

Electrodynamics of a Power Switch — Multibody Version

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 non-moving 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: ACDC_Module/Electromagnetics_and_Mechanics/ power_switch_multibody

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 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 \bigcirc Study.
- 7 In the Select Study tree, select Preset Studies for Some Physics Interfaces> Coil Geometry Analysis.
- 8 Click **M** Done.

GLOBAL DEFINITIONS

Parameters 1

Import the geometric and physical parameters from an external file.

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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

- I 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)

- I 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)

- I 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 11.
- 6 In the y text field, type -h1.

Difference I (dif1)

- I In the Geometry toolbar, click 📃 Booleans and Partitions and choose Difference.
- 2 Select the object rI only.
- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Click to select the **Calculate Selection** toggle button.
- 5 Select the object r2 only.

Mirror I (mir I)

- I 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 difl only.
- 7 Locate the Input section. Select the Keep input objects check box.
- 8 Click 틤 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

- I 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)

I 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 I (c1)

- I In the **Geometry** toolbar, click \bigcirc **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)

- I 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 I (dell)

- I 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 cl, select Domain 1 only.
- 5 Click 틤 Build Selected.
- 6 Click the 4 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

- I 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 I (wp1)

- I 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 I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

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

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

Nonlinear Core

- I In the Model Builder window, under Global Definitions>Geometry Parts>Solid Parts rightclick Work Plane I (wpl) 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

- I 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 extl, select Domain 2 only.

Work Plane 2 (wp2)

I 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 (pil)

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

Coil

- I In the Model Builder window, under Global Definitions>Geometry Parts>Solid Parts rightclick 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 threedimensional 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 1 (pil)

- I In the Geometry toolbar, click \land 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	Кеер	Physics	Contribute to
Nonlinear Core	\checkmark	\checkmark	None
Upper Core	\checkmark	\checkmark	None
Coil	\checkmark	\checkmark	None

4 Click 🟢 Build All Objects.

Block I (blk1)

- I 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 11+g1+d1+g2+12+2[cm].
- 4 In the **Depth** text field, type 15+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 -h1-13-2[cm].

Union I (uniI)

- I In the Geometry toolbar, click Pooleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select all objects.

Partition Domains I (pard I)

- I In the Geometry toolbar, click booleans and Partitions and choose Partition Domains.
- **2** On the object **unil**, 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 unil, select Boundaries 7, 10, 11, 24, and 33 only.

Work Plane I (wp1)

- I 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 **CALC** Activate Selection toggle button.
- 5 On the object pard I, 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 Del Activate Selection toggle button.
- 8 On the object pard I, 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 \bigcirc Cross Section.

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

- I 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 crol, select Domains 1, 2, 4, and 7 only.
- 5 Click 틤 Build Selected.

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

- I 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 **dell**, 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 I (wp1)>Polygon I (pol1)

- I 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 틤 Build Selected.

Extrude I (extI)

- I In the Model Builder window, right-click Geometry I 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 pard I, select Point 11 only.
- 6 Click 틤 Build Selected.

Form Union (fin)

- I 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)

- I 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 1 (cmd1)

- I 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 틤 Build Selected.

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

DEFINITIONS

Plunger

- I In the **Definitions** toolbar, click **H 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

- I In the **Definitions** toolbar, click **The 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

- I 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

- I In the **Definitions** toolbar, click http://www.explicit.
- 2 In the Settings window for Explicit, type Top Domain in the Label text field.
- **3** Select Domain 5 only.

Domain in between

- I 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

- I 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

- I 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

- I 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

- I 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

- I 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

- I 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

- I 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

- I In the **Definitions** toolbar, click **herefore 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.
- 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

- I 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 I) in the Input selections list.
- 5 Click OK.

Boundary layer mesh

I 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.

II Click OK.

MATERIALS

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

Air

- I In the Model Builder window, under Component I (compl) 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	I	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	I	Basic

Coil Material

I 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	I	Basic

ADD MATERIAL

- I 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 I (compl)**.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Soft Iron (With Losses) (mat3)

- I In the Settings window for Material, locate the Geometric Entity Selection section.
- 2 From the Selection list, choose Nonlinear Core (Solid Parts I).

MESH I

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

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- 3 From the Element size list, choose Coarser.

Swept I

- I In the Mesh toolbar, click A 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

- I Right-click Swept I 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 I

- I In the Model Builder window, right-click Swept I 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

- I Right-click Swept I and choose Distribution.
- 2 Select Domains 7, 12, 15, 18, 22, and 25 only.

Free Tetrahedral I

In the Mesh toolbar, click \land Free Tetrahedral.

Boundary Layers 1

- I In the Mesh toolbar, click Moundary 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 I).

Boundary Layer Properties

- I 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 I and choose Build All.

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

COMPONENT I (COMPI)

Deforming Domain 1

- I 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 Normal Mesh Displacement I

- I 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 I

- I 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

I Right-click Prescribed Mesh Displacement I and choose Duplicate.

- **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	Х
0	Y
disp	Z

Continue by setting up the physics of the model.

MAGNETIC FIELDS (MF)

- I In the Model Builder window, under Component I (compl) 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 I

- I 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 I).
- 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_{coil} text field, type a_coil.

Geometry Analysis I

- In the Model Builder window, expand the Component I (compl)>Magnetic Fields (mf)>
 Coil I>Geometry Analysis I node, then click Geometry Analysis I.
- 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

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

Geometry Analysis I

In the Model Builder window, click Geometry Analysis I.

Output I

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

Ampère's Law 2

- I 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 I).

Force Calculation 1

- I 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 I).

Gauge Fixing for A-field 1

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

MULTIBODY DYNAMICS (MBD)

- I In the Model Builder window, under Component I (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 I

- I 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 ρ list, choose **User defined**.

Mass and Moment of Inertia 1

- I 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 I

In the Model Builder window, click Rigid Material I.

Applied Force 1

- I 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	у
<pre>min(mf.Forcez_0+k0*(x0-disp),0)</pre>	z

Prismatic Joint 1

- I 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 I.
- 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 у

1 z

Locking I

- I In the **Physics** toolbar, click **C** 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

- I In the Model Builder window, under Component I (compl) 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	mbd.rd1.w	m	Plunger displacement
vel	mbd.rd1.wt	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)

- I 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

- I In the Model Builder window, under Study I (Preprocessing) click Step I: 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

- I 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

- I 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

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

Streamline 1

I In the Model Builder window, right-click

Preprocessing: Normalized Air Gap Parameterization and Coil Direction and choose Streamline.

- 2 Select Boundary 97 only.
- In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 Magnetic Fields>Coil parameters>mf.coill.eCoilx,...,mf.coill.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

- I In the Home toolbar, click 2 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 2 Add Study to close the Add Study window.

STUDY 2 (TIME DEPENDENT)

- I 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

- I In the Model Builder window, under Study 2 (Time Dependent) click Step I: 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 I (Preprocessing), Coil Geometry Analysis.

Solution 2 (sol2)

I In the Study toolbar, click The 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 I 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.
- IO In the Model Builder window, under Study 2 (Time Dependent)>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1 click Coil current (compl.mf.coill.lCoil_ode).
- II In the Settings window for State, locate the Scaling section.
- 12 From the Method list, choose Manual.
- I3 In the Model Builder window, expand the Study 2 (Time Dependent)> Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver I node.
- I4 Right-click Study 2 (Time Dependent)>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver I>Direct and choose Enable.
- 15 In the Settings window for Direct, locate the General section.
- **I6** 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

I 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

- I Right-click Volume I and choose Selection.
- 2 In the Settings window for Selection, locate the Selection section.
- 3 From the Selection list, choose Deformed Domains.

Multislice I

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

- I In the Model Builder window, click Streamline Multislice I.
- 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

- I 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 **O** 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.
- II 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

- I 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

- I 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

- I 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 1

- I 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/mm².
- 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.
- IO Click OK.
- II 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 1

- I 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 1

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

Current Density and Magnetic Flux Lines

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

Streamline 1

- I 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 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

- I 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

- I 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

- I 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

- I 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

- I 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

- I 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 2

- I In the Model Builder window, under Results>Core Losses right-click Volume I 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

- I 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

- I 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 I.

Volume 3

- I 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

- I In the Model Builder window, expand the Volume 3 node, then click Translation I.
- 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

- I In the Core Losses toolbar, click i 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]
-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

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 I

- I 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
<pre>mf.ICoil_1*mf.RCoil_1/ mf.VCoil_1</pre>	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

- I In the Model Builder window, right-click Dynamics of the System Before Switching and choose Duplicate.
- 2 In the Settings window for ID Plot Group, type Dynamics of the System During Switching in the Label text field.

Global I

- I In the Model Builder window, expand the Dynamics of the System During Switching node, then click Global I.
- 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
<pre>mf.ICoil_1*mf.RCoil_1/ mf.VCoil_1</pre>	1	Current normalized to DC current
1+disp/maxdisp	1	Normalized gap size
<pre>4*mass*d(vel,t)*vel/at(1, mf.ICoil_1*mf.VCoil_1)</pre>	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

- I In the Model Builder window, right-click Dynamics of the System During Switching and choose Duplicate.
- 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. In the associated text field, type .

Global I

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

- I 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

- I 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 I).

Global 2

- I 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 I.
- 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 currents losses, normalized to DC coil power

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

Locking Force

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

- I In the Model Builder window, expand the Locking Force node, then click Global I.
- 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
mbd.prj1.lk1.F_u	Ν	Locking force

Global 2

- I 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

- I 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

- I 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.

Global Loads

- I In the Model Builder window, expand the Results>Applied Loads (mbd) node, then click Global Loads (mbd).
- 2 In the Settings window for 3D Plot Group, type Global Loads in the Label text field.
- **3** Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.