Inductor in an Amplifier Circuit

Introduction

This example studies a finite element model of an inductor inserted into an electrical amplifier circuit.

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 constantly changes, 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 application 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. Importing a SPICE circuit netlist brings in the circuit elements along with their model parameters and location in the circuit. All elements can be edited in COMSOL Multiphysics, and any pair of nodes can connect to the finite element model.

Model Definition

The inductor model uses the Magnetic Fields interface solving for the magnetic potential **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 coil has a large number of turns it is not efficient to model the separate wires and a homogenized formulation, where eddy currents within each wire are neglected, is used for the coil cross section. A dedicated coil feature is used for this purpose.

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 Figure 1.
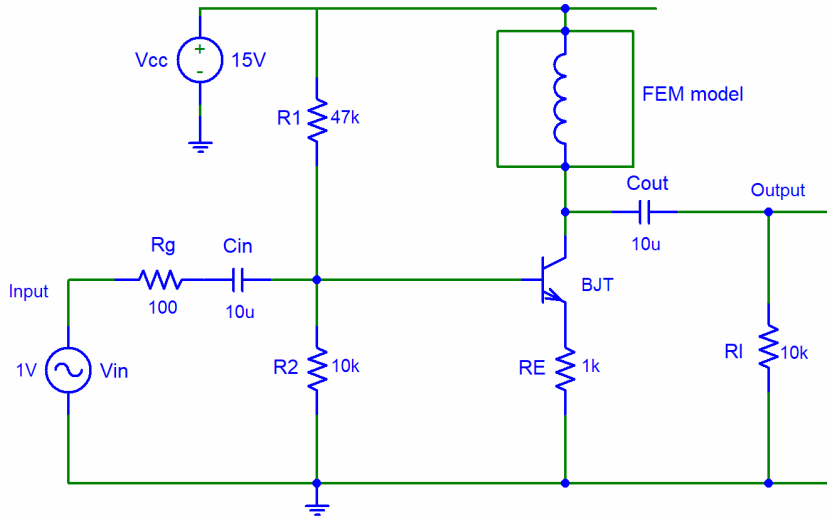

Figure 1: Circuit diagram.

The input is a sine signal of **1** V and **10** kHz. The following listing shows the SPICE netlist for this circuit:

```
* 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
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)
 .END
```

A dedicated circuit device is then added in COMSOL Multiphysics to connect the circuit with the inductor model. The model parameters of the transistor do not correspond to a real device, but the numbers are nevertheless chosen to be realistic.

The import of the SPICE netlist does not fully support the SPICE format; especially for the semiconductor device models it only supports a limited set of parameters. Supplying unsupported parameters results in those parameters not being used in the circuit model. For example, transit time capacitance and temperature dependence are not supported for the transistor model.

Results and Discussion

A first version of this application lets you compute the magnetic flux density distribution from a 1 A current through the inductor, without the circuit connection taken into consideration.
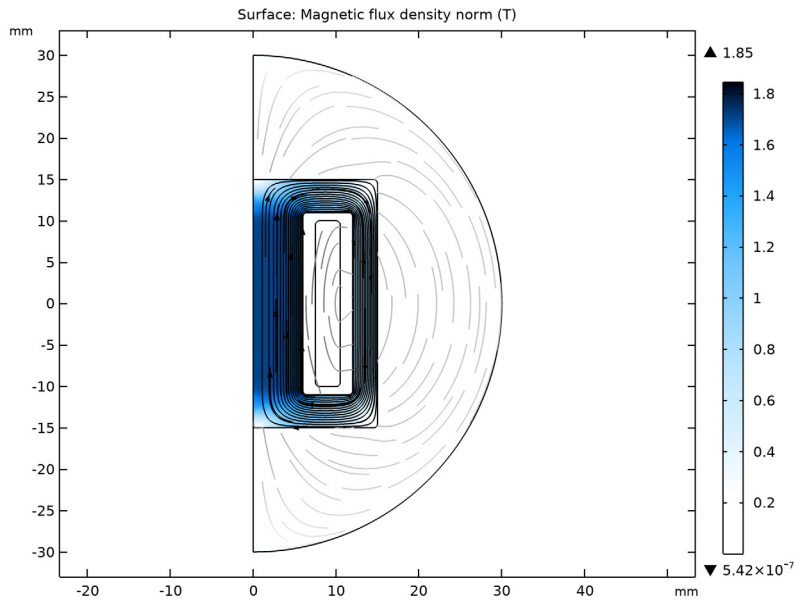

Figure 2: Magnetic flux density distribution as the coil is driven by a 1 A current source.

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, because 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 application 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 you can use this solution as initial condition for the transient simulation.

Using a global variables plot, you can easily plot input signal, output signal, and inductor voltage in the same figure.
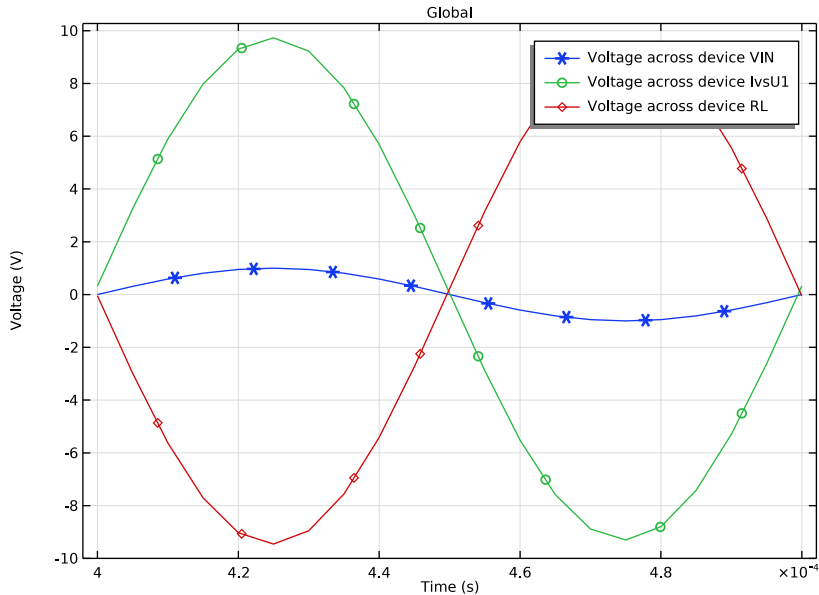

Figure 3: Input signal (cir.VIN_v), output signal (cir.RL_v), and inductor voltage (cir.X1_v) as functions of time.

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.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/`
`inductor_in_circuit`

Modeling Instructions

From the **File** menu, choose **New**.

In the **New** window, click **Model Wizard**.

M O D E L   W I Z A R D
1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

G E O M E T R Y   I
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)
1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type 30.

4 In the **Sector angle** text field, type 180.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Rectangle 1 (r1)
1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 15.

4 In the **Height** text field, type 30.

5 Locate the **Position** section. In the **z** text field, type -15.

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click ▭ **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 3.

4 In the **Height** text field, type 20.

5 Locate the **Position** section. In the **r** text field, type 7.5.

6 In the **z** text field, type -10.

7 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click ▭ **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 6.

4 In the **Height** text field, type 22.

5 Locate the **Position** section. In the **r** text field, type 6.

6 In the **z** text field, type -11.

7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click ◿ **Fillet**.

Next, select all ten points in the internal of the geometry as follows:

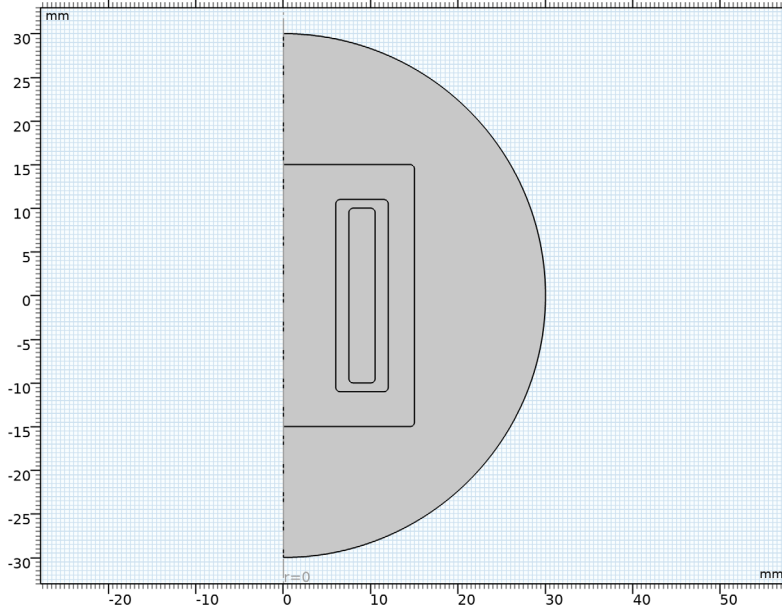**2** Click the **Select Box** button in the **Graphics** toolbar.

3 Using the mouse, enclose the internal vertices in a box to select them.
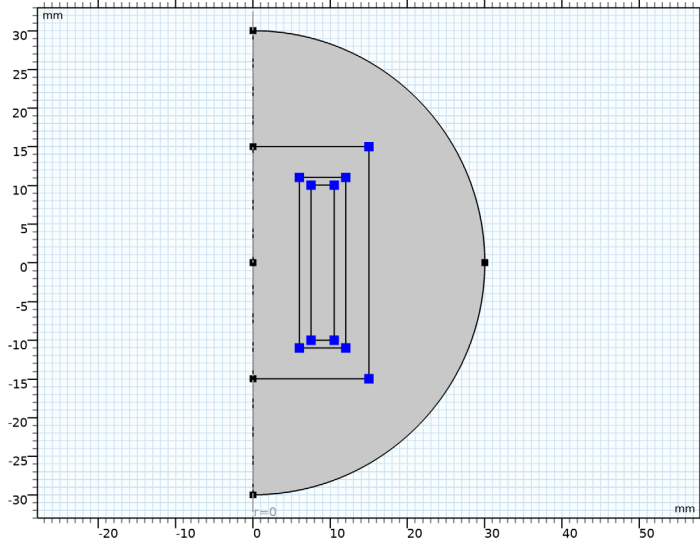
4 In the **Settings** window for **Fillet**, locate the **Radius** section.

5 In the **Radius** text field, type 0.5.

6 Click ▦ **Build All Objects**.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
t	0[s]	0 s	Time for stationary solution
N	1e3	1000	Coil turns
d_coil	0.1[mm]	1E-4 m	Coil wire diameter
sigma_coil	5e7[S/m]	5E7 S/m	Wire conductivity
Vappl	15[V]	15 V	Supply voltage

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>Air**.

4 Click **Add to Component** in the window toolbar.

5 In the tree, select **Nonlinear Magnetic>Low Carbon Steel>Low Carbon Steel 1006**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click 🔧 **Add Material** to close the **Add Material** window.

MATERIALS

Low Carbon Steel 1006 (mat2)

1 Select Domain 2 only.

This leaves your model with material data for low carbon steel in the core and air elsewhere. Note that the behavior of the coil is determined by the applied current and the resulting voltage.

MAGNETIC FIELDS (MF)

First, give the solved-for magnetic potential an initial value with a nonzero gradient. This helps the nonlinear solver avoid an otherwise singular linearization before it takes the first step.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **A** vector as

0	r
1[uWb/m^2]*r	phi
0	z

Here, the prefix u in uWb stands for micro (μWb).

Next, set up the coil.

Ampère's Law 2

1 In the **Physics** toolbar, click 🔵 **Domains** and choose **Ampère's Law**.

2 Select Domain 2 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.

4 From the **Magnetization model** list, choose **B-H curve**.

5 Locate the **Constitutive Relation Jc-E** section. From the σ list, choose **User defined**. In the associated text field, type 1000.

Usually, laminated core is used to decrease the eddy current loss. Here we set a user defined conductivity to simulate such an effect.

Coil 1

1 In the **Physics** toolbar, click ⬤ **Domains** and choose **Coil**.

2 Select Domain 4 only.

3 In the **Settings** window for **Coil**, locate the **Coil** section.

4 From the **Conductor model** list, choose **Homogenized multiturn**.

5 Locate the **Homogenized Multiturn Conductor** section. In the σ_{coil} text field, type sigma_coil.

6 Locate the **Coil** section. In the I_{coil} text field, type 0.1.

7 Locate the **Homogenized Multiturn Conductor** section. In the N text field, type N.

8 From the **Coil wire cross-section area** list, choose **From round wire diameter**.

9 In the d_{coil} text field, type d_coil.

In the I_{coil} text field, change the default value to 0.1[A].

MESH 1

The steepest field gradients and consequently the most important challenges to the convergence of this model are expected to occur in the vicinity of the fillets. You can increase the accuracy and help the solver by using a high resolution of narrow regions.

Free Triangular 1

In the **Mesh** toolbar, click ◿ **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Resolution of narrow regions** text field, type 4.
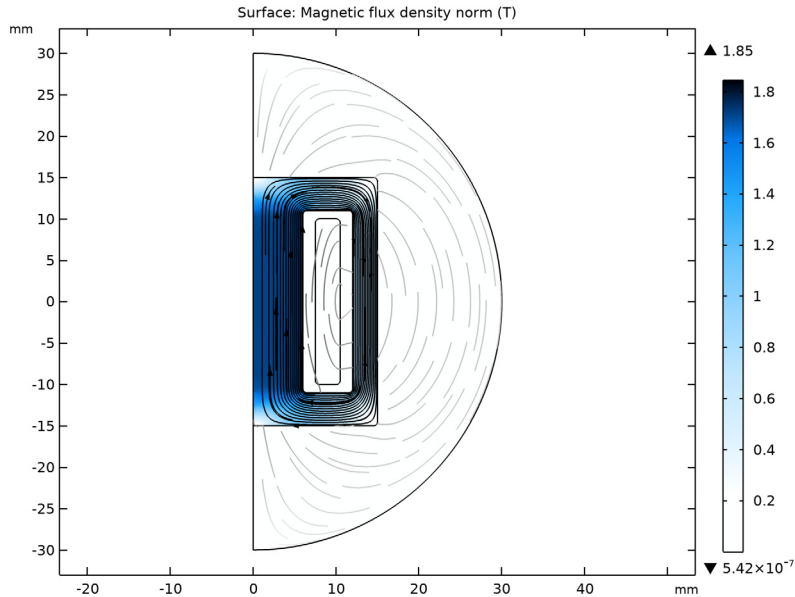
4 Click ▦ **Build All**.

STUDY 1

In the **Home** toolbar, click ▬ **Compute**.

Magnetic Flux Density Norm (mf)

The default plot shows the resulting magnetic flux density distribution from the applied 0.1 A current.

Surface: Magnetic flux density norm (T)

COMPONENT 1 (COMP1)

It is now time to add the circuit. Although you are eventually looking for transient results, the first solution step will use the stationary solver to ramp up the voltage from the voltage generator. You will therefore select a stationary study in the Model Wizard. First, prepare for the import by making the coil circuit-driven.

MAGNETIC FIELDS (MF)

To be able to keep the first version of the application fully intact, create a new **Coil** node for the circuit version of the application.

Coil 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** right-click **Coil 1** and choose **Duplicate**.

2 In the **Settings** window for **Coil**, locate the **Coil** section.

3 From the **Coil excitation** list, choose **Circuit (current)**.

4 In the **Coil name** text field, type 1.

ADD PHYSICS

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

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

3 In the tree, select **AC/DC>Electrical Circuit (cir)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 1** in the window toolbar.

6 In the **Home** toolbar, click   **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 In the **Home** 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>Stationary**.

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

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

ELECTRICAL CIRCUIT (CIR)

Voltage Source VIN (VIN)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Import SPICE Netlist**.

2 Browse to the model's Application Libraries folder and double-click the file `amplifier.cir`.

The SPICE netlist is imported in the Circuit physics. In order to couple the amplifier with the inductor, an **External I vs U** feature must be connected between nodes 4 and 5.

3 In the **Model Builder** window, click **Electrical Circuit (cir)**.

External I vs. U 1 (IvsU1)

1 In the **Electrical Circuit** toolbar, click   **External I vs. U**.

2 In the **Settings** window for **External I vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	4
n	5

Now prepare for the ramping of the voltage generator by changing the 15 V used in the voltage supply VCC to a parameter that the solver can sweep.

Voltage Source VCC (VCC)

1 In the **Model Builder** window, click **Voltage Source VCC (VCC)**.

2 In the **Settings** window for **Voltage Source**, locate the **Device Parameters** section.

3 In the v_{src} text field, type Vappl.

STUDY 1

Disable the new **Coil** node and the **Electrical Circuits** interface for **Study 1**. Conversely, you will disable the original node for the steps of **Study 2** shortly.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Magnetic Fields (mf)>Coil 2**.

5 Click ⊘ **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Electrical Circuit (cir)**.

7 Click ⊘ **Disable in Model**.

The new study already contains a node for the initial stationary solution.

STUDY 2

Time Dependent

1 In the **Study** toolbar, click ⟋ **Study Steps** and choose **Time Dependent> Time Dependent**.

To get accurate results, you need to tighten the tolerances.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type range(0,5e-6,5e-4).

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type 1e-4.

For the steps in this study, disable the original **Coil** node.

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

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Magnetic Fields (mf)>Coil 1**.

8 Click ⊘ **Disable**.

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click ＋ **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Vappl (Supply voltage)	range(1,15)	V

Using continuation rather than a parametric sweep lets you begin with a parametric solution and then use the result for the final parameter as the initial value for the time-dependent solver. In contrast, adding a parametric sweep would mean performing a transient solution for each parameter value.

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

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Magnetic Fields (mf)>Coil 1**.

8 Click ⊘ **Disable**.

Solution 2 (sol2)

1 In the **Study** toolbar, click ⊓⊓ **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-6`.

This model requires a somewhat tighter tolerance in the stationary solver than the default on account of the strong magnetic nonlinearity in the core material. A relative tolerance of 1×10^{-6} gives a very well-converged result, which is important for maintaining stability in the final time-dependent solver step.

5 In the **Model Builder** window, click **Time-Dependent Solver 1**.

6 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

7 From the **Tolerance method** list, choose **Manual**.

8 In the **Absolute tolerance** text field, type `1e-6`.

ELECTRICAL CIRCUIT (CIR)

External I vs. U 1 (IvsU1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrical Circuit (cir)** click **External I vs. U 1 (IvsU1)**.

2 In the **Settings** window for **External I vs. U**, locate the **External Device** section.

3 From the V list, choose **Coil voltage (mf)**.

STUDY 2

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

RESULTS

Magnetic Flux Density Norm (mf) 1

The new default plot shows the flux density distribution at $t = 5\times10^{-4}$ s.

Follow the instructions below to plot the input and output signals as well as the inductor voltage versus time.

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

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

4 From the **Time selection** list, choose **From list**.

5 In the **Time** list, click and Shift-click to select all times between `4e-4` and `5e-4`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Time (s)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type Voltage (V).

Global 1

1 Right-click **ID Plot Group 5** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
cir.VIN_v	V	Voltage across device VIN
cir.IvsU1_v	V	Voltage across device IvsU1
cir.RL_v	V	Voltage across device RL

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Voltages

1 In the **Model Builder** window, right-click **ID Plot Group 5** and choose **Rename**.

2 In the **Rename ID Plot Group** dialog box, type Voltages in the **New label** text field.

3 Click **OK**.

The plot should now look like that in Figure 3.

Finish the modeling session by saving a representative model thumbnail.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Presentation** section.

3 Find the **Thumbnail** subsection. Click **Set from Graphics Window**.