Model created in COMSOL Multiphysics 6.4

# Vector Hysteresis Modeling

## *Introduction*

This model reproduces the TEAM (Testing Electromagnetic Analysis Method) problem 32, which aims to evaluate numerical methods for the simulation of anisotropic magnetic hysteresis. A hysteretic three-limbed laminated iron core is subject to a time-varying magnetic field generated by two coils. The Jiles–Atherton material model (available in the Magnetic Fields interface) is used to simulate the response of the material, reproducing published experimental and numerical data.

## *Model Definition*

The geometry of the simulated experimental setup of TEAM problem 32 is represented in Figure 1.

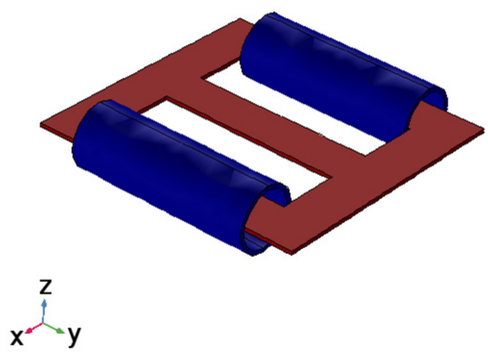



*Figure 1: The geometry of the device. The coils are colored in blue and the core in red.*

The system is composed of a three-limbed magnetic core with two feeding coils on the two outer limbs. A low excitation frequency (10 Hz) and a finite lamination of the frame prevent skin effects in the core. The frame is composed of 5 layers with a thickness of 0.48 mm.

The applied magnetic field is mainly oriented in the $xy$-plane; the material is anisotropic and react differently to fields applied along the $x$ or the $y$ direction. In the experimental setup there are a series of pick-up coils used to accurately probe the magnetic field; these coils are not included in the model as point measurements can easily be made by means of direct numerical evaluations.

Ref. 1 details four analysis cases that differ in the applied excitations. The case represented numerically in this model is the third one, in which the two coils are excited with an AC source with a peak value of 14.5 V and in quadrature phase. The coils have a total DC resistance of 11.42 Ω which includes an externally applied resistance. The field generated by this setup is strong enough to drive the material to saturation, while the phase shift creates a rotating field at the junction between the central limb and the frame.

In the literature, experimental results have been compared favorably with vector hysteresis models (Ref. 1 and 4). This model follows Ref. 4 in using the empirical Jiles-Atherton magnetic hysteresis model to simulate the core material. The values of the parameters for the empirical model are presented in Table 1. For an anisotropic material, the parameters are all diagonal matrices; the table reports the values on the diagonal.

TABLE 1: PARAMETER FOR THE JILES-ATHERTON MODEL.

| PARAMETER | SYMBOL | VALUES ON THE DIAGONAL |
|---|---|---|
| Saturation magnetization | `Ms` | `1.31e6 A/m, 1.33e6 A/m, 1.31e6 A/m` |
| Domain wall density | `a` | `233.78 A/m, 172.856 A/m, 233.78 A/m` |
| Pinning loss | `k` | `374.975 A/m, 232.652 A/m, 374.975 A/m` |
| Magnetization reversibility | `c` | `736e-3, 652e-3, 736e-3` |
| Inter-domain coupling | `alpha` | `562e-6, 417e-6, 562e-6` |

The Jiles-Atherton model is particularly suitable for AC feeding and requires only a limited number of parameters: `a` and `Ms` control the slope of the hysteretic B-H curve respectively at zero field and at saturation; `c` and `k` control the strength of the hysteretic effects — with the limit of no hysteresis for `c = 1` or for large `k`. The values presented in Table 1are taken from Ref. 1 and are obtained by fitting the model to experimental data.

## *Results and Discussion*

Figure 2 shows the magnetic flux at two different time instants, $t$ = 275 ms (top) and $t$ = 300 ms (bottom), at which the current in respectively the left and the right coil is at the peak value. The images show how the magnetic field rotates in the $xy$-plane at the junction between the central limb and the outer frame.

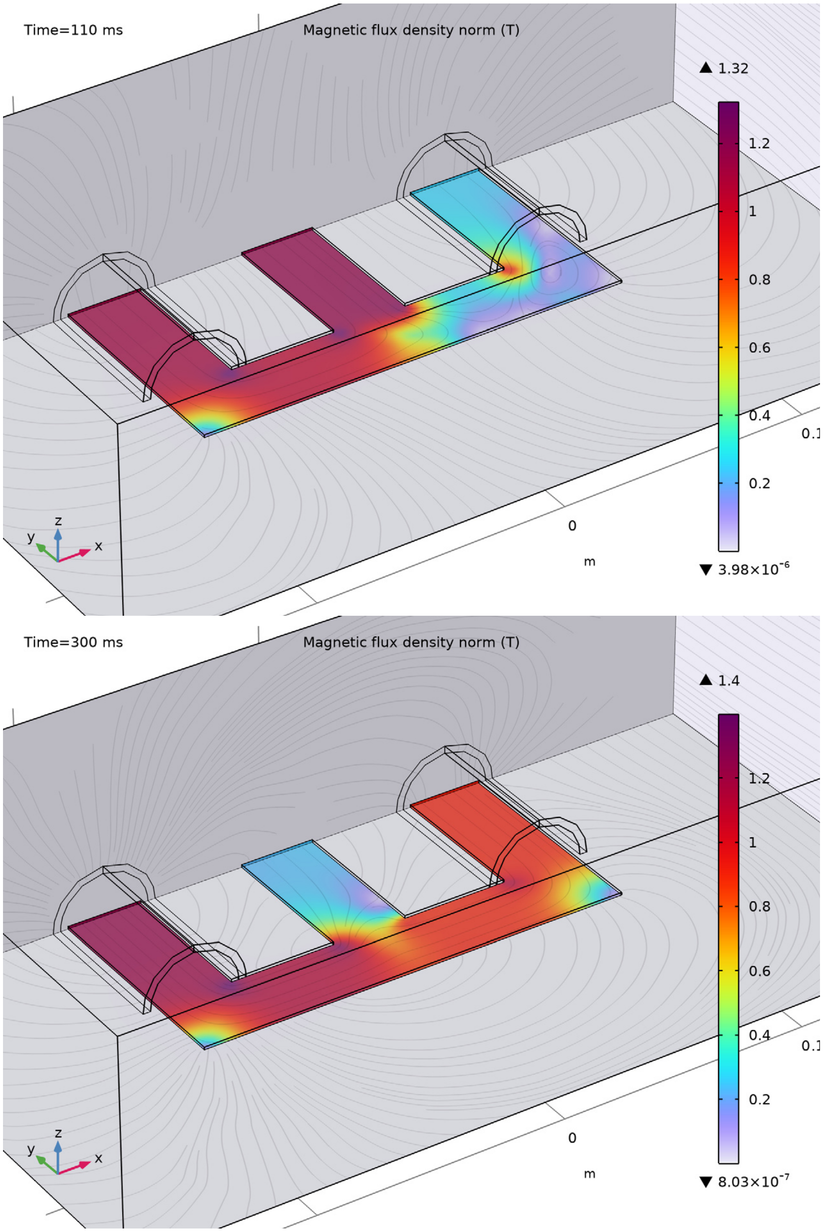



*Figure 2: Magnetic flux density at t = 275 ms (top) and t = 300 ms (bottom).*

The hysteretic behavior can be displayed by plotting the magnetic flux density as a function of the magnetic field during one AC cycle (corresponding to one hysteresis loop). Figure 3 shows the hysteresis loop obtained by averaging the quantities on a cross section of the central limb.

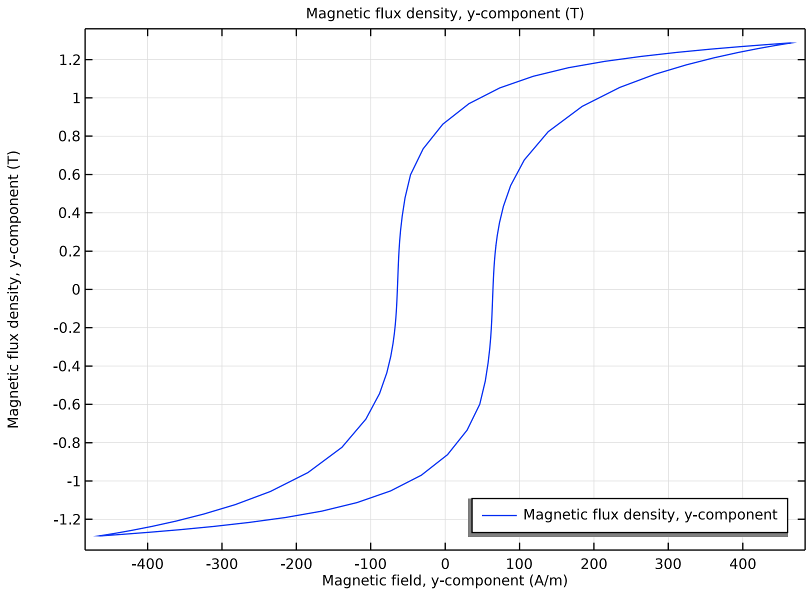



*Figure 3: Hysteresis (B-H) loop in the central limb.*

Finally, a representation of instantaneous magnetization field is shown in Figure 4. In those figures the red and blue vectors represent the instantaneous fields respectively at the time $t$ = 300 ms and $t$ = 275 ms, when the fields are expected to be in quadrature. The plots highlight how the fields at the junction are rotating in the *xy*-plane.

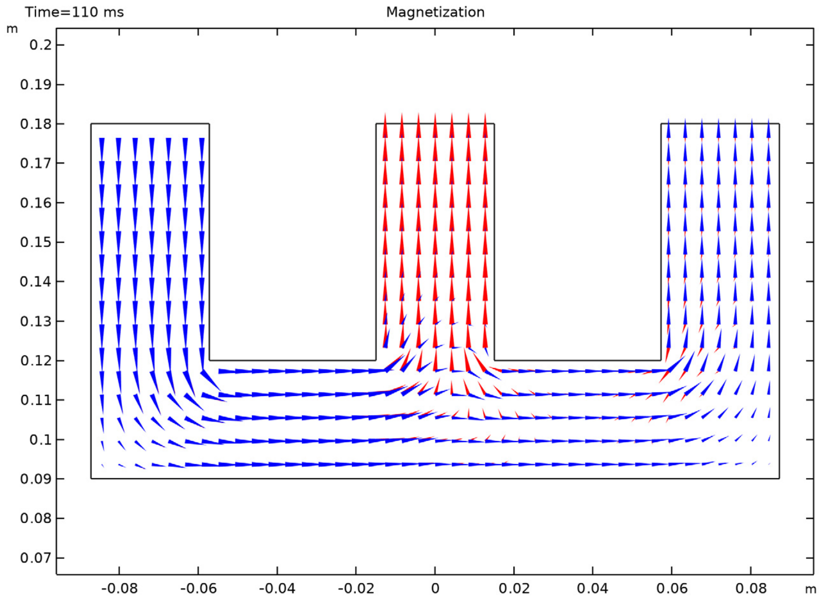



*Figure 4: Magnetization vector field at t = 275 ms (red) and t = 300 ms (blue).*

The spatial distribution of hysteresis losses can also be calculated, as illustrated in Figure 5. Example hysteresis loops at different locations are presented in Figure 6.

For further details, see the modeling instructions below.

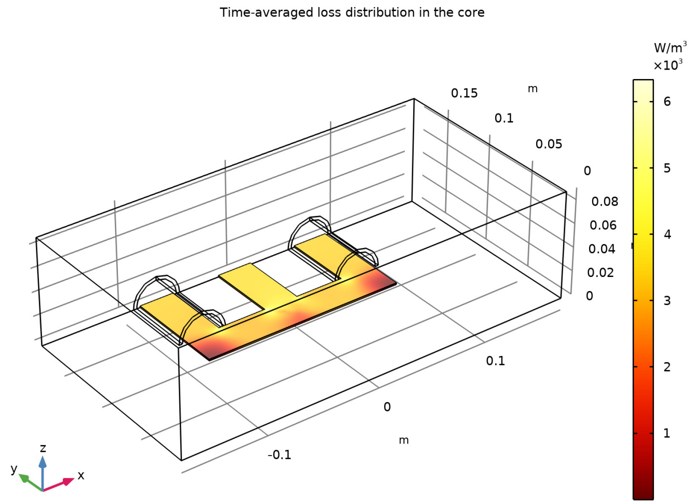



*Figure 5: Time-averaged loss distribution in the core.*

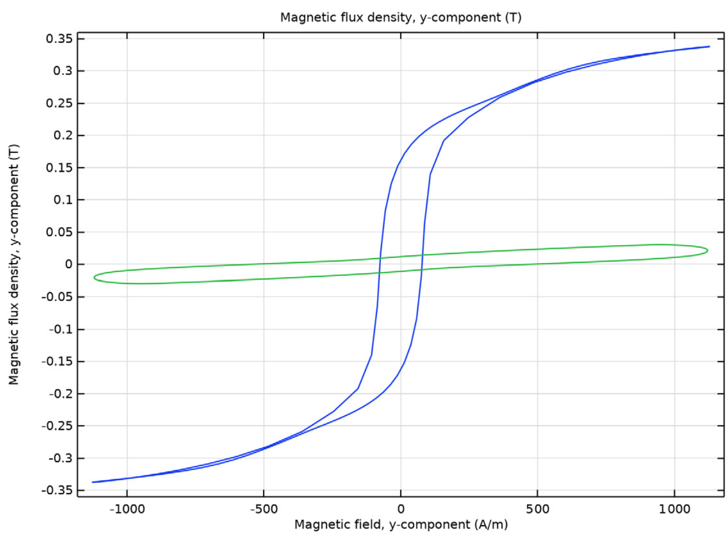



*Figure 6: Two different hysteresis loops for two different locations.*

## *Notes About the COMSOL Implementation*

The application uses the Jiles-Atherton hysteresis model available in the **Magnetic Fields** physics interface. The anisotropic material is constructed starting from the default isotropic Jiles-Atherton material (available in the **AC/DC Module** material library) and modifying the properties appropriately.

To obtain a good compromise between an accurate solution, robust convergence and efficient solving, the following settings are used:

- A direct solver (**PARDISO**) is used instead of the default iterative solver. In order to solve a **Magnetic Fields** problem with a direct solver it is necessary to apply the **Gauge Fixing for A-Field** feature.
- The discretization order for the magnetic vector potential A is set to use **Linear** elements. The discretization order of the Jiles-Atherton auxiliary dependent variables is then automatically set to zero.
- The scales of the dependent variables are set manually, in order to take advantage of the information on the maximum expected value of the magnetic field and the magnetization in the hysteretic material.

## *References*


1. www.compumag.org/wp/

2. http://www.cadema.polito.it/team32

3. A.J. Bergqvist, "A Simple Vector Generalization of the Jiles-Atherton Model of Hysteresis," *IEEE Transactions on Magnetics*, vol. 32, no. 5, p. 4213, 1996.

4. J.P.A. Bastos and N. Sadowski, *Magnetic Materials and 3D Finite Element Modeling*, CRC Press 2014.

5. S. Yan and J.-M. Jin, "Theoretical Formulation of a Time-Domain Finite Element Method for Nonlinear Magnetic Problems in Three Dimensions," *Progress In Electromagnetics Research*, vol. 153, pp. 33–55, 2015.


---

**Application Library path:** `ACDC_Module/Verifications/vector_hysteresis_modeling`

---

## *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 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Coil Geometry Analysis**.

6 Click **Done**.

### 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 |
|---|---|---|---|
| W | 174.5[mm] | 0.1745 m | Width of the core |
| H | 180[mm] | 0.18 m | Height of the core |
| w | 30[mm] | 0.03 m | Width of the central limb |
| h1 | H-2*w | 0.12 m | Height of the windows |
| w1 | (W-3*w)/2 | 0.04225 m | Width of the windows |
| Th | 5*0.48[mm] | 0.0024 m | Thickness of the core |
| f | 10[Hz] | 10 Hz | Feeding voltage frequency |
| R_coil | 11.42[ohm] | 11.42 Ω | Coil resistance |
| p | 1/f | 0.1 s | Time period |

## DEFINITIONS

### *Step 1 (step1)*

In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

### *Variables 1*

1 In the **Model Builder** window, 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 |
|---|---|---|---|
| Q_core | (mf.Hx*d(mf.Bx,t))+ (mf.Hy*d(mf.By,t))+ (mf.Hz*d(mf.Bz,t)) | W/m³ | Core hysteretic loss density |
| P_core | intop1((mf.Hx*d(mf.Bx, t))+(mf.Hy*d(mf.By, t))+(mf.Hz*d(mf.Bz, t))) | | Core hysteretic loss |
| P_coils | mf.ICoil_1^2* mf.RCoil_1 + mf.ICoil_2^2* mf.RCoil_2 | | Coil resistive losses |
| P_rem | intop2((mf.Hx*d(mf.Bx, t))+(mf.Hy*d(mf.By, t))+(mf.Hz*d(mf.Bz, t))) | | Remaining Power |
| P_input | mf.PCoil_1 + mf.PCoil_2 | | Input power |

### *Integration 1 (intop1)*

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

### *Step 1 (step1)*

1 In the **Model Builder** window, click **Step 1 (step1)**.

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

3 In the **Location** text field, type `0.5`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1`.

## GEOMETRY 1

### *Work Plane 1 (wp1)*

In the **Geometry** toolbar, click **Work Plane**.

*Work Plane 1 (wp1) > Plane Geometry*

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

*Work Plane 1 (wp1) > Rectangle 1 (r1)*

1 In the **Work Plane** toolbar, click **Rectangle**.

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

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

4 In the **Height** text field, type `H/2`.

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

6 In the **yw** text field, type `3*H/4`.

*Work Plane 1 (wp1) > Rectangle 2 (r2)*

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `w1`.

4 In the **Height** text field, type `h1/2`.

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

6 In the **xw** text field, type `-(w+w1)/2`.

7 In the **yw** text field, type `H-h1/4`.

*Work Plane 1 (wp1) > Rectangle 3 (r3)*

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `w1`.

4 In the **Height** text field, type `h1/2`.

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

6 In the **xw** text field, type `(w+w1)/2`.

7 In the **yw** text field, type `H-h1/4`.

*Work Plane 1 (wp1) > Difference 1 (dif1)*

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

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

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

5 Select the objects **r2** and **r3** only.

*Extrude 1 (ext1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

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

3 In the table, enter the following settings:

| Distances (m) |
|---|
| Th/2 |

*Cylinder 1 (cyl1)*

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

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

3 In the **Radius** text field, type w*0.7.

4 In the **Height** text field, type h1/2.

5 Locate the **Position** section. In the **x** text field, type -w-w1.

6 In the **y** text field, type H-h1/2.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

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

| Layer name | Thickness (m) |
|---|---|
| Layer 1 | 0.1*w |

*Delete Entities 1 (del1)*

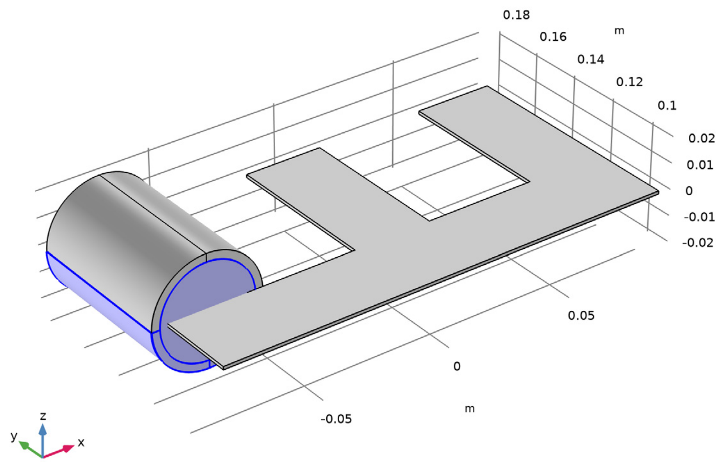
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

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

**5** On the object **cyl1**, select Domains 3–5 only.




*Copy 1 (copy1)*

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

**2** Select the object **del1** only.

**3** In the **Settings** window for **Copy**, locate the **Displacement** section.

**4** In the **x** text field, type `(w+w1)*2`.

*Block 1 (blk1)*

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

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

**3** In the **Width** text field, type `2*W`.

**4** In the **Depth** text field, type `H`.

**5** In the **Height** text field, type `3*w`.

**6** Locate the **Position** section. In the **x** text field, type `-W`.

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

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

**9** Click the **Wireframe Rendering** button in the **Graphics** toolbar to get a better view.

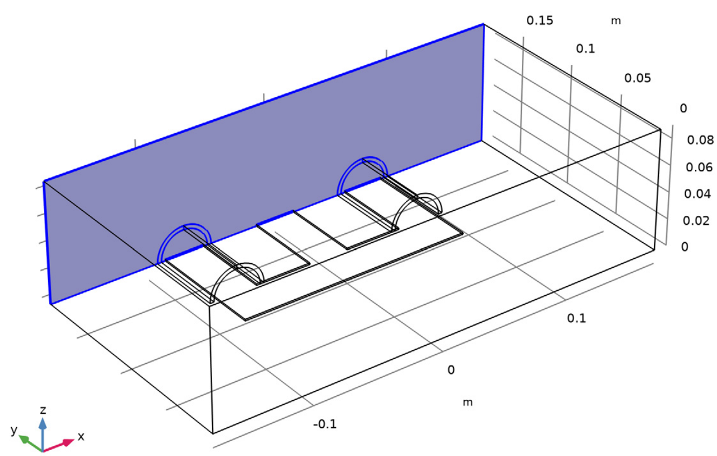
The volume around the components is modeled using the **Free Space** feature. This applies a small but nonzero conductivity in 3D **Magnetic Fields** simulations to obtain consistent equations.

### MAGNETIC FIELDS (MF)

Apply a **Symmetry Plane** on the antisymmetry cut to set the appropriate boundary condition (zero tangential magnetic field). The default **Magnetic Insulation** is the correct boundary condition for the symmetry cut boundaries (zero normal magnetic field).

*Symmetry Plane 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.
2 In the **Model Builder** window, click **Symmetry Plane 1**.
3 Select Boundaries 5, 9, 11, 16, 21, 30, 37, 42, 44, and 49 only.
4 In the **Settings** window for **Symmetry Plane**, locate the **Symmetry Plane** section.
5 From the **Symmetry type for the magnetic flux density** list, choose **Antisymmetry** (all the boundaries at $x = 0$).

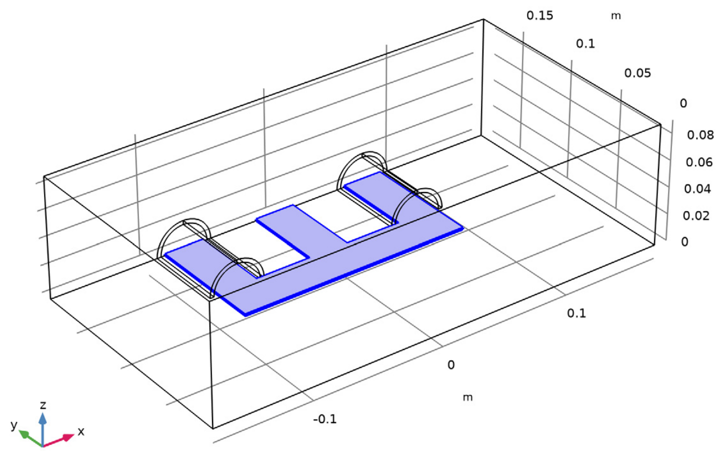



*Ampère's Law in Solids 1*

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

**2** Select Domain 3 only (the core).

It might be easier to select the correct domain by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)




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

**4** From the **Magnetization model** list, choose **Hysteresis Jiles–Atherton model**.

*Domain Coil 1*

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

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

**3** From the **Conductor model** list, choose **Homogenized multiturn**.

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

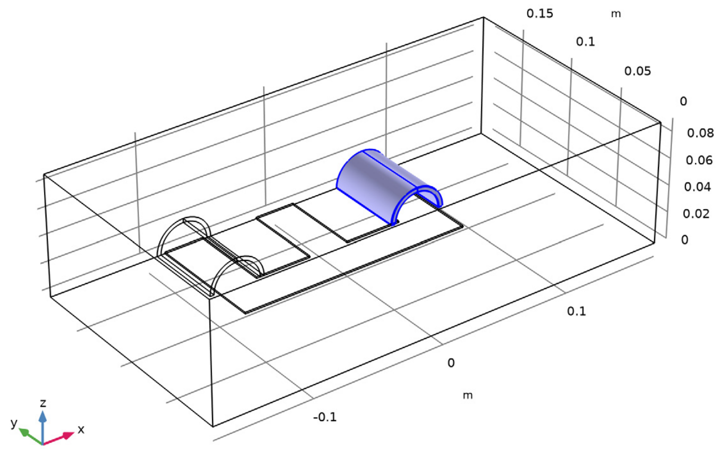
**5** In the $V_{\mathrm{coil}}$ text field, type `14.5[V]*sin(2*pi*f*t)*step1(f*t)`.

**6** Locate the **Homogenized Conductor** section. In the $N$ text field, type `90`.

**7** From the list, choose **From coil resistance**.

**8** In the $R_{\mathrm{coil}}$ text field, type `R_coil`.

**9** Select Domains 5 and 6 only.




*Geometry Analysis 1*

1. In the **Model Builder** window, click **Geometry Analysis 1**.
2. In the **Settings** window for **Geometry Analysis**, click to expand the **Symmetry Specification** section.
3. In the $F_L$ text field, type 2.
4. In the $F_A$ text field, type 2.

*Input 1*

1. In the **Model Builder** window, expand the **Geometry Analysis 1** node, then click **Input 1**.
2. Select Boundary 51 only.

*Geometry Analysis 1*

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

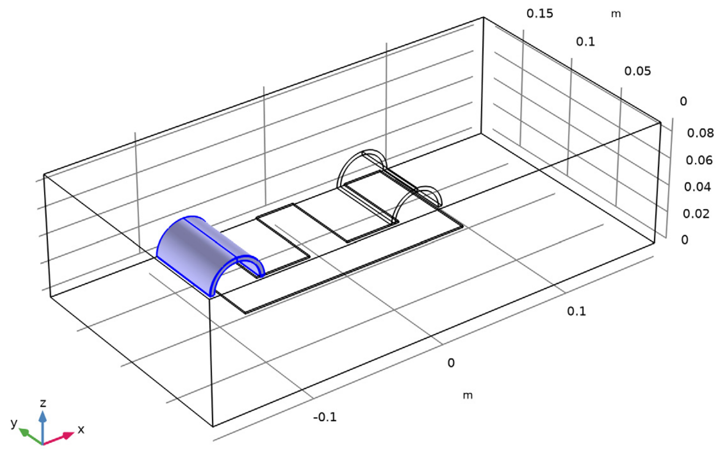
*Output 1*

1. In the **Physics** toolbar, click **Attributes** and choose **Output**.
2. Select Boundary 35 only.

*Domain Coil 2*

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

**2** Select Domains 2 and 4 only.




**3** In the **Settings** window for **Domain Coil**, locate the **Coil** section.

**4** From the **Conductor model** list, choose **Homogenized multiturn**.

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

**6** In the $V_{coil}$ text field, type `14.5[V]*cos(2*pi*f*t)*step1(f*t)`.

**7** Locate the **Homogenized Conductor** section. In the $N$ text field, type `90`.

**8** From the list, choose **From coil resistance**.

**9** In the $R_{coil}$ text field, type `R_coil`.

*Geometry Analysis 1*

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

**2** In the **Settings** window for **Geometry Analysis**, locate the **Symmetry Specification** section.

**3** In the $F_L$ text field, type `2`.

**4** In the $F_A$ text field, type `2`.

*Input 1*

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

**2** Select Boundary 26 only.

*Geometry Analysis 1*

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

*Output 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Output**.
2 Select Boundary 6 only.

Apply a **Gauge Fixing for A-Field** feature to improve the stability of the computation and in order to use a direct solver.

*Gauge Fixing for A-Field 1*

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

   Using lower order shape functions improves the robustness of the solution process for a nonlinear problem such as the Jiles–Atherton hysteresis model. Lowering the order also reduces the size of the problem, making it easier to solve with **Gauge Fixing** and a direct solver.

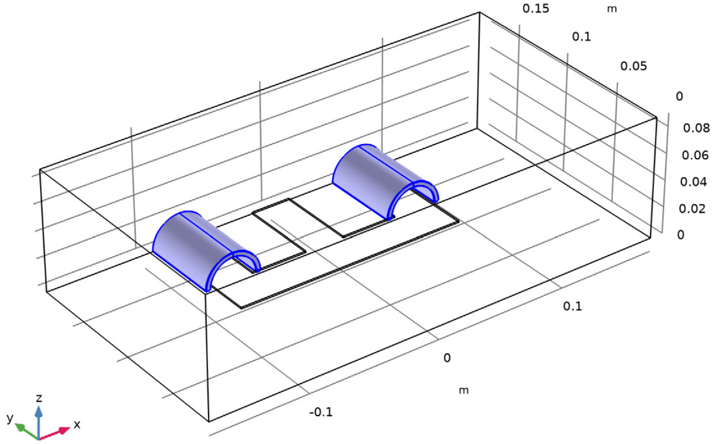
2 In the **Model Builder** window, click **Magnetic Fields (mf)**.
3 In the **Settings** window for **Magnetic Fields**, click to expand the **Discretization** section.
4 From the **Magnetic vector potential** list, choose **Linear**.

## MATERIALS

*Coil*

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

2 Select Domains 2 and 4–6 only.




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

4 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 |
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 1 | 1 | Basic |

5 In the **Label** text field, type `Coil`.

The following steps create the material for the Jiles–Atherton hysteresis model. First add the isotropic default material, then modify it to make it anisotropic.

### 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** > **Jiles–Atherton Hysteretic Material**.

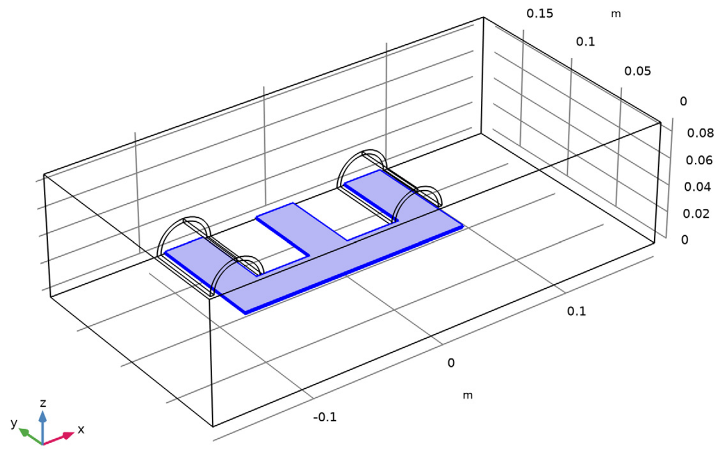
**4** Right-click and choose **Add to Component 1 (comp1)**.

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

### MATERIALS

*Jiles–Atherton Hysteretic Material (mat2)*

**1** Select Domain 3 only.




Start by specifying the basic material properties. Since the **Jiles–Atherton model** will be used for the magnetic behavior of the material it is not necessary to specify a magnetic permeability.

**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** |
|---|---|---|---|---|
| Electric conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 1 | S/m | Basic |
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 1 | 1 | Basic |

Now update the parameters specific to the Jiles–Atherton model. For each parameter in the **Output properties** table, perform the following steps:

**4** In the **Model Builder** window, expand the **Jiles–Atherton Hysteretic Material (mat2)** node, then click **Jiles–Atherton model parameters (ja)**.

**5** Click the corresponding row.

**6** Click the **Edit** button below the table.

**7** Choose **Diagonal** and enter the diagonal elements according to the following table:

| **Parameter** | **Values on the diagonal** |
|---|---|
| Saturation magnetization | `1.31e6[A/m], 1.33e6[A/m], 1.31e6[A/m]` |
| Domain wall density | `233.78[A/m], 172.856[A/m], 233.78[A/m]` |
| Pinning loss | `374.975[A/m], 232.652[A/m], 374.975[A/m]` |
| Magnetization reversibility | `736e-3, 652e-3, 736e-3` |
| Interdomain coupling | `562e-6, 417e-6, 562e-6` |

**8** Click **OK**.

*Jiles–Atherton Isotropic Hysteretic Material*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** right-click **Jiles–Atherton Hysteretic Material (mat2)** and choose **Rename**.

**2** In the **Rename Material** dialog, type `Jiles-Atherton Anisotropic Hysteretic Material` in the **New label** text field.

**3** Click **OK**.

## MESH 1

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

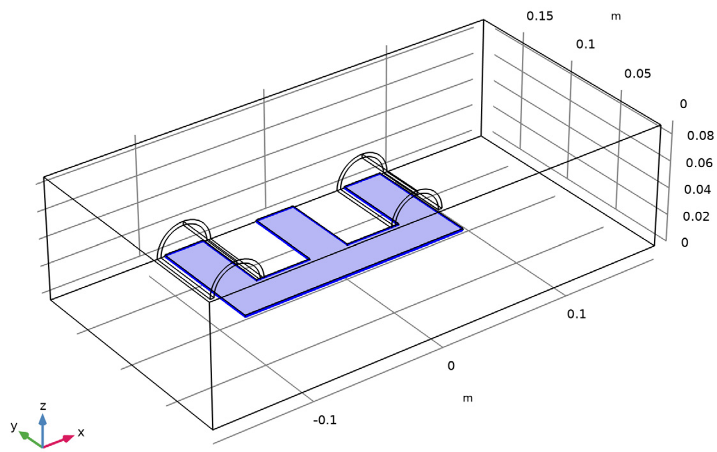
**2** In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

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

### *Free Triangular 1*

**1** In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

**2** Select Boundary 14 only.




### *Size 1*

**1** Right-click **Free Triangular 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.

**5** Select the **Maximum element size** checkbox. In the associated text field, type `w/10`.

**6** Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.3`.

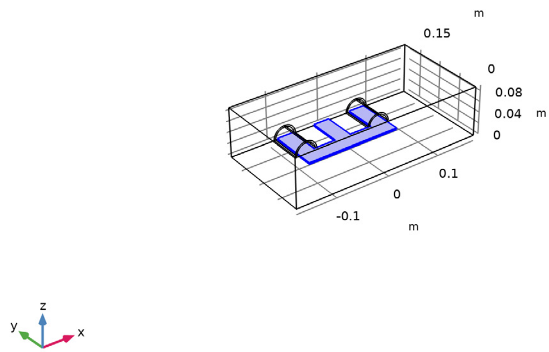
### *Swept 1*

**1** In the **Mesh** toolbar, click **Swept**.

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

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

**4** Select Domain 3 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 `1`.

*Free Tetrahedral 1*

**1** In the **Mesh** toolbar, click **Free Tetrahedral**.

**2** In the **Settings** window for **Free Tetrahedral**, click **Build All**.

## STUDY 1

*Step 2: Time Dependent*

**1** In the **Study** toolbar, click **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(0,1,300)`.

To improve the robustness and the performance of the solution, generate the default solvers and adjust some settings.

*Solution 1 (sol1)*

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

Set a manual scaling for the magnetic vector potential and the internal states used in the Jiles–Atherton model (magnetization and magnetic field). An appropriate value would be the maximum expected value for these quantities.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** node, then click **Magnetic Vector Potential (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 `5e-3`.

Similarly set **Scaling** to **Manual** for the other variables with the **Scaling** set according to the following table.

7 In the table, enter the following settings:

| Dependent variable | Scale |
|---|---|
| Magnetic Field | 1e4 |
| Magnetization | 1e6 |
| Divergence condition variable | 1 |
| Both Coil current | 1 |

## DEFINITIONS

*Integration 1 (intop1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Integration 1 (intop1)**.

2 Select Domain 3 only.

*Integration 2 (intop2)*

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

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

3 From the **Selection** list, choose **All domains**.

4 In the list box, select **3**.

5 Click **Remove from Selection**.

6 Select Domains 1, 2, and 4–6 only.

## STUDY 1

### *Solution 1 (sol1)*

1 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Fully Coupled 1**.

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

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

4 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Direct**.

5 In the **Settings** window for **Direct**, locate the **General** section.

6 From the **Solver** list, choose **PARDISO**.

7 In the **Home** toolbar, click **= Compute**.

## RESULTS

### *Magnetic Flux Density (mf)*

When the computation is completed, the default plot is generated and shown. Follow these steps to replicate Figure 2.

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

2 From the **View** list, choose **New view**.

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

This generates a dedicated view for the plot group. Next set it up to zoom in on the magnet.

4 Click **Go to Source**.

### *View 3D 3*

1 In the **Model Builder** window, under **Results** > **Views** click **View 3D 3**.

2 Use the mouse buttons to zoom in and pan to get a closer view of the magnet.

3 In the **Settings** window for **View 3D**, locate the **View** section.

4 Select the **Lock camera** checkbox.

### *Multislice 1*

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

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

3 Find the **x-planes** subsection. In the **Coordinates** text field, type `W`.

**4** Find the **y-planes** subsection. In the **Coordinates** text field, type `H`.

**5** Find the **z-planes** subsection. In the **Coordinates** text field, type `0`.

### *Streamline Multislice 1*

**1** In the **Model Builder** window, click **Streamline Multislice 1**.

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

**3** Find the **x-planes** subsection. In the **Coordinates** text field, type `W`.

**4** Find the **y-planes** subsection. In the **Coordinates** text field, type `H`.

**5** Find the **z-planes** subsection. In the **Coordinates** text field, type `0`.

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

### *Magnetic Flux Density (mf)*

Use the **Time** list to visualize the results at different times. Select 275 ms and 300 ms to reproduce Figure 2.

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

**2** Click **Plot**.

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

**4** From the **Time (ms)** list, choose **110**.

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

Create some auxiliary datasets to use in the other plots.

### *Cut Point 3D 1*

**1** In the **Results** toolbar, click **Cut Point 3D**.

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

**3** In the **x** text field, type `0`.

**4** In the **y** text field, type `H-61.5[mm]`.

**5** In the **z** text field, type `0`.

### *Average 1*

In the **Results** toolbar, click **More Datasets** and choose **Evaluation** > **Average**.

### *Selection*

**1** In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

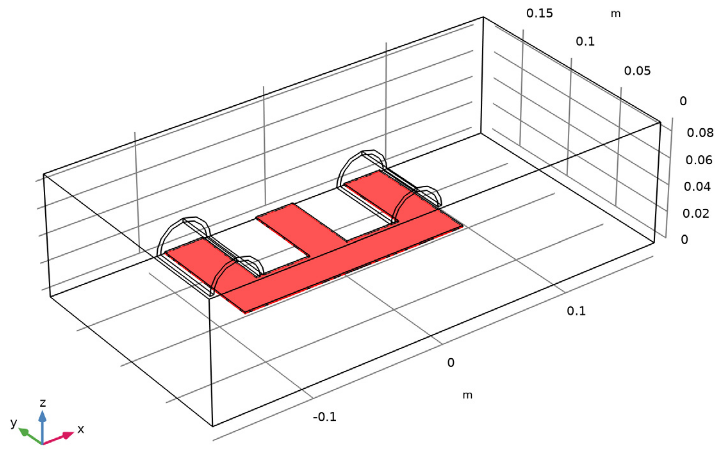
**3** From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundary 30 only.

*Surface 1*

**1** In the **Results** toolbar, click **More Datasets** and choose **Surface**.

**2** Select Boundary 14 only.




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

**4** From the **x- and y-axes** list, choose **xy-plane**.

*1D Plot Group 2*

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

**2** In the **Settings** window for **1D Plot Group**, locate the **Data** section.

**3** From the **Dataset** list, choose **Cut Point 3D 1**.

*Point Graph 1*

**1** Right-click **1D Plot Group 2** and choose **Point Graph**.

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

**3** In the **Expression** text field, type `mf.By`.

**4** Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

**5** In the **Expression** text field, type `mf.Bx`.

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

*Rotating Field*

**1** In the **Model Builder** window, click **1D Plot Group 2**.

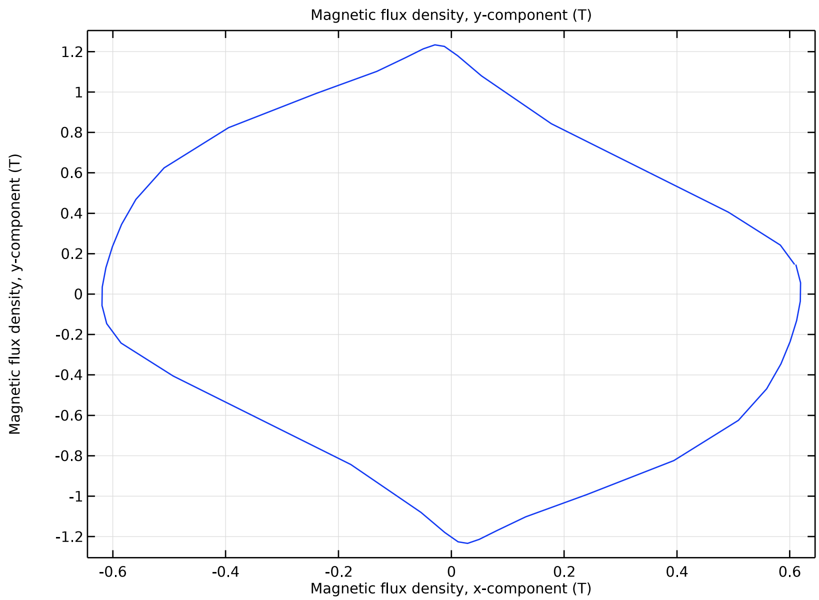
**2** In the **Settings** window for **1D Plot Group**, locate the **Data** section.

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

**4** In the **Times (ms)** text field, type `range(200,2.5,300)`.

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

**6** In the **Label** text field, type `Rotating Field`.




### *1D Plot Group 3*

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

### *Global 1*

**1** Right-click **1D Plot Group 3** and choose **Global**.

Plot the current flowing in the first coil.

**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)** > **Magnetic Fields** > **Coil parameters** > **mf.ICoil_1 - Coil current - A**.

### *Coil Current*

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

**2** In the **Settings** window for **1D Plot Group**, locate the **Data** section.

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

**4** Click **Range**.

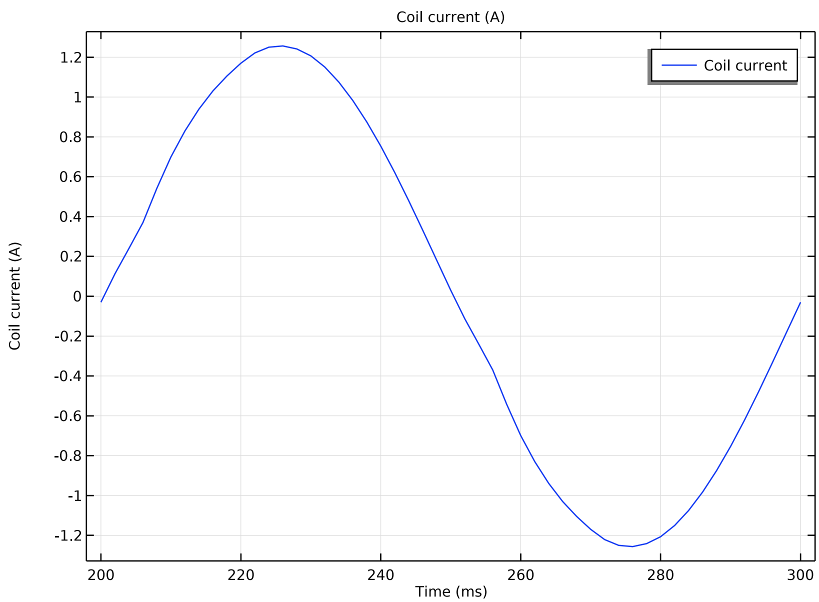
**5** In the **Range** dialog, type 200 in the **Start** text field.

**6** In the **Step** text field, type 2.

**7** In the **Stop** text field, type 300.

**8** Click **Replace**.

**9** In the **1D Plot Group 3** toolbar, click **Plot**.




**10** In the **Settings** window for **1D Plot Group**, type `Coil Current` in the **Label** text field.

*Electric Current Harmonic Pollution*

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

**2** In the **Settings** window for **1D Plot Group**, type `Electric Current Harmonic Pollution` in the **Label** text field.

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

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

*Global 1*

**1** Right-click **Electric Current Harmonic Pollution** and choose **Global**.

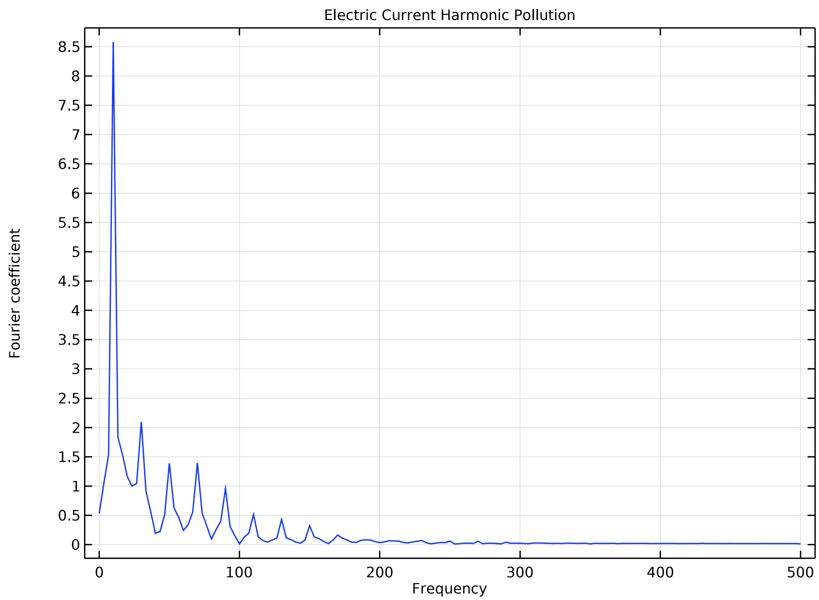
In the **Expression** enter `mf.ICoil_1-mf.VCoil_1/mf.RCoil_1`.

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

3 From the **Parameter** list, choose **Discrete Fourier transform**.

4 From the **Show** list, choose **Frequency spectrum**.

5 In the **Electric Current Harmonic Pollution** toolbar, click **Plot**.




*1D Plot Group 5*

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

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

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

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

6 Click **Range**.

7 In the **Range** dialog, type 200 in the **Start** text field.

8 In the **Step** text field, type 1.

9 In the **Stop** text field, type 300.

10 Click **Replace**.

### Global 1

1 Right-click **1D Plot Group 5** and choose **Global**.

Enter the quantities to be averaged on the boundary.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `mf.By` | `T` | `Magnetic flux density, y-component` |

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `mf.Hy`.

6 In the **1D Plot Group 5** toolbar, click **Plot**.

### Hysteresis

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

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

This reproduces Figure 3.

### 2D Plot Group 6

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

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

3 From the **Time (ms)** list, choose **110**.

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

### Arrow Surface 1

1 Right-click **2D Plot Group 6** and choose **Arrow Surface**.

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

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

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

5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `41`.

6 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

### Arrow Surface 2

1 Right-click **Arrow Surface 1** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `7e-9`.

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

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

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

### *Arrow Surface 1*

1 In the **Model Builder** window, click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `7e-9`.

4 In the **2D Plot Group 6** toolbar, click **Plot**.

### *Magnetization*

1 In the **Model Builder** window, under **Results** click **2D Plot Group 6**.

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

This reproduces Figure 4. Plot the magnetic field as a vector field to visualize the rotation of the field at the junction.

### *Magnetic Field*

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

2 Right-click **Magnetization 1** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog, type `Magnetic Field` in the **New label** text field.

4 Click **OK**.

### *Arrow Surface 1*

1 In the **Model Builder** window, expand the **Magnetic Field** node, then click **Arrow Surface 1**.

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

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

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

5 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `2e-5`.

### *Arrow Surface 2*

1 In the **Model Builder** window, click **Arrow Surface 2**.

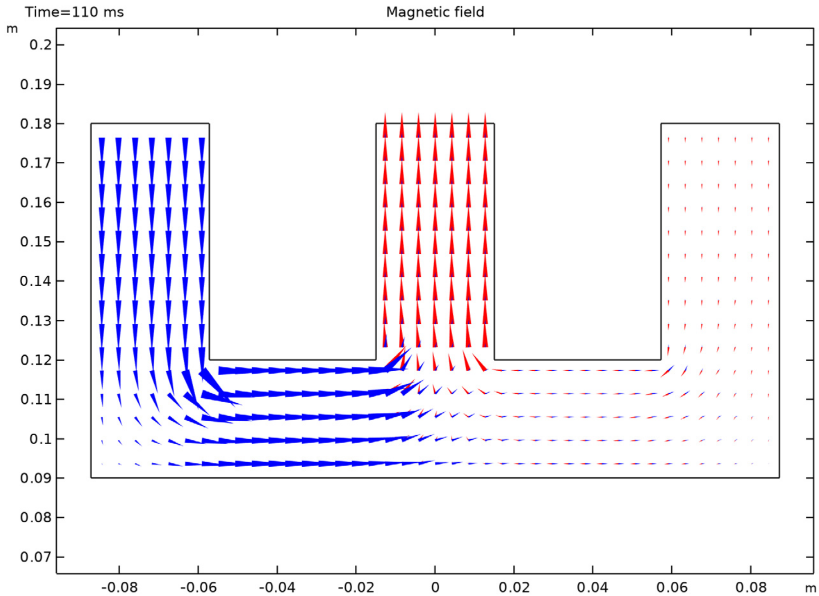
2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

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

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

5 Locate the **Coloring and Style** section. In the **Scale factor** text field, type `2e-5`.

**6** In the **Magnetic Field** toolbar, click **Plot**.




*Averaged Losses over the Last Period*

**1** In the **Results** toolbar, click **3D Plot Group**.

**2** In the **Settings** window for **3D Plot Group**, type `Averaged Losses over the Last Period` in the **Label** text field.

*Volume 1*

**1** Right-click **Averaged Losses over the Last Period** and choose **Volume**.

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

**3** In the **Expression** text field, type `timeavg(2*p,3*p,Q_core)`.

**4** Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

*Selection 1*

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

*Averaged Losses over the Last Period*

**1** In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

**2** From the **Title type** list, choose **Manual**.

**3** In the **Title** text area, type `Time-averaged loss distribution in the core`.

**4** Locate the **Color Legend** section. Select the **Show units** checkbox.

**5** Locate the **Title** section. Clear the **Parameter indicator** text field.

**6** In the **Averaged Losses over the Last Period** toolbar, click **Plot**.

*Selection 1*

**1** In the **Model Builder** window, under **Results > Averaged Losses over the Last Period > Volume 1** click **Selection 1**.

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

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

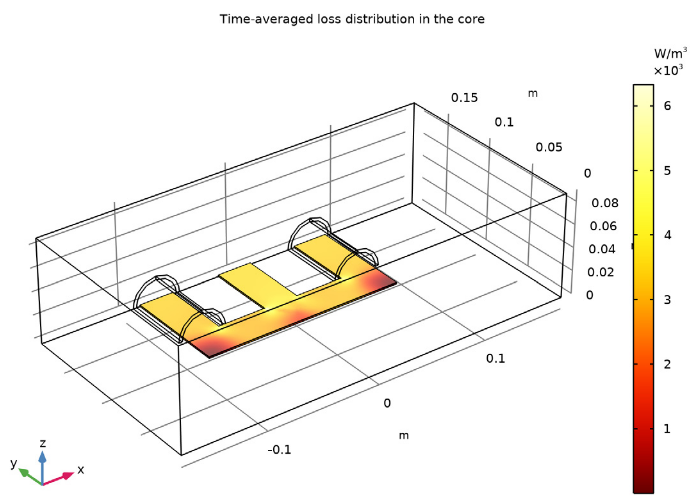
**4** From the **Selection** list, choose **All domains**.

**5** In the list box, select **1**.

**6** Click **Remove from Selection**.

**7** Select Domain 3 only.

**8** In the **Averaged Losses over the Last Period** toolbar, click **Plot**.




*Two Different Hysteresis Loops for Two Different Locations*

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

**2** In the **Settings** window for **1D Plot Group**, type `Two Different Hysteresis Loops for Two Different Locations` in the **Label** text field.

**3** Locate the **Data** section. From the **Time selection** list, choose **Manual**.

**4** In the **Time indices (1-301)** text field, type `range(201,1,301)`.

*Point Graph 1*

1 Right-click **Two Different Hysteresis Loops for Two Different Locations** and choose **Point Graph**.

2 In the **Two Different Hysteresis Loops for Two Different Locations** toolbar, click **Plot**.

3 In the **Model Builder** window, click **Point Graph 1**.

4 In the **Settings** window for **Point Graph**, locate the **Selection** section.

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

6 Select Points 32 and 50 only.

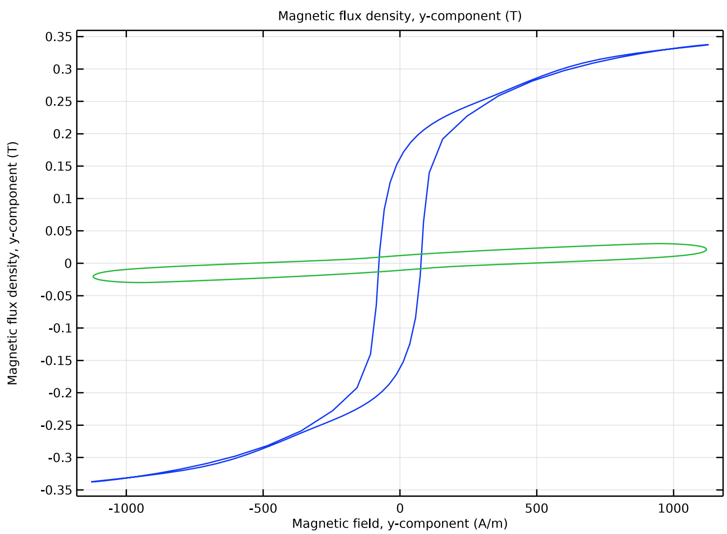
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

8 Locate the **y-Axis Data** section. In the **Expression** text field, type `mf.By`.

9 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

10 In the **Expression** text field, type `mf.Hy`.

11 Click to expand the **Legends** section. In the **Two Different Hysteresis Loops for Two Different Locations** toolbar, click **Plot**.




*Evaluation Group 1*

In the **Results** toolbar, click **Evaluation Group**.

Finally, let us check whether energy is conserved in our simulation. The input power should equal the output power, meaning that the sum of hysteresis losses in the core, resistive losses, and the remaining power must match the input power.

*Global Evaluation 1*

1 Right-click **Evaluation Group 1** and choose **Global Evaluation**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

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

5 In the **Time indices (1-301)** text field, type `range(201,1,301)`.

6 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| `100*(P_input-P_core-P_coils-P_rem)/P_input` | `1` | `Power Error in %` |

7 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

You should find that the average power balance error is about 0.6%. In theory, if everything were modeled perfectly and computed exactly, the error would be zero. In practice, however, it is non-zero due to factors such as numerical approximations from coarse meshing and time-stepping.