Created in COMSOL Multiphysics 5.6

Modeling of a 3D Inductor

Introduction

Inductors are used in many applications for low-pass filtering or for impedance matching of predominantly capacitive loads. They are used in a wide frequency range from near static up to several MHz. An inductor usually has a magnetic core to increase the inductance while keeping its size small. The magnetic core also reduces the electromagnetic interference with other devices as the magnetic flux tends to stay within it. Because there are only crude analytical or empirical formulas available to calculate impedances, computer simulations or measurements are necessary during the design stage. In general, inductor modeling is more complex than modeling resistors and capacitors but similar principles apply. The model geometry is designed using an external CAD software, then it is imported into the AC/DC Module for static and frequency domain analysis. The inductor geometry is shown in Figure 1.

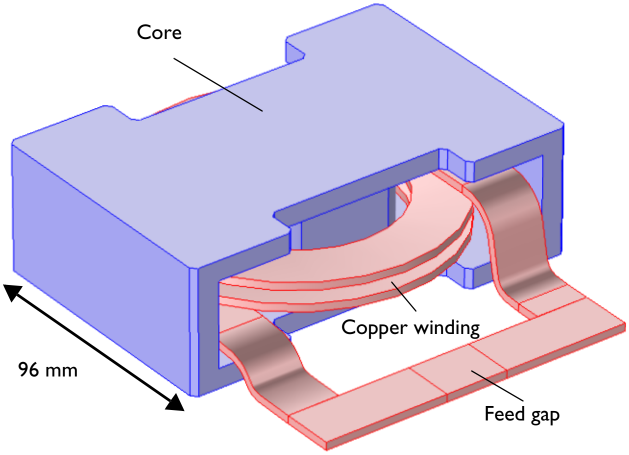


Figure 1: The inductor geometry.

First a magnetostatic simulation is performed to get the DC inductance. At low frequencies capacitive effects are negligible. Thus, the relevant equivalent circuit model is an ideal inductor in a series with an ideal resistor. The inductance and the resistance are both computed in the magnetostatic simulation. At a high frequency, capacitive effects and skin effect become significant and the equivalent circuit model involves connecting an ideal capacitor in parallel with the DC circuit. The skin effect modifies the current distribution in the winding so the resistance increases and the inductance also changes. The circuit parameters are obtained by analyzing the frequency dependent impedance obtained from a frequency domain simulation. In this tutorial, the AC analysis is done up to the point when the frequency-dependent impedance is computed.

Model Definition

The application uses the **Magnetic Fields** interface, which supports stationary, transient, and frequency-domain modeling. The following table lists the material properties used in this application:

MATERIAL PARAMETER	COPPER WINDING	CORE	AIR
σ	$5.998\cdot 10^7$ S/m	0 S/m	0 S/m
ε_r	1	1	1
μ_r	1	10^3	1

The outer boundaries are set to the default magnetic insulation,

$$\mathbf{n}\times\mathbf{A} = \mathbf{0}$$

which from the inductive perspective is equivalent to a perfect electric conductor. In the magnetostatic analysis, the coil is modeled by a **Coil** feature, which computes the current flow by means of a **Coil Geometry Analysis** preprocessing step and then applies a total current of 1 A. For the frequency-domain analysis, a **Lumped Port** with a fixed a current of 1 A is applied to the feed gap instead.

Results and Discussion

The magnetostatic analysis yields an inductance of 0.11 mH and a DC resistance of 0.29 mΩ. Figure 2 shows the magnetic flux density norm and the direction of the current flow.

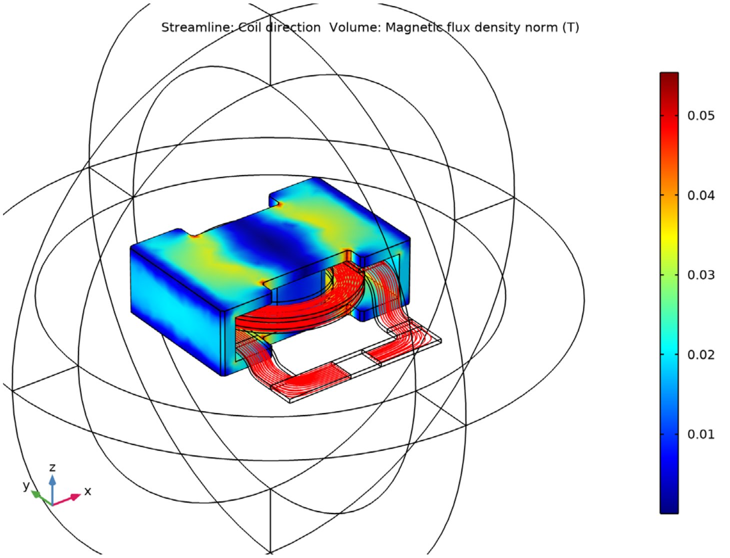


Figure 2: Magnetic flux density norm and current direction for the magnetostatic analysis.

In the static (DC) limit, the potential drop along the winding is purely resistive and could in principle be computed separately and before the magnetic flux density is computed. When increasing the frequency, inductive effects start to limit the current and skin effect makes it increasingly difficult to resolve the current distribution in the winding. At sufficiently high frequency, the current is mainly flowing in a thin layer near the conductor surface. When increasing the frequency further, capacitive effects come into play and current is flowing across the winding as displacement current density. When going through the resonance frequency, the device goes from behaving as an inductor to become predominantly capacitive. At the self resonance, the resistive losses peak due to the large internal currents. Figure 4 shows the surface current distribution at 1 MHz. Typical for high frequency, the currents are displaced toward the edges of the conductor.

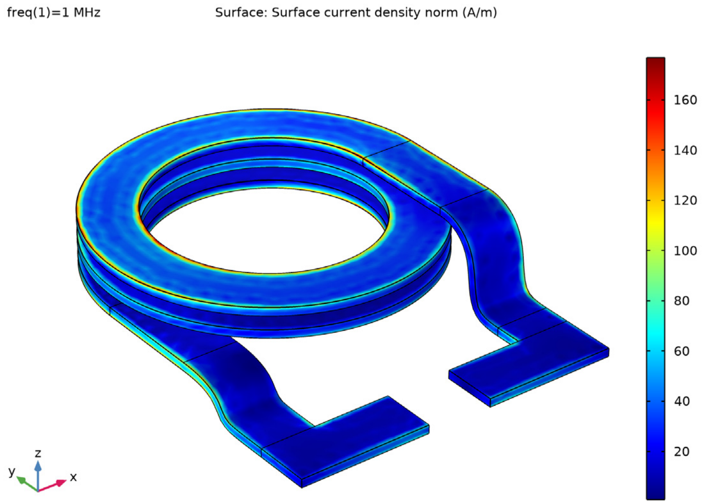


Figure 3: Surface current density at 1 MHz (below the resonance frequency).

Figure 4 shows how the resistive part of the coil impedance peaks at the resonance frequency near 6 MHz whereas Figure 5 shows how the reactive part of the coil impedance changes sign and goes from inductive to capacitive when passing through the resonance.

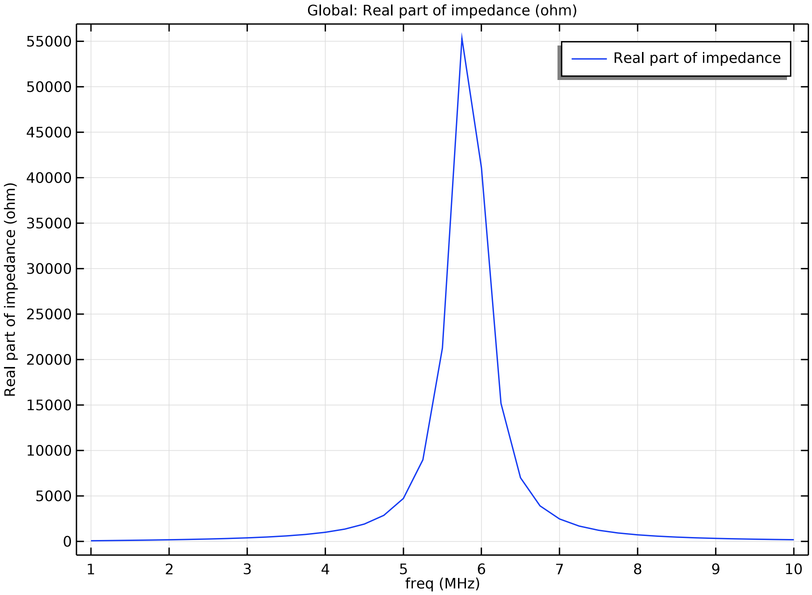


Figure 4: The real part of the coil impedance peaks at the resonance frequency.

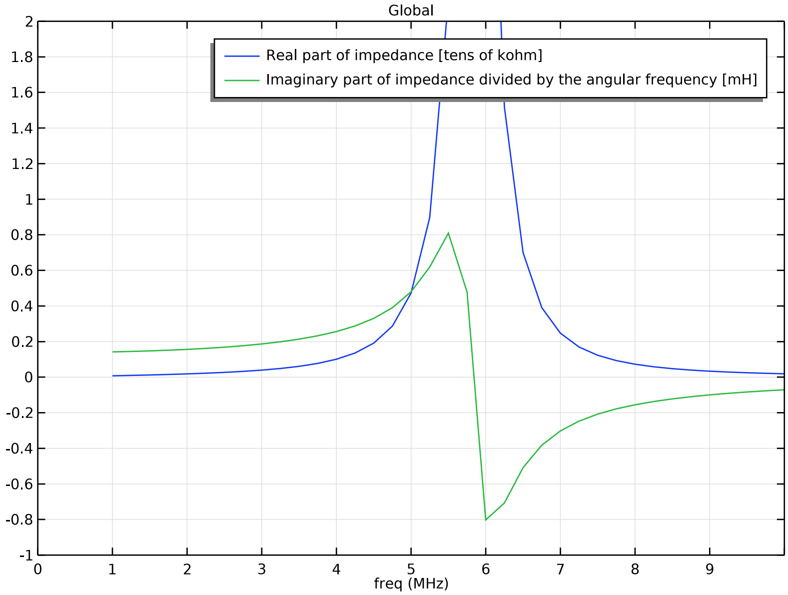


Figure 5: The reactive part of the coil impedance changes sign when passing through the resonance frequency, going from inductive to capacitive.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/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 **3D**.
2. In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.
3. Click **Add**.

4 In the **Added physics interfaces** tree, select **Magnetic Fields (mf)**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies>Stationary**.

7 Click **Done**.

GEOMETRY 1

The main geometry is imported from file. Air domains are typically not part of a CAD geometry so they usually have to be added later. For convenience three additional domains have been defined in the CAD file. These are used to define a narrow feed gap where an excitation can be applied.

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**.

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type `0.2`.

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

Layer name	Thickness (m)
`Layer 1`	`0.05`

5 Click **Build All Objects**.

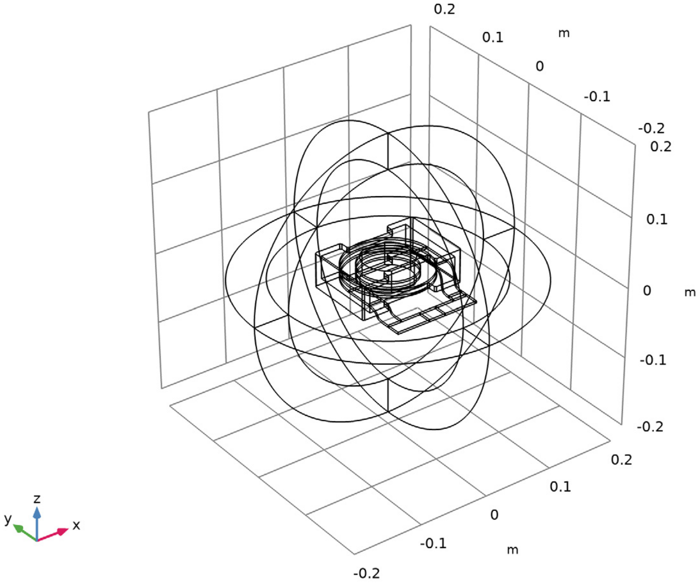
Form Union (fin)

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

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

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

The geometry should now look as in the figure below.



MATERIALS

Next, define selections to be used when setting up materials and physics. Start by defining the domain group for the inductor winding and continue by adding other useful selections.

DEFINITIONS

Winding

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

2 Select Domains 7, 8, and 14 only.

3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Winding` in the **New label** text field.

5 Click **OK**.

Gap

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

2 Select Domain 9 only.

3. Right-click **Explicit 2** and choose **Rename**.
4. In the **Rename Explicit** dialog box, type `Gap` in the **New label** text field.
5. Click **OK**.

Core

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domain 6 only.
3. Right-click **Explicit 3** and choose **Rename**.
4. In the **Rename Explicit** dialog box, type `Core` in the **New label** text field.
5. Click **OK**.

Infinite Elements

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 1–4 and 10–13 only.
3. Right-click **Explicit 4** and choose **Rename**.
4. In the **Rename Explicit** dialog box, type `Infinite Elements` in the **New label** text field.
5. Click **OK**.

Nonconducting

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 1–6 and 9–13 only.
3. Right-click **Explicit 5** and choose **Rename**.
4. In the **Rename Explicit** dialog box, type `Nonconducting` in the **New label** text field.
5. Click **OK**.

Nonconducting without IE

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 5, 6, and 9 only.
3. Right-click **Explicit 6** and choose **Rename**.
4. In the **Rename Explicit** dialog box, type `Nonconducting without IE` in the **New label** text field.
5. Click **OK**.

Use infinite elements to emulate an infinite open space surrounding the inductor.

Infinite Element Domain 1 (ie1)

1. In the **Definitions** toolbar, click **Infinite Element Domain**.

2. In the **Settings** window for **Infinite Element Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Infinite Elements**.
4. Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

Now define the material settings.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **AC/DC>Copper**.
4. Click **Add to Component** in the window toolbar.
5. In the tree, select **Built-in>Air**.
6. Click **Add to Component** in the window toolbar.
7. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat1)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Winding**.

Air (mat2)

1. In the **Model Builder** window, click **Air (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Nonconducting**.

The core material is not part of the material library so it is entered as a user-defined material.

Core

1. In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Core**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1e3	1	Basic
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

5 Right-click **Material 3 (mat3)** and choose **Rename**.

6 In the **Rename Material** dialog box, type `Core` in the **New label** text field.

7 Click **OK**.

MAGNETIC FIELDS (MF)

Select Domains 1–8 and 10–14 only.

Coil 1

1 Right-click **Component 1 (comp1)>Magnetic Fields (mf)** and choose the domain setting **Coil**.

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

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

4 In the **Model Builder** window, expand the **Coil 1** node.

Input 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1>Geometry Analysis 1** node, then click **Input 1**.

2 Select Boundary 58 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 79 only.

The physics-controlled meshing functionality in the **Magnetic Fields** interface automatically creates a mesh suited for the requirements of the physics interface. In this case, the functionality creates a swept mesh in the infinite element region. This kind of mesh is able to handle the steep radial scaling with high accuracy using a limited number of elements.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

Add a **Coil Geometry Analysis** study step before the **Stationary** step to compute the direction of the current applied in the windings.

STUDY 1

Coil Geometry Analysis

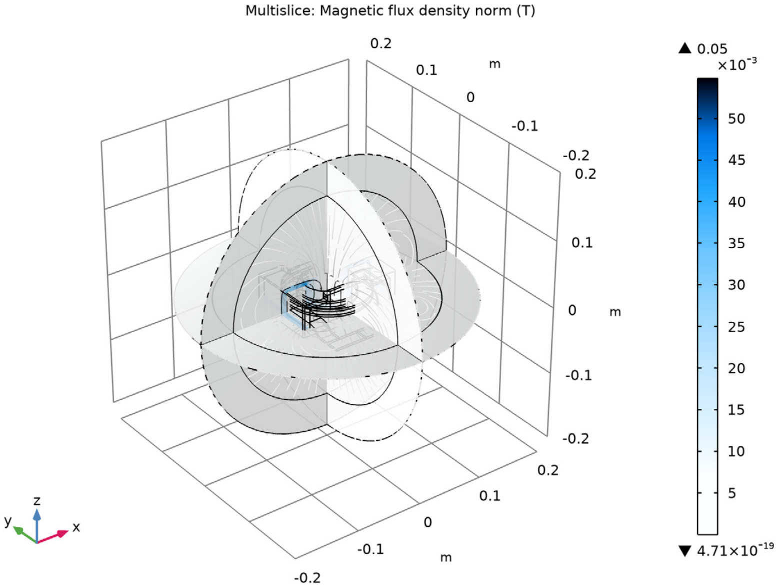
1. In the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.
2. Right-click **Study 1>Step 2: Coil Geometry Analysis** and choose **Move Up**.

   The magnetostatic model is now ready to solve.
3. In the **Study** toolbar, click = **Compute**.

RESULTS

Magnetic Flux Density Norm (mf)

The default plot group shows the magnetic flux density norm and helps in detecting possible modeling errors.



DEFINITIONS

View 1

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

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

3 Clear the **Show grid** check box.

RESULTS

Additional plots, more specific to this problem, can be obtained by manipulating the datasets as described in the next paragraphs.

Study 1/Solution 1 (3) (sol1)

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

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** and choose **Duplicate**.

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 **Domain**.

4 From the **Selection** list, choose **Winding**.

Study 1/Solution 1 (4) (sol1)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 1/ Solution 1 (1) (sol1)** and choose **Duplicate**.

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 **Domain**.

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

3D Plot Group 2

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

Streamline 1

1 Right-click **3D Plot Group 2** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Fields>Coil parameters>mf.coil1.eCoilx,...,mf.coil1.eCoilz - Coil direction**.

3 Select Boundary 58 only.

Volume 1

1 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Volume**.

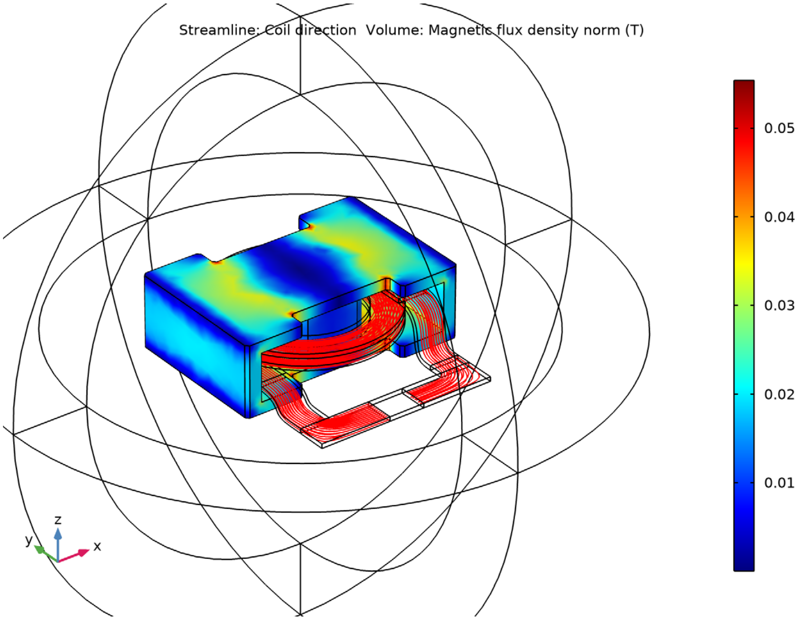
2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

4 In the **3D Plot Group 2** toolbar, click **Plot**.

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

6 Click the 🔍 **Zoom In** button in the **Graphics** toolbar.



Next, evaluate the coil inductance and resistance and compare them respectively to the inductance estimated from the total magnetic energy and to the resistance defined using Ohm's law.

Global Evaluation 1

1. In the **Results** toolbar, click (8.5) **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	**Unit**	**Description**
`mf.RCoil_1`	Ω	`Coil resistance (DC)`
`mf.LCoil_1`	`H`	`Coil inductance`
`mf.VCoil_1/mf.ICoil_1`	Ω	`Voltage drop definition`
`2*mf.intWm/mf.ICoil_1^2`	`H`	`Inductance via magnetic energy density`

4. Click 〓 **Evaluate**.

TABLE

1 Go to the **Table** window.

The results should be about 0.11 mH for the inductance and 0.29 mΩ for the resistance.

Now, try solving the model without the infinite elements. Since most of the magnetic flux resides inside the core region, the effect of the **Infinite Elements** is rather limited. In absence of the core, the effect of using infinite elements would be more significative.

MAGNETIC FIELDS (MF)

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

2 Select Domains 5–8 and 14 only.

STUDY 1

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

RESULTS

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, click = **Evaluate**.

COMPONENT 1 (COMP1)

Next, connect a simple circuit to the model.

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 **AC/DC>Electrical Circuit (cir)**.

4 Click **Add to Component 1** in the window toolbar.

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

ELECTRICAL CIRCUIT (CIR)

Change the coil excitation so it can connect to the circuit.

MAGNETIC FIELDS (MF)

Coil 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Coil 1**.
2. In the **Settings** window for **Coil**, locate the **Coil** section.
3. From the **Coil excitation** list, choose **Circuit (current)**.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Voltage Source 1 (V1)

1. In the **Electrical Circuit** toolbar, click **Voltage Source**.
2. In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
n	0

Resistor 1 (R1)

1. In the **Electrical Circuit** toolbar, click **Resistor**.
2. In the **Settings** window for **Resistor**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	1
n	2

4. Locate the **Device Parameters** section. In the R text field, type `100[mohm]`.

In order to connect the circuit to the finite elements model use the dedicated circuit element.

External I vs. U 1 (IvsU1)

1. In the **Electrical Circuit** toolbar, click **External I vs. U**.
2. In the **Settings** window for **External I vs. U**, locate the **External Device** section.
3. From the V list, choose **Coil voltage (mf/coil1)**.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	0

STUDY 1

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

RESULTS

Next, evaluate the current.

Global Evaluation 2

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`mf.ICoil_1`	A	`Coil current`

4 Click = **Evaluate**.

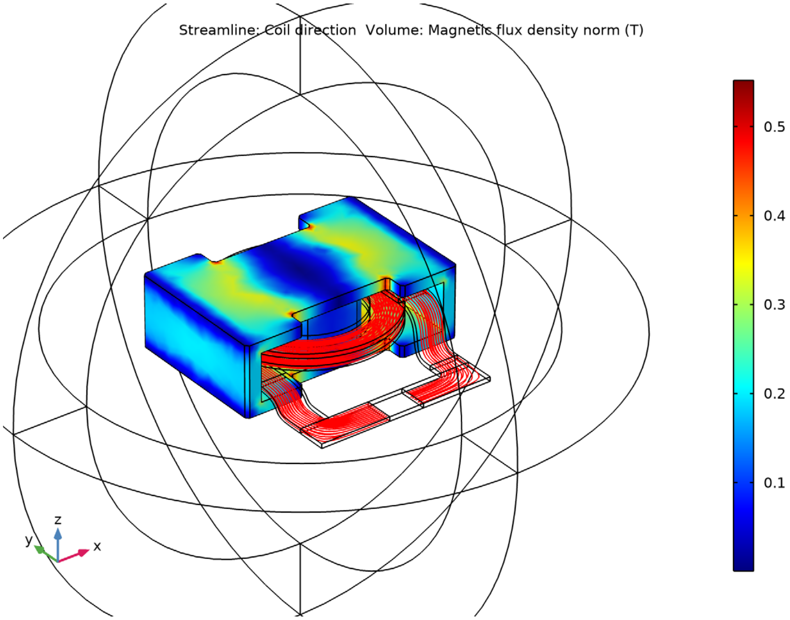
TABLE

1 Go to the **Table** window.

The current is limited to approximately 10 A by the external resistor, which is much larger than the internal resistance of the winding.

RESULTS

3D Plot Group 2



Now set up the model for computing the frequency-dependent impedance.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electrical Circuit (cir)**.
4. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.
5. Click **Add Study** in the window toolbar.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

DEFINITIONS

At high frequency the skin depth of the conductor is so small that it is not convenient to resolve it. Instead of modeling the entire domain, use a lossy boundary condition with the appropriate material information, specified in a boundary **Material** node. Create also a **Selection** node to simplify the specification of the boundary condition.

Conductor Boundaries

1. In the **Definitions** toolbar, click **Explicit**.
2. Select Domains 7, 8, and 14 only.
3. In the **Settings** window for **Explicit**, locate the **Output Entities** section.
4. From the **Output entities** list, choose **Adjacent boundaries**.
5. Right-click **Explicit 7** and choose **Rename**.
6. In the **Rename Explicit** dialog box, type `Conductor Boundaries` in the **New label** text field.
7. Click **OK**.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **AC/DC>Copper**.
4. Click **Add to Component** in the window toolbar.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper 1 (mat4)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Geometric entity level** list, choose **Boundary**.
3. From the **Selection** list, choose **Conductor Boundaries**.

MAGNETIC FIELDS (MF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.
2. In the **Settings** window for **Magnetic Fields**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Nonconducting without IE**.

Impedance Boundary Condition 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.
2. In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Conductor Boundaries**.

Coil 1

The **Coil** feature no longer applies to an active domain and it has no effect. Disable it for clarity.

1. In the **Model Builder** window, right-click **Coil 1** and choose **Disable**.

Without additional information about the external feeding, the electric potential is not a well defined quantity at high frequency. In order to provide missing information, use the **Lumped Port** boundary feature to excite the system.

Lumped Port 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.
2. Select Boundaries 59–62 only.

   Enter the geometrical parameters of the boundary.
3. In the **Settings** window for **Lumped Port**, locate the **Lumped Port Properties** section.
4. From the **Type of lumped port** list, choose **User defined**.
5. In the h_{port} text field, type `0.024`.
6. In the w_{port} text field, type `0.046`.
7. Specify the $\mathbf{a}_{\mathrm{h}}$ vector as

1	x
0	y
0	z

8. From the **Terminal type** list, choose **Current**.

Apart from the surface losses in the copper conductor, there will also be losses in the core due to eddy currents. These losses are introduced by specifying an effective complex permeability for the magnetic iron in the core, a quantity which is often available from the manufacturer. There is a specific constituive relation for this.

Ampère's Law 2

1. In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.
2. In the **Settings** window for **Ampère's Law**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Core**.
4. Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Magnetic losses**.
5. From the μ′ list, choose **User defined**. From the μ″ list, choose **User defined**. In the μ′ text field, type `1200`.

6. In the μ″ text field, type `100`.

STUDY 2

Step 1: Frequency Domain

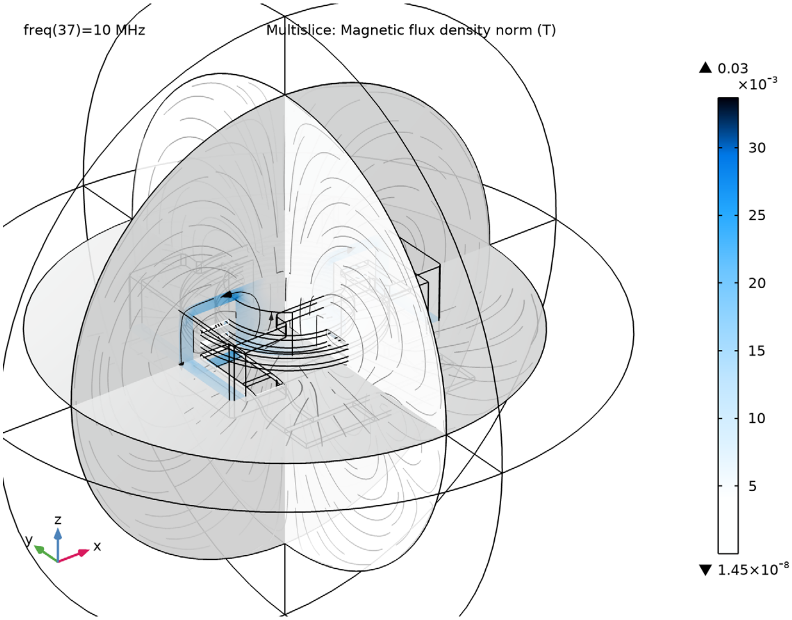
Set up a frequency sweep from 1 MHz to 10 MHz in steps of 0.25 MHz.

1. In the **Model Builder** window, under **Study 2** 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. Click **Range**.
5. In the **Range** dialog box, type `1` in the **Start** text field.
6. In the **Stop** text field, type `10`.
7. In the **Step** text field, type `0.25`.
8. Click **Replace**.
9. In the **Home** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf) 1

Check for possible modeling errors by comparing the obtained results with the following figure.



Proceed to look at the surface current distribution in the winding.

Study 2/Solution 3 (6) (sol3)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 2/Solution 3 (sol3)** and choose **Duplicate**.

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. From the **Selection** list, choose **Conductor Boundaries**.

3D Plot Group 4

1. In the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2/Solution 3 (6) (sol3)**.

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

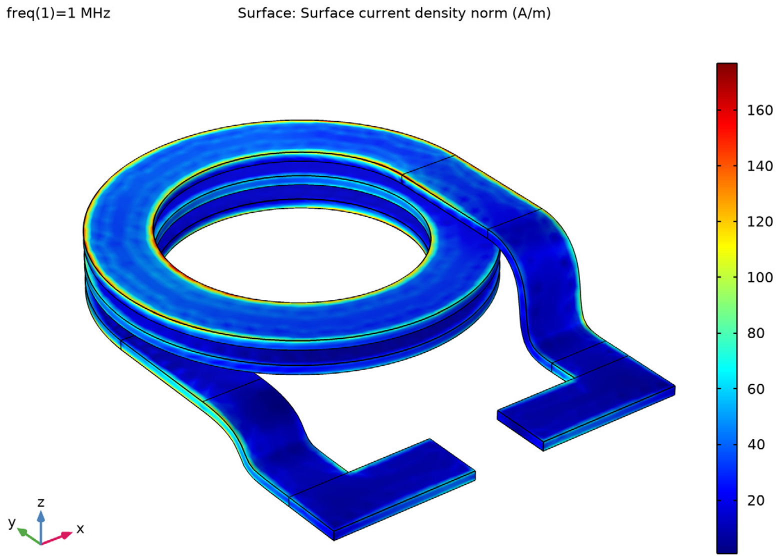
Surface 1

1. Right-click **3D Plot Group 4** and choose **Surface**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields> Currents and charge>mf.normJs - Surface current density norm - A/m**.

3D Plot Group 4

1. In the **Model Builder** window, click **3D Plot Group 4**.
2. In the **3D Plot Group 4** toolbar, click **Plot**.

   This is the surface current distribution.



Finish the modeling session by plotting the real and imaginary parts of the coil impedance.

1D Plot Group 5

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2/Solution 3 (5) (sol3)**.

Global 1

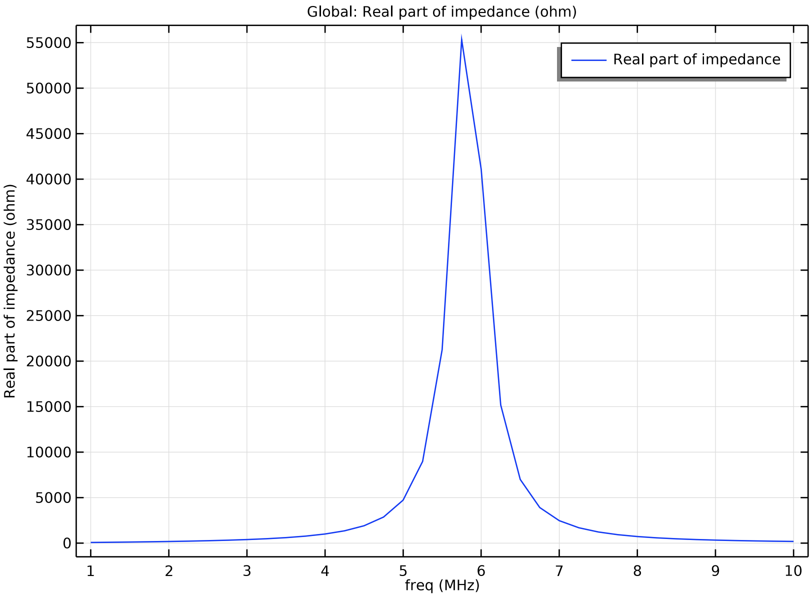
1. Right-click **1D Plot Group 5** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Ports>mf.Zport_1 - Lumped port impedance - Ω**.

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

Expression	Unit	Description
`real(mf.Zport_1)`	`ohm`	`Real part of impedance`

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



The resistive part of the coil impedance peaks at the resonance frequency.

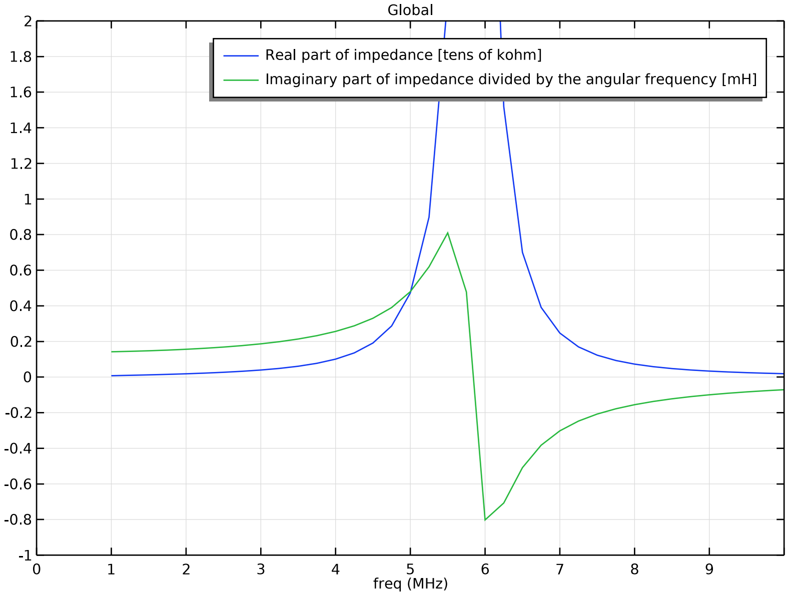
5 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Ports>mf.Zport_1 - Lumped port impedance - Ω**.

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

Expression	Unit	Description
`real(mf.Zport_1)/10[kohm]`	`1`	`Real part of impedance [tens of kohm]`
`real(mf.Zport_1/mf.iomega)`	`mH`	`Imaginary part of impedance divided by the angular frequency [mH]`

1D Plot Group 5

1. In the **Model Builder** window, click **1D Plot Group 5**.
2. In the **Settings** window for **1D Plot Group**, locate the **Axis** section.
3. Select the **Manual axis limits** check box.
4. In the **x minimum** text field, type `0`.
5. In the **x maximum** text field, type `10`.
6. In the **y minimum** text field, type `-1`.
7. In the **y maximum** text field, type `2`.
8. In the **1D Plot Group 5** toolbar, click **Plot**.



The reactive part of the coil impedance approaches the static inductance value at low frequency. With increasing frequency, the quantity increases and then changes sign at the resonance. This happens since the inductor is entering a capacitive regime where the net current is dominated by displacement currents flowing perpendiculary to the coil windings.

Before saving the application, specify a plot to be used as a thumbnail.