Inductor in Amplifier Circuit

SOLVED WITH COMSOL MULTIPHYSICS 3.3
**Inductor in Amplifier Circuit**

This model studies a finite element inductor inserted in an electrical amplifier circuit.

**Introduction**

Modern electronic systems are very complex and depend heavily on computer aided design in the development and manufacturing process. Common tools for such calculations are based on the SPICE format originally developed at Berkeley University (Ref. 1). The SPICE format consists of a standardized set of models for describing electrical devices—especially semiconductor devices such as transistors, diodes, and thyristors. SPICE also includes a simple, easy-to-read text format for circuit netlists and model parameter specifications. Although the netlist format is essentially the same as it was from the beginning, the set of models and model parameters are constantly changing, with new models being added according to the latest achievements in semiconductor device development. When the devices are scaled down, new effects appear that have to be properly modeled. The new models are the result of ongoing research in device modeling.

When an engineer is designing a new electronic component, like a capacitor or an inductor, the SPICE parameters for that device are not known. They are either extracted from finite element tools, such as COMSOL Multiphysics, or from measurements on a prototype. To speed up the design process it can be convenient to include the finite element model in the SPICE circuit simulation, calculating the device behavior in an actual circuit.

This model takes a simple amplifier circuit and exchanges one of its components with a finite element model of an inductor with a magnetic core. COMSOL Multiphysics calculates the transient behavior of the entire system. A script adds the circuit elements as ODE equations to the inductor model along with the necessary model parameters for the SPICE devices in the circuit.

**Model Definition**

The inductor model uses the Azimuthal induction currents application mode of the Electromagnetics module, solving for the magnetic potential $\mathbf{A}$.

\[
\sigma \frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1} \mu_r^{-1} \nabla \times \mathbf{A}) = \mathbf{J}_e
\]
where \( \mu_0 \) is the permeability of vacuum, \( \mu_r \) the relative permeability, and \( \sigma \) the electrical conductivity.

Because the inductor has a large number of turns it is not efficient to model each turn as a separate wire. Instead the entire coil is treated as a block with a constant external current density corresponding to the current in each wire. The conductivity in this block is zero to avoid eddy currents, which is motivated with the fact that no currents can flow between the individual wires. The eddy currents within each wire is neglected.

**Connection to a SPICE Circuit**

The electrical circuit is a standard amplifier circuit with one bipolar transistor, biasing resistors, input filter, and output filter, see the figure below.

![SPICE Circuit Diagram](image)

The input is a sine signal of 1 V and 10 kHz. The SPICE netlist for this circuit is shown below.

```
* BJT Amplifier circuit
.OPTIONS TNOM=27
.TEMP 27
Vin  1  0  sin(0 1 10kHz)
Vcc  4  0  15
Rg   1  2  100
Cin  2  3  10u
R1   4  3  47k
R2   3  0  10k
X1   4  5  inductor
RE   7  0  1k
```
The device X1 refers to a subcircuit defined at the end of the file. The subcircuit definition is part of the SPICE standard to define blocks of circuits that can be reused in the main circuit. The special implementation used here defines a subcircuit that really is a Comsol Multiphysics file, referenced with the option COMSOL: <file name>. The parameters V_coil and I_coil are the variables that links the model file with the circuits. These variables must be defined in the model in a certain way. The variable V_coil must give the voltage over the device, defined in the global scope. I_coil must be a global variable used in the model as a current through the device.

The script can also handle model files with more than two terminals. It is then necessary to define each terminal as a floating-potential boundary condition. Note that any use of the ground boundary condition will connect those boundaries directly to the ground node of the circuit, labeled 0.

The model parameters of the transistor do not correspond to a real device, but the numbers are nevertheless chosen so as to be realistic.

The script to import the SPICE netlist is called spiceimport.m. This script does not fully support the SPICE format; especially for the semiconductor device models, it only supports a limited set of parameters. Supplying unsupported parameters does not give an error message, but the parameter is not used in the circuit model. For example, the transit time capacitance and the temperature dependence are not supported for the transistor model.

Results and Discussion

Biasing of an amplifier is often a complicated compromise, especially if you only use resistors. Adding an inductor as the collector impedance simplifies the biasing design, since the instantaneous voltage on the collector of the transistor can be higher than the supply voltage, which is not possible with resistors. Amplifiers using inductors can be quite narrow banded.
Before starting the transient simulation, proper initial conditions have to be calculated. For this model it is sufficient to ramp the supply voltage to 15 V with the nonlinear parametric solver. After the ramp, the DC bias conditions have been calculated properly and this solution can be used as initial condition for the transient simulation. After 0.2 ms the magnetic field and contour lines of the vector potential should look like Figure 1 below.

![Figure 1: Magnetic flux density (color) after the bias point calculation.](image)

Figure 1: Magnetic flux density (color) after the bias point calculation.
Using the global variable plot, input signal, output signal, and inductor voltage can easily be plotted in the same graph.

![Graph showing input signal (U_VIN_cir), output signal (U_RL_cir), and inductor voltage (U_X1_cir) as a function of time.]

**Figure 2:** Input signal (U_VIN_cir), output signal (U_RL_cir), and inductor voltage (U_X1_cir) as a function of time.

The output signal is about 1.5 times the input signal in amplitude.

**Reference**


**Model Library path:**

AC/DC_Module/Electrical_Components/amplifier_and_inductor
Modeling Using the Graphical User Interface

**MODEL NAVIGATOR**
1. In the **Model Navigator**, choose **Axial symmetry (2D)** in the **Space dimension** list.
2. In the **AC/DC Module** folder, select **Statics>Magnetostatics>Azimuthal Induction Currents, Vector Potential**.
3. Click **OK** to close the **Model Navigator**.

**OPTIONS AND SETTINGS**
1. From the **Options** menu, choose **Constants**.
2. In the **Constants** dialog box, define the following constants with names and expressions. The description field is optional.

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>t</td>
<td>0</td>
<td>Time for stationary solution (s)</td>
</tr>
<tr>
<td>N</td>
<td>1e3</td>
<td>Turns in coil</td>
</tr>
<tr>
<td>freq</td>
<td>10e3[Hz]</td>
<td>Frequency</td>
</tr>
<tr>
<td>r_coil</td>
<td>0.05e-3[m]</td>
<td>Wire radius in coil</td>
</tr>
<tr>
<td>sigma_coil</td>
<td>5e7[S/m]</td>
<td>Wire conductivity</td>
</tr>
</tbody>
</table>
3. Click **OK**.
4. From the **Options** menu, choose **Expressions>Global Expressions**.
5. In the **Global Expressions** dialog box, define the following variables with names and expressions. The description field is optional.

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>I_coil</td>
<td>1[A]</td>
<td>Current in coil</td>
</tr>
<tr>
<td>J_coil</td>
<td>I_coil*N/A</td>
<td>Equivalent current density in coil</td>
</tr>
</tbody>
</table>
6. Click **OK**.

**GEOMETRY MODELING**
1. Choose **Draw>Specify Objects>Circle** to create a circle with the following properties:

<table>
<thead>
<tr>
<th>NAME</th>
<th>RADIUS</th>
<th>BASE</th>
<th>R</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>0.03</td>
<td>Center</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
2 Choose `Draw>Specify Objects>Rectangle` to create rectangles with the following properties:

<table>
<thead>
<tr>
<th>NAME</th>
<th>WIDTH</th>
<th>HEIGHT</th>
<th>BASE</th>
<th>R</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>R1</td>
<td>5e-3</td>
<td>2e-2</td>
<td>Corner</td>
<td>0</td>
<td>-1e-2</td>
</tr>
<tr>
<td>R2</td>
<td>3e-3</td>
<td>2e-2</td>
<td>Corner</td>
<td>7.5e-3</td>
<td>-1e-2</td>
</tr>
<tr>
<td>R3</td>
<td>0.04</td>
<td>0.08</td>
<td>Corner</td>
<td>0</td>
<td>-0.04</td>
</tr>
</tbody>
</table>

3 Click the `Zoom Extents` toolbar button to see the objects.
4 Select the circle and the third rectangle (C1 and R3), and click the `Intersection` toolbar button.
5 Open the `Fillet/Chamfer` dialog box from the `Draw` menu.
6 In the dialog box, expand rectangle R1 and select Vertices 2 and 3. Then expand rectangle R2 and select all vertices there. Hold down the Ctrl key to do multiple selections.
7 In the `Radius` edit field type `5e-4`.
8 Click `OK`.

**PHYSICS SETTINGS**

**Variables**
1 From the `Options` menu, point to `Integration Coupling Variables` and click `Subdomain Variables`.
2 In the `Subdomain Integration Variables` dialog box, define two variables with integration order 4 and global destination according to the table below.

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION IN SUBDOMAIN 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>V_coil</td>
<td>N<em>2</em>pi<em>r</em>I_coil/(sigma_coil<em>pi</em>r_coil^2)</td>
</tr>
<tr>
<td>A</td>
<td>1</td>
</tr>
</tbody>
</table>

3 Click `OK`.

**Subdomain Settings**
1 Open the `Subdomain Settings` from the `Physics` menu.
2 Select Subdomain 2 and click the `Load` button to open the `Materials/Coefficients Library` dialog box.
3 In the dialog box, select the `Soft Iron` material under the `acdc_lib.txt` library. (You may have to scroll both vertically and horizontally to see the library name.)
4 Click OK to close the Materials/Coefficients Library dialog box.
5 Select Subdomain 3 and enter \( J_{\text{coil}} \) in the External current density edit field.
6 Click the Init tab, select all subdomains, and type 1 in the edit field for the initial value.
7 Click OK.

**Boundary Settings**
1 Open the Boundary Settings from the Physics menu.
2 Select Boundaries 1, 2, and 4, then choose Axial symmetry from the Boundary condition list. For all other boundaries keep the default condition, Magnetic insulation.
3 Click OK.

**Mesh Generation**
1 Open the Free Mesh Parameters dialog box from the Mesh menu.
2 Click the Custom mesh size option button and type 4 in the Resolution of narrow regions edit field.
3 Click Remesh and then OK.

**Computing the Solution**
1 Click the Solve button.
2 From the File menu, choose Save As. Save the model to the file name amplifier_and_inductor_nocircuit.mph. It is important that this file name is the same as the file name specified in the circuit file. Also remember the location where you save the file.

This completes the first part of the model, which is to create the model file of the inductor that is connected to the circuit file later.

**SPICE Import**
These steps require that you have access to COMSOL Script or MATLAB. If you use MATLAB, COMSOL Multiphysics must be opened with MATLAB.
1 First create the circuit file in an editor of your choice. Create the text content as it is shown below:

```plaintext
* BJT Amplifier circuit
.OPTIONS TNOM=27
.TEMP 27
Vin 1 0 sin(0 1 10kHz)
```
Vcc  4  0  15
Rg  1  2  100
Cin  2  3  10u
R1  4  3  47k
R2  3  0  10k
X1  4  5  inductor
RE  7  0  1k
Cout  5  6  10u
Rl  6  0  10k
Q1  5  3  7  BJT

.model BJT NPN(Is=15f Ise=15f Isc=0 Bf=260 Br=6.1
+ Ikf=.3 Xtb=.5 Xtg=.35 Vjc=.75 Fo=.5 Cj=.2 Cje=0.4 Vje=.75
+ Vaf=.75 Xtf=3 Xti=3)

.subckt inductor V_coil I_coil ...

.comsol: amplifier_and_inductor_nocircuit.mph
.ends

2  Save the file as amplifier.cir in the same location as the model file you saved
earlier. Then go back to the COMSOL Multiphysics window.

3  Go to the COMSOL Script window. You open the window with the File>COMSOL Script
menu item.

4  At the command prompt, first make sure that you are in the correct folder where
the model file and circuit file are stored, then enter the following command.

    fem = spiceimport('amplifier.cir');

5  Go back to the COMSOL Multiphysics window and choose Import>FEM structure from
the File menu. Type fem in the dialog box that appears and click OK.

The model is the same as before but with constants, global variables, and ODE
variables added that represent the amplifier circuit. The syntax of these variables are
related to the names in the file. The variable I_R1_cir stands for the current in resistor
R1, value_VCC_cir is the voltage value of the supply voltage generator VCC, and
U_X1_cir is the voltage over the device X1, which is the inductor model you created
earlier. The variables beginning with the letter V are node potentials referenced to the
ground node, 0, of the circuit.

Computing the Solution with a Circuit

1  Open Solver Parameters from the Solve menu.

2  In the Solver Parameters dialog box, select Parametric from the Solver list.

   1. ‘...’ Indicates that the line break should be omitted in the amplifier.cir file. I.e. “.SUBCKT
      inductor V_coil I_coil COMSOL: amplifier_and_inductor_nocircuit.mph” must appear on one line.
3 In the Name of parameter edit field type value_VCC_cir, and in the List of parameter values edit field type 1:15.

4 Click OK.

5 Click the Restart button.

This solving step ramps the supply voltage stored in value_VCC_cir up to 15 V. The ramp is necessary to handle the highly nonlinear effects in the bipolar transistor model. The circuit is now at its bias point, and the plot should look like Figure 1 on page 4.

1 From the Options menu point to Integration Coupling Variables and click Subdomain Variables.

2 In the Subdomain Integration Variables dialog box change the expression for the \( V_{\text{coil}} \) variable so it matches the expression below.

\[
V_{\text{coil}} = \frac{N \times 2 \pi r I_{\text{coil}}}{\sigma_{\text{coil}} \pi r_{\text{coil}}^2 - E_{\Phi_{\text{emqa}}}/A}
\]

3 Click OK.

4 Open Solver Parameters from the Solve menu.

5 In the Solver Parameters dialog box select Transient from the Analysis list.

6 Type \texttt{linspace(0,5*10^{-4},101)} in the Times edit field, type \texttt{1*10^{-4}} in the Relative tolerance edit field, and type \texttt{1*10^{-6}} in the Absolute tolerance edit field. The last steps are because the accuracy obtained with default error tolerances is not sufficient in this model.

7 Click OK.

8 Click the Restart toolbar button.
**POSTPROCESSING AND VISUALIZATION**

1. Open **Global Variables Plot** from the **Postprocessing** menu.

2. From the **Predefined quantities** list select the variables **U_X1_cir**, **U_VIN_cir**, and **U_RL_cir**. Click the > button to add the selected variables to the **Quantities to plot** list.

3. From the **Solutions to use** list, select all the time steps from **4E-4** to **5E-4**.

4. Click the **Line settings** button. In the dialog box that appears, select **Cycle** from the **Line marker** list, and select the **Legend** check box. Click **OK**.

5. Click **OK** to see the plot in Figure 2 on page 5.