Created in COMSOL Multiphysics 5.5

Topology Optimization of a Magnetic Circuit

Introduction

This model presents an example of topology optimization of the magnetic circuit of a loudspeaker driver. Topology optimization is used to find the shape of a nonlinear iron yoke that maximizes its performance while minimizing the weight, obtaining a smaller and lighter design.

The geometry and simulation parameters are similar to the ones used in the Loudspeaker Driver application available in the Acoustic Module Application Library. More insight on the acoustics of that geometry can be found in the documentation relative to that model.

This model illustrates:

- the use of a nonlinear magnetic material specified by a provided B-H curve, representing a nonlinear relative magnetic permeability definition;
- the use of this permeability as variable field which is being optimized by a topology optimization process;
- the reconstruction of an optimized geometry from the topology optimization solution.

The final geometry features a loudspeaker with similar performance whose weight is 25% less than the original.

Model Definition

The magnetic circuit of a loudspeaker is used to concentrate the magnetic flux generated by a permanent magnet into an air gap in which a coil is placed, with the coil windings perpendicular to magnetic flux lines. The coil is mechanically connected to the membrane of the loudspeaker. When a current is run through the coil, electromagnetic forces act on it inducing movement. The movement of the coil is transferred to the membrane, which interacts with the air producing sound waves.

In a real device, the iron yoke that constitutes the magnetic circuit must be designed to maximize the flux concentrated at the coil, and to provide a uniform field across the coil. In order to estimate the performance of a loudspeaker magnetic circuit, it is useful to introduce the BL parameter, defined as the product of magnetic flux in the air gap B and coil length L. The larger this parameter, the higher the performance of the magnetic circuit.

If the magnetic circuit is axisymmetric and the coils are winded in the azimuthal direction inside the air gap, as in the present model, the BL parameter is defined as

$$BL = \frac{2\pi N_0}{A}\int rB_r dA$$

where B_r is the r-component of the magnetic flux density, N_0 is the number of turns of the coil, and A the cross-section area of the coil.

In the present model, the electromagnetic problem is solved using the **Magnetic Fields** interface. The iron yoke is made of a soft iron with a specified B-H curve, in order to take into account magnetic saturation. In the model, it is explicitly shown that using a given magnetic B-H curve is equivalent to using a relative permeability:

$$\mu_{r,\,BH}(|\mathbf{B}|) = \frac{H(|\mathbf{B}|)}{\mu_0|\mathbf{B}|}$$

Using the relative permeability instead of the B-H curve is useful for the purpose of topology optimization. The permeability of the yoke region is defined in terms of a control field p, with the formula:

$$\mu_r(p) = 1 - p^2 + p^2\mu_{r,\,BH}(|\mathbf{B}|)$$

The variable reduces to the permeability of air for $p = 0$ and the permeability of the nonlinear iron for $p = 1$. Topology optimization is performed by searching for the values of p in the interval $[0, 1]$ at each point of the yoke, while constraining the integral of p. The latter is proportional to the total weight of the yoke, since the field p indicates the regions consisting of iron.

In the model three domains are made of iron, and it is shown that optimization can be performed just on one of them. For this domain, the best configuration for p is found in order to have a volume of about 37 cm^3, which is the volume of the original geometry, or about 26 cm^3. By scanning the optimal shape as function of the weight, it is possible to derive the Pareto front for a optimization process which uses both the BL parameter and the weight in the objective.

Results and Discussion

The magnetic fields for the initial configuration are investigated in the first two analyses computed in Study 1. In this configuration, the lower arm of the yoke has a volume of about 37 cm^3. The first solution illustrates using the B-H curve functionality, readily available in the **Ampère's Law** feature in the **Magnetic Fields** interface, while the second solution uses the approach of specifying an equivalent nonlinear relative permeability. The results are identical in the two cases. The norm of the magnetic flux density and the magnetic flux lines for this initial configuration are shown in Figure 1 and Figure 2.

The lowest part of the magnetic circuit is enclosed in a cylindrical annulus whose volume is about 52 cm^3. Figure 3 illustrates the result of the optimization process, performed in Study 2. The plot shows the value of the control variable field p, with the constraint that the integral of p over the whole domain is the same as the initial configuration (about 37 cm^3). The computed geometry differs marginally from the original configuration indicated by a gray line, meaning that the original configuration was already nearly optimal. Figure 4 shows the result of the optimization process with the constraint of a volume of yoke equal to 26 cm^3.

The model also illustrates how to export the optimized shape as an interpolation function in a text file and import it as a geometry. The result is shown in Figure 5. The red line in Figure 5 represents the level set for p = 0.5. The line is explicitly added to the geometry, dividing the lower rectangle in two domains. For this final simulation, the H-B curve constitutive relation is used once again. The results are presented in Figure 6 and Figure 7.

A 3D representation of the final geometry and its magnetic fields is represented in Figure 8.

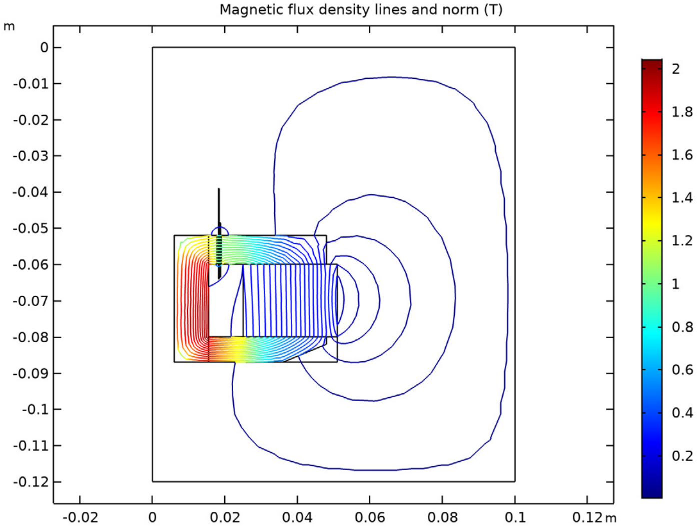

Figure 1: Magnetic flux lines for the initial configuration. The lower rectangle is divided in two domains: the left part, with volume of 37 cm^3*, is made of a nonlinear magnetic iron characterized by a B-H curve.*

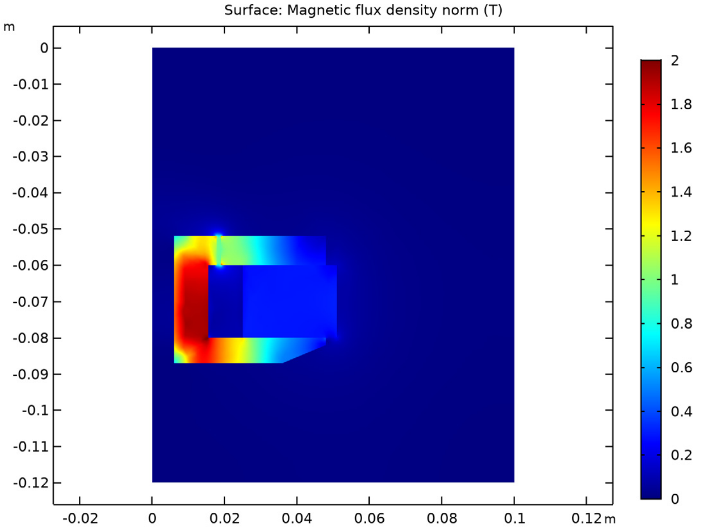

Figure 2: Magnetic flux density norm for the initial configuration.

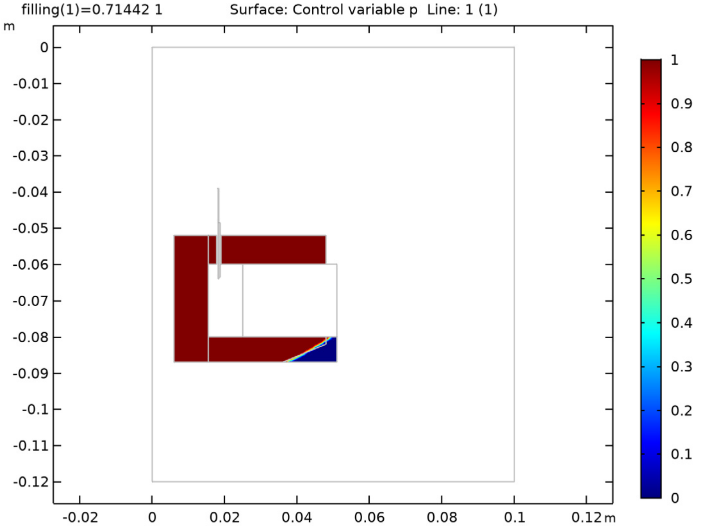

Figure 3: Results of the optimization with the constraint that the iron volume must be $37\ \mathrm{cm}^3$*. The regions with* $p = 1$ *(red) are made of iron, the region with* $p = 0$ *(blue) is air. The original shape is indicated by a solid gray line. The plot shows how the original configuration is nearly optimal.*

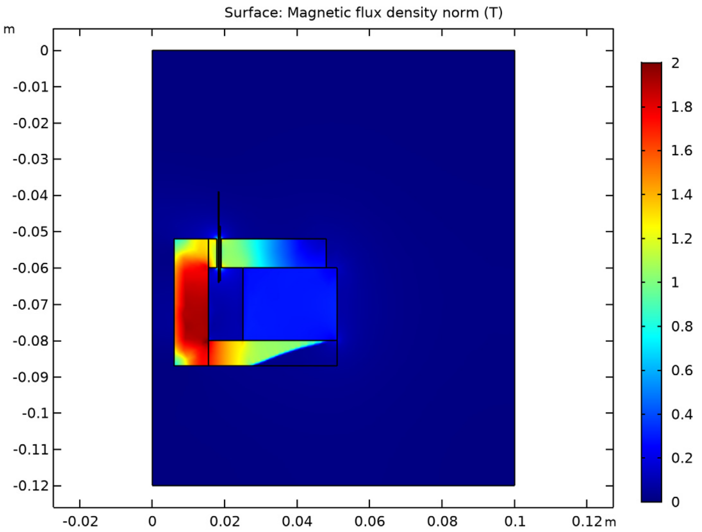

Figure 4: Magnetic flux density norm for the configuration with the constraint on volume of $26\ \mathrm{cm}^3$.

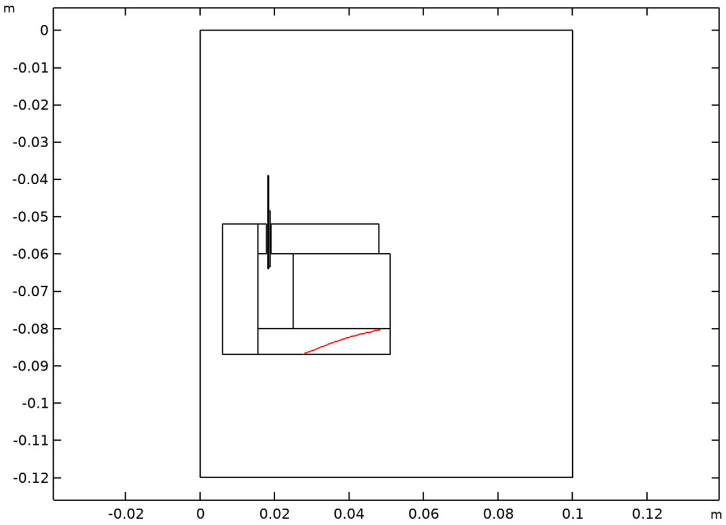

Figure 5: Contour line at $p = 0.5$ *(red line). This line is added in the geometry to divide the lower arm of the yoke in two domains.*

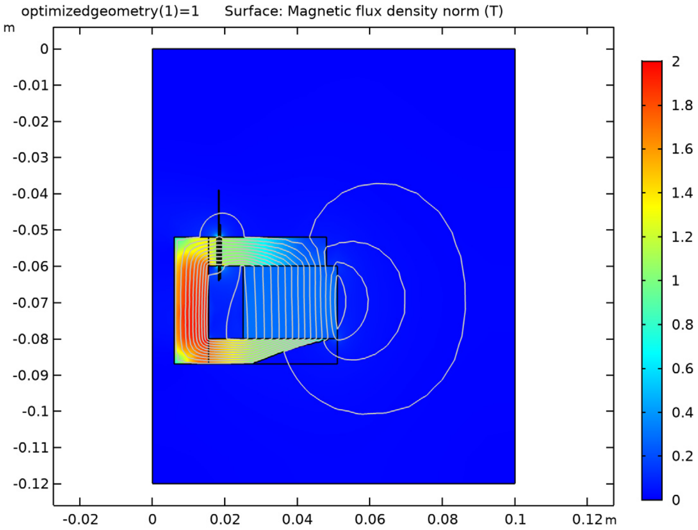

Figure 6: Magnetic flux density for norm for the geometry in Figure 5. The volume of the lower arm of the iron yoke (left of the division line) is $29\ \mathrm{cm}^3$*. The magnetic performance of this new lighter geometry is comparable with that of the configuration presented in Figure 1 and Figure 2.*

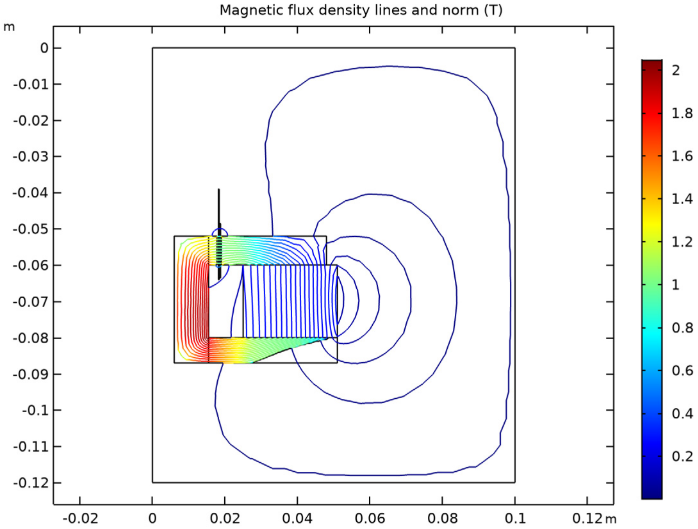

Figure 7: Magnetic flux lines colored proportionally to the magnetic flux density norm for the optimized configuration.

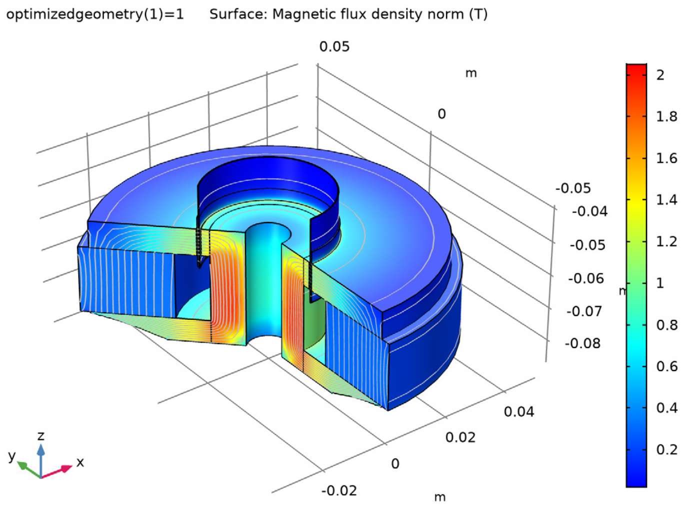

Figure 8: 3D revolution plot of the magnetic flux density norm in the final optimized geometry.

Notes About the COMSOL Implementation

USING SMART SELECTIONS

As the studies solve for different kinds of geometries, the entity naming and numbering differs between studies. In order to consistently address certain domains and boundaries for each study without having to manually reselect them, a smart use of selection features is implemented. This includes the selection of domains based on their coordinates or their adjacency to coordinate selected entities. The advantage of this approach is that for each of the five studies the corresponding geometry can be automatically generated with the proper selections for mesh and physics settings included. The state of the geometry is controlled by the parameters `originalgeometry` and `optimizedgeometry`. You can investigate the different states manually by setting these parameters to either 0 or 1 (and rebuilding the geometry). Optimization studies need the parameters to be set to 0.

USING COMPLEX SPLIT

Some studies include a relative permeability that is a function of the local magnetic flux density norm. Without solver adjustments, these studies would show poor nonlinear solver convergence. This happens as the magnetic flux density norm definition contains an operator that is not exactly differentiable; the `realdot()` operator[1]. In order to improve convergence, the solver is set to *split complex variables in real and imaginary parts*. This causes the variables to be reinterpreted so that the expression including the `realdot()` operator can be properly differentiated. Proper differentiated quantities result in an exact problem Jacobian. This is required for a good convergence rate. The *complex split* setting is needed in most gradient based electromagnetic optimization problems.

SMOOTHING THE OPTIMIZED PROFILE

The optimized geometry is obtained using a discontinuous element-wise constant shape function for the optimizing variable field `p`. The resulting material interface shows a saw-tooth profile reminiscent of the underlying mesh. This is good for topological optimization. When extracting the geometrical shape however, a smooth shape is preferred. In order to achieve this, the solution is mapped on continuous linear shape functions. After exporting the curve, the discontinuous constant elements are set back to their original value (the solution corresponding to the optimization study).

Application Library path: `ACDC_Module/Other_Industrial_Applications/magnetic_circuit_topology_optimization`

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 Axisymmetric**.
2. In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

---

[1] For more information on the realdot operator, see the COMSOL Multiphysics reference manual.

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. Specify the parameters of the loudspeaker driver magnetic circuit.

Name	Expression	Description
`B0`	`0.4[T]`	`Magnet remanent flux`
`N0`	`100`	`Number of turns`
`originalgeometry`	`1`	`Logical parameter which is controlling the drawing of original geometry`
`optimizedgeometry`	`0`	`Logical parameter which is controlling the importing of optimization generated geometry`

GEOMETRY 1

Draw the yoke, the magnet and the coil as a series of rectangles.

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node, then click **Geometry 1**.
2. Right-click **Geometry 1** and add eight **Rectangle** nodes.
3. Specify the sizes and position of the rectangles according to the following specification:

Name	Width [m]	Height [m]	xw [m]	yw [m]
Rectangle 1 (r1)	`2e-4`	`0.025`	`0.0182`	`-0.064`
Rectangle 2 (r2)	`6e-4`	`0.015`	`0.0182`	`-0.0635`
Rectangle 3 (r3)	`0.026`	`0.02`	`0.025`	`-0.08`
Rectangle 4 (r4)	`0.1`	`0.12`	`0`	`-0.12`
Rectangle 5 (r5)	`0.0023`	`0.008`	`0.0155`	`-0.06`
Rectangle 6 (r6)	`0.029`	`0.008`	`0.019`	`-0.06`
Rectangle 7 (r7)	`0.0355`	`0.007`	`0.0155`	`-0.087`
Rectangle 8 (r8)	`0.0095`	`0.035`	`0.006`	`-0.087`

Rectangle 1 (r1)

As different geometries are going to be built automatically from the study, introduce an "If" instance, which is building an element depending on the value of a parameter.

If 1 (if1)

1 In the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `originalgeometry`.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

r (m)	**z (m)**
`0.0155`	`-0.087`
`0.036`	`-0.087`
`0.048`	`-0.082`
`0.048`	`-0.08`
`0.0155`	`-0.08`

4 Click **Build Selected**.

Generate some selection which will be used during the modeling. Some of these are empty depending on the value of parameters.

DEFINITIONS

Disk 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Selections>Disk**.

3 In the **Settings** window for **Disk**, locate the **Disk Center** section.

4 In the **r** text field, type `0.03`.

5 In the **z** text field, type `-0.084`.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.

Disk 2

1 Right-click **Disk 1** and choose **Duplicate**.

2 In the **Settings** window for **Disk**, locate the **Disk Center** section.

3 In the **r** text field, type `0.05`.

Box 1

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

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **r minimum** text field, type `0.01`.

4 In the **r maximum** text field, type `0.06`.

5 In the **z minimum** text field, type `-0.09`.

6 In the **z maximum** text field, type `-0.07`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Difference 1

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

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

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **Box 1** in the **Selections to add** list.

5 Click **OK**.

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

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

8 In the **Add** dialog box, select **Disk 2** in the **Selections to subtract** list.

9 Click **OK**.

10 In the **Settings** window for **Difference**, type `Left domain` in the **Label** text field.

Left domain 1

1 Right-click **Left domain** and choose **Duplicate**.

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

3 Locate the **Input Entities** section. In the **Selections to subtract** list, select **Disk 2**.

4 Under **Selections to subtract**, click **Delete**.

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

6 In the **Add** dialog box, select **Disk 1** in the **Selections to subtract** list.

7 Click **OK**.

Difference 3

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

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

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

4 In the **Add** dialog box, select **Box 1** in the **Selections to add** list.

5 Click **OK**.

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

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

8 In the **Add** dialog box, in the **Selections to subtract** list, choose **Left domain** and **Right domain**.

9 Click **OK**.

Explicit 1

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

2 Select Domains 2, 4, and 9 only.

3 In the **Settings** window for **Explicit**, type `Fixed Yoke` in the **Label** text field.

Union 1

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

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

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

4 In the **Add** dialog box, in the **Selections to add** list, choose **Left domain** and **Fixed Yoke**.

5 Click **OK**.

Explicit 2

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

2 Select Domains 5–8 and 10 only.

3 In the **Settings** window for **Explicit**, type `Other solid parts` in the **Label** text field.

Union 2

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

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

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

4 In the **Add** dialog box, in the **Selections to add** list, choose **Optimizing region**, **BH relationship region**, and **Other solid parts**.

5 Click **OK**.

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

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 **Built-in>Air**.

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

MATERIALS

Air (mat1)

The magnetic circuit will be simulated using the BH curve constitutive relation and the magnet is set to use the Remanent Flux constitutive relation. Add materials in order to provide required quantities.

ADD MATERIAL

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

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

3 Click **Add to Component** in the window toolbar.

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

MATERIALS

Soft Iron (With Losses) (mat2)

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

2 From the **Selection** list, choose **BH relationship region**.

Material 3 (mat3)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Select Domain 10 only.

In order to have a faster convergence of the optimization process, use linear shape order elements for the discretization.

MAGNETIC FIELDS (MF)

Proceed to the physics setup.

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**.

Ampère’s Law 1

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

2 In the **Settings** window for **Ampère's Law**, type `Nonmagnetic domains` in the **Label** text field.

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**, type `BH curve domains` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **BH relationship region**.

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

Ampère's Law 3

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

2 In the **Settings** window for **Ampère's Law**, type `Permanent Magnet` in the **Label** text field.

3 Select Domain 10 only.

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

5 Specify the **e** vector as

0	r
0	phi
1	z

MATERIALS

Material 3 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 3 (mat3)**.

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

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Recoil permeability	murec_iso ; murecii = murec_iso, murecij = 0	1	1	Basic
Remanent flux density norm	normBr	B0	S/m	Basic

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	1	Remanent flux density
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	T	Remanent flux density

Mesh is created using parametric selections so to be independent of current geometry. Mapped mesh will be created in the following when optimization study will be setup.

MESH 1

Proceed to the mesh setup.

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 7, 10, 32, and 37 only.

Size 1

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

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

3 Click the **Custom** button.

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

5 In the associated text field, type `5e-4`.

Mapped 1

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

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

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

4 From the **Selection** list, choose **Optimizing region**.

Free Triangular 1

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

2 Click **Build All**.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Original geometry BH curve description` in the **Label** text field.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
originalgeometry (Logical parameter which is controlling the drawing of original geometry)	1	

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

RESULTS

The following instructions create a plot in which contour lines are used to visualize the magnetic flux density lines.

2D Plot Group 1

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Magnetic Flux Density, Direct Model` 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 `Magnetic flux density lines and norm (T)`.

Contour 1

1 Right-click **Magnetic Flux Density, Direct Model** and choose **Contour**.

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

3 In the **Expression** text field, type `Aphi*r`.

Color Expression 1

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

2 In the **Magnetic Flux Density, Direct Model** toolbar, click **Plot**.

Contour 1

Create a plot for the magnetic flux density norm.

2D Plot Group 2

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Magnetic Flux Density, Direct Model (original geometry)` in the **Label** text field.

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

Surface 1

1 Right-click **Magnetic Flux Density, Direct Model (original geometry)** and choose **Surface**.

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

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

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

5 In the **Magnetic Flux Density, Direct Model (original geometry)** toolbar, click **Plot**.

Compute the loudspeaker's BL according to definition presented in documentation.

Surface Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 Select Domain 8 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	**Unit**	**Description**
`mf.Br*NO*2*pi*r`	`Wb/m`	`BL factor`

5 Locate the **Integration Settings** section. Clear the **Compute volume integral** check box.

6 Click **New Table**.

ORIGINAL GEOMETRY BH CURVE DESCRIPTION

Solver Configurations

Proceed now to solving the same problem using a nonlinear relative permeability equivalent to the B-H curve. In order to preserve the solution just computed, create a copy of it using the **Copy Solution** functionality.

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>Stationary**.
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, expand the **Original geometry BH curve description>Solver Configurations** node, then click **Study 2**.
2. In the **Settings** window for **Study**, type `Original geometry nonlinear mur description` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

1. In the **Study** toolbar, click **Parametric Sweep**.
2. In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3. Click **Add**.
4. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
originalgeometry (Logical parameter which is controlling the drawing of original geometry)	1	

Import the nonlinear relative permeability as a function of B from an external file. The interpolation data is computed from the Soft Iron BH curve as $B/(\mu_0 H)$.

GLOBAL DEFINITIONS

Interpolation 1 (int1)

1. In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
2. In the **Settings** window for **Interpolation**, locate the **Definition** section.
3. In the **Function name** text field, type `murOfB`.
4. In the table, enter the following settings:

t	f(t)
0	1200
1	1200
1.1	820

t	f(t)
1.2	560
1.3	420
1.4	290
1.5	220
1.6	160
1.7	110
1.8	70
1.9	47
2	26
2.1	15
2.2	10
2.3	7
2.4	6

5 Locate the **Units** section. In the **Arguments** text field, type `T`.

6 In the **Function** text field, type `1`.

MAGNETIC FIELDS (MF)

Ampère's Law 4

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

2 In the **Settings** window for **Ampère's Law**, type `Manual BH relationship` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Box 1**.

4 Locate the **Constitutive Relation B-H** section. From the μ_r list, choose **User defined**. In the associated text field, type `murOfB(mf.normB)`.

Ampère's Law 5

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

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

3 From the **Selection** list, choose **Right domain**.

4 In the **Label** text field, type `Extra Air Domain`.

Adjust study 1 so that solution will not change because of the added nodes.

ORIGINAL GEOMETRY BH CURVE DESCRIPTION

Step 1: Stationary

1. In the **Model Builder** window, under **Original geometry BH curve description** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. Select the **Modify model configuration for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)> Magnetic Fields (mf)>Manual BH relationship**.
5. Click **Disable**.

Solve Study 2 and verify that the results are the same of the ones in the previous study. Before solving activate manual spit of complex variable to improve robustness of the nonlinear solution.

ORIGINAL GEOMETRY NONLINEAR MUR DESCRIPTION

Solution 4 (sol4)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Compile Equations: Stationary**.
3. In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.
4. Select the **Split complex variables in real and imaginary parts** check box.
5. In the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density, Direct Model

1. In the **Model Builder** window, under **Results** click **Magnetic Flux Density, Direct Model**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Original geometry nonlinear mur description/ Solution 4 (sol4)**.
4. In the **Magnetic Flux Density, Direct Model** toolbar, click **Plot**.
5. In the **Results** toolbar, click **Evaluate** and choose **Evaluate All**.

Derived Values

Compute the volume of the lower arm of the yoke, as a reference for the next steps.

Surface Integration 1

1. In the **Results** toolbar, click **More Derived Values** and choose **Integration> Surface Integration**.
2. Select Domain 3 only.
3. In the **Settings** window for **Surface Integration**, locate the **Expressions** section.
4. In the table, enter the following settings:

Expression	Unit	Description
`1`	`cm^3`	`Volume`

5. Click **New Table**.

TABLE

1. Go to the **Table** window.
2. Select Domains 3 and 11 only.
3. Right-click **Surface Integration 1** and choose **Evaluate>Table 2 - Surface Integration 1**.

   The lower arm of the yoke has a volume of about 37 cm^3, out of about 52 cm^3 of the bottom region.

Add the optimization.

In the following, the solver is set so to optimize the topology of the yoke in the lower region, while keeping the volume of the arm fixed. First change the parameters which is making the original geometry disappear.

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	Description
`originalgeometry`	`0`	`Logical parameter which is controlling the drawing of original geometry`

GEOMETRY 1

In the **Home** toolbar, click **Build All**.

Thanks to the added Parametric Sweep, Study 1 and 2 will continue using the original geometry, if solved again. Now add the Optimization Interface.

ADD PHYSICS

1. In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Mathematics>Optimization and Sensitivity>Optimization (opt)**.
4. Click **Add to Component 1** in the window toolbar.
5. In the **Model Builder** window, click **Component 1 (comp1)**.
6. In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

OPTIMIZATION (OPT)

Control Variable Field 1

1. In the **Physics** toolbar, click **Domains** and choose **Control Variable Field**.

   The control variable `p` is used as an indicator to determine if iron or air is present at a certain point in space.
2. Select Domain 3 only.
3. In the **Settings** window for **Control Variable Field**, locate the **Control Variable** section.
4. In the **Initial value** text field, type `1`.

   The control variable field `p` is a logical variable and does not correspond to a physical field, so it is not necessary to require it to be continuous across elements. Moreover, it is expected to be piecewise constant. Choose a suitable shape function and discretization order for this purpose.
5. Locate the **Discretization** section. From the **Shape function type** list, choose **Discontinuous Lagrange**.
6. Find the **Base geometry** subsection. From the **Element order** list, choose **Constant**.

Specify that `p` must be bounded between 0 and 1.

Control Variable Bounds 1

1. Right-click **Control Variable Field 1** and choose **Control Variable Bounds**.
2. In the **Settings** window for **Control Variable Bounds**, locate the **Bounds** section.
3. In the **Upper bound** text field, type `1`.

Integral Inequality Constraint 1

1 In the **Physics** toolbar, click **Domains** and choose **Integral Inequality Constraint**.

Constrain the total volume of the iron region to a given fraction of the available 51.916 cm^3 using an integral constraint. Fraction is specified via the parameter "filling".

2 Select Domain 3 only.

3 In the **Settings** window for **Integral Inequality Constraint**, locate the **Constraint** section.

4 In the p text field, type `p/51.916e-6`.

5 Locate the **Bounds** section. In the **Lower bound** text field, type `filling`.

6 In the **Upper bound** text field, type `filling`.

Set the value of filling to 37.090 cm^3.

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	Description
`filling`	`37.090[cm^3]/51.916[cm^3]`	`Filling percentage of optimized piece`

OPTIMIZATION (OPT)

Integral Objective 1

1 In the **Physics** toolbar, click **Domains** and choose **Integral Objective**.

Specify the objective function so that BL has the largest absolute value. Note that BL is negative.

2 Select Domain 8 only.

3 In the **Settings** window for **Integral Objective**, locate the **Objective** section.

4 In the q text field, type `mf.Br*N0/6.0E-6[m^2]`.

Define the variable `p` as 1 in all the domains where the optimization is not active.

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 **Geometric Entity Selection** section.

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

4 Select Domains 2, 4, and 9 only.

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

Name	Expression	Unit	Description
p	1		

Enter the permeability according to the formula Equation .

MAGNETIC FIELDS (MF)

Manual BH relationship

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Manual BH relationship**.

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

3 In the μ_r text field, type `1-p^2+p^2*murOfB(mf.normB)`.

ROOT

Proceed with the setup of the optimization study. Add a third **Stationary** study step, controlled by an **Optimization** step.

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>Stationary**.

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

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

STUDY 3

Step 1: Stationary

Specify again the value of filling so that this study will not change solution when filling is updated in parameters.

1. In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.
2. Select the **Auxiliary sweep** check box.
3. Click **Add**.
4. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
filling (Filling percentage of optimized piece)	`37.090[cm^3]/ 51.916[cm^3]`	1

5. In the **Model Builder** window, click **Study 3**.
6. In the **Settings** window for **Study**, type `Optimization with target 37[cm^3]` in the **Label** text field.

Optimization

1. In the **Study** toolbar, click **Optimization**.

   The typical optimization solver used for topology optimization is MMA.
2. In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
3. From the **Method** list, choose **MMA**.
4. In the **Model Builder** window, click **Optimization with target 37[cm^3]**.
5. In the **Settings** window for **Study**, locate the **Study Settings** section.
6. Clear the **Generate default plots** check box.

Solution 7 (sol7)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 7 (sol7)** node, then click **Compile Equations: Stationary**.
3. In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.
4. Select the **Split complex variables in real and imaginary parts** check box.
5. In the **Model Builder** window, expand the **Optimization with target 37[cm^3]>Solver Configurations>Solution 7 (sol7)>Optimization Solver 1>Stationary 1** node, then click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Constant (Newton)**.

8 In the **Damping factor** text field, type `0.5`.

RESULTS

2D Plot Group 3

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 From the **Dataset** list, choose **Optimization with target 37[cm^3]/Solution 7 (sol7)**.

4 In the **Label** text field, type `Topology Variable Distribution for Optimization with Target 37[cm^3]`.

Surface 1

1 Right-click **Topology Variable Distribution for Optimization with Target 37[cm^3]** and choose **Surface**.

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

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

OPTIMIZATION WITH TARGET 37[CM^3]

Optimization

1 In the **Model Builder** window, under **Optimization with target 37[cm^3]** click **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Topology Variable Distribution for Optimization with Target 37[cm^3]**.

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

RESULTS

Topology Variable Distribution for Optimization with Target 37[cm^3]

Add a gray line showing the original shape on top of the optimized shape.

Line 1

1 Right-click **Topology Variable Distribution for Optimization with Target 37[cm^3]** and choose **Line**.

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

3 From the **Dataset** list, choose **Original geometry BH curve description/ Parametric Solutions 1 (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

7 In the **Topology Variable Distribution for Optimization with Target 37[cm^3]** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Repeat the same operation for adding an optimization study. New study will search for an optimal speaker lower core geometry which is half in weight with respect the full available space.

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**	**Description**
`filling`	`0.5`	`Filling percentage of optimized piece`

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>Stationary**.

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

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

STUDY 4

Step 1: Stationary

1. In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
2. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
3. From the **Method** list, choose **Solution**.
4. From the **Study** list, choose **Optimization with target 37[cm^3], Stationary**.
5. In the **Model Builder** window, click **Study 4**.
6. In the **Settings** window for **Study**, type `Optimization with target filling=0.5` in the **Label** text field.

Optimization

1. In the **Study** toolbar, click **Optimization**.
2. In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
3. From the **Method** list, choose **MMA**.
4. In the **Model Builder** window, click **Optimization with target filling=0.5**.
5. In the **Settings** window for **Study**, locate the **Study Settings** section.
6. Clear the **Generate default plots** check box.

Solution 8 (sol8)

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

RESULTS

Topology Variable Distribution for Optimization with Target 37[cm^3] 1

1. In the **Model Builder** window, expand the **Solution 8 (sol8)** node.
2. Right-click **Topology Variable Distribution for Optimization with Target 37[cm^3]** and choose **Duplicate**.
3. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
4. From the **Dataset** list, choose **Optimization with target filling=0.5/Solution 8 (sol8)**.
5. In the **Label** text field, type `Topology Variable Distribution for Optimization with Target Filling 50%`.

OPTIMIZATION WITH TARGET FILLING=0.5

Optimization

1. In the **Model Builder** window, under **Optimization with target filling=0.5** click **Optimization**.
2. In the **Settings** window for **Optimization**, locate the **Output While Solving** section.
3. Select the **Plot** check box.
4. From the **Plot group** list, choose **Topology Variable Distribution for Optimization with Target Filling 50%**.
5. In the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density, Direct Model (original geometry) 1

1. In the **Model Builder** window, right-click **Magnetic Flux Density, Direct Model (original geometry)** and choose **Duplicate**.
2. In the **Settings** window for **2D Plot Group**, type `Magnetic Flux Density Norm for Optimization with Target Filling 50%` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Optimization with target filling=0.5/Solution 8 (sol8)**.
4. Locate the **Plot Settings** section. Select the **Plot dataset edges** check box.
5. In the **Magnetic Flux Density Norm for Optimization with Target Filling 50%** toolbar, click **Plot**.

Exporting the optimized profile, then use it to modify the geometry.

OPTIMIZATION (OPT)

Control Variable Field 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Optimization (opt)** click **Control Variable Field 1**.
2. In the **Settings** window for **Control Variable Field**, locate the **Discretization** section.
3. From the **Shape function type** list, choose **Lagrange**.
4. Find the **Base geometry** subsection. From the **Element order** list, choose **Linear**.

OPTIMIZATION WITH TARGET FILLING=0.5

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

OPTIMIZATION (OPT)

Control Variable Field 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Optimization (opt)** click **Control Variable Field 1**.
2. In the **Settings** window for **Control Variable Field**, locate the **Discretization** section.
3. From the **Shape function type** list, choose **Discontinuous Lagrange**.
4. Find the **Base geometry** subsection. From the **Element order** list, choose **Constant**.

RESULTS

Contour 1

1. In the **Results** toolbar, click **More Datasets** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Data** section.
3. From the **Dataset** list, choose **Optimization with target filling=0.5/Solution 8 (sol8)**.
4. Locate the **Expression** section. In the **Expression** text field, type `p`.
5. Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
6. In the **Levels** text field, type `0.5`.
7. Click **Plot**.

Data 1

1. Right-click **Contour 1** and choose **Add Data to Export**.
2. In the **Filename** text field, enter a temporary filename, or use the **Browse** button to choose one.
3. In the **Settings** window for **Data**, locate the **Output** section.
4. From the **Data format** list, choose **Sectionwise**.
5. From the **Space dimension** list, choose **2**.
6. From the **Geometry level** list, choose **Line**.
7. Click the **Export** button.

The data has been exported to a text file. Reimport it as an interpolation curve in the geometry. This interpolation will be put under the logical condition "optimizedgeometry" so that each study will work separately.

GEOMETRY 1

Else If 1 (elseif1)

1. In the **Geometry** toolbar, click **Programming** and choose **Add After Selected>Else If**.
2. In the **Settings** window for **Else If**, locate the **Else If** section.
3. In the **Condition** text field, type `optimizedgeometry`.
4. Click **Build Selected**.

Interpolation Curve 1 (ic1)

1. In the **Geometry** toolbar, click **More Primitives** and choose **Interpolation Curve**.
2. In the **Settings** window for **Interpolation Curve**, locate the **Interpolation Points** section.
3. From the **Data source** list, choose **File**.
4. In the **Filename** text field, enter the path to the temporary file just exported, or click **Browse** and navigate to it.
5. From the **Data format** list, choose **Sectionwise**.
6. In the **Relative tolerance** text field, type `1e-3`.

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>Stationary**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5

Step 1: Stationary

1. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
2. Select the **Modify model configuration for study step** check box.
3. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Magnetic Fields (mf)>Manual BH relationship**.
4. Click **Disable**.

Parametric Sweep

1. In the **Study** toolbar, click **Parametric Sweep**.
2. In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
optimizedgeometry (Logical parameter which is controlling the importing of optimization generated geometry)	1	

5 In the **Model Builder** window, click **Study 5**.

6 In the **Settings** window for **Study**, type `Direct solution built with surface extracted from optimization result` in the **Label** text field.

7 In the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf)

In the **Settings** window for **2D Plot Group**, type `Magnetic Flux Density for Optimization with Target Filling 50%` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results> Magnetic Flux Density for Optimization with Target Filling 50%** node, then click **Surface 1**.

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

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

4 In the **Minimum** text field, type `0`.

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

6 In the **Magnetic Flux Density for Optimization with Target Filling 50%** toolbar, click **Plot**.

Magnetic Flux Density, Direct Model

1 In the **Model Builder** window, click **Magnetic Flux Density, Direct Model**.

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

3 From the **Dataset** list, choose **Direct solution built with surface extracted from optimization result/Solution 9 (sol9)**.

4 In the **Magnetic Flux Density, Direct Model** toolbar, click **Plot**.

Selection

1. In the **Model Builder** window, right-click **Direct solution built with surface extracted from optimization result/ Parametric Solutions 3 (sol10)** 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 **Solid parts**.
5. Select the **Propagate to lower dimensions** check box.

Magnetic Flux Density Norm, Revolved Geometry (mf)

1. In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm, Revolved Geometry (mf)**.
2. In the **Settings** window for **3D Plot Group**, type `Revolved Representation of Magnetic Flux Density for Optimization with Target Filling 50%` in the **Label** text field.
3. Click the **Zoom Extents** button in the **Graphics** toolbar.
4. In the **Revolved Representation of Magnetic Flux Density for Optimization with Target Filling 50%** toolbar, click **Plot**.