Created in COMSOL Multiphysics 6.2

Axisymmetric Approximation of 3D Inductor

Introduction

When the frequency is high enough, capacitative effects can become important also for devices that are inductive and/or resistive at lower frequencies. Modeling this effect in an inductor requires accounting for electric field components both parallel and perpendicular to the wire. This easily leads to the conclusion that a 3D model is always necessary even if the coil is a low slant helix. This is not always the case, as shown in this tutorial.

Starting from the 3D inductor model described in the *Introduction to AC/DC Module* manual, a 2D axisymmetric model able to describe a self resonating inductor is created. In order to build an equivalent 2D axisymmetric model, an effective axisymmetric core is drawn and the RLC Coil Group feature is used.

The method shown here is particularly suitable for studying systems with thousands of turns, such as sensors or transformers, with limited computational power.

Model Definition

The 3D solution of the power inductor of Figure 1 is presented in the manual *Introduction to AC/DC Module*. We are considering the same system.

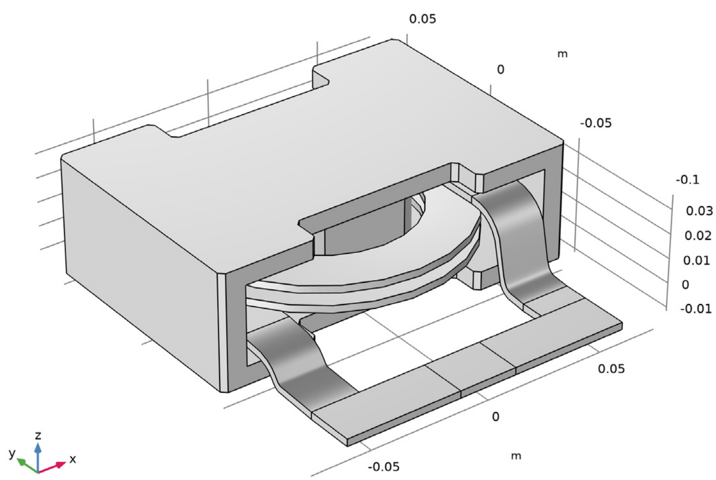


Figure 1: The 3D geometry representation of the power inductor under study.

In that device, the coil can to a reasonable level of approximation be treated as axisymmetric. Therefore, any of its cross-sections are suitable for performing a 2D axisymmetric analysis from a geometric point of view.

On the other hand, the core geometry shape indicates that a 3D analysis is required. However, in order to have a description which is correct from a magnetic point of view, the core does not have to be rotationally symmetric as long as:

- it is not supporting 3D nonaxisymmetric eddy currents
- an equivalent axisymmetric geometry respecting the original core reluctance is used

Such a construction, that is, replacing the 3D core with a 2D axisymmetric representation that preserves the crucial area that influences the magnetic circuit, is shown in Figure 2.

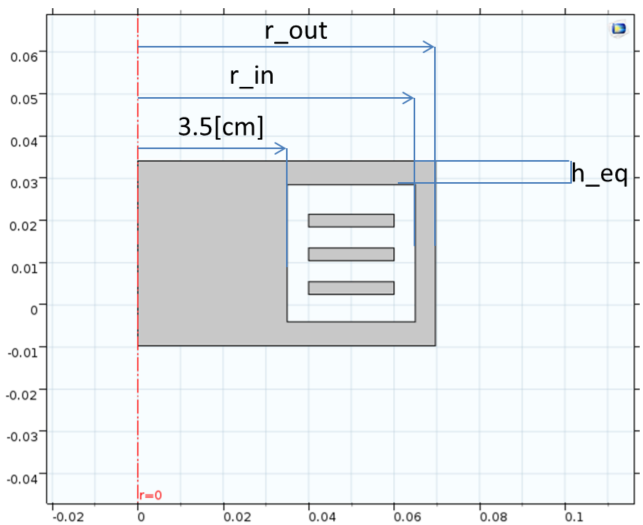


Figure 2: The 2D axisymmetric geometry equivalent to the power inductor under study.

The parameters are defined in Table 1. The original external column area (as given by the *Outer column section area* variable in Table 1) is the area represented in Figure 3. It is exactly $\pi(r_{out}^2 - r_{in}^2)$ in the 2D axisymmetric geometry. A similar consideration is valid for the *Upper magnetic circuit closure*.

The coil geometry representation in 2D axial symmetry is directly taken from the midplane cross section in the *xz*-plane in 3D. Figure 4 shows the final 2D axisymmetric geometry.

TABLE 1: QUOTA OF THE 3D AND THE 2D AXISYMMETRIC EQUIVALENT.

	3D	**2D axial symmetry**	**2D axisymmetric variable name**
Inner column radius	3.5 cm	3.5 cm	-
Outer column internal radius	6.5 cm	6.5 cm	`r_in`
Outer column section area	19.92 cm^2	Identical by construction	`external_area`
Outer column external radius	-	`sqrt(external_area/pi+r_in^2)`	`r_out`
Upper magnetic circuit closure area	7.5 cm^2	Identical by construction	`upper_area`
Upper magnetic circuit closure	-	`upper_area/pi/r_in`	`h_eq`

The main challenge of the 2D axisymmetric coil model is to account for the way that the electric currents will flow. The RLC Coil Group feature available in the Magnetic and Electric Fields interface automatically takes into account that the currents are balanced among:

1 conduction currents and induced currents that are flowing in the azimuthal direction

2 displacement currents that flow in the *rz*-plane from one turn to the other

In the low-frequency limit, currents are all of the first type. In the high-frequency limit, the currents are all of the second type. At some intermediate frequency there is a resonant frequency where inductive and capacitive effects perfectly balance and the coil self-resonates. At this frequency the inductor is purely resistive and the total loss peaks as a function of frequency.

In this model the core is not grounded and there is no other external electric ground. Compared to a real system where the inductor often is mounted on or near a ground plane, this may cause a shift in the resonance frequency. This is easily added, for example by applying one or more electric boundary conditions to the core boundaries.

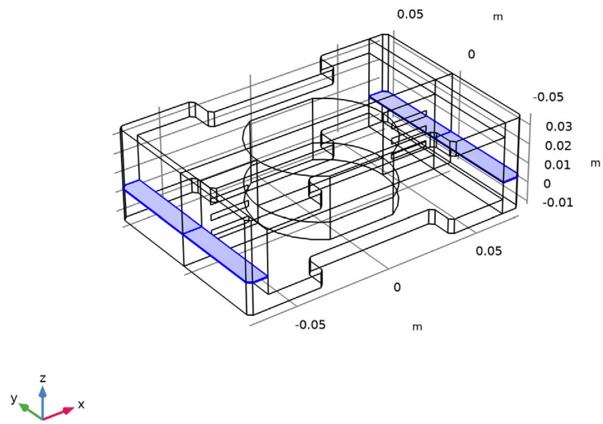


Figure 3: The 3D cross sectional area of the outer columns used to compute the 2D axisymmetric outer columns width.

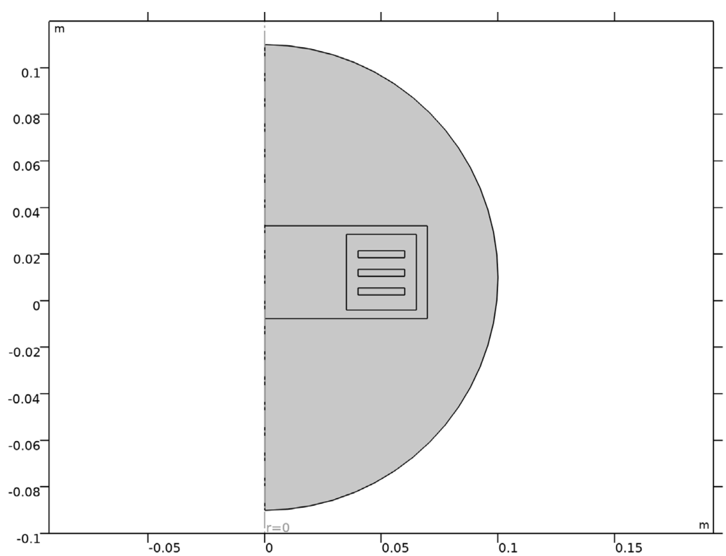


Figure 4: The 2D axisymmetric geometry including some surrounding air.

Results and Discussion

The field and loss plots can be compared to the 3D model discussed in the *Introduction to AC/DC Module* manual.

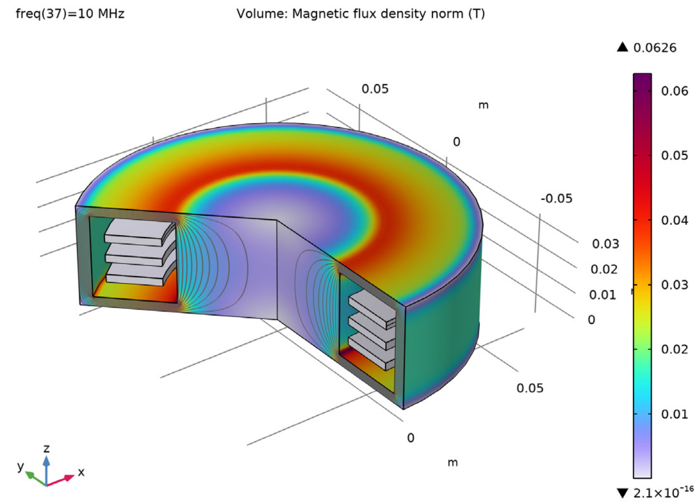


Figure 5: Magnetic flux density in the revolved 2D geometry.

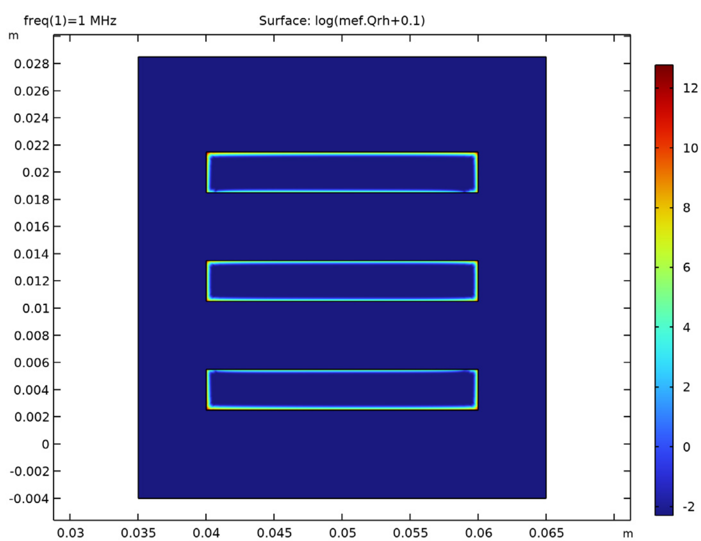


Figure 6: The eddy current losses are shown (log scale with zero offset).

The in-plane electric field magnitude strongly depends on the frequency, as is clearly shown from comparing Figure 7 and Figure 8.

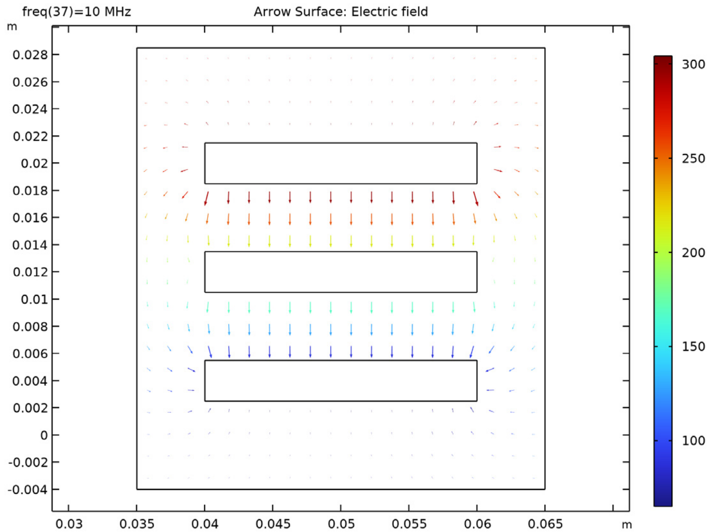


Figure 7: Electric field between turns far from the resonance frequency.

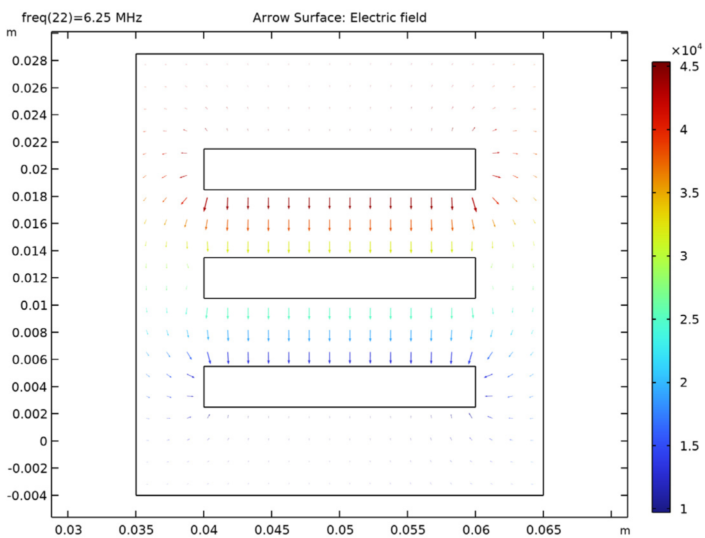


Figure 8: Electric field between turns at the resonance frequency.

The most striking sign of the resonance is shown in Figure 9, where the real and imaginary parts of the coil impedance are shown. The resistance (blue curve) is shown in tens of kΩ together with the reactance divided by the angular frequency (green curve). The green curve starts at low frequencies from the static inductance, grows, and then becomes negative. When the green curve becomes negative, it means that the coil behaves like a capacitor. The real part of the impedance accounts for losses, and peaks at the frequency where the capacitative and the inductive contributions to the overall system impedance balance out. The position and value of the peak featured by the 2D axisymmetric model also match the ones of the original 3D model in the *Introduction to AC/DC Module* manual fairly well.

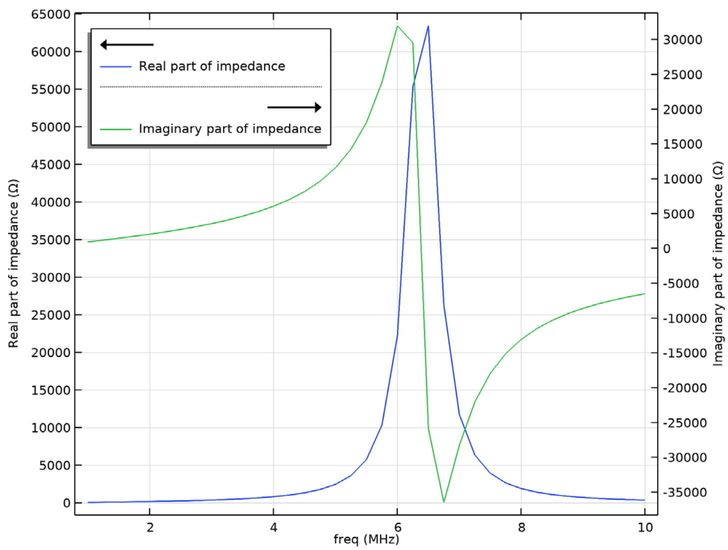


Figure 9: The inductor impedance as a function of frequency. Compare this plot to the corresponding plot of the 3D Inductor in the Introduction to AC/DC Module manual.

Application Library path: `ACDC_Module/Devices,_Inductive/axisymmetric_approximation_of_inductor_3d`

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>Vector Formulations>Magnetic and Electric Fields (mef)**.
3 Click **Add**.
4 In the **Added physics interfaces** tree, select **Magnetic and Electric Fields (mef)**.
5 Click **Study**.
6 In the **Select Study** tree, select **General Studies>Frequency Domain**.
7 Click **Done**.

First import the 3D geometry and take some measurements to generate the axisymmetric equivalent.

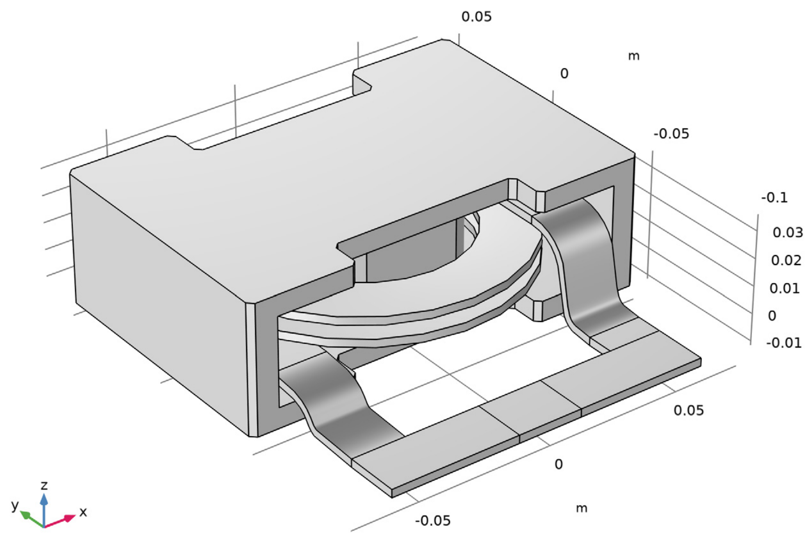
PART I

In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts>3D Part**.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `inductor_3d.mphbin`.

5 Click Import.



Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 Click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Cross Section 1 (cro1)

1 In the **Work Plane** toolbar, click **Cross Section**.

2 In the **Settings** window for **Cross Section**, click **Build Selected**.

Work Plane 1 (wp1)

1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Part 1** click **Work Plane 1 (wp1)**.

2 In the **Settings** window for **Work Plane**, click **Build Selected**.

Delete Entities 1 (del1)

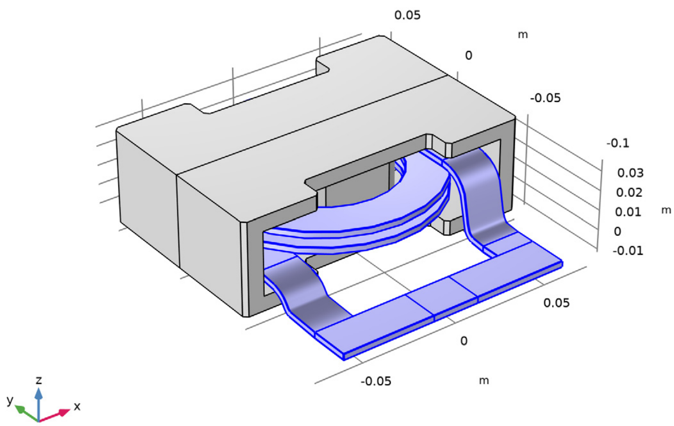
1 In the **Model Builder** window, right-click **Part 1** and choose **Delete Entities**.

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 On the object **imp1**, select Domains 2–5 only.

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



6 Click **Build Selected**.

Work Plane 2 (wp2)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `5[mm]`.

4 Click **Go to Plane Geometry**.

Work Plane 2 (wp2)>Cross Section 1 (cro1)

1 In the **Work Plane** toolbar, click **Cross Section**.

2 In the **Settings** window for **Cross Section**, click **Build Selected**.

Work Plane 2 (wp2)

1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Part 1** click **Work Plane 2 (wp2)**.

2 In the **Settings** window for **Work Plane**, click **Build Selected**.

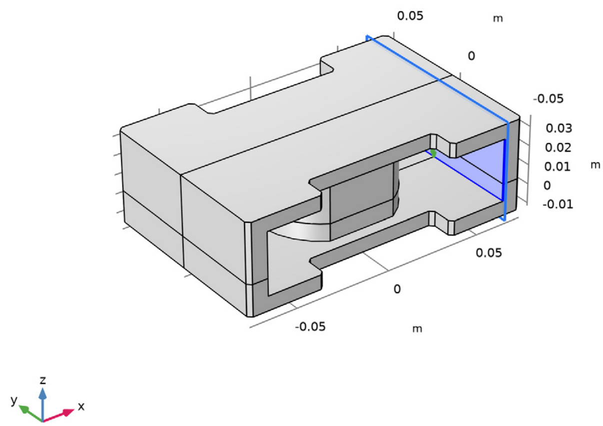
Work Plane 3 (wp3)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **del1**, select Boundary 46 only.



5 Click **Go to Plane Geometry**.

Work Plane 3 (wp3)>Cross Section 1 (cro1)

1 In the **Work Plane** toolbar, click **Cross Section**.

2 In the **Settings** window for **Cross Section**, click **Build Selected**.

Work Plane 3 (wp3)

1 In the **Model Builder** window, under **Global Definitions>Geometry Parts>Part 1** click **Work Plane 3 (wp3)**.

2 In the **Settings** window for **Work Plane**, click **Build Selected**.

The next step returns in the message log the relevant quota for the 3D to 2Daxi reduction.

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

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

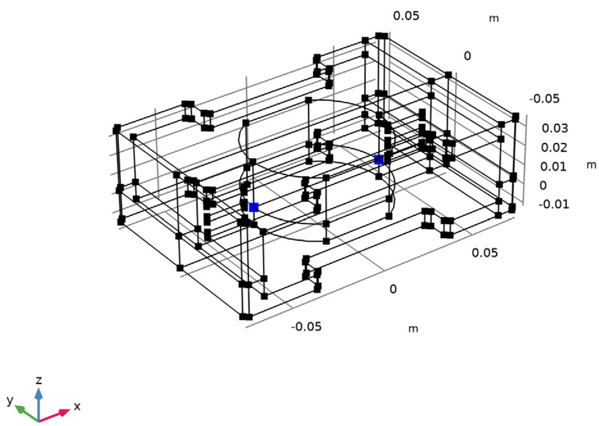
Next, make some measurements for use when creating the axisymmetric geometry.

5 In the **Model Builder** window, click **Part 1**.

6 In the **Graphics** window toolbar, click ▾ next to **Select Boundaries**, then choose **Select Points**.

7 On the object **wp2**, select Points 9 and 14 only.

It might be easier to select the points 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.)

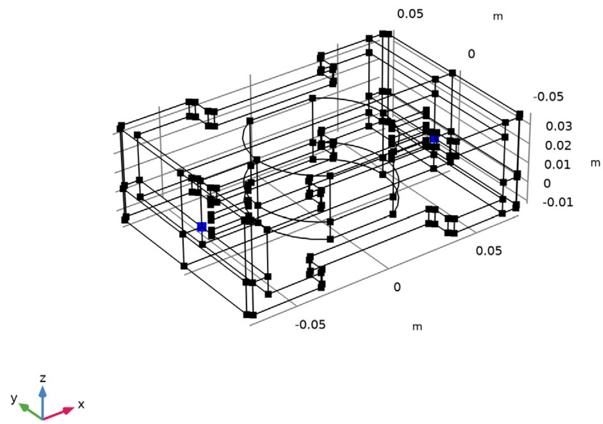


8 In the **Geometry** toolbar, click **Measure**.

Note the distance value, 0.07 m, and that the points have the same y- and z-coordinates.

9 On the object **wp2**, select Point 7 only.

10 On the object **wp3**, select Point 8 only.

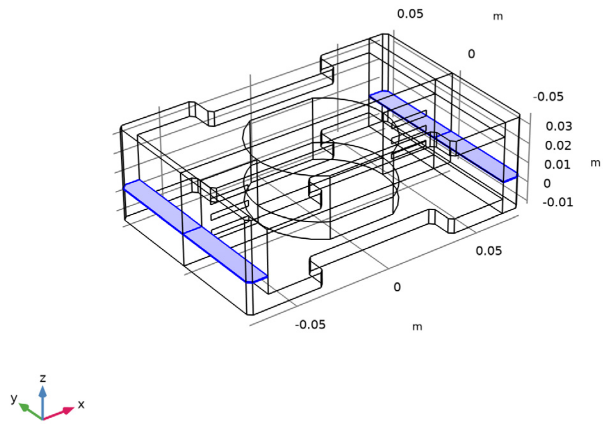


11 In the **Geometry** toolbar, click **Measure**.

The distance between the points is 0.13 m, again along an axis parallel to the x-axis.

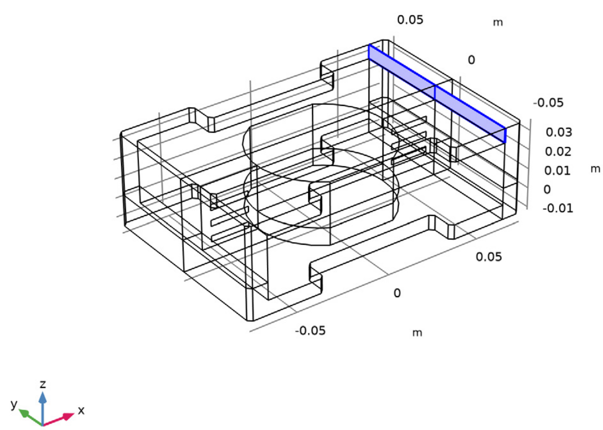
12 In the **Graphics** window toolbar, click next to **Select Points**, then choose **Select Boundaries**.

13 On the object **wp2**, select Boundaries 1, 2, 5, and 6 only.



14 In the **Geometry** toolbar, click **Measure**.

15 On the object **wp3**, select Boundaries 1 and 5 only.



16 In the **Geometry** toolbar, click **Measure**.

The combined area of the two faces is $7.5 \cdot 10^{-4}$ m^2. You will later use this value when drawing the 3D area of the upper magnetic circuit closure of the axisymmetric equivalent core.

GLOBAL DEFINITIONS

Add parameters for drawing the axisymmetric equivalent core.

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
inner_diameter	0.07[m]	0.07 m	3D diameter of central column
outer_diameter	0.13[m]	0.13 m	3D inner distance between external columns
external_area	0.001992 [m^2]	0.001992 m²	3D area of one of the two lateral columns
upper_area	7.5e-4[m^2]	7.5E-4 m²	3D area of the upper magnetic circuit closure
r_in	outer_diameter/2	0.065 m	2Daxi equivalence for external radius of lateral column
r_out	sqrt(external_area /pi+r_in^2)	0.069707 m	2Daxi equivalence for external radius of lateral column
h_eq	upper_area/pi/r_in	0.0036728 m	2Daxi equivalence for height of upper magnetic circuit closure

GEOMETRY 1

Import 1 (imp1)

1. In the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Work Plane 1 (wp1), Part 1**.

5 Click **Import**.

Rectangle 1 (r1)

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

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

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

4 In the **Height** text field, type `0.0325+2*h_eq`.

5 Locate the **Position** section. In the **z** text field, type `-0.004-h_eq`.

Circle 1 (c1)

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

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

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

4 Locate the **Position** section. In the **z** text field, type `0.01`.

5 Locate the **Size and Shape** section. In the **Sector angle** text field, type `180`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

Intersection 1 (int1)

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

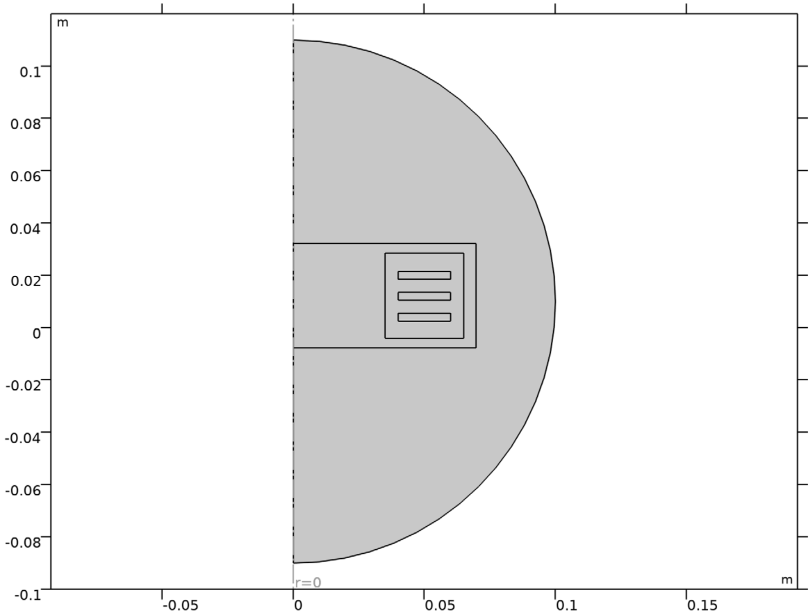
2 Select the objects **imp1** and **r1** only.

Form Union (fin)

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

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

3 In the **Model Builder** window, click **Form Union (fin)**.



MATERIALS

Add material data for the air and for the copper windings.

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic

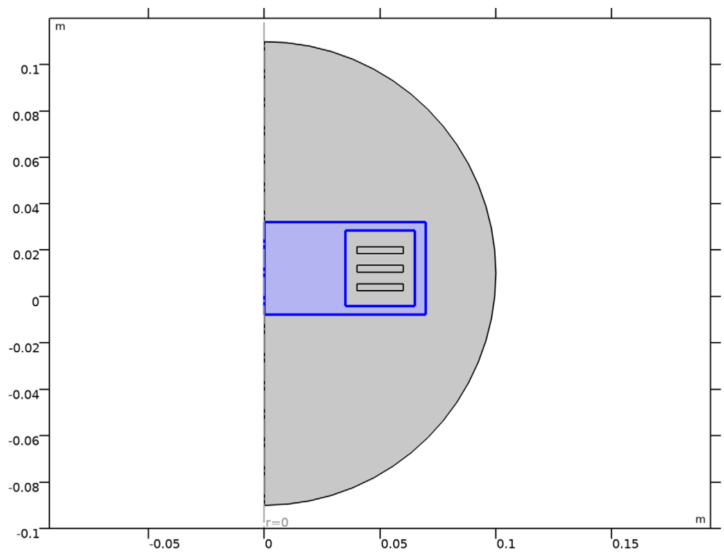
Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Add the constitutive relationship and the material information for the lossy iron core.

Material 2 (mat2)

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

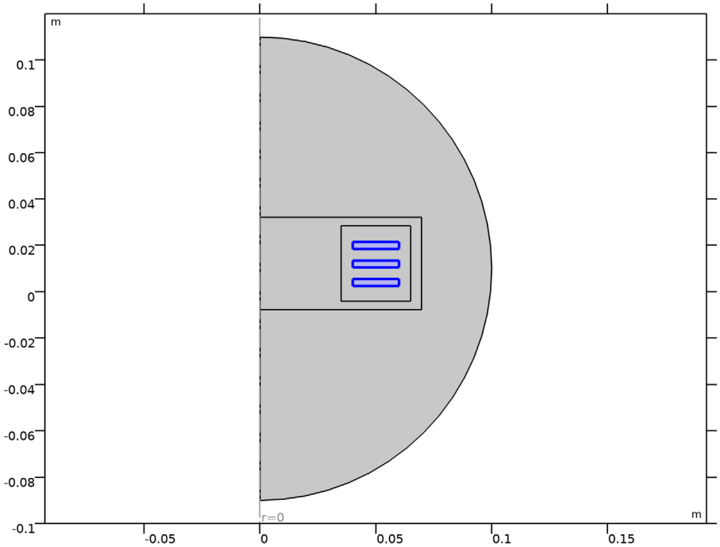
2 Select Domain 2 only.



Material 3 (mat3)

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

2 Select Domains 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
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	6e7	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

MAGNETIC AND ELECTRIC FIELDS (MEF)

Ampère's Law and Current Conservation 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic and Electric Fields (mef)** and choose **Ampère's Law and Current Conservation**.

2 Select Domain 2 only.

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

4 From the **Magnetization model** list, choose **Magnetic losses**.

Magnetic Insulation 1

In the **Model Builder** window, click **Magnetic Insulation 1**.

Electric Insulation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Insulation**.

2 Click the **Select All** button in the **Graphics** toolbar.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 2 (mat2)**.

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

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability (real part)	murPrim	1200	1	Magnetic losses
Relative permeability (imaginary part)	murBis	100	1	Magnetic losses
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

MAGNETIC AND ELECTRIC FIELDS (MEF)

First add the Ampere's law, which is necessary for adding the RLC Coil Feature.

Ampère's Law 1

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

2 Select Domains 4–6 only.

Add the RLC Coil Feature, which takes care of coupling the transverse currents inside the conductors to the capacitative current flowing perpendicularly to the windings.

RLC Coil Group 1

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

2 Select Domains 4–6 only.

As the coils are connected from the bottom to the top and domain ordering is not the same, a manual ordering is necessary.

3 In the **Settings** window for **RLC Coil Group**, locate the **Geometry** section.

4 From the **Domain ordering** list, choose **Manual**.

5 Locate the **Domain Selection** section. In the list, select **5**.

6 Locate the **Geometry** section. In the **Domain list** text field, type `5, 4, 6`.

In order to improve variable scaling, set a ground voltage slightly different from zero.

7 Locate the **RLC Coil Group** section. In the V_0 text field, type `1[mV]`.

MESH 1

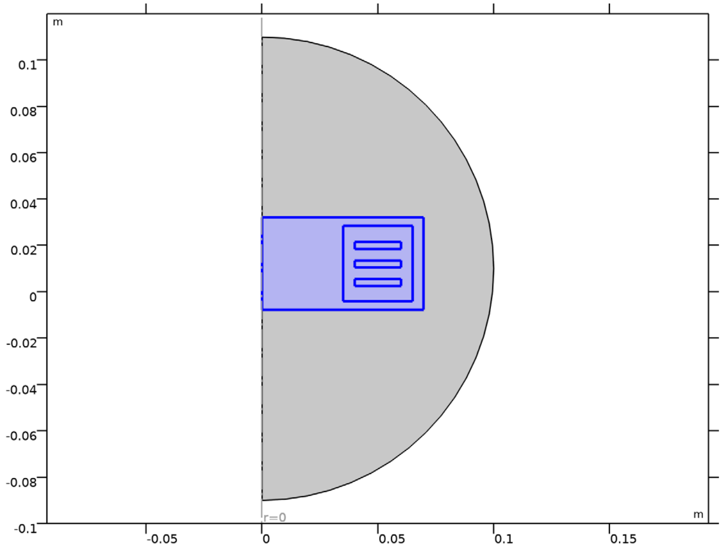
Free Triangular 1

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

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

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

4 Select Domains 2–6 only.



Size 1

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

2 Select Domains 2 and 3 only.

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

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Maximum element size** check box. In the associated text field, type `2e-3`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

3 Click **Clear Selection**.

4 Select Domains 4–6 only.

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

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `5e-4`.

Free Triangular 2

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

2 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

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

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

4 Select Domains 4–6 only.

A boundary layer mesh is added in order to resolve the skin depth.

Boundary Layer Properties

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

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

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

4 Locate the **Layers** section. From the **Thickness specification** list, choose **First layer**.

5 In the **Number of layers** text field, type `12`.

6 In the **Stretching factor** text field, type `1.3`.

7 In the **Thickness** text field, type `10[um]`.

8 Click **Build Selected**.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

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

3 From the **Frequency unit** list, choose **MHz**.

4 In the **Frequencies** text field, type `range(1,0.25,10)`.

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

RESULTS

Magnetic Flux Density Norm (mef)

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

Selection

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

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Selection**.

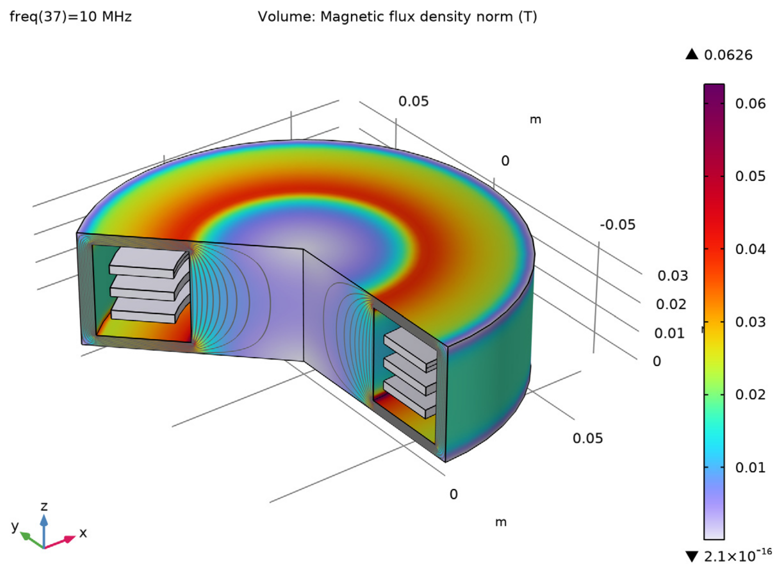
3 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

5 Select Domains 2 and 4–6 only.

Magnetic Flux Density Norm, Revolved Geometry (mef)

In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm, Revolved Geometry (mef)**.



Study 1/Solution 1 (2) (sol1)

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

Selection

1 Right-click **Study 1/Solution 1 (2) (sol1)** 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 Select Domains 3–6 only.

Resistive Losses

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 From the **Parameter value (freq (MHz))** list, choose **1**.

Surface 1

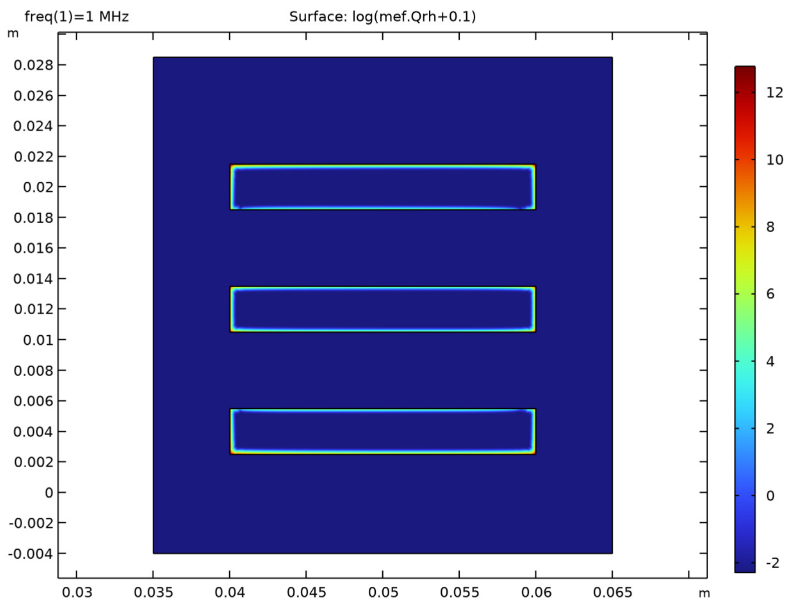
1 Right-click **Resistive Losses** and choose **Surface**.

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

3 In the **Expression** text field, type `log(mef.Qrh+0.1)`.

4 In the **Resistive Losses** toolbar, click **Plot**.

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



Electric Field

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

Arrow Surface 1

1 Right-click **Electric Field** and choose **Arrow Surface**.

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

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

4 Find the **r grid points** subsection. In the **Points** text field, type 20.

5 Find the **z grid points** subsection. In the **Points** text field, type 20.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic and Electric Fields>Electric>mef.Er, mef.Ez - Electric field**.

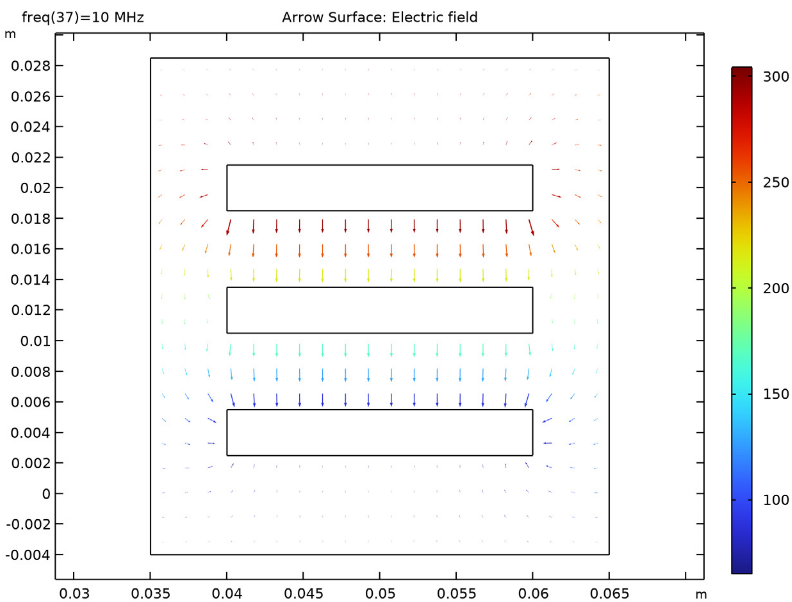
Color Expression 1

1 Right-click **Arrow Surface 1** and choose **Color Expression**.

2 In the **Electric Field** toolbar, click **Plot**.

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

4 In the **Model Builder** window, click **Color Expression 1**.



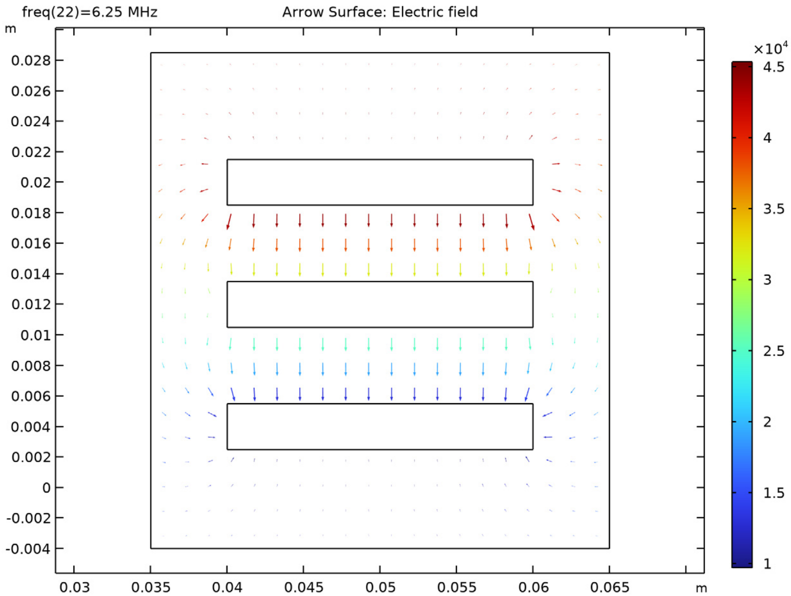
Electric Field

1 In the **Model Builder** window, under **Results** click **Electric Field**.

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

3 From the **Parameter value (freq (MHz))** list, choose **6.25**.

4 In the **Electric Field** toolbar, click **Plot**.



Impedance

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

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

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

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

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

Generate a plot with the real and the imaginary part of the impedance, which can be read at low frequency as impedance and resistance. The resonant peak is clearly visible and compares well with the original 3D geometry.

Global 1

Right-click **Impedance** and choose **Global**.

Global 2

In the **Model Builder** window, right-click **Impedance** and choose **Global**.

Global 1

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

2 In the table, enter the following settings:

Expression	**Unit**	**Description**
`real(mef.VCoil_1/1[A])`	Ω	`Real part of impedance`

Global 2

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

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

3 Select the **Plot on secondary y-axis** check box.

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

Expression	**Unit**	**Description**
`imag(mef.VCoil_1/1[A])`	Ω	`Imaginary part of impedance`

5 In the **Impedance** toolbar, click **Plot**.

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