Created in COMSOL Multiphysics 6.1

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

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

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, see the comparisons in Figure 2. Only the electric and magnetic fields are independent of the gauge, as shown in Figure 3.

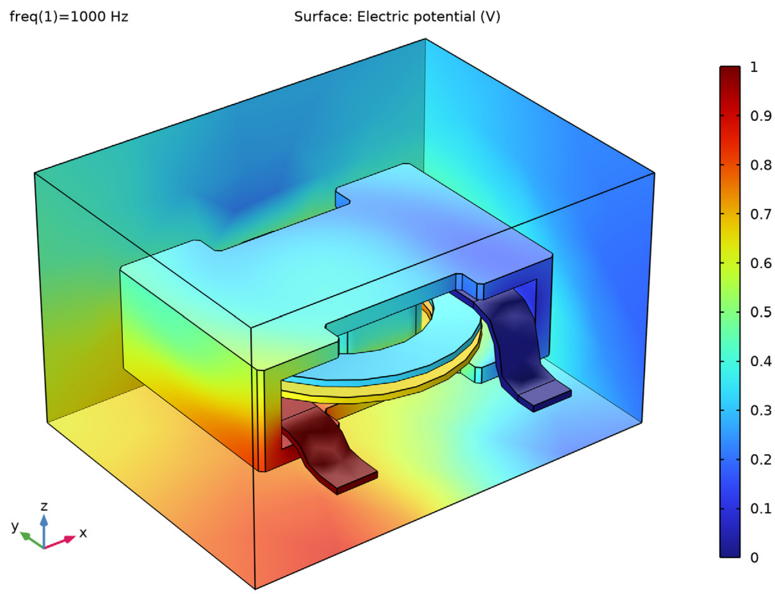

Figure 1: Real part of the electric potential distribution.

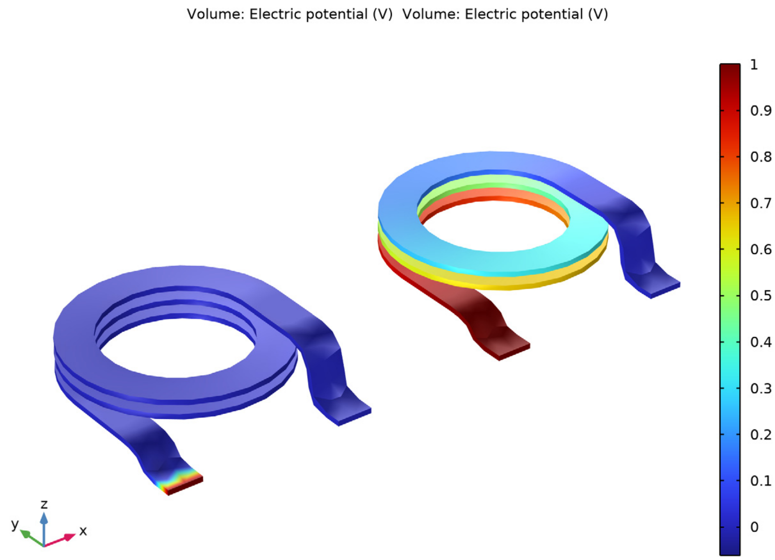

Figure 2: Comparison of the electric potential distribution on the coil between ungauged formulation (left) and gauged formulation (right).

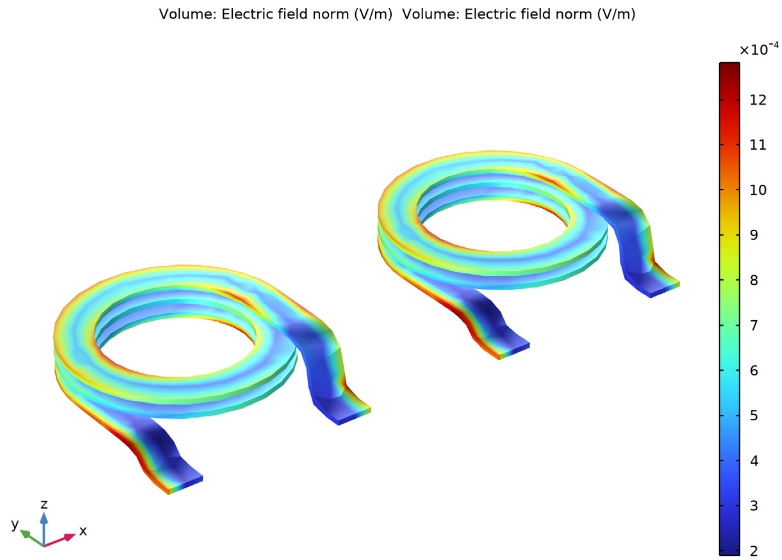

Figure 3: Comparison of the electric field distribution on the coil between ungauged formulation (left) and gauged formulation (right).

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

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**.

Magnetic Insulation 1

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

Electric Insulation 1

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

2 Select Boundaries 1–5 and 79 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**.

Gauge Fixing for A-field 1

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

MATERIALS

Core Material

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

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

3 Select Domain 2 only.

4 Locate the **Material Contents** section. 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

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

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.

7 Select the **Maximum element size** check box. In the associated text field, type `5[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 **Layers** section.

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

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `0.5[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]`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Magnetic and Electric Fields (mef)>Gauge Fixing for A-field 1**.

6 Right-click and choose **Disable**.

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**, locate the **General** section.

5 From the **Solver** list, choose **GMRES**.

6 In the **Number of iterations before restart** text field, type `500`.

7 From the **Preconditioning** list, choose **Right**.

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.

8 Click to expand the **Error** section. In the **Factor in error estimate** text field, type `1e7`.

9 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** and choose **SOR**.

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

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

12 In the **Label** text field, type `Ungauged Formulation`.

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

ADD STUDY

Next, solve the model with Gauge Fixing for A-field. Gauge fixing is compatible with direct solver which essentially does not require any tuning.

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Frequency Domain**.

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

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

STUDY 2

Step 1: Frequency Domain

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

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

Solution 2 (sol2)

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 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

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

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

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, type `Gauged Formulation` in the **Label** text field.

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

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

Next instructions show how to extract the conductance and inductance.

RESULTS

Global Evaluation 1

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

2 Right-click **Results>Derived Values** and choose **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).

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

4 In the table, enter the following settings:

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

5 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 **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 next to **Evaluate**, then choose **Table 1 - Global Evaluation 1**.

Global Evaluation 1

Next compute the new estimate of conductance and inductance with Gauged Formulation.

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

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

3 From the **Dataset** list, choose **Gauged Formulation/Solution 2 (sol2)**.

4 Click next to **Evaluate**, then choose **Table 1 - Global Evaluation 1**.

TABLE

1 Go to the **Table** window.

2 Right-click **Results>Derived Values>Global Evaluation 1** and choose **Plot**.

RESULTS

Next compare the electric potential and electric field distributions between results from Ungauged and Gauged formulations.

Electric Potential, Comparison

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

2 In the **Settings** window for **3D Plot Group**, type `Electric Potential, Comparison` in the **Label** text field.

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

Volume 1

1 Right-click **Electric Potential, Comparison** and choose **Volume**.

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

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

Selection 1

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

2 Select Domain 3 only.

Volume 2

1 In the **Model Builder** window, under **Results>Electric Potential, Comparison** right-click **Volume 1** and choose **Duplicate**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Dataset** list, choose **Gauged Formulation/Solution 2 (sol2)**.
4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

Translation 1

1 Right-click **Volume 2** and choose **Translation**.
2 In the **Settings** window for **Translation**, locate the **Translation** section.
3 In the **x** text field, type `0.2`.
4 Click the **Go to Default View** button in the **Graphics** toolbar.
5 Click the **Show Grid** button in the **Graphics** toolbar.
6 In the **Electric Potential, Comparison** toolbar, click **Plot**.

Electric Field, Comparison

1 In the **Model Builder** window, right-click **Electric Potential, Comparison** and choose **Duplicate**.
2 In the **Model Builder** window, click **Electric Potential, Comparison 1**.
3 In the **Settings** window for **3D Plot Group**, type `Electric Field, Comparison` in the **Label** text field.

Volume 1

1 In the **Model Builder** window, click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 In the **Expression** text field, type `mef.normE`.

Volume 2

1 In the **Model Builder** window, click **Volume 2**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 In the **Expression** text field, type `mef.normE`.
4 In the **Electric Field, Comparison** toolbar, click **Plot**.

Finally visualize the potential distribution in a new plot group.

Electric Potential, Gauged Formulation

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

2 In the **Settings** window for **3D Plot Group**, type `Electric Potential, Gauged Formulation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Gauged Formulation/ Solution 2 (sol2)**.

Surface 1

Right-click **Electric Potential, Gauged Formulation** 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.

Electric Potential, Gauged Formulation

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

1 In the **Model Builder** window, under **Results** click **Electric Potential, Gauged Formulation**.

2 In the **Electric Potential, Gauged Formulation** toolbar, click **Plot**.

3 Click **Plot**.