolbar, click  **Difference**.
- 2 In the **Settings** window for **Difference**, type **Domain in between** in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Selections to add**, click  **Add**.
- 4 In the **Add** dialog box, select **Deformed 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*

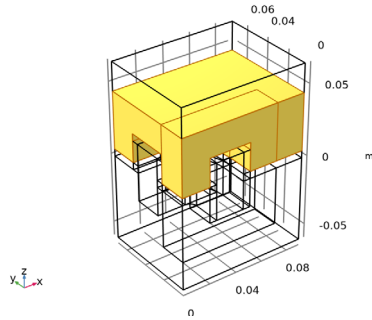
- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type Ext. boundaries to Deformed Domains in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Input selections**, click  **Add**.
- 4 In the **Add** dialog box, select **Deformed Domains** in the **Input selections** list.
- 5 Click **OK**.





*Ext. Boundaries to Plunger*

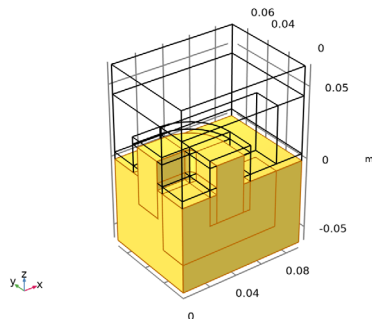
- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type Ext. Boundaries to Plunger in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Input selections**, click  **Add**.
- 4 In the **Add** dialog box, select **Plunger** in the **Input selections** list.

5 Click **OK**.





#### *Ext. Boundaries to Fixed Domains*

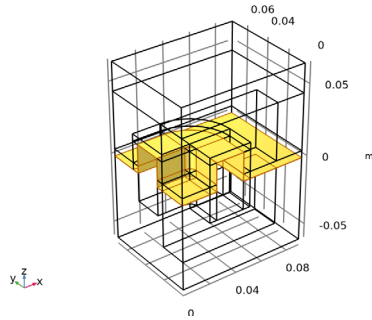
- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type **Ext. Boundaries to Fixed Domains** in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Input selections**, click  **Add**.
- 4 In the **Add** dialog box, select **Fixed Domains** in the **Input selections** list.
- 5 Click **OK**.




#### *Fixed Boundaries at Plunger*

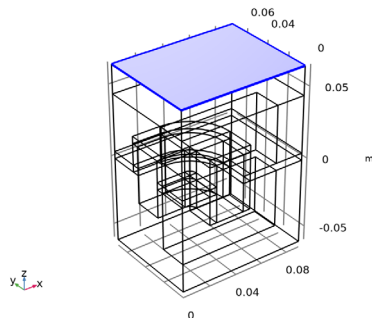
- 1 In the **Definitions** toolbar, click  **Intersection**.
- 2 In the **Settings** window for **Intersection**, type **Fixed Boundaries at Plunger** in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the **Input Entities** 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*

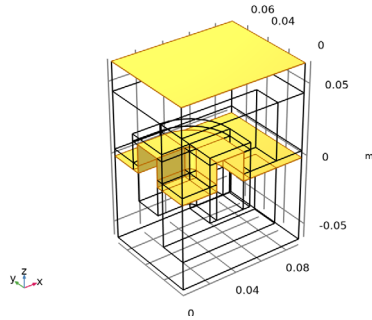
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Top boundary** in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select **Boundary 16** only.





#### *Fixed Boundaries*

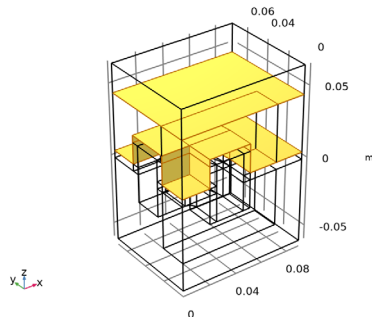
- 1 In the **Definitions** toolbar, click  **Union**.
- 2 In the **Settings** window for **Union**, type **Fixed Boundaries** in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the **Input Entities** section. Under **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*

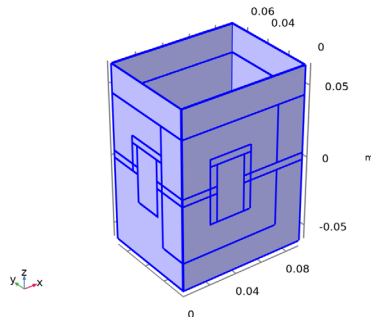
- 1 In the **Definitions** toolbar, click  **Intersection**.
- 2 In the **Settings** window for **Intersection**, type Moving Boundaries in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the **Input Entities** section. 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*

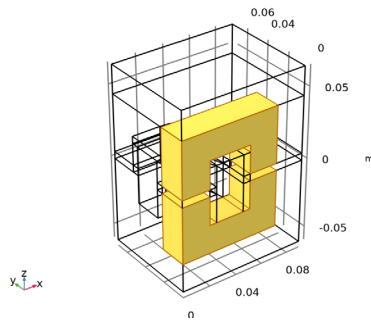
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Side boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select the **Group by continuous tangent** 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*

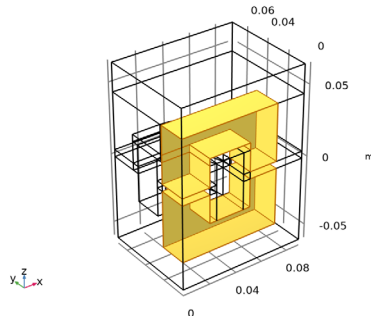
- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type *Ext. Boundaries to cores* in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Input selections**, click  **Add**.
- 4 In the **Add** dialog box, select **Nonlinear 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**.



## ADD MATERIAL

- 1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.  
Next, add the materials: copper for the coil material and soft iron for the core.
- 3 In the tree, select **AC/DC>Copper**.
- 4 Click **Add to Component** in the window toolbar.
- 5 In the tree, select **AC/DC>Soft Iron (With Losses)**.
- 6 Right-click and choose **Add to Component 1 (comp1)**.
- 7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

### *Coil Material*

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

### *Soft Iron (With Losses) (mat2)*


- 1 In the **Model Builder** window, click **Soft Iron (With Losses) (mat2)**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Nonlinear Core (Solid Parts 1)**.

## MESH 1

Move now to the mesh setup, using the **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**.

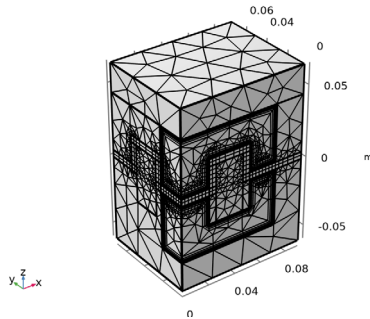


### *Boundary Layers 1*

- 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 **Physics** toolbar, click  **Moving Mesh** and choose **Free Deformation**.
- 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 Normal Mesh Displacement 1*

- 1 In the **Moving Mesh** toolbar, click  **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 **Moving Mesh** toolbar, click  **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**.
- 4 Right-click **Prescribed Mesh Displacement 1** and choose **Duplicate**.

#### *Prescribed Mesh Displacement 2*

- 1 In the **Model Builder** window, click **Prescribed Mesh Displacement 2**.
- 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 by setting 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  $V_{\text{coil}}$  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  $a_{\text{wire}}$  text field, type a\_coil.

#### *Geometry Analysis I*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1>Geometry Analysis 1** node, then 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 I*

- 1 In the **Model Builder** window, click **Input 1**.
- 2 Select Boundary 97 only.


#### *Geometry Analysis I*

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


#### *Output I*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Output**.
- 2 Select Boundary 31 only.

#### *Ampère's Law in Solids I*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Ampère's Law in Solids**.
- 2 In the **Settings** window for **Ampère's Law in Solids**, 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 I*

- 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 I*

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


**MULTIBODY DYNAMICS (MBD)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.
- 2 In the **Settings** window for **Multibody Dynamics**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Plunger**.

*Rigid Material 1*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Rigid Material**.
- 2 In the **Settings** window for **Rigid Material**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Plunger**.
- 4 Locate the **Density** section. From the  $\rho$  list, choose **User defined**.

*Mass and Moment of Inertia 1*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Mass and Moment of Inertia**.
- 2 In the **Settings** window for **Mass and Moment of Inertia**, locate the **Mass and Moment of Inertia** section.
- 3 In the  $m$  text field, type mass.

*Rigid Material 1*


In the **Model Builder** window, click **Rigid Material 1**.

*Applied Force 1*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Applied Force**.
- 2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.
- 3 Specify the **F** vector as

0	x
0	y
$\min(\text{mf.Forcez}_0 + k_0 * (x_0 - \text{disp}), 0)$	z


*Prismatic Joint 1*

- 1 In the **Physics** toolbar, click  **Global** and choose **Prismatic Joint**.
- 2 In the **Settings** window for **Prismatic Joint**, locate the **Attachment Selection** section.
- 3 From the **Source** list, choose **Fixed**.
- 4 From the **Destination** list, choose **Rigid Material 1**.
- 5 Locate the **Center of Joint** section. From the list, choose **User defined**.

6 Locate the **Axis of Joint** section. Specify the  $\mathbf{e}_0$  vector as

0	x
0	y
1	z

#### Locking I

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Locking**.
- 2 In the **Settings** window for **Locking**, locate the **Translational Locking** section.
- 3 In the  $u_{\min}$  text field, type `-maxdisp`.
- 4 From the **Locking parameters** list, choose **User defined**.
- 5 In the  $p_{u1}$  text field, type `(0.01*mbd.prj1.lk1.Eequ)*mbd.diag/10`.
- 6 In the  $p_{u2}$  text field, type `mbd.prj1.lk1.p_u1*10[ms]/50`.

## DEFINITIONS

#### Variables I

- 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
disp	<code>mbd.rd1.w</code>	m	Plunger displacement
vel	<code>d(mbd.rd1.w,t)</code>	m/s	Velocity of plunger


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

## STUDY I (PREPROCESSING)

- 1 In the **Model Builder** window, click **Study I**.
- 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 I (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 boxes for **Multibody Dynamics (mbd)** and **Moving mesh (Component I)**.
- 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 I*

- 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 I*

- 1 Right-click **Volume I** 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 I*

- 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 I (compI)>**



**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, expand the **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** node.
- 14 Right-click **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1>Direct** and choose **Enable**.
- 15 In the **Settings** window for **Direct**, locate the **General** section.
- 16 From the **Solver** list, choose **PARDISO**.
- 17 In the **Model Builder** window, under **Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.
- 18 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 19 Click to expand the **Absolute Tolerance** section. From the **Tolerance method** list, choose **Manual**.
- 20 In the **Absolute tolerance** text field, type  $0.01$ .



- 21 Right-click **Study 2 (Time Dependent)**>**Solver Configurations**>**Solution 2 (sol2)**>**Time-Dependent Solver 1** and choose **Fully Coupled**.
- 22 In the **Settings** window for **Fully Coupled**, locate the **General** section.
- 23 From the **Linear solver** list, choose **Direct**.
- 24 Click to expand the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.
- 25 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  $(z-Z)/\text{disp}$ .

### *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 I*

- 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 I*

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

### *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 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 I*

- 1 Right-click **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 **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 I*

- 1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D I**.
- 2 In the **Settings** window for **Mirror 3D**, click to expand the **Advanced** section.
- 3 Select the **Define variables** check box.

### *Surface I*

- 1 In the **Model Builder** window, under **Results>Current Density and Magnetic Flux Lines** click **Surface I**.
- 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 **Current Density and Magnetic Flux Lines** toolbar, click  **Plot**.

#### *Current Density and Magnetic Flux Lines*

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


#### *Streamline I*

- 1 In the **Model Builder** window, right-click **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  $\text{mf.Bz}$ .
- 4 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 5 In the **x** text field, type  $\text{range}(-0.018, 0.004, 0.018)$ .
- 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 I*

- 1 Right-click **Streamline I** 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.


#### *Current Density and Magnetic Flux Lines*

- 1 In the **Model Builder** window, under **Results** click **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 **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 I*

- 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  $m_f.Jx$ .
- 4 In the **y-component** text field, type  $m_f.Jy$ .
- 5 In the **z-component** text field, type  $m_f.Jz$ .
- 6 Select Boundary 97 only.

#### *Volume I*

- 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  $m_f.Qrh$ .
- 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 I*

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

#### *Volume I*

In the **Model Builder** window, right-click **Volume I** and choose **Duplicate**.

### Volume 2

- 1 In the **Model Builder** window, click **Volume 2**.
- 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**.
- 4 Right-click **Volume 2** and choose **Duplicate**.

### Volume 3

- 1 In the **Model Builder** window, click **Volume 3**.
- 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**.

### Core Losses

In the **Model Builder** window, under **Results** click **Core Losses**.


### Table Annotation 1

- 1 In the **Core Losses** toolbar, click  **More Plots** and choose **Table Annotation**.
- 2 In the **Settings** window for **Table Annotation**, locate the **Data** section.
- 3 From the **Source** list, choose **Local table**.
- 4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
0.03	0	0.07	50 [ms]

x-coordinate	y-coordinate	z-coordinate	Annotation
-0.07	0	0.07	100 [ms]
-0.17	0	0.07	200 [ms]


5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 In the **Core Losses** toolbar, click  **Plot**.

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 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
$\text{mf.ICoil}_1 \cdot \text{mf.RCoil}_1 / \text{mf.VCoil}_1$	1	Current normalized to DC current
$1 + \text{disp}/\text{maxdisp}$	1	Normalized gap size
$1 - \exp(-t/50[\text{ms}])$		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.

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

*Dynamics of the System During Switching*

- 1 In the **Model Builder** window, under **Results** click **Dynamics of the System Before Switching 1**.
- 2 In the **Settings** window for **ID 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 ID 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.

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

*Dynamics of the Completed System*

- 1 In the **Model Builder** window, under **Results** click **Dynamics of the System During Switching 1**.




- 2 In the **Settings** window for **ID 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. Clear the associated text field.


#### 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
$4 \cdot m_f \cdot Q_{rh} / at(1, m_f \cdot I_{Coil\_1} \cdot m_f \cdot V_{Coil\_1})$	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
$4 * m f . Q r h / a t ( 1 , m f . I C o i l _ { 1 } * m f . V C o i l _ { 1 } )$	1	Induction currents losses, normalized to DC coil power

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


#### *Dynamics of the Completed System*

Right-click **Dynamics of the Completed System** and choose **Duplicate**.

#### *Locking Force*


- 1 In the **Model Builder** window, under **Results** click **Dynamics of the Completed System 1**.
- 2 In the **Settings** window for **ID Plot Group**, type Locking Force 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 **Plot Settings** section. Clear the **y-axis label** check box.
- 5 Select the **Two y-axes** check box.
- 6 In the table, select the **Plot on secondary y-axis** check box for **Global 2**.

#### *Global 1*

- 1 In the **Model Builder** window, expand the **Locking Force** node, then click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **From parent**.
- 4 Locate the **y-Axis Data** section. Click  **Clear Table**.
- 5 In the table, enter the following settings:


Expression	Unit	Description
$m b d . p r j 1 . l k 1 . F _ { u }$	N	Locking force

#### *Global 2*

- 1 In the **Model Builder** window, click **Global 2**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **From parent**.
- 4 Locate the **y-Axis Data** section. Click  **Clear Table**.

5 In the table, enter the following settings:

Expression	Unit	Description
1+disp/maxdisp	1	Normalized gap size

6 In the **Locking Force** toolbar, click  **Plot**.

#### *Displacement Magnitude*

- 1 In the **Model Builder** window, under **Results** click **Displacement (mbd)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Title** section.
- 3 From the **Title type** list, choose **Label**.
- 4 In the **Label** text field, type Displacement Magnitude.
- 5 Locate the **Color Legend** section. Select the **Show units** check box.
- 6 Select the **Show maximum and minimum values** check box.

#### *Velocity*

- 1 In the **Model Builder** window, click **Velocity (mbd)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Title** section.
- 3 From the **Title type** list, choose **Label**.
- 4 In the **Label** text field, type Velocity.

