AC Contactor with Shading Coil

Introduction

An AC contactor is a particular type of magnetic switch device that is activated by a primary coil fed by an alternating current. Unlike DC switches, such devices can suffer from a tendency to reopen when the AC current crosses zero. The addition of a shading coil that supports retarded induced currents with respect to those of the feeding coil makes it possible to always have a nonzero pulling force, thus providing a more stable closure.

This example studies the closing dynamics of an AC contactor and consequent establishing of a mechanical contact. The changing distance between the moving and stationary parts of the device has an influence on the distribution of the magnetic field. This effect is accounted for using a moving mesh in the surrounding air. The model is set up using the Magnetomechanics multiphysics interface.

Model Definition

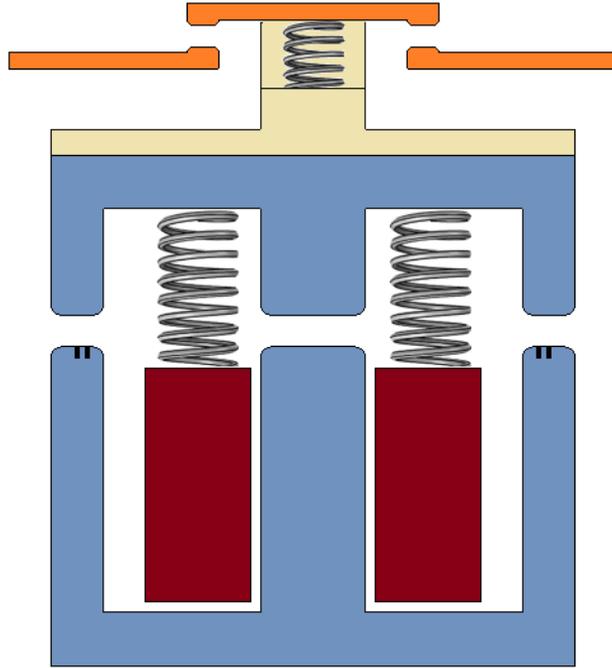shows a cross section of the geometry of the device.

Figure 1: Full geometry of the contactor. The primary coil is shown in red, the shading coil is shown in black, the copper contacts are shown in orange, the iron core is shown in blue, and the plastic parts are shown in yellow.

The fixed part of the iron core is winded by the primary feeding coil, which together can act as a strong electric magnet that will pull down the moving part of the iron core. The moving part is suspended on springs known as return springs. When there is no current through the main coil, the return springs will keep the device opened. The electrical contact is made of copper and is split up into fixed parts, as well as a part that is attached to the moving part of the core. It is attached by another spring called the contact spring. The initial gap between the electrical contact parts is smaller than that between the iron pieces. When the core becomes fully closed, the contact spring will be slightly extended, thus providing an extra pulling force to keep the electrical contact closed.

The primary coil is fed by an AC voltage of 48 V at 50 Hz. Two shading coils (rings) made of aluminum are embedded into the side limbs of the fixed part of the iron core. The

current induced in the shading coils will have a phase shift with respect to that in the primary coil. This will provide some pulling magnetic force even when the current in the primary coil is close to zero, preventing the contactor from opening.

The core is made of soft iron with nonlinear magnetization properties as specified by the BH curve shown in Figure 2.
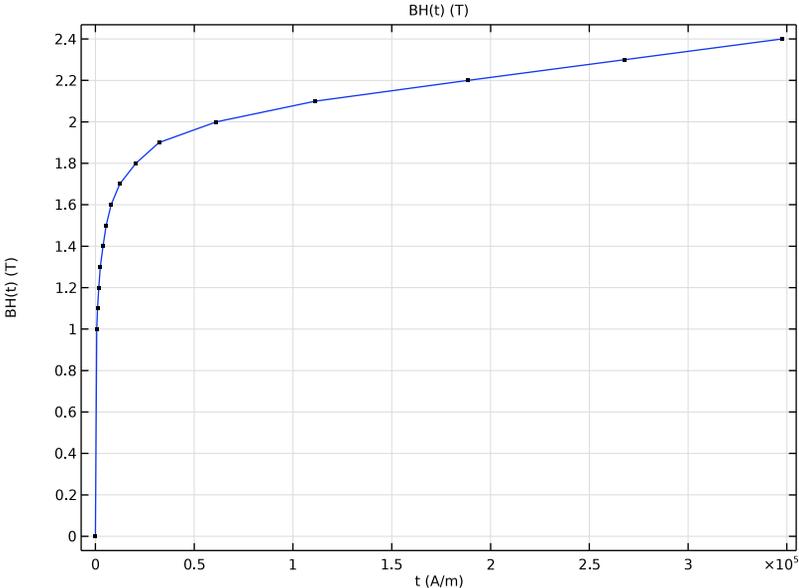

Figure 2: The BH curve for soft iron.

Because of the two-fold mirror symmetry of the geometry along the vertical axis, it is sufficient to model only one half of the device. This will reduce the time needed to solve the model. In addition, the magnetic field needs to be computed in the air surrounding

the contactor. This can be done by creating a bounding box for the geometry, as shown in Figure 3.
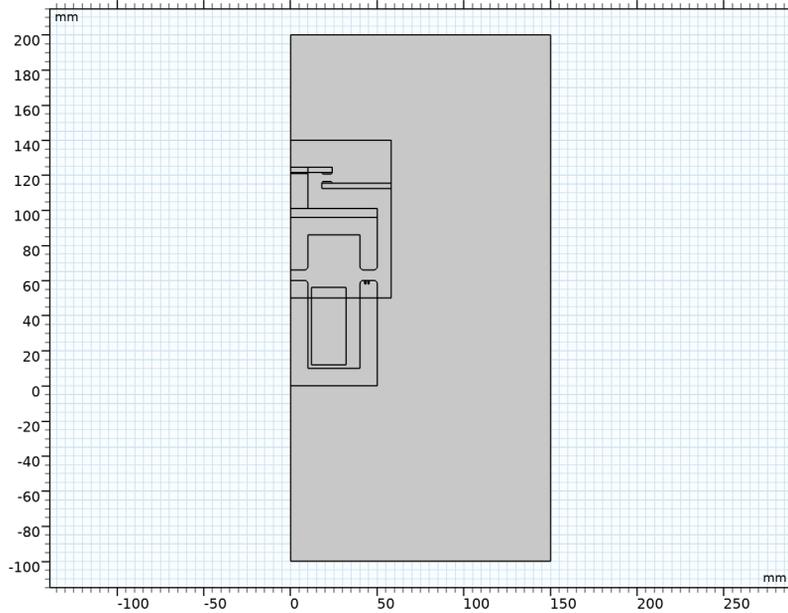

Figure 3: Model geometry, when cut in half and including the bounding box.

To include the effect of the changing distance between the parts, when the upper part of the device is deflected by the magnetic field, you set up a moving mesh in a domain surrounding the moving part. This is shown in Figure 3. The mesh is free to deform inside that domain, and it can also slide on the symmetry cut boundaries.

For the structural mechanics problem, the magnetic field, and the moving mesh, symmetry conditions are used at the symmetry cut plane. This allows you to retrieve the solution for the full geometry by mirroring that of the half that is being modeled.

Results and Discussion

Figure 4 shows how the sizes of the two different air gaps evolve in time. Here, it is shown that the initial air gap in the central limb is larger than that between the electrical contacts, even though both become zero after extending the contact spring. Figure 5 instead shows

the current in the primary coil, along with the corresponding induced current in the shading coil. Figure 6 and Figure 7 show the magnetic fields generated by each coil.
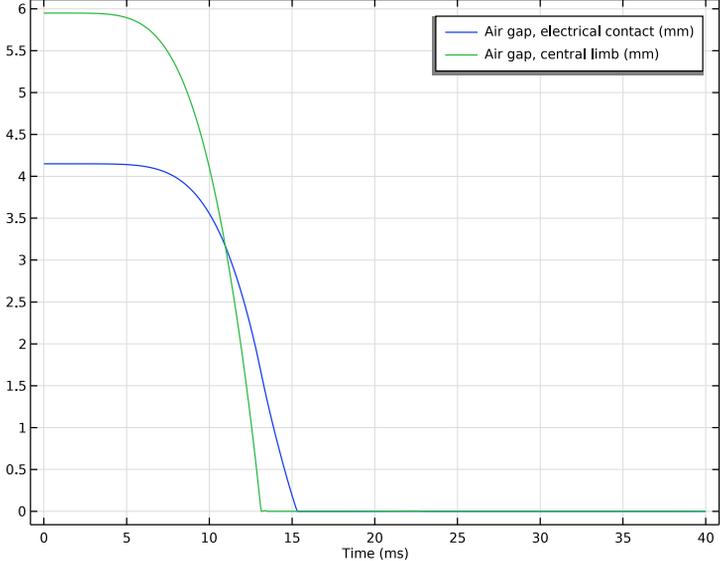

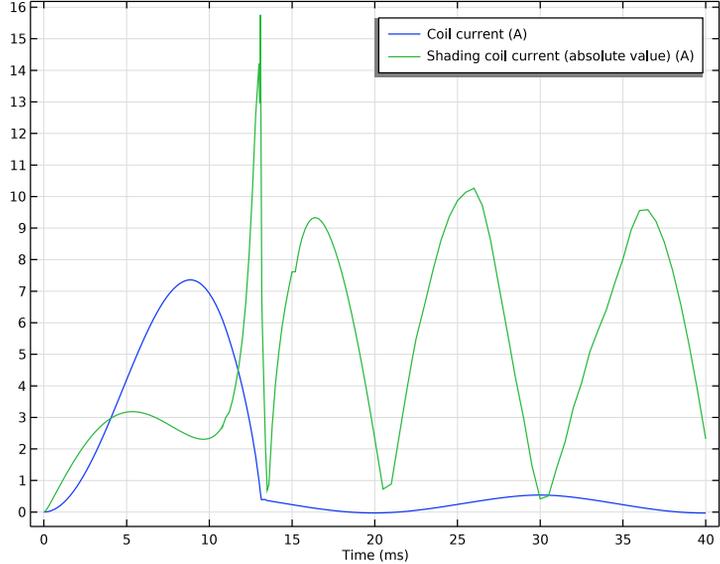
Figure 4: Air gap evolution for the closing switch.

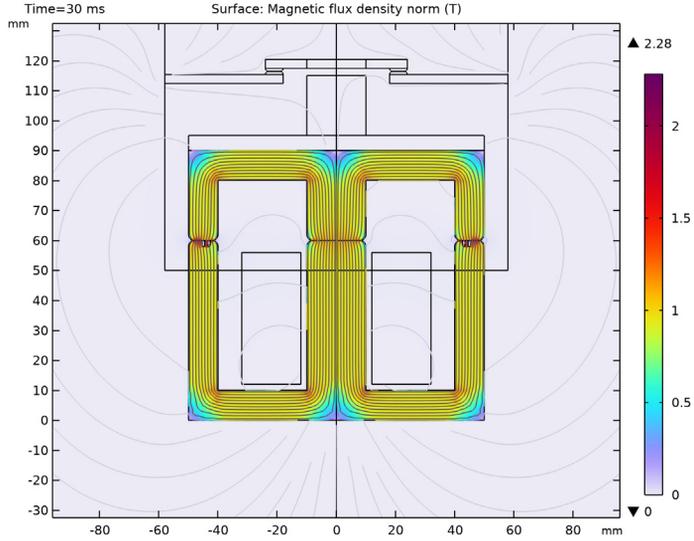*Figure 5: Currents in the primary and shading coils.*

Figure 6: Magnetic flux density and magnetic flux streamlines at t = 30 ms. The magnetic field is generated by the primary coil.
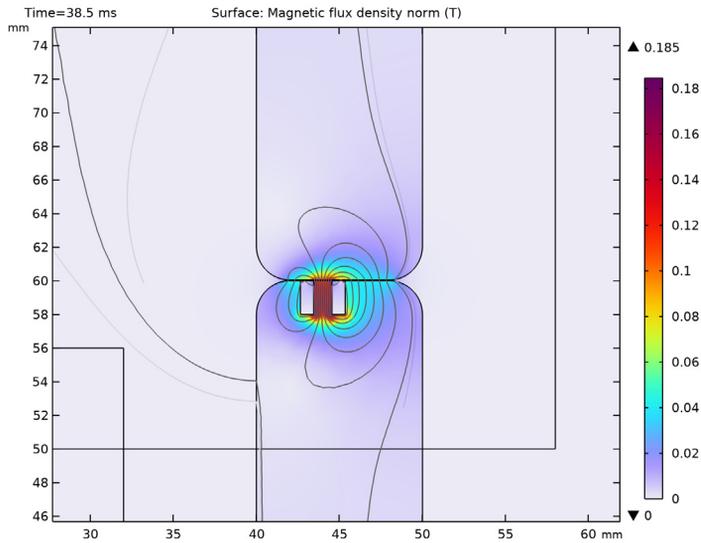
Figure 7: Magnetic flux density and magnetic flux streamlines at t = 38.5 ms. The magnetic field is due to the induced current in the shading coil.

Figure 8 and Figure 9 show the contact forces acting on the electrical contact and the central limb, as evaluated by the Global Variable Probes. By comparing with the air gap sizes in Figure 4, it can be noted that the contact force on the central limb continues to oscillate even after the two parts has come into contact, while the force on the electrical contact remains nearly constant due to the contact spring.

.

Figure 8: Mechanical contact force variation in time.

Figure 9: Closer look at the contact force variation at the electric contact.

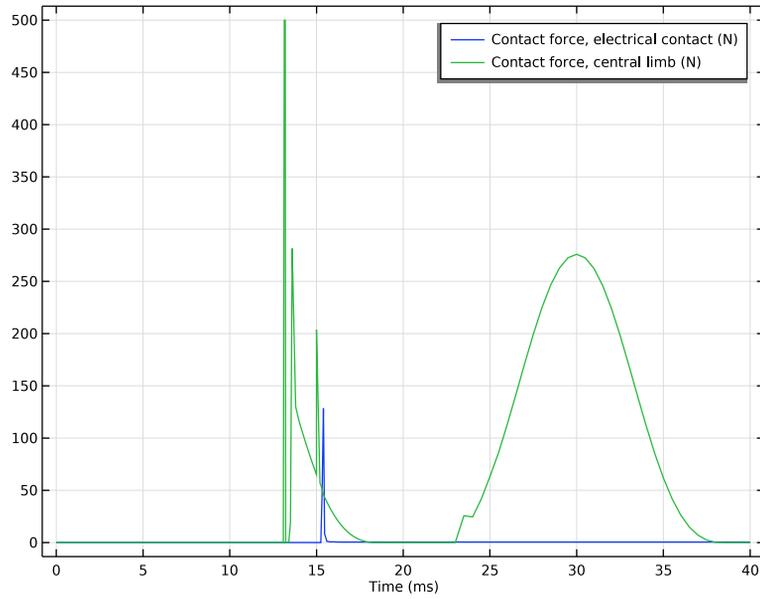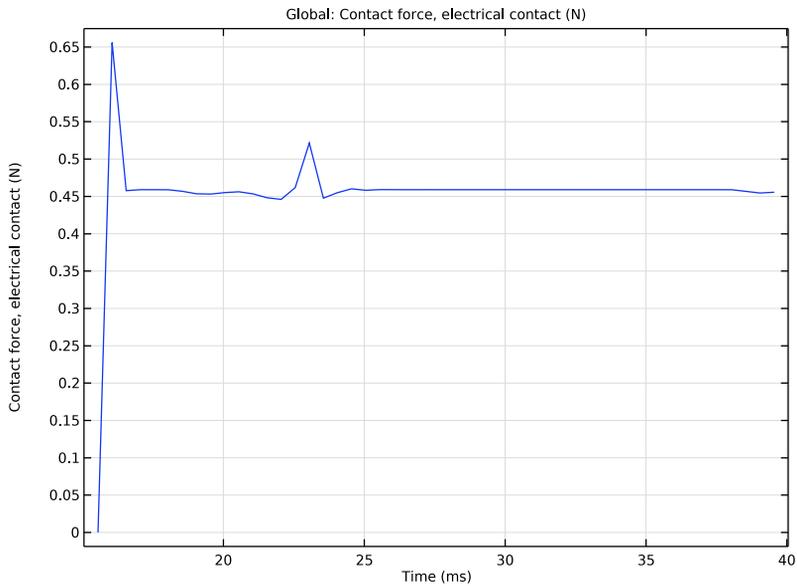Finally, Figure 10 shows how the vertical component of the total pulling electromagnetic force evolves in time, as evaluated by the Force Calculation feature. Comparing with the previous plot, it can be seen that this pulling force corresponds to the contact force on the central limb. However, due to the shading coil, the device never reopens.
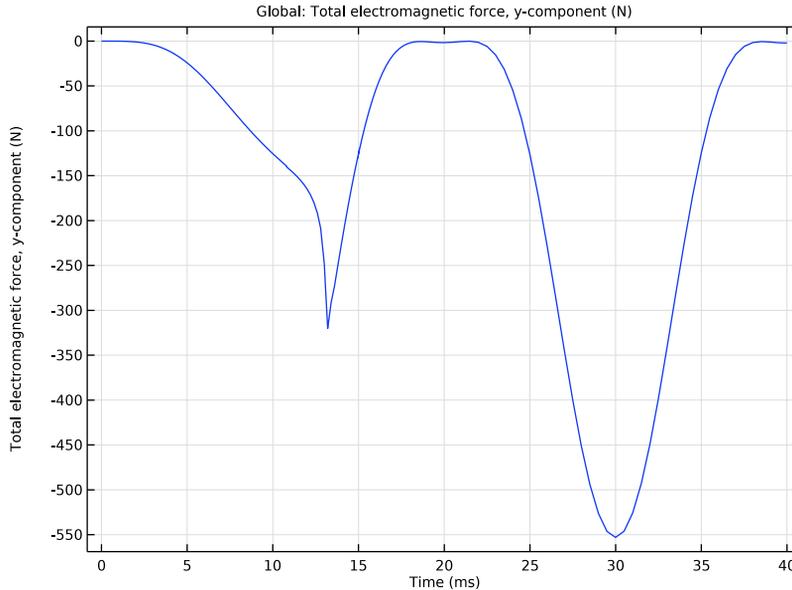
Figure 10: The total pulling electromagnetic force (recomputed for the full geometry).

Notes About the COMSOL Implementation

The model is set up using the Magnetomechanics multiphysics interface. When this interface is added to a model, one Solid Mechanics and one Magnetic Fields physics interface are automatically added, along with a Magnetomechanics multiphysics coupling feature. Two moving-mesh-related nodes are also added automatically: a Deforming Domain and a Symmetry/Roller boundary condition.

The magnetic body and surface forces acting on the plate are applied automatically by the coupling feature on its selection. Later on in the modeling, you will also add a Force Calculation node under the Magnetic Fields physics interface. This node is only used in postprocessing for computing the total force acting on the plate.

Model the return springs by using a Spring Foundation feature in the domain representing the moving part of the core. For the contact spring, use a special Spring-Damper feature

that connects the moving part of the electrical contact with the plastic cover attached to the moving core.

Use an offset on the mechanical Contact feature to prevent the complete closing of the air gaps, because such a change in the topology would be incompatible with the moving mesh in the air gaps. Furthermore, since the mesh will be more and more distorted when the gaps are closing, you need to activate the automatic remeshing option on the study step node.

Application Library path: ACDC_Module/Electromagnetics_and_Mechanics/ contactor_shading_coil

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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Electromagnetics– Structure Interaction** > **Magnetomechanics** > **Magnetomechanics**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

6 Click **Done**.

First, define some parameters that are used when creating the model.

GLOBAL DEFINITIONS

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
em_w	100[mm]	0.1 m	Width, electromagnet
em_h	60[mm]	0.06 m	Height, electromagnet
core_w	20[mm]	0.02 m	Width, core
pl_h	30[mm]	0.03 m	Height, plunger
plunger_travel	6[mm]	0.006 m	Total stroke, plunger
N	500	500	Number of turns, primary coil
d	1.35[N*s/m]	1.35 N·s/m	Damping constant, return spring
kr	24.2[N/m]	24.2 N/m	Spring constant, return spring
l0	plunger_travel*0.1	6E-4 m	Undeformed length, contact spring
op	plunger_travel*0.7	0.0042 m	Opening, electrical contact
dc	2.7[mm]	0.0027 m	Diameter, shading coil
kc	255[N/m]	255 N/m	Spring constant, contact spring
c	5[N*s/m]	5 N·s/m	Damping coefficient, contact spring

Now, start building the geometry of the model.

GEOMETRY I

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

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

3 From the **Length unit** list, choose **mm**.

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 em_w/2.

4 In the **Height** text field, type em_h.

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 em_w/2.

4 In the **Height** text field, type pl_h.

5 Locate the **Position** section. In the **y** text field, type em_h.

Rectangle 3 (r3)

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 (em_w-2*core_w)/2.

4 In the **Height** text field, type em_h+pl_h -core_w.

5 Locate the **Position** section. In the **x** text field, type core_w/2.

6 In the **y** text field, type core_w/2.

Difference 1 (dif1)

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

2 Select the objects **r1** and **r2** only.

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

4 Click to select the ▭ **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **r3** only.

Split 1 (spl1)

1 In the **Geometry** toolbar, click ▧ **Conversions** and choose **Split**.

2 Select the object **dif1** only.

Rectangle 4 (r4)

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 20.

4 In the **Height** text field, type 44.

5 Locate the **Position** section. In the **x** text field, type 12.

6 In the **y** text field, type 12.

Rectangle 5 (r5)

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 50.

4 In the **Height** text field, type 5.

5 Locate the **Position** section. In the **y** text field, type 90.

Rectangle 6 (r6)

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 10.

4 In the **Height** text field, type 20.

5 Locate the **Position** section. In the **y** text field, type 95.

Rectangle 7 (r7)

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 24.

4 In the **Height** text field, type 3.

5 Locate the **Position** section. In the **y** text field, type 115.

6 Click to expand the **Layers** section. Select the **Layers to the left** checkbox.

7 Clear the **Layers on bottom** checkbox.

8 In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	10

Rectangle 8 (r8)

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 6.

4 Locate the **Position** section. In the **x** text field, type 18.

5 In the **y** text field, type 114.

Rectangle 9 (r9)

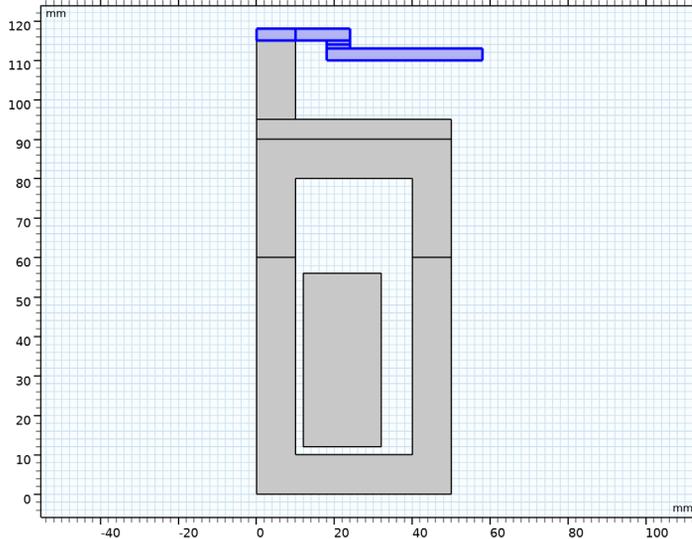**1** In the **Geometry** toolbar, click ▭ **Rectangle**.

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

3 In the **Width** text field, type 6.

4 Locate the **Position** section. In the **x** text field, type 18.

5 In the **y** text field, type 113.

Rectangle 10 (r10)

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 40.

4 In the **Height** text field, type 3.

5 Locate the **Position** section. In the **x** text field, type 18.

6 In the **y** text field, type 110.

Move 1 (mov1)

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

2 In the **Settings** window for **Move**, locate the **Displacement** section.

3 In the **y** text field, type 10.

4 Select the objects **r10**, **r7**, **r8**, and **r9** only.

Move 2 (mov2)

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

2 In the **Settings** window for **Move**, locate the **Displacement** section.

3 In the **y** text field, type `plunger_travel`.

4 Select the objects **mov1(1)**, **mov1(2)**, **mov1(3)**, **mov1(4)**, **r5**, **r6**, and **spl1(2)** only.
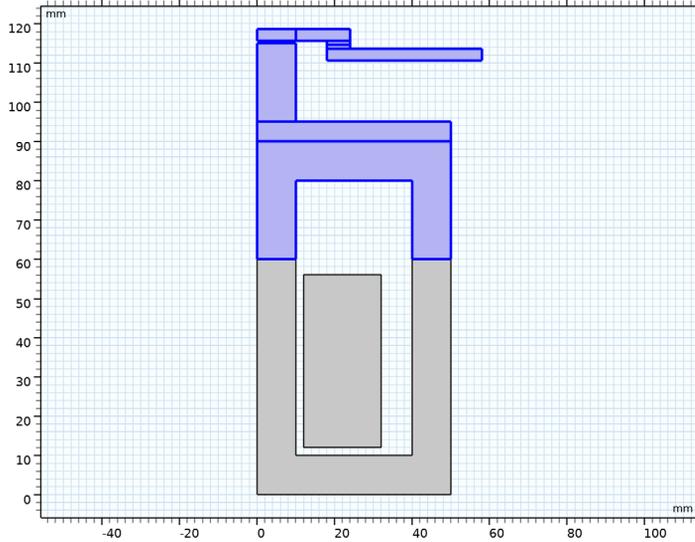

Move 3 (mov3)

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

2 In the **Settings** window for **Move**, locate the **Displacement** section.

3 In the **y** text field, type `-op`.

4 Select the objects **mov2(1)** and **mov2(4)** only.
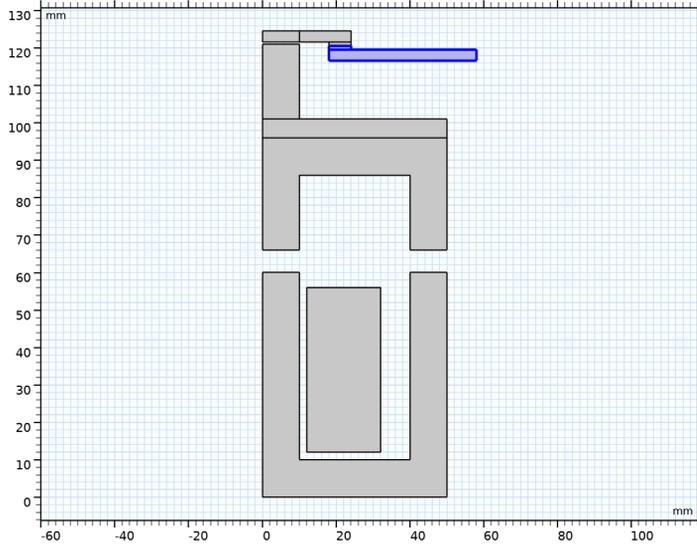
Fillet 1 (fil1)

1 In the **Geometry** toolbar, click ◰ **Fillet**.

2 In the **Settings** window for **Fillet**, locate the **Radius** section.

3 In the **Radius** text field, type `core_w*0.1`.

4 On the object **mov2(7)**, select Points 3, 5, and 7 only.

5 On the object **spl1(1)**, select Points 4, 6, and 8 only.
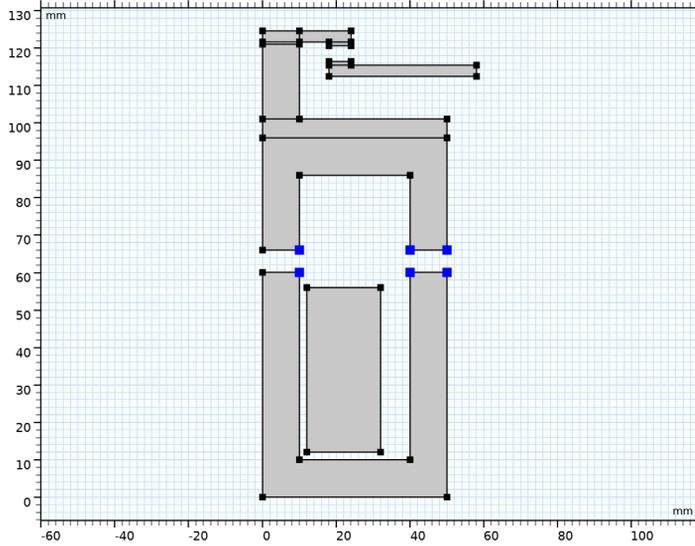
Fillet 2 (fil2)

1 In the **Geometry** toolbar, click ▨ **Fillet**.

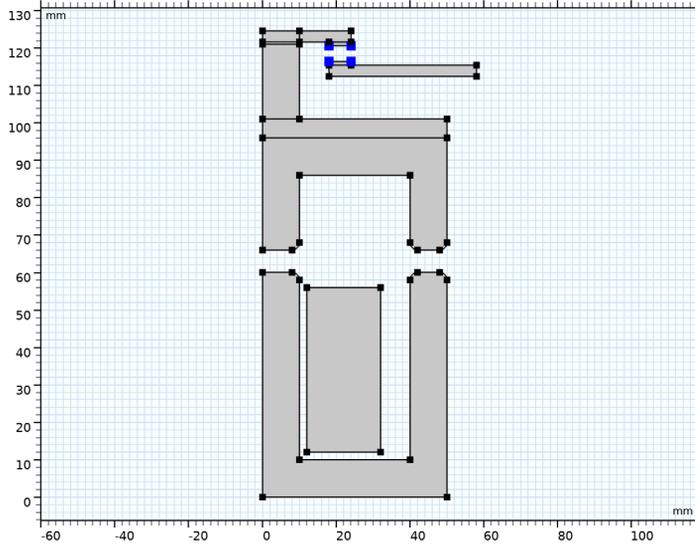**2** In the **Settings** window for **Fillet**, locate the **Radius** section.

3 In the **Radius** text field, type 1.

4 On the object **mov2(3)**, select Points 1 and 2 only.

5 On the object **mov3(2)**, select Points 3 and 4 only.

To finish the building of the geometry, set up a bounding box for it, which will define the surrounding air domains.

Rectangle 11 (r11)

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 3*em_w/2.

4 In the **Height** text field, type 3*em_w.

5 Locate the **Position** section. In the **y** text field, type -em_w.

Next, define domains representing the shading coil cross section.

Rectangle 12 (r12)

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 dc.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **x** text field, type 44.

7 In the **y** text field, type 59.

8 Locate the **Layers** section. Clear the **Layers on bottom** checkbox.

9 Select the **Layers to the right** checkbox.

10 Select the **Layers to the left** checkbox.

11 In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.8

Delete Entities 1 (del1)

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

2 On the object **r12**, select Boundary 5 only.

Finally, define a domain where the moving mesh will be used.

Rectangle 13 (r13)

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 58.

4 In the **Height** text field, type 90.

5 Locate the **Position** section. In the **y** text field, type 50.

6 Click ▦ **Build All Objects**.

In order to treat the boundaries that start off separated but eventually come into contact, create two contact pairs. One of them is for the copper electrical contact, while the other one is for the iron pieces in the core.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click ▯◀ **Pairs** and choose **Contact Pair**.

2 Select Boundaries 44, 78, and 80 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Click to select the ▭ **Activate Selection** toggle button.

5 Select Boundaries 45, 79, and 81 only.

6 Locate the **Advanced** section. From the **Search method** list, choose **Distance tracking**.

7 From the **Search distance** list, choose **Manual**.

8 In the **Distance** text field, type 6.

Contact Pair 2 (p2)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 8, 55, 59, 61, 64, 66, 76, 82, and 84 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 10, 56, 77, 83, and 85 only.

6 Locate the **Advanced** section. From the **Search method** list, choose **Distance tracking**.

7 From the **Search distance** list, choose **Manual**.

8 In the **Distance** text field, type 6.

In order to account for the distance between the parts decreasing, moving mesh functionality will be used.

MOVING MESH

Deforming Domain 1

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

2 Select Domain 4 only.

Symmetry/Roller 1

1 In the **Model Builder** window, click **Symmetry/Roller 1**.

2 Select Boundaries 7, 15, 19, 72, and 74 only.

ADD MATERIAL

1 In the **Materials** 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 (Without Losses)**.

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

MATERIALS

Soft Iron (Without Losses) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Soft Iron (Without Losses) (mat1)**.

2 Select Domains 2, 3, 5, and 16 only.
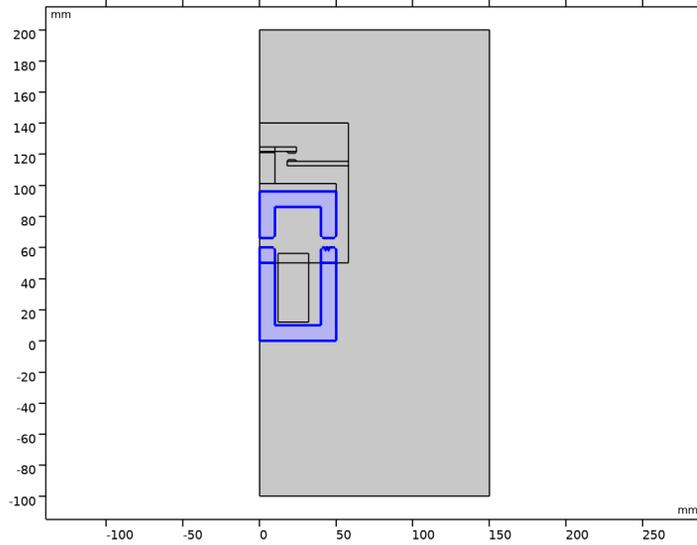
ADD MATERIAL

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

2 In the tree, select **Built-in** > **Copper**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click
   Copper (mat2).

2 Select Domains 8, 10, and 13–15 only.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Acrylic plastic**.

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

MATERIALS

Acrylic plastic (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click
Acrylic plastic (mat3).

2 Select Domains 6 and 7 only.
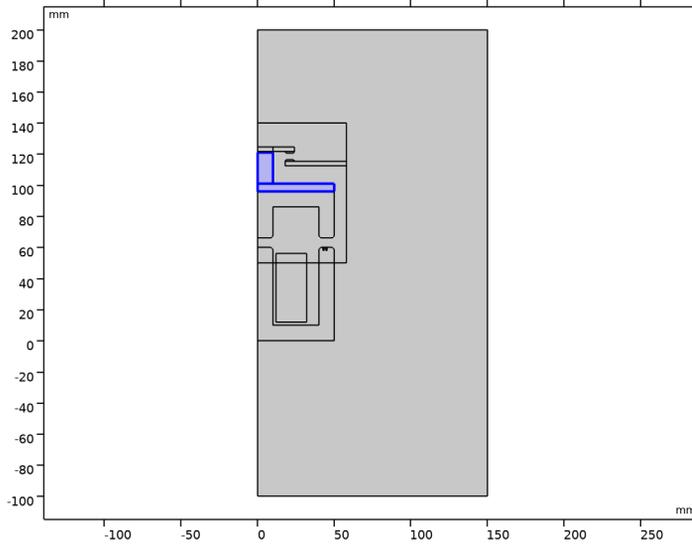
ADD MATERIAL

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

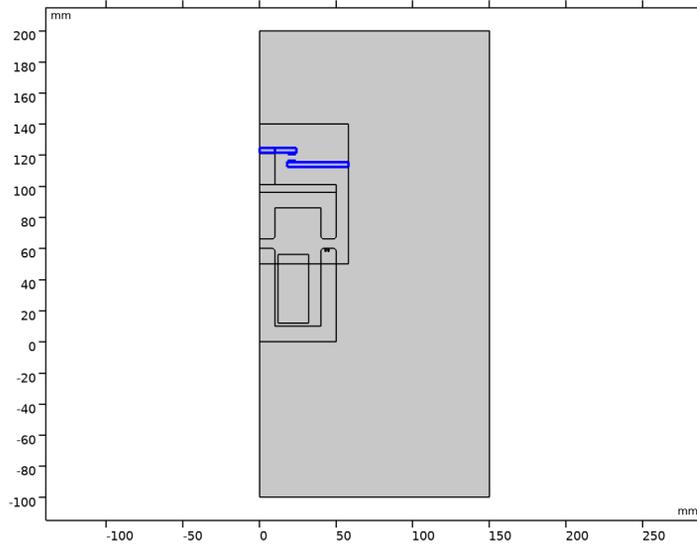**2** In the tree, select **Built-in** > **Aluminum**.

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

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat4)
Select Domains 11, 12, 17, and 18 only.

Now, it is time to add the physics to the model, using the **Solid Mechanics** interface and the **Magnetic Fields** interface. Both of these were added automatically when the multiphysics coupling was added.

SOLID MECHANICS (SOLID)

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

2 In the **Show More Options** dialog, click **Select All**.

3 Click **OK**.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

5 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

6 In the d text field, type 30[mm].

7 Select Domains 2, 3, 5–8, 10, and 13–18 only.
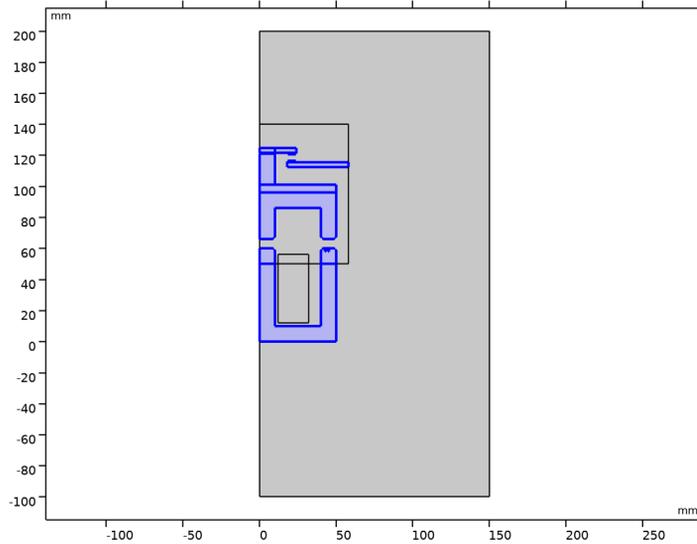
8 Click to expand the **Advanced Settings** section. Find the **Group ODE variables in solver** subsection. Clear the **Attachments** checkbox.

Fixed Constraint 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 4, 41, and 73 only.

Spring Foundation 1

1 In the **Physics** toolbar, click 🔵 **Domains** and choose **Spring Foundation**.

2 Select Domain 5 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 In the $\mathbf{k}_{tot}$ text field, type kr.

6 Click to expand the **Viscous Damping** section. From the **Damping type** list, choose **Total damping constant**.

7 In the $\mathbf{d}_{tot}$ text field, type d.

Roller 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Roller**.

2 Select Boundaries 47, 67, 68, 70, and 71 only.

Attachment 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Attachment**.

2 Select Boundary 18 only.

Attachment 2

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Attachment**.

2 Select Boundary 16 only.

Spring–Damper 1

1 In the **Physics** toolbar, click ⚛ **Global** and choose **Spring–Damper**.

2 In the **Settings** window for **Spring–Damper**, locate the **Attachment Selection** section.

3 From the **Source** list, choose **Attachment 1**.

4 From the **Destination** list, choose **Attachment 2**.

5 Locate the **Spring–Damper** section. In the k text field, type kc.

6 In the c text field, type c.

Symmetry 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Symmetry**.

2 Select Boundaries 3, 5, 9, 11, 13, and 17 only.

Contact 1

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

2 In the **Settings** window for **Contact**, click to expand the
   Contact Surface Offset and Adjustment section.

3 In the $d_{\text{offset,d}}$ text field, type 0.05[mm].

MAGNETIC FIELDS (MF)

Ampère's Law in Solids 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Magnetic Fields (mf)** click
   Ampère's Law in Solids 1.

2 In the **Settings** window for **Ampère's Law in Solids**, locate the **Domain Selection** section.

3 Click 🖊 **Clear Selection**.

4 Select Domains 6–8, 10, and 13–15 only.

5 In the **Model Builder** window, click **Magnetic Fields (mf)**.

6 In the **Settings** window for **Magnetic Fields**, locate the **Thickness** section.

7 In the d text field, type 30[mm].

Ampère's Law 2, Iron

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

2 In the **Settings** window for **Ampère's Law in Solids**, type Ampère's Law 2, Iron in the **Label** text field.

3 Select Domains 2, 3, 5, and 16 only.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **B-H curve**.

Coil 1, primary

1 In the **Physics** toolbar, click ⬤ **Domains** and choose **Coil**.

2 In the **Settings** window for **Coil**, type Coil 1, primary in the **Label** text field.

3 Select Domains 11 and 12 only.

4 Locate the **Material Type** section. From the **Material type** list, choose **Solid**.

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

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

7 In the V_{coil} text field, type 48[V]*sin(2*pi*50[Hz]*t)/2.

8 Locate the **Homogenized Multiturn Conductor** section. In the N text field, type N.

9 From the **Coil wire cross-section area** list, choose **From round wire diameter**.

10 In the d_{wire} text field, type 1.5[mm].

Coil 2, shading

1 In the **Physics** toolbar, click ⬤ **Domains** and choose **Coil**.

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

3 Select Domains 17 and 18 only.

4 Locate the **Material Type** section. From the **Material type** list, choose **Solid**.

5 Locate the **Coil** section. In the I_{coil} text field, type 0[A].

Magnetic Insulation 2

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Magnetic Insulation**.

2 Select Boundaries 18, 20, 33, 34, 40, 41, 44–47, 73, and 78–81 only.

Symmetry Plane 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Symmetry Plane**.

2 Select Boundaries 1, 3, 5, 7, 9, 11, 13, 15, 17, 19, and 21 only.

Finally, the selection for the multiphysics coupling between the two physics interfaces needs to be specified.

MULTIPHYSICS

Magnetomechanics 1 (mmcp11)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Magnetomechanics 1 (mmcp11)**.

2 Select Domains 2, 3, 5, and 16–18 only.
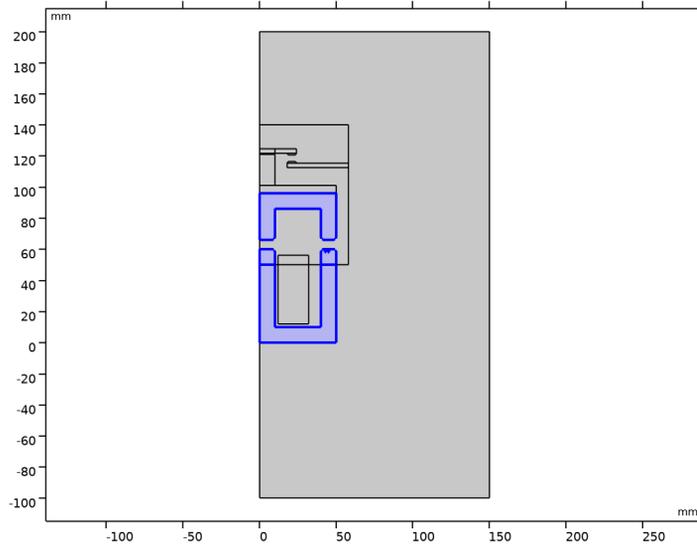
MATERIALS

Acrylic plastic (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Acrylic plastic (mat3)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 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
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	3.5	1	Basic

Soft Iron (Without Losses) (mat1)

1 In the **Model Builder** window, click **Soft Iron (Without Losses) (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	204e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.39	1	Young's modulus and Poisson's ratio
Density	rho	7840	kg/m³	Basic

In order to show the results during the computation, prepare some probes. The first two of these are point probes, which are placed in the air gaps between the electrical contact and the core, where they evaluate certain variables.

DEFINITIONS

Air gap, electrical contact

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type Air gap, electrical contact in the **Label** text field.

3 In the **Variable name** text field, type gap_p1.

4 Select Point 30 only.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.gap_p1`.

6 Select the **Description** checkbox. In the associated text field, type `Electrical contact air gap`.

7 Click to expand the **Table and Window Settings** section. Click ✛ **Add Plot Window**.

Air gap, central limb

1 Right-click **Air gap, electrical contact** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `Air gap, central limb` in the **Label** text field.

3 In the **Variable name** text field, type `gap_p2`.

4 Select Point 4 only.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.gap_p2`.

6 In the **Description** text field, type `Central limb air gap`.

Contact force, electrical contact

1 In the **Definitions** toolbar, click 🖉 **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Contact force, electrical contact` in the **Label** text field.

3 In the **Variable name** text field, type `T_toty_p1`.

4 Locate the **Expression** section. In the **Expression** text field, type
`solid.dcnt1.T_toty_p1*(solid.dcnt1.T_toty_p1<500[N])+500[N]*`
`(solid.dcnt1.T_toty_p1>500[N])`.

5 Select the **Description** checkbox. In the associated text field, type `Contact force, electrical contact`.

6 Click to expand the **Table and Window Settings** section. Click ✛ **Add Plot Window**.

Contact force, central limb

1 Right-click **Contact force, electrical contact** and choose **Duplicate**.

2 In the **Settings** window for **Global Variable Probe**, type `Contact force, central limb` in the **Label** text field.

3 In the **Variable name** text field, type `T_toty_p2`.

4 Locate the **Expression** section. In the **Description** text field, type `Contact force, central limb`.

5 In the **Expression** text field, type `solid.dcnt1.T_toty_p2*`
`(solid.dcnt1.T_toty_p2<500[N])+500[N]*(solid.dcnt1.T_toty_p2>500[N])`.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 Select Domain 18 only.

Total current, coil

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type I_coil in the **Variable name** text field.

3 In the **Label** text field, type Total current, coil.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields** > **Coil parameters** > **mf.ICoil_1 - Coil current - A**.

5 Locate the **Table and Window Settings** section. Click ─ **Add Plot Window**.

Total current, shading coil

1 Right-click **Total current, coil** and choose **Duplicate**.

2 In the **Settings** window for **Global Variable Probe**, type I_shading_coil in the **Variable name** text field.

3 In the **Label** text field, type Total current, shading coil.

4 Locate the **Expression** section. In the **Expression** text field, type intop1(abs(mf.Jz)).

5 Select the **Description** checkbox. In the associated text field, type Shading coil current (absolute value).

STUDY 1

Step 1: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **ms**.

4 In the **Output times** text field, type range(0,0.2,20) range(20.5,0.5,40).

5 Click to expand the **Study Extensions** section. Select the **Automatic remeshing** checkbox.

6 In the **Study** toolbar, click **Show Default Plots**.

RESULTS

Probe Plot Group 1

Set up a dataset to visualize the results for the full geometry. Since only half of the geometry was modeled due to the symmetry, a mirror operation is required to retrieve the full model. Also, create a view that can be used for the different plots.

Mirror 2D 1

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

2 Right-click **Results** > **Datasets** and choose **More 2D Datasets** > **Mirror 2D**.

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

4 From the **Dataset** list, choose **Study 1/Remeshed Solution 1 (sol2)**.

DEFINITIONS

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **View 1**.

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

3 Select the **Lock axis** checkbox.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 In the **x minimum** text field, type -90.

4 In the **x maximum** text field, type 90.

5 In the **y minimum** text field, type -20.

6 In the **y maximum** text field, type 160.

Also, set up the plots and the study so that the full contactor is shown when solving.

RESULTS

Magnetic Flux Density (mf)

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

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 1**.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Solver Configurations** node.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

4 From the **Method** list, choose **BDF**.

5 Select the **Initial step** checkbox. In the associated text field, type 0.05.

Step 1: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.

3 Select the **Plot** checkbox.

4 From the **Plot group** list, choose **Magnetic Flux Density (mf)**.

5 In the **Study** toolbar, click ▬ **Compute**.

RESULTS

Air Gap

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, type Air Gap in the **Label** text field.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Air Gap** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

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

4 In the table, enter the following settings:

Legends
Air gap, electrical contact (mm)
Air gap, central limb (mm)

Contact Force

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, type Contact Force in the **Label** text field.

Coil Current

I In the **Model Builder** window, under **Results** click **Probe Plot Group 3**.

2 In the **Settings** window for **ID Plot Group**, type Coil Current in the **Label** text field.

Calculate the total electromagnetic force acting on the moving part by adding a **Force Calculation** feature to the model. This feature is only used for postprocessing.

MAGNETIC FIELDS (MF)

Force Calculation, for Postprocessing

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

2 In the **Settings** window for **Force Calculation**, type Force Calculation, for Postprocessing in the **Label** text field.

3 Select Domain 5 only.

STUDY I

In the **Study** toolbar, click ↻ **Update Solution**.

RESULTS

ID Plot Group 7

In the **Results** toolbar, click ∿ **ID Plot Group**.

Global 1

Right-click **ID Plot Group 7** and choose **Global**.

Total Electromagnetic Force

I In the **Settings** window for **ID Plot Group**, type Total Electromagnetic Force in the **Label** text field.

2 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Remeshed Solution 1 (sol2)**.

3 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

I In the **Model Builder** window, click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
2*mf.Forcey_0	N	Total electromagnetic force, y-component

4 In the **Total Electromagnetic Force** toolbar, click ▣ **Plot**.

Contact Force, Electrical Contact

1 In the **Results** toolbar, click ∿ **ID Plot Group**.

2 In the **Settings** window for **ID Plot Group**, type Contact Force, Electrical Contact in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study I/ Remeshed Solution 1 (sol2)**.

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

5 In the **Times (ms)** text field, type range(15.55,0.5,40).

6 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Contact Force, Electrical Contact** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **T_toty_p1 - Contact force, electrical contact - N**.

3 In the **Contact Force, Electrical Contact** toolbar, click ▣ **Plot**.

Magnetic Flux Density (mf)

Finally, reproduce Figure 6 and Figure 7.

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

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

3 From the **Time (ms)** list, choose **38.5**.

4 In the **Magnetic Flux Density (mf)** toolbar, click ▣ **Plot**.

Zoom in on the shading coil on the right to reproduce Figure 7.

5 From the **Time (ms)** list, choose **30**.

6 In the **Magnetic Flux Density (mf)** toolbar, click ▣ **Plot**.

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

8 Click the ⊕ **Zoom In** button in the **Graphics** toolbar.

The plot should now look like Figure 6.