COMSOL MULTIPHYSICS™

Inductor in Amplifier Circuit

SOLVED WITH COMSOL MULTIPHYSICS 3.3

COMSOL

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 $\mathbf{A}$.

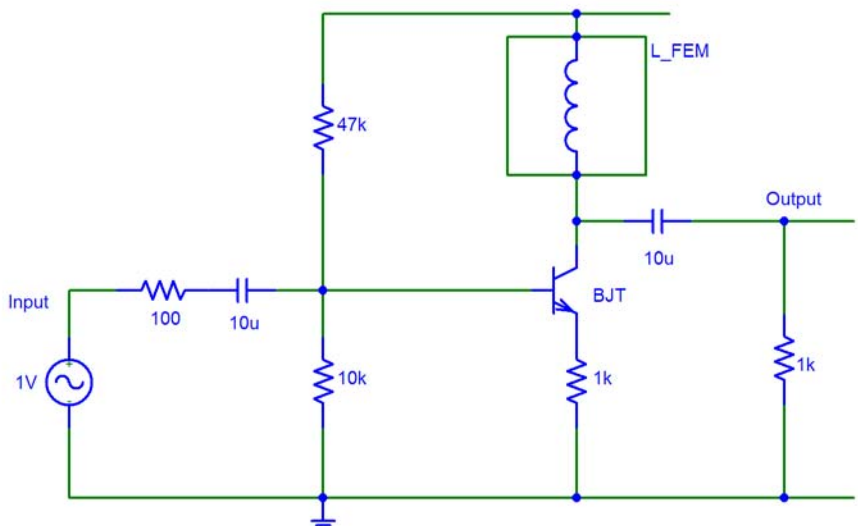
$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \mathbf{J}_e$$

where μ_0 is the permeability of vacuum, μ_r the relative permeability, and σ 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.



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
```

```
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=1.5 Ne=1.3 Nc=2 Rc=1  Rb=10 Eg=1.11
+ Cjc=7.5p Mjc=.35 Vjc=.75 Fc=.5 Cje=20p Mje=0.4 Vje=0.75
+ Vaf=75 Xtf=3 Xti=3)
.SUBCKT inductor V_coil I_coil COMSOL: amplifier_and_inductor.mph
.ENDS
.END
```

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.

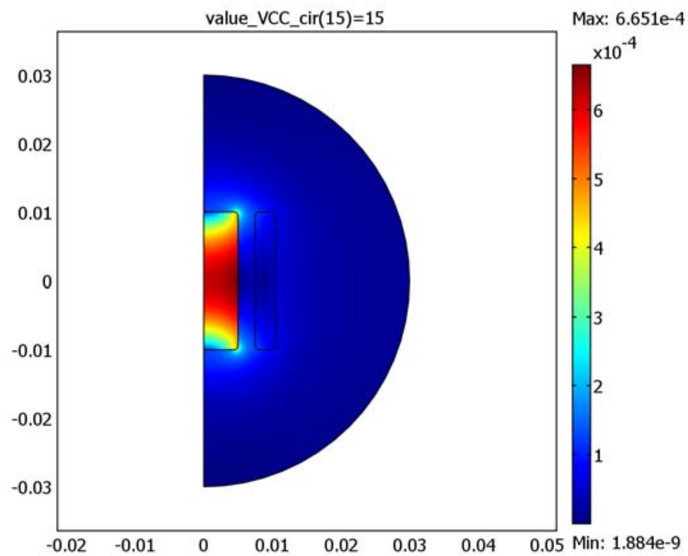


Figure 1: Magnetic flux density (color) after the bias point calculation.

Using the global variable plot, input signal, output signal, and inductor voltage can easily be plotted in the same graph.

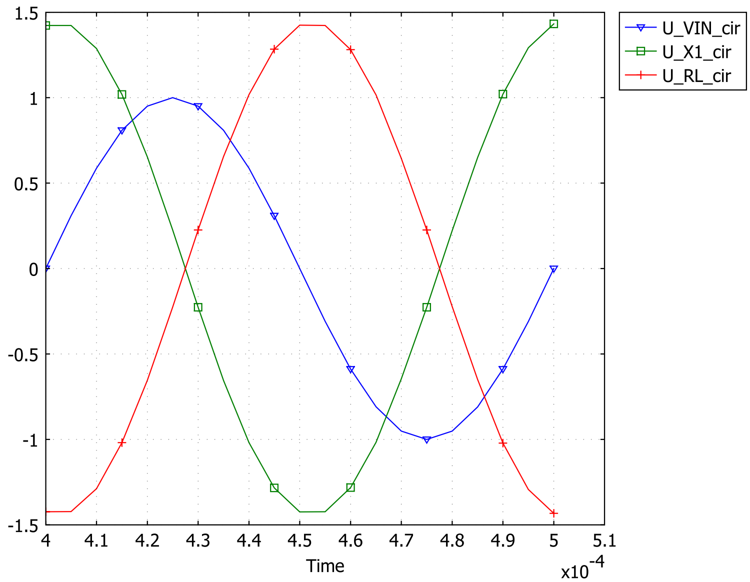


Figure 2: Input signal (`U_VIN_cir`*), output signal (*`U_RL_cir`*), and inductor voltage (*`U_X1_cir`*) as a function of time.*

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

Reference

1. “The SPICE home page,” http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE.

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

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**.

1 In the **Constants** dialog box, define the following constants with names and expressions. The description field is optional.

NAME	EXPRESSION	DESCRIPTION
t	0	Time for stationary solution (s)
N	1e3	Turns in coil
freq	10e3[Hz]	Frequency
r_coil	0.05e-3[m]	Wire radius in coil
sigma_coil	5e7[S/m]	Wire conductivity

2 Click **OK**.

3 From the **Options** menu, choose **Expressions>Global Expressions**.

4 In the **Global Expressions** dialog box, define the following variables with names and expressions. The description field is optional.

NAME	EXPRESSION	DESCRIPTION
I_coil	1[A]	Current in coil
J_coil	I_coil*N/A	Equivalent current density in coil

5 Click **OK**.

GEOMETRY MODELING

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

NAME	RADIUS	BASE	R	Z
C1	0.03	Center	0	0

2 Choose **Draw>Specify Objects>Rectangle** to create rectangles with the following properties:

NAME	WIDTH	HEIGHT	BASE	R	Z
R1	5e-3	2e-2	Corner	0	-1e-2
R2	3e-3	2e-2	Corner	7.5e-3	-1e-2
R3	0.04	0.08	Corner	0	-0.04

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.

NAME	EXPRESSION IN SUBDOMAIN 3
V_coil	N*2*pi*r*I_coil/(sigma_coil*pi*r_coil^2)
A	1

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_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.

```
* 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=1.5 Ne=1.3 Nc=2 Rc=1  Rb=10 Eg=1.11
+ Cjc=7.5p Mjc=.35 Vjc=.75 Fc=.5 Cje=20p Mje=0.4 Vje=0.75
+ Vaf=75 Xtf=3 Xti=3)
.SUBCKT inductor V_coil I_coil ...[1]
COMSOL: amplifier_and_inductor_nocircuit.mph
.ENDS
.END
```

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`.

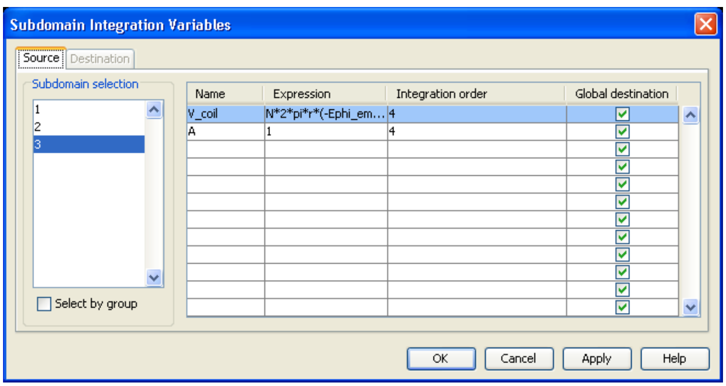
4 Click **OK**.

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_coil variable so it matches the expression below.



NAME	EXPRESSION IN SUBDOMAIN 3
V_coil	`N*2*pi*r*I_coil/(sigma_coil*pi*r_coil^2-Ephi_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 `linspace(0,5e-4,101)` in the **Times** edit field, type `1e-4` in the **Relative tolerance** edit field, and type `1e-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**.

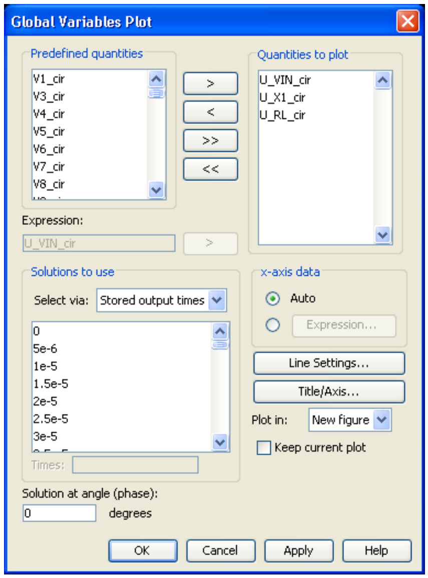
8 Click the **Restart** toolbar button.

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.