Created in COMSOL Multiphysics 6.1

Integrated Square-Shaped Spiral Inductor

Introduction

This example presents a model of a microscale square inductor, used for LC bandpass filters in microelectromechanical systems (MEMS).

The purpose of the application is to calculate the self-inductance of the microinductor. Given the magnetic field, you can compute the self-inductance, L, from the relation

$$L = \frac{2W_m}{I^2}$$

where W_m is the magnetic energy and I is the current. The application uses the Terminal boundary condition, which sets the current to 1 A and automatically computes the self-inductance. The self-inductance L becomes available as the L_{11} component of the inductance matrix.

Model Definition

The model geometry consists of the spiral-shaped inductor and the air surrounding it. Figure 1 shows the inductor and air domains used in the model. The outer dimensions of the model geometry are around 0.3 mm.

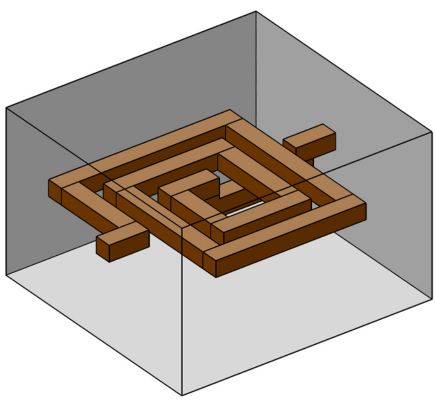

Figure 1: Inductor geometry and the surrounding air domain.

The model equations are the following:

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}_\mathrm{e}) = 0$$

$$\nabla \times (\mu_0^{-1} \mu_\mathrm{r}^{-1} \nabla \times \mathbf{A}) + \sigma \nabla V = \mathbf{J}_\mathrm{e}$$

In the equations above, σ denotes the electrical conductivity, $\mathbf{A}$ the magnetic vector potential, V the electric scalar potential, $\mathbf{J}_e$ the externally generated current density vector, μ_0 the permeability in vacuum, and μ_r the relative permeability.

The electrical conductivity in the coil is set to 10^6 S/m, and in the air it is set to 1 S/m. The conductivity of air is set to a small nonzero value in order to avoid singularities in the model. The resulting error is negligible as long as the value of the conductivity in the air is small compared to the other conductivities in the model.

The constitutive relation is given by the expression

$$\mathbf{B} = \mu_0\mu_r\mathbf{H}$$

where $\mathbf{H}$ denotes the magnetic field.

The boundary conditions are of three different types corresponding to the three different boundary groups; see Figure 2 (a), (b), and (c).

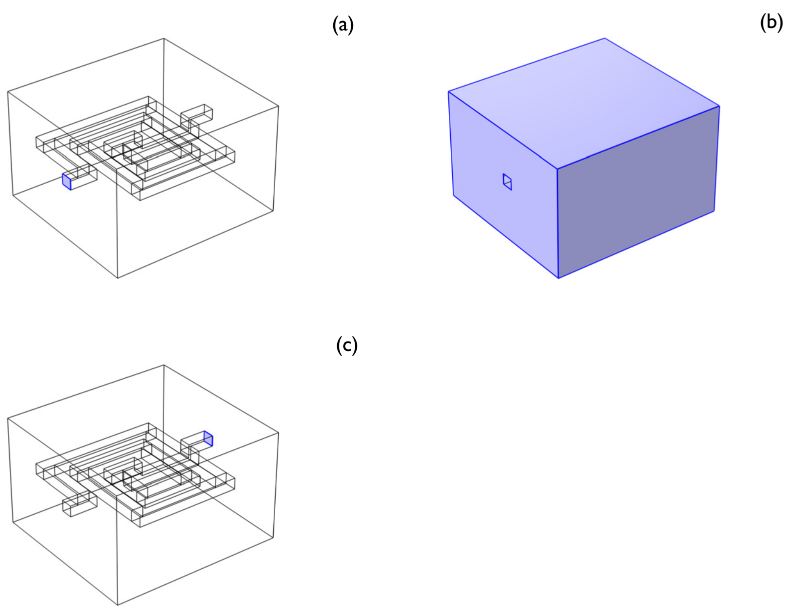

Figure 2: Boundaries with the same type of boundary conditions.

The boundary condition for the boundary highlighted in Figure 2 (a) is a magnetic insulation boundary with a terminal boundary condition. For the boundaries in Figure 2 (b), both magnetic and electric insulation prevail. The condition for the last boundary, Figure 2 (c), is magnetic insulation set to a constant electric potential of 0 V (ground).

Results

Figure 3 shows the electric potential in the inductor and the electric field lines.

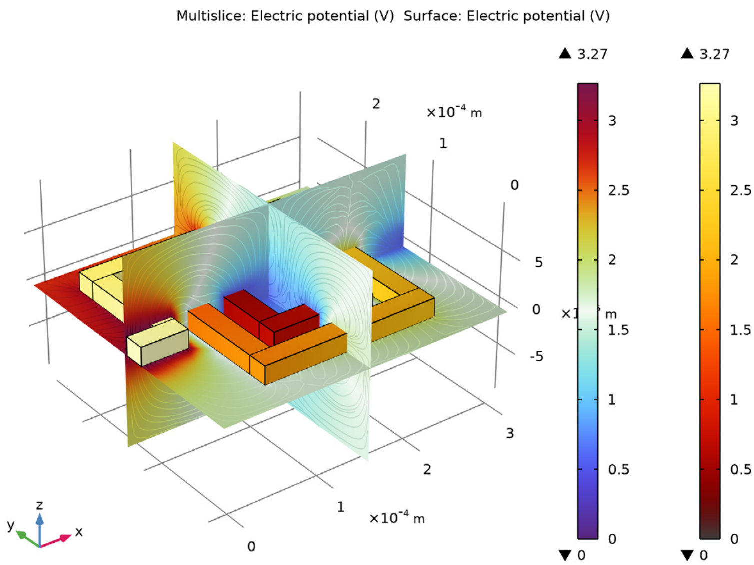

Figure 3: Electric potential in the device and electric field lines around the device.

The computed self-inductance is 0.75 nH.

Application Library path: `ACDC_Module/Introductory_Electromagnetics/spiral_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>Stationary**.

6 Click **Done**.

GEOMETRY 1

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 `spiral_inductor.mphbin`.

5 Click **Import**.

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

This geometry would be relatively straightforward to create from scratch; here it is imported for convenience.

MATERIALS

Conductor

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 `Conductor` 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	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1e6	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Air

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

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

3 Select Domain 1 only.

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	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Setting the conductivity to zero in the air would lead to a numerically singular problem. You can avoid this problem by using a small nonzero value. As 1 S/m is much less than the electric conductivity in the inductor, the fields will only be marginally affected.

MAGNETIC AND ELECTRIC FIELDS (MEF)

Magnetic Insulation 1

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

Terminal 1

1 In the **Physics** toolbar, click **Attributes** and choose **Terminal**.
2 Select Boundary 5 only.
3 In the **Settings** window for **Terminal**, locate the **Terminal** section.
4 In the I_0 text field, type 1.

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 Select Boundaries 1–4, 10, and 75 only.

   This concludes the boundary settings. Note that the boundaries that you have not assigned are electrically grounded and magnetically insulated by default.

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 **Coarser**.
4 In the **Home** toolbar, click **Build Mesh**.

STUDY 1

Click **Compute**.

One of the default plots shows the electric potential distribution in three cross sections. There are plenty of other ways of visualizing the solution. The following instructions detail how to also show the potential distribution inside the coil.

RESULTS

Surface 1

1 In the **Model Builder** window, right-click **Electric Potential (mef)** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Thermal>GrayBody** in the tree.

5 Click **OK**.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

Selecting the boundaries of the inductor is most effectively done by first selecting all boundaries, then removing the exterior boundaries of the air box from the 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 **All boundaries**.

5 Select Boundaries 5–9, 11–74, and 76 only.

Electric Potential (mef)

1 In the **Model Builder** window, under **Results** click **Electric Potential (mef)**.

2 In the **Electric Potential (mef)** toolbar, click **Plot**.

Global Evaluation 1

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

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Magnetic and Electric Fields>Terminals>mef.L11 - Inductance - H**.

3 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The inductance evaluates to 0.75 nH.