Created in COMSOL Multiphysics 5.6

Inductance of a Power Inductor

Introduction

Power inductors are a central part of many low-frequency power applications. They are, for example, used in switched power supplies and DC-DC converters. The inductor is used in conjunction with a high-power semiconductor switch that operates at a certain frequency, stepping up or down the voltage on the output. The relatively low voltage and high power consumption put high demands on the design of the power supply and especially on the inductor, which must be designed with respect to switching frequency, current rating, and hot environments.

A power inductor usually has a magnetic core to increase its inductance value, reducing the demands for a high frequency while keeping the sizes small. The magnetic core also reduces the electromagnetic interference with other devices. There are only crude analytical formulas or empirical formulas available for calculating impedances, so computer simulations or measurements are necessary in the design of these inductors. This application uses a design drawn in an external CAD software, imports the geometry to COMSOL Multiphysics, and finally calculates the inductance from the specified material parameters and frequency.

Model Definition

The application uses the Magnetic and Electric Fields interface, taking electric and magnetically induced currents into account. This formulation, often referred to as an AV-formulation, solves both for the magnetic vector potential $\mathbf{A}$ and the electric potential V. The model is solved first with a free gauge, using an iterative, residual minimizing method and, in a second step with the Coulomb gauge

$$\nabla \cdot \mathbf{A} = 0$$

applied to the magnetic vector potential. See the *Theory of Magnetic Fields* and *Gauge Fixing for A-field* sections in the *AC/DC Module User's Guide* for more details.

For very high frequencies capacitive effects yield a frequency-dependent coil reactance as shown in the related example described in the manual *Introduction to AC/DC Module*. There it is also shown how to handle a very small skin depth by, instead of volumetric meshing, using an asymptotic high frequency impedance boundary condition.

In this model the frequency is much lower and the coil is almost purely inductive with an inductance that is close to that computed in the static limit. Still the resistance is frequency dependent due to skin effect. Volumetric meshing is applied but with the twist of using a boundary layer mesh to resolve the skin depth.

The following table lists the material properties used in this application:

MATERIAL PARAMETER	COPPER	CORE	AIR
σ	$5.997 \cdot 10^7$ S/m	0 S/m	0 S/m
ε_r	1	1	1
μ_r	1	10^3-10i	1

The losses in the copper coil are purely resistive, while the core loss is described using a complex relative permeability. The latter information is commonly provided by magnetic material (for example ferrites) manufacturers. In COMSOL Multiphysics the value can easily be made frequency or temperature dependent if needed by means of using interpolation tables, and so on.

The outer boundaries are mainly the default magnetic insulation and electric insulation,

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

The copper winding is grounded in one end. The other end uses a Terminal boundary condition to apply an electric potential of 1 V. The Terminal generates an admittance variable for the inductor that can be accessed in postprocessing. You can calculate the inductance from the formula

$$L_{11} = \operatorname{Im}\left(\frac{1}{\omega Y_{11}}\right)$$

where ω is the angular frequency, and Y_{11} is the coil/Terminal admittance. The effective conductance that is due to resistive losses in the coil and magnetic losses in the core is evaluated as the real part of Y_{11}.

Results and Discussion

The model is solved for a frequency of 1 kHz. It yields an inductance of 115 µH similar to the static value computed in the manual *Introduction to AC/DC Module*. The conductance evaluates to 0.015 S and is shown to be balanced by losses in model. Figure 1 shows the distribution of the real part of the electric potential distribution for the case with the Coulomb gauge applied. Note that the electric potential is not gauge invariant so it will look different with the free gauge. Only the electric and magnetic fields are independent of the gauge.

Figure 2 shows the magnetic flux density, both its norm as a slice plot and its local direction and strength at zero phase as an arrow plot.

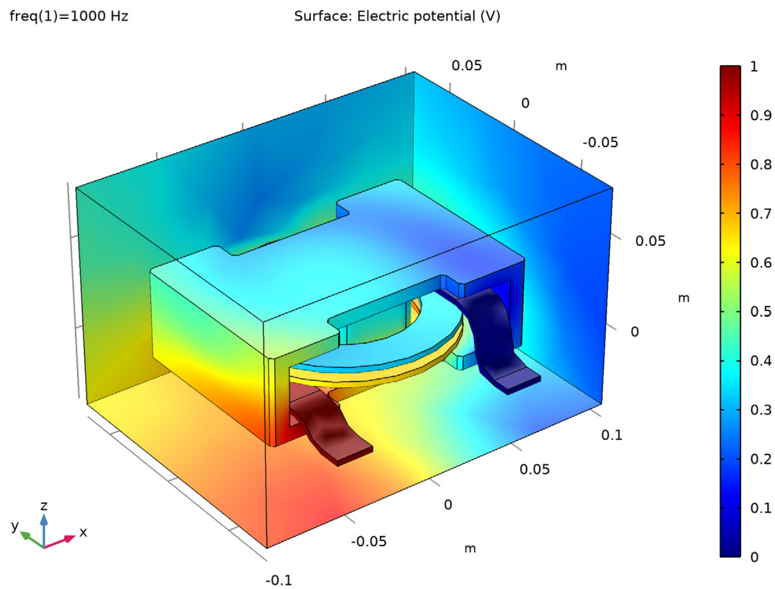

Figure 1: Real part of the electric potential distribution.

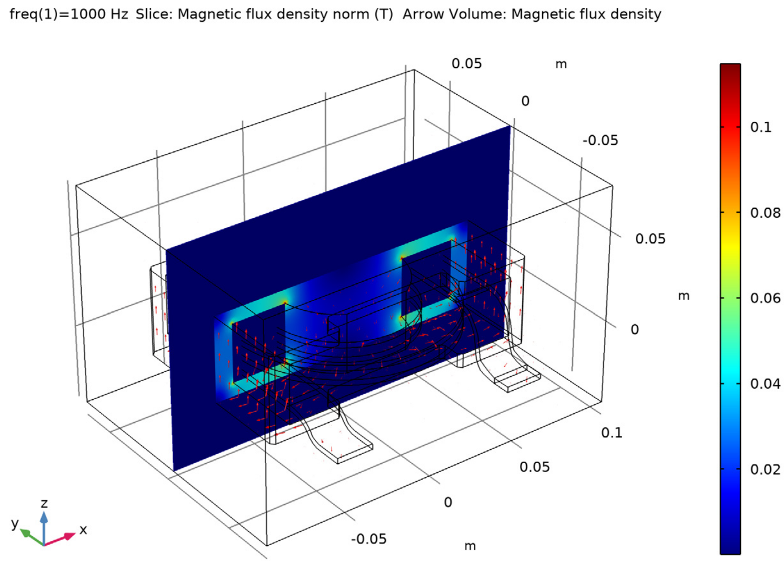

Figure 2: The final plot of the power inductor, showing the potential on the coil, the magnitude of the flux density inside the ferrite core, and the direction of the same as arrows.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/power_inductor`

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>Vector Formulations>Magnetic and Electric Fields (mef)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

The geometry of the inductor is available as a CAD file. Import it and create a surrounding air box.

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 `power_inductor.mphbin`.

5 Click **Import**.

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 `0.2`.

4 In the **Depth** text field, type `0.15`.

5 In the **Height** text field, type `0.12`.

6 Locate the **Position** section. In the **x** text field, type `-0.1`.

7 In the **y** text field, type `-0.08`.

8 In the **z** text field, type `-0.04`.

9 Click **Build All Objects**.

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

MATERIALS

This application uses two materials that are already available in the Material Library and one defined from a Blank material.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>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 Click **Clear Selection**.

4 Select Domain 3 only.

Air (mat2)

1 In the **Model Builder** window, click **Air (mat2)**.

2 Select Domain 1 only.

MAGNETIC AND ELECTRIC FIELDS (MEF)

Proceed now with the setup of the physics interface and set up the **Terminal** and **Ground** conditions driving the current through the model.

Magnetic Insulation 1

In the **Model Builder** window, expand the **Component 1 (comp1)> Magnetic and Electric Fields (mef)>Magnetic Insulation 1** node, then click **Magnetic Insulation 1**.

Ground 1

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

2 Select Boundary 63 only.

Magnetic Insulation 1

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

Terminal 1

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

2 Select Boundary 17 only.

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

4 From the **Terminal type** list, choose **Voltage**.

In order to model a finite loss core, it is added a constitutive relationship where the complex relative permeability is specified with the corresponding material providing these data.

Ampère's Law and Current Conservation 2

1 In the **Physics** toolbar, click **Domains** 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**.

MATERIALS

Core Material

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

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

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

4 Click **OK**.

5 Select Domain 2 only.

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

7 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability (real part)	murPrim	1000	1	Magnetic losses
Relative permeability (imaginary part)	murBis	10	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

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 **Extra coarse**.

Size

Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Free Tetrahedral 1

As in the following linear elements are going to be chosen, a finer mesh in the core is added even though this is unnecessary for the first solution where default quadratic elements are used.

Size 1

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

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

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

4 Select Domain 2 only.

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

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

7 In the associated text field, type `4[mm]`.

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

Boundary Layers 1

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

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

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

4 Select Domain 3 only.

Boundary Layer Properties

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

2 Select Boundaries 16, 18–26, 62, and 64–74 only.

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

4 In the **Number of boundary layers** text field, type `2`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `1[mm]`.

7 Click **Build All**.

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 In the **Frequencies** text field, type `1[kHz]`.

The model is now ready for solving. In order to compute the admittance with sufficient accuracy in this case with fairly low frequency, the error estimate must be increased so that the iterative solver will refine the solution more than the standard value.

Solution 1 (sol1)

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

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)>Stationary Solver 1** node, then click **Iterative 1**.

4 In the **Settings** window for **Iterative**, click to expand the **Error** section.

5 In the **Factor in error estimate** text field, type `1e7`.

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

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

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

RESULTS

3D Plot Group 1

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Slice 1

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

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic and Electric Fields>Magnetic>mef.normB - Magnetic flux density norm - T**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type `1`.

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

Arrow Volume 1

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

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic and Electric Fields>Magnetic>mef.Bx,...,mef.Bz - Magnetic flux density**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 20.

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

5 Find the **z grid points** subsection. In the **Points** text field, type 10.

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

The plot should now look like Figure 2 .

Next instructions show how to extract the conductance and inductance.

Global Evaluation 1

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

Compute the conductance as real part of admittance, and inductance as the inverse of admittance imaginary part divided by the angular frequency. Specify the output unit as uH (microhenry, µH).

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

3 In the table, enter the following settings:

Expression	Unit	Description
real(mef.Y11)	S	
real(1/mef.Y11/mef.iomega)	uH	

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

Real part of admittance evaluates to about 0.015[S] while inductance 115[uH] similarly to the static limit.

RESULTS

Finite value of real part of inductance is due to losses in the material. In the present case, most of the losses are in the lossy core, but some are also in the conducting coil. In the next it is shown how effective conductance of the coil as evaluated by the terminal is consistent with the balance of losses in the whole system.

Volume Integration 1

1 In the **Results** toolbar, click **8.85 e-12** **More Derived Values** and choose **Integration> Volume Integration**.

2 Select Domains 2 and 3 only.

3 In the **Settings** window for **Volume Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`2*mef.Qh/1[V^2]`	`S`	

5 Click **Evaluate (Table 1 - Global Evaluation 1)**.

MAGNETIC AND ELECTRIC FIELDS (MEF)

In the following it is shown that an alternative to increasing solver tolerances is add gauge fixing. Gauge fixing is compatible with direct solver which essentially does not require any tuning. As direct solver implies a considerably higher computational burden, in the next, linear element are chosen. The logical step are then, adding the Gauge Fixing node, set solver to Direct, and solve. In the following this is done, but first a modified label is added to the former solver and the new one.

Gauge Fixing for A-field 1

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

STUDY 1

Model without gauge fixing - default iterative solver tweaked

1 In the **Model Builder** window, under **Study 1>Solver Configurations** click **Solution 1 (sol1)**.

2 In the **Settings** window for **Solution**, type `Model without gauge fixing - default iterative solver tweaked` in the **Label** text field.

Model with gauge fixing - direct solver

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Direct** and choose **Enable**.

5 In the **Model Builder** window, click **Solution 2 (sol2)**.

6 In the **Settings** window for **Solution**, type `Model with gauge fixing - direct solver` in the **Label** text field.

MAGNETIC AND ELECTRIC FIELDS (MEF)

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

2 In the **Settings** window for **Magnetic and Electric Fields**, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

4 From the **Electric potential** list, choose **Linear**.

STUDY 1

Model with gauge fixing - direct solver (sol2)

In the **Model Builder** window, under **Study 1>Solver Configurations** right-click **Model with gauge fixing - direct solver (sol2)** and choose **Compute**.

RESULTS

Next compute the new estimate of conductance and inductance, and update the former plot.

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**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Model with gauge fixing - direct solver (sol2)**.

4 Click **Evaluate (Table 1 - Global Evaluation 1)**.

3D Plot Group 1

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

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

3 From the **Dataset** list, choose **Study 1/Model with gauge fixing - direct solver (sol2)**.

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

Finally visualize the potential distribution in a new plot group.

3D Plot Group 2

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 From the **Dataset** list, choose **Study 1/Model with gauge fixing - direct solver (sol2)**.

Surface 1

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

Take a look inside by hiding a few of the exterior boundaries.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 3 and 5–79 only. The quickest way to do this is to select **All boundaries** from the **Selection** list, then remove Boundaries 1, 2, and 4.

3D Plot Group 2

Click **3D Plot Group 2** to visualize the plot reproducing Figure 1 .

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

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

3 Click **Plot**.