Created in COMSOL Multiphysics 5.5

Electrodynamics of a Power Switch — Multibody Version

Introduction

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

Circuit breakers can be classified according to features like voltage rating, construction type, structural features, and interruption technique. The main purpose of this model is to illustrate the working principle and some possible solutions for modeling 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 towards 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.

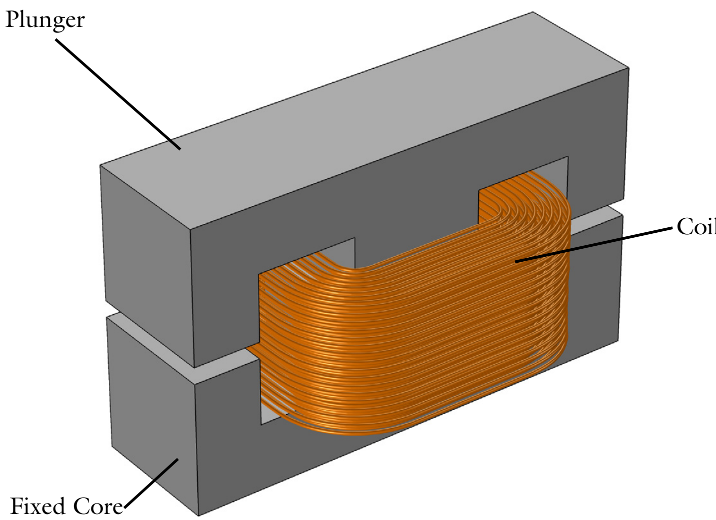

Figure 1: Geometry of the switch.

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.

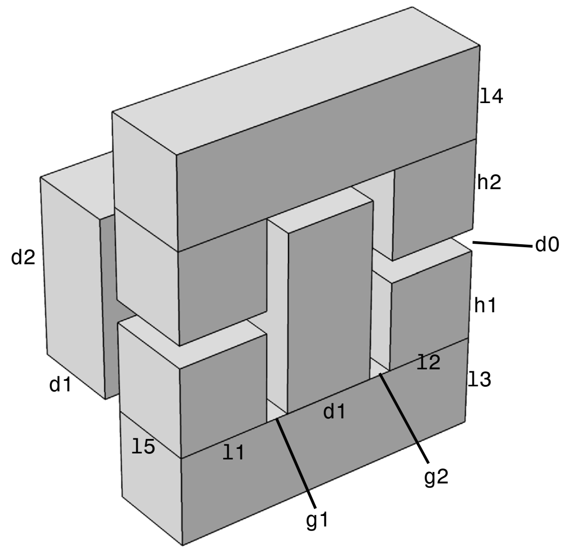

Figure 2: Switch specifications.

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 multi-turn 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.

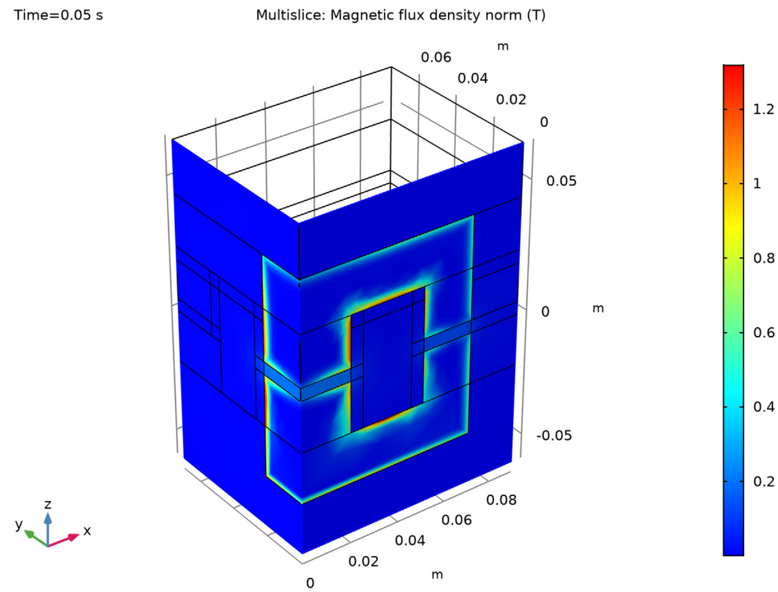

Figure 3: Magnetic flux density norm at t = 0.05 s, when the gap is open.

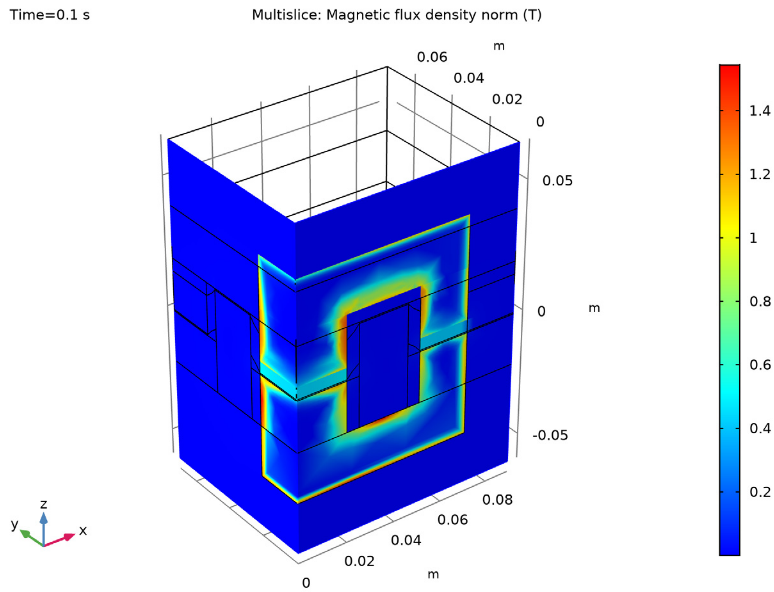

Figure 4: Magnetic flux density norm at t = 0.1 s, when the gap is closed.

Figure 5 shows the evolution of magnetic field streamlines at different instants of the simulation. The top-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 (top right) until it reaches its maximum compression (lower-left). Well before the final time of the simulation, the spring is completely compressed and the induced currents in the core have decayed (lower-right).

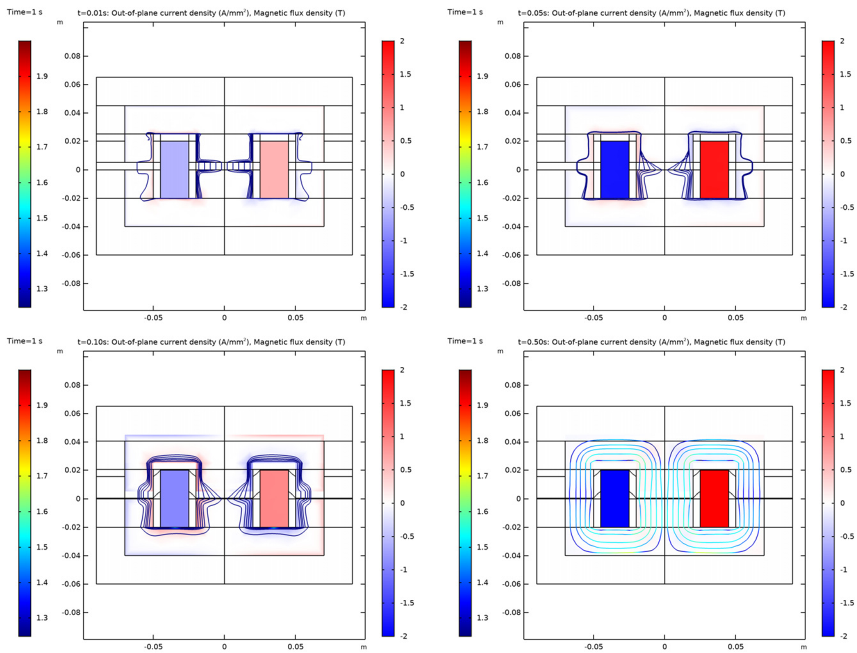

Figure 5: Evolution of current density (surface) and magnetic flux density (streamlines) at different times.

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

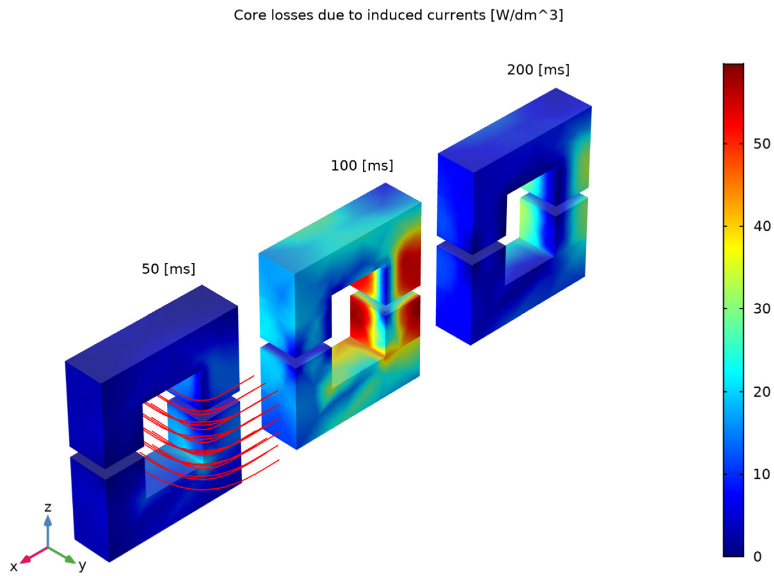

Figure 6: Core losses due to induced currents at different time instants.

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.

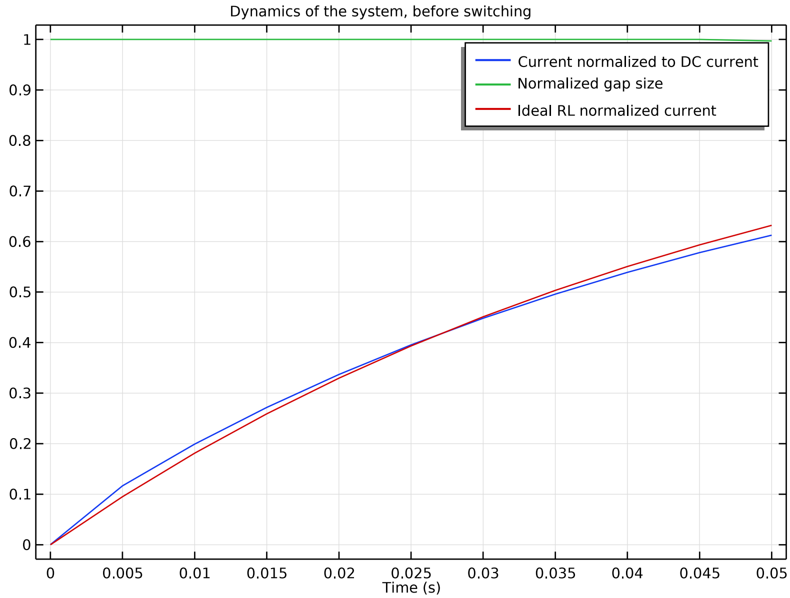

Figure 7: Magnetic flux density norm at t = 0.1 s, when the gap is closed.

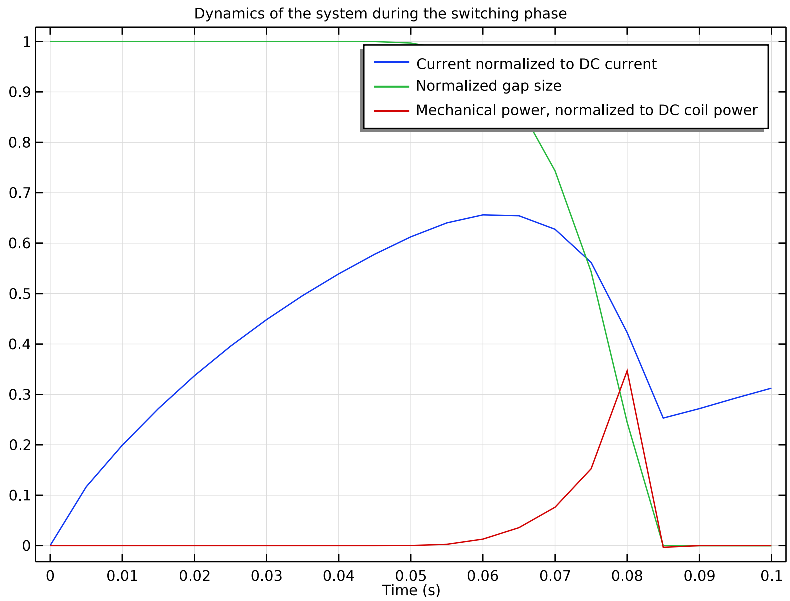

Figure 8: Magnetic flux density norm at t = 0.1 s, when the gap is closed.

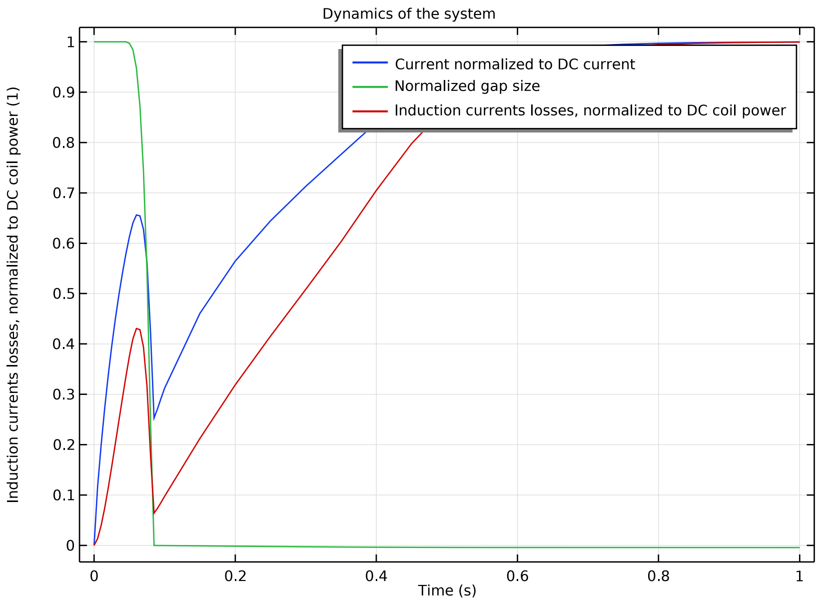

Figure 9: Magnetic flux density norm at t = 0.1 s, when the gap is closed.

It is worth recalling that blue and green curves in Figure 7, Figure 8, and Figure 9 are representing the same physical quantities, respectively the normalized current and the normalized gap size. 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.

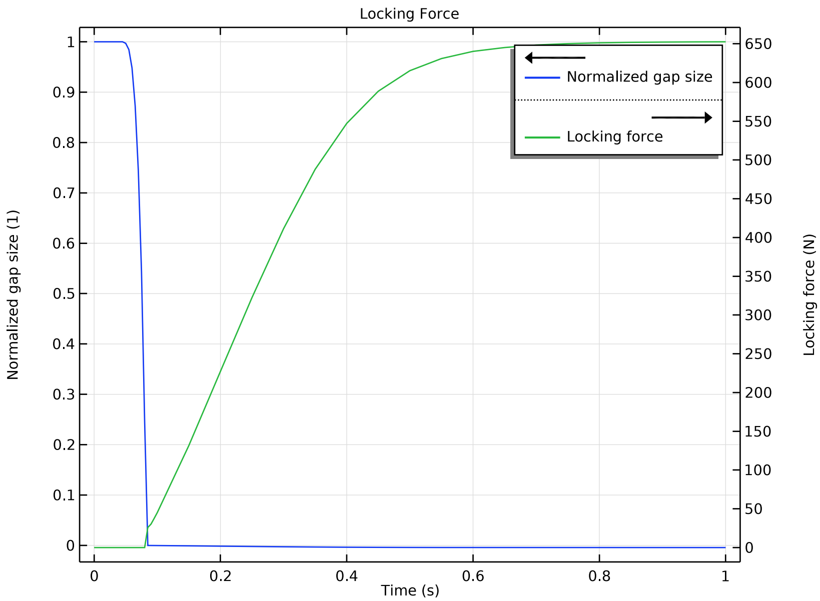

Figure 10: Variation of locking force and gap with time.

Application Library path: `ACDC_Module/Motors_and_Actuators/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 1

Import the geometric and physical parameters from an external file.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `power_switch_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 in a third 3D part.

PART 1

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.

The core section is made up of six rectangles.

3 Right-click **Core Section** and choose **Rectangle** six times, entering the following settings in each **Rectangle** node:

Name	**Width**	**Height**	**xw**	**yw**
Rectangle 1 (r1)	l1	h1	0	h1
Rectangle 2 (r2)	l1+g1+d1+g2+l2	l3	0	-h1-l3
Rectangle 3 (r3)	l2	h1	l1+g1+d1+g2	-h1
Rectangle 4 (r4)	l1	h2	0	d0

Name	Width	Height	xw	yw
Rectangle 5 (r5)	`l1+g1+d1+g2+ l2`	`l4`	`0`	`d0+h2`
Rectangle 6 (r6)	`l2`	`h2`	`l1+g1+d1+g2`	`d0`

Proceed to create the geometry part for the coil.

PART 2

1 Right-click **Core Section** 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 `l5`.

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

Circle 1 (c1)

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

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

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

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

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

6 In the **y** text field, type `l5`.

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

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

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `l1`.

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

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

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

7 Click **Build Selected**.

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**.

GLOBAL DEFINITIONS

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

PART 3

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**.

Plane Geometry

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

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

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

Work Plane 1 (wp1)

In the **Model Builder** window, click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (m)
15

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

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`.

Plane Geometry

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

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

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

Work Plane 2 (wp2)

In the **Model Builder** window, click **Work Plane 2 (wp2)**.

Extrude 2 (ext2)

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

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

3 In the table, enter the following settings:

Distances (m)
d2

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

5 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, several other solid parts have to be drawn in order to partition properly the air domain surrounding the coil and the core. Refer to the saved `.mph` file to see a possible parameterized partitioning. The purpose of partitioning the air gap is to minimize as much as possible the distortion of the mesh during the deformation.

GEOMETRY 1

Solid Parts 1 (pi1)

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

2 Right-click **Solid Parts 1 (pi1)** and choose **Build All Objects**.

In order to have a robust geometric parameterization, create a number of **Selection** nodes.

3 In the **Definitions** toolbar, use the buttons to create **Selection** nodes according to the following table:

Selection label	**Selection type**	**Entity level**	**Entity numbers/ Selection settings**
Deformed Domains	Explicit, sel1	`domain`	`4, 7, 9-11, 14, 16-19, 23-25, 27-29, 31-34, 36-39, 41`
Plunger	Explicit, sel2	`domain`	`5-6, 12, 20-21, 42-43`
Fixed Domains	Explicit, sel3	`domain`	`1-3, 8, 13, 15, 22, 26, 30, 35, 40`
Ext. Boundaries to Deformed Domains	Adjacent, adj1	`domain`	`sel1, exterior boundaries`
Ext. Boundaries to Plunger	Adjacent, adj2	`domain`	`sel2, exterior boundaries`
Ext. Boundaries to Fixed Domains	Adjacent, adj3	`domain`	`sel3, exterior boundaries`
Fixed Boundaries at Plunger	Intersection	`boundary`	`adj1, adj3`
Top Boundary	Explicit, sel4	`boundary`	`22`
Fixed Boundaries	Union, uni1	`boundary`	`int1, sel4`
Moving Boundaries	Intersection, int2	`boundary`	`adj1, adj2`
Nonlinear Core	Explicit, sel5	`domain`	`2-3, 5-6, 40, 42`
Ext. Boundaries to Nonlinear Core	Adjacent, adj4	`domain`	`sel5, exterior boundaries`
Coil Domain	Explicit, sel6	`domain`	`13, 30, 35`

MATERIALS

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

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Air` in the **Label** text field.

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

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

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Coil Domain**.

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

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **AC/DC>Soft Iron (With Losses)**.

4 Click **Add to Component 1 (comp1)**.

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

MATERIALS

Soft Iron (With Losses) (mat3)

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

2 From the **Selection** list, choose **Nonlinear Core**.

MESH 1

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

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

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

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

Free Triangular 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 12, 29, 54, 64, 84, 97, 117, 144, 160, and 174 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 Select Boundaries 12, 29, 54, 84, 97, 117, 144, and 160 only.

3 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

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

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

6 In the associated text field, type `d0*0.5`.

7 Select the **Maximum element growth rate** check box.

8 In the associated text field, type `1.2`.

Swept 1

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

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

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

4 Select Domains 4, 9, 16, 19, 23, 27, 32, 37, and 41 only.

Distribution 1

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

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

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

Swept 2

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

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

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

4 Select Domains 10, 17, 24, 28, 33, and 38 only.

Distribution 1

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

Free Triangular 2

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

2 Select Boundaries 47, 112, and 135 only.

Swept 3

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

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

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

4 Select Domains 11, 14, 18, 25, 29, 31, 34, 36, and 39 only.

Distribution 1

1 Right-click **Swept 3** and choose **Distribution**.

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

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

Free Tetrahedral 1

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

Use a boundary layer mesh to resolve the skin depth inside the core.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

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

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

4 From the **Selection** list, choose **Nonlinear Core**.

Boundary Layer Properties

Select the boundaries interior to the geometry and adjacent to the nonlinear core. The easiest way to do this is as follows:

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

2 From the **Selection** list box, choose **Ext. Boundaries to Nonlinear Core** .

3 Deselect the boundaries exterior to the geometry, leaving only the interior boundaries.

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

5 In the **Number of boundary layers** text field, type `7`.

6 In the **Boundary layer stretching factor** text field, type `1.4`.

7 From the **Thickness of first layer** list, choose **Manual**.

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

9 Click **Build All**.

DEFINITIONS

Deforming Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **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 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Normal Mesh Displacement**.

2 Select Boundaries 10, 11, 19, 20, 27, 30, 33, 45, 52, 55, 58, 62, 74, 78, 83, 86, 89, 134, 143, 146, 149, 159, 173, 186, 190, and 191 only.

Prescribed Mesh Displacement 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Mesh Displacement**.

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

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

Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Prescribed Mesh Displacement 1** 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	X
0	Y
disp	Z

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 Domain**.
4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.
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 Multi-Turn 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 1

1 In the **Model Builder** window, expand the **Coil 1** node.
2 In the **Settings** window for **Geometry Analysis**, locate the **Coil Geometry** section.
3 Find the **Symmetry specification** subsection. In the F_L text field, type `4`.

Input 1

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

2 Select Boundary 129 only.

Output 1

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

2 Select Boundary 40 only.

Ampère's Law 2

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.

3 From the **Magnetization model** list, choose **B-H curve**.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Nonlinear Core**.

Force Calculation 1

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

2 Select Domains 5, 6, and 42 only.

Gauge Fixing for A-field 1

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 Domain 1

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

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

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

4 Locate the **Density** section. From the ρ list, choose **User defined**. In the **Model Builder** window, click **Rigid Domain 1**.

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`.

Applied Force 1

1 In the **Model Builder** window, right-click **Rigid Domain 1** and choose **Applied Force**.

2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.

3 Specify the $\mathbf{F}$ vector as

0	x
0	y
min(mf.Forcez_0+k0*(x0-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 Domain 1**.

5 In the **Settings** window for **Prismatic Joint**, locate the **Center of Joint** section.

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

7 In the **Model Builder** window, click **Prismatic Joint 1**.

8 In the **Settings** window for **Prismatic Joint**, locate the **Axis of Joint** section.

9 Specify the $\mathbf{e}_0$ vector as

0	x
0	y
1	z

Locking 1

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 In the **Settings** window for **Locking**, locate the **Translational Locking** section.

5 From the **Locking parameters** list, choose **User defined**.

6 In the **Model Builder** window, click **Locking 1**.

7 In the **Settings** window for **Locking**, locate the **Translational Locking** section.

8 In the p_{u1} text field, type `(0.01*mbd.prj1.lk1.Eequ)*mbd.diag/10`.

9 In the p_{u2} text field, type `mbd.prj1.lk1.p_u1*10[ms]/50`.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
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 **Coil Geometry Analysis** which is needed in order to compute the direction of the coil.

STUDY 1

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

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

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

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

RESULTS

3D Plot Group 1

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

4 In the **Title** text area, type `Preprocessing: Air Gap Parameterization and Coil Direction`.

Volume 1

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

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

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

Filter 1

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

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

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

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

Streamline 1

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

2 Select Boundary 129 only.

3 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>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: 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

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 **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, 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, 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 **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, click **Time-Dependent Solver 1**.

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

19 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

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

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

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

23 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

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

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

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

27 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 significanly 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: Air Gap Parameterization and Coil Direction** click **Volume 1**.

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

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

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

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

Selection 1

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

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

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

Multislice 1

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

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

3 Find the **Z-planes** subsection. In the **Planes** text field, type 0.

4 Find the **X-planes** subsection. From the **Entry method** list, choose **Coordinates**.

5 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 Find the **X-planes** subsection. In the **Coordinates** text field, type 0.

7 Find the **Y-planes** subsection. In the **Coordinates** text field, type 0.

Magnetic Flux Density Norm (mf)

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

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

3 From the **Frame** list, choose **Spatial (x, y, z)**.

4 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.

5 Locate the **Data** section. From the **Time (s)** list, choose **0.05**.

6 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.

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

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

9 From the **Time (s)** list, choose **0.1**.

10 In the **Magnetic Flux Density Norm (mf)** 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.

Datasets

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

Mirror 3D 1

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

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

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

Cut Plane 1

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

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

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

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

2D Plot Group 6

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

2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

3 In the **Label** text field, type `Current Density and Magnetic Flux Lines`.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `t=0.01s: Out-of-plane current density (A/mm<sup> 2</sup>), Magnetic flux density (T)`.

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

Surface 1

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. From the **Color table** list, choose **WaveLight**.

9 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

1 In the **Model Builder** window, click **Mirror 3D 1**.

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

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

Surface 1

1 In the **Model Builder** window, 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 **Plot**.

Current Density and Magnetic Flux Lines

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

Streamline 1

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

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

3 In the **x component** text field, type `mf.Bx`.

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

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

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

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

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

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

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

Current Density and Magnetic Flux Lines

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

3D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

4 In the **Label** text field, type `Core Losses`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Core losses due to induced currents [W/dm^3]`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Streamline 1

1 Right-click **Core Losses** and choose **Streamline**.

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

3 In the **X component** text field, type `mf.Jx`.

4 In the **Y component** text field, type `mf.Jy`.

5 In the **Z component** text field, type `mf.Jz`.

6 Select Boundary 129 only.

Volume 1

1 In the **Model Builder** window, right-click **Core Losses** and choose **Volume**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `mf.Qrh`.

5 In the **Unit** field, type `W/dm^3`.

6 Locate the **Data** section. From the **Time (s)** list, choose **0.05**.

Selection 1

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

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

3 From the **Selection** list, choose **Nonlinear Core**.

Volume 2

1 In the **Model Builder** window, under **Results>Core Losses** right-click **Volume 1** and choose **Duplicate**.

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

3 From the **Time (s)** list, choose **0.1**.

Deformation 1

1 Right-click **Volume 2** and choose **Deformation**.

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

3 In the **X component** text field, type `-0.1`.

4 In the **Y component** text field, type `0`.

5 In the **Z component** text field, type `0`.

6 Locate the **Scale** section. Select the **Scale factor** check box.

7 In the associated text field, type `1`.

Volume 2

1 In the **Model Builder** window, click **Volume 2**.

2 In the **Settings** window for **Volume**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Volume 1**.

Volume 3

1 Right-click **Results>Core Losses>Volume 2** and choose **Duplicate**.

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

3 From the **Time (s)** list, choose **0.2**.

Deformation 1

1 In the **Model Builder** window, expand the **Volume 3** node, then click **Deformation 1**.

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

3 In the **X component** text field, type `-0.2`.

Annotation 1

1 In the **Model Builder** window, right-click **Core Losses** and choose **Annotation**.

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

3 In the **Text** text field, type `50 [ms]`.

4 Locate the **Position** section. In the **X** text field, type `0.03`.

5 In the **Z** text field, type `0.07`.

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

Annotation 2

1 Right-click **Core Losses** and choose **Annotation**.

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

3 In the **Text** text field, type `100 [ms]`.

4 Locate the **Position** section. In the **X** text field, type `-0.07`.

5 In the **Z** text field, type `0.07`.

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

Annotation 3

1 Right-click **Core Losses** and choose **Annotation**.

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

3 In the **Text** text field, type `200 [ms]`.

4 Locate the **Position** section. In the **X** text field, type `-0.17`.

5 In the **Z** text field, type `0.07`.

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

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

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

1D Plot Group 8

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

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

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

4 In the **Title** text area, type `Dynamics of the system, before switching`.

Global 1

1 Right-click **Dynamics, Before Switching** and choose **Global**.

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

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

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

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

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

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

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

Dynamics, 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, Before Switching 1

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

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

3 Locate the **Title** section. In the **Title** text area, type `Dynamics of the system during the switching phase`.

Global 1

1 In the **Model Builder** window, expand the **Results>Dynamics 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 During Switching** toolbar, click **Plot**.

Dynamics 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 During Switching 1

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

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

3 Locate the **Title** section. In the **Title** text area, type `Dynamics of the system`.

Global 1

1 In the **Model Builder** window, expand the **Results>Dynamics, Complete** 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 Right-click **Integral 1** 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 Domain**.

Global 2

1 In the **Model Builder** window, right-click **Dynamics, Complete** 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/m^3	

4 Locate the **Data** section. From the **Dataset** list, choose **Integral 1**.

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

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

6 In the **Dynamics, Complete** toolbar, click **Plot**.

Dynamics, Complete 1

1 Right-click **Dynamics, Complete** and choose **Duplicate**.

2 In the **Settings** window for **1D 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 **Title** section. In the **Title** text area, type `Locking Force`.

5 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

6 In the table, enter the following settings:

Plot	Plot on secondary y-axis
Global 2	√

Global 1

1 In the **Model Builder** window, expand the **Results>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 **Model Builder** window, click **Global 1**.

6 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

7 In the table, enter the following settings:

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

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 **Model Builder** window, click **Global 2**.

6 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

7 In the table, enter the following settings:

Expression	Unit	Description
mbd.prj1.lk1.F_u	N	Locking force

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