COMSOL MULTIPHYSICS™

Inductance of a Power Inductor

SOLVED WITH COMSOL MULTIPHYSICS 3.3

COMSOL

Inductance of a Power Inductor

The model shows an inductance calculation on a large 3D geometry using higher-order vector elements and memory-efficient iterative solver settings.

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 relative low voltage and high power consumption puts 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 warm 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 model 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 model uses the Quasi-static application mode taking electric induced and inductively 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. In addition, it must also solve for the Gauge fixing on the vector potential,

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

At low frequencies the inductance is almost constant. For high frequencies the capacitive effects play a role, and the permeability usually decreases, causing a frequency-dependent inductance. A model limiting factor in increasing the frequency is the skin depth, so at a frequency in the vicinity of 10 kHz the model either needs a finer mesh at the conductor boundary or an impedance boundary condition. The following table lists the material properties used in this model.

MATERIAL PARAMETER	COPPER	CORE
σ	5.997e7	10
ε_r	1	1
μ_r	1	1e3

Using a low conductivity for the surrounding air improves the stability of the iterative solver. This has negligible impact on the solution.

The outer boundaries are mainly magnetic insulation and electric insulation,

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

For the boundaries to the conductor, one end is grounded, and the other end has a port boundary condition. The port boundary condition gives the impedance of the inductor, and you can calculate the inductance from the formula,

$$L_{11} = \frac{\mathrm{Im}(Z_{11})}{\omega}$$

where ω is the angular frequency, and $\mathrm{Im}(Z_{11})$ is the imaginary part of the impedance.

COMPUTING THE SOLUTION

An analysis using the AV formulation with gauge fixing turned on consumes large amounts of memory when you use direct solvers like UMFPACK or SPOOLES. This model has almost 200,000 degrees of freedom, so an iterative solver is necessary. The geometric multigrid (GMG) solver with the Vanka pre- and postsmoother solves this problem efficiently. Different element order defines the multigrid hierarchy, which means that a direct solver solves the problem on linear vector and Lagrange elements. The iterative solver then produce the solution for the quadratic versions of these elements.

Results and Discussion

At a frequency of 1 kHz the inductance is 97 μH, and the figure below shows the electric potential and the magnetic flux density in a combined plot.

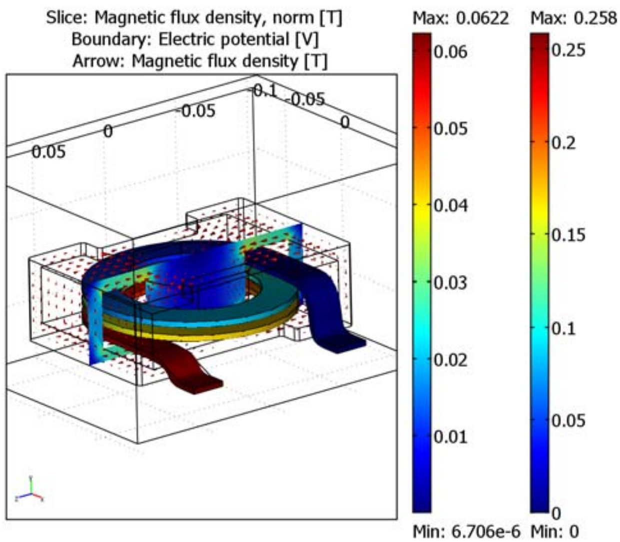


Figure 1: The final plot of the power inductor, showing the potential on the coil, the magnitude of the flux density inside the ferrite core, and the direction of the same as arrows.

Model Library path: `AC/DC_Module/Electrical_Components/power_inductor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **3D** in the **Space dimension** list.

2 In the list of application modes, select **AC/DC Module>Quasi-Statics, Electromagnetic>Electric and Induction Currents**.

3 From the **Element** list, select **Vector, Lagrange - Quadratic**.

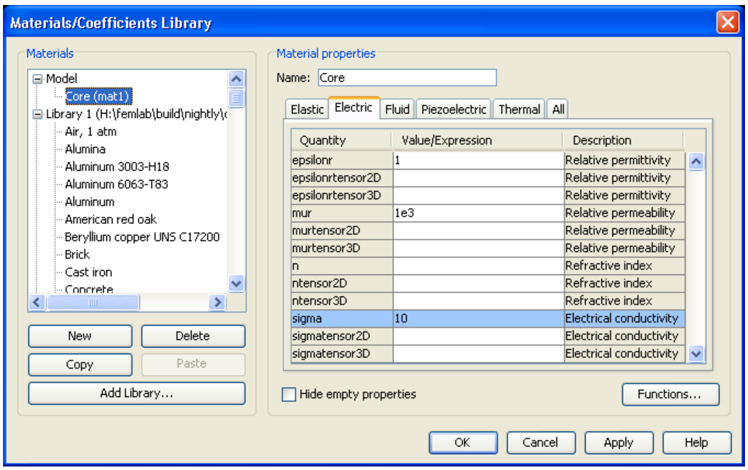
4 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Material/Coefficient Library**. In the dialog box that appears, click the **New** button.

2 Type `Core` in the **Name** edit field for the new material.

3 Click the **Electric** tab and type the values for each material parameter listed in the table below.



MATERIAL PARAMETER	CORE
σ	10
ε_r	1
μ_r	1e3

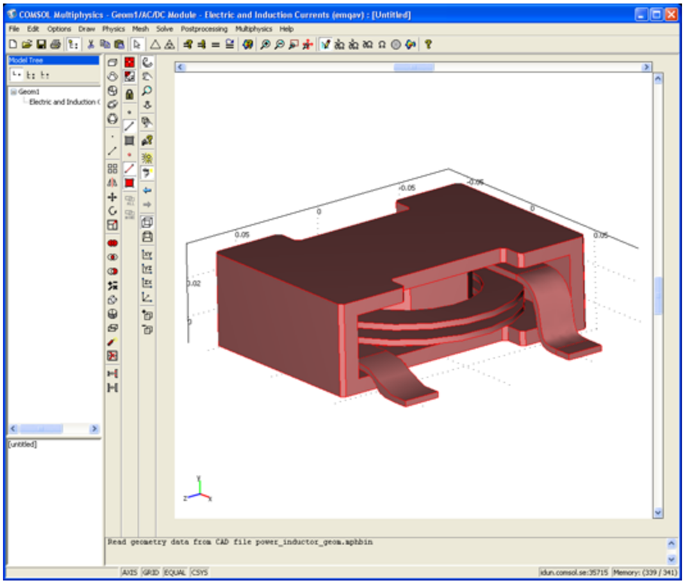
4 Click **OK**.

GEOMETRY MODELING

1 From the **File** menu, choose **Import>CAD Data From File**.

2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.

3 Locate the `power_inductor_geom.mphbin` file in the model path specified on page 3, and click **Import**.



4 From the **Draw** menu, choose **Block**.

5 In the dialog box that appears, define the block properties according to the table below.

LENGTH X	LENGTH Y	LENGTH Z	BASE	AXIS BASE POINT (X,Y,Z)
0.15	0.12	0.2	Corner	(-0.07,-0.031,-0.1)

PHYSICS SETTINGS

Scalar Variables

1 Open the **Scalar Variables** dialog box from the **Physics** menu.

2 Type `1e3` for the frequency, **nu_emqav**. Click **OK**.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Select all boundaries and make sure that the default **Magnetic insulation** is selected from the **Boundary condition** list.

3 Click the **Electric Parameters** tab. Select **Electric insulation** from the **Boundary condition** list.

4 Select boundary number 78, which is one of the boundaries for the inductor coil. Choose the **Ground** boundary condition from the **Boundary condition** list.

5 Select the other coil boundary, which is number 79. Select the **Port** boundary condition.

6 Next, click the **Port** tab, and select the **Use port as input** check box. Leave the other settings at their defaults, which is `1` for **Port number** and **Fixed current density** for **Input property**.

7 Click **OK**.

Subdomain Settings

1 Open the **Subdomain Settings** dialog box from the **Physics** menu.

2 Select subdomain 1 and click **Electric Parameters** tab. Type `1` in the edit field for the electric conductivity.

3 Select subdomain 2 and select **Core** from the **Materials** list. This is the material you defined in the Materials library earlier.

4 Select subdomain 3 and click the **Load** button. Locate and select **Copper** in the dialog box and click **OK**.

5 Click **OK** again to close the **Subdomain Settings** dialog box.

MESH GENERATION

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

2 Select **Coarse** mesh size from the **Predefined mesh sizes** list, and click the **Custom mesh size** option button.

3 Type 1.8 in the **Element growth rate** edit field, `0.4` in the **Mesh curvature factor** edit field, and type `0.02` in the **Mesh curvature cutoff** edit field.

4 Click **Remesh** and then click **OK**.

COMPUTING THE SOLUTION

The program can use the linear order element combination for a coarse solution and solve for the quadratic elements using the geometric multigrid solver. The linear solution is then used in the preconditioning step. COMSOL Multiphysics does this automatically with the setting selected in step 3.

1 Open the **Solver Parameters** dialog box from the **Solve** menu.

2 Select **Geometric multigrid** from the **Linear system solver** list.

3 Click the **Settings** button. In the dialog box that appears, make sure that **Linear system solver** is selected in the field to the left. Then select **Lower element order first** from the **Hierarchy generation method** list.

4 Enter 5 in the **Factor in error estimate** edit field.

5 Select **Coarse solver** from the field. Choose **PARDISO** from the **Coarse solver** list. Set the **Tolerance** to 1.0E-5.

6 All other settings can be left at their default values. For details on the default settings, see "Solving Large 3D Problems" on page 67 of the *AC/DC Module User's Guide*.

7 Click **OK** to close the **Linear System Solver Settings** dialog.

8 Click the **Stationary** tab, and then select **Linear** in the **Linearity** list. This model is linear, but the solver interprets the model as nonlinear, because of the coupling variable that the port condition introduces. Furthermore, with the additional dependent variable for the gauge fixing, the model has large differences in scale, which makes it difficult for the nonlinear solver to converge.

9 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Make sure that the **Slice**, **Boundary**, **Arrow**, and **Geometry edges** check boxes are selected under the **General** tab.

3 Click on the **Slice** tab, and select **Magnetic flux density, norm** from the **Predefined quantities** list.

4 Type 1 in the **x levels** edit field.

5 Click on the **Boundary** tab, and select **Electric potential** from the **Predefined quantities** list.

6 Click the **Arrow** tab, and select **Magnetic flux density** from the **Predefined quantities** list.

7 In the **x points**, **y points**, and **z points** edit fields for the **Number of points**, type 20, 7, and 20, respectively.

8 Select the **Cone** in the **Arrow type** list.

9 Click **OK**.

10 It is now necessary to remove some boundaries and subdomains from the plot. From the **Options** menu choose **Suppress>Boundaries**. Select all boundaries that are

part of the coil. Click **Apply**, then click the **Invert Suppression** button, and finally click **Cancel**.

11 From the **Options** menu, choose **Suppress>Subdomains**. Select subdomain 1 and click **OK**.

12 Click on the **Postprocessing mode** button, and the plot in Figure 1 on page 3 should appear after proper rotation and zoom operations.

13 To get the inductance value, choose **Data Display>Global** from the **Postprocessing** menu. Type `imag(Z11_emqav)/omega_emqav` in the **Expression** edit field.

You should get a value close to 97 μH in the message field when you click **OK**.