# E-Core Transformer

## *Introduction*

Transformers are electrical components that are used for power transmission. Most transformers work on the principle of electromagnetic induction. A typical transformer consists of a primary coil, a secondary coil and a ferromagnetic core. The primary coil receives the AC electrical input signal. As a result of mutual induction, an induced voltage is obtained across the secondary coil. The ferromagnetic core serves the purpose of a magnetic flux concentrator thereby minimizing losses due to flux leakage.

Commercial transformers use several types of cores, which are named based on their geometric shapes, for example I-core, U-core, E-core, pot core, toroidal, and planar. This model uses a pair of E-cores for magnetic flux concentration.

This model demonstrates how to perform transient simulations of a single-phase E-core transformer. Including the effect of a nonlinear B-H curve in the soft-iron core, the model computes the spatial distribution of the magnetic and electric fields, the magnetic saturation effect, the transient response, and the flux leakage to the surroundings. Two different versions of the transformer are simulated: the first one with a turn ratio of unity and the second one with a turn ratio of 1000.

## *Model Definition*

The core of the single-phase E-core transformer considered here consists of a pair of E-cores, which form a closed magnetic flux path. The primary and secondary coils in the transformer are placed around the central leg of the core as shown in Figure 1.

A nonlinear B-H curve that includes saturation effects is used to simulate the magnetic behavior of the soft-iron core. Hysteresis effects in the core are neglected. The model assumes that the primary and secondary windings are made of thin wire and have multiple turns. Using the assumptions that the wire diameter is less than the skin depth and that there are many turns, these windings are modeled with Multi-Turn Coil Domain features. Furthermore, the model does not account for eddy currents in the individual turns of the coil. The primary winding is connected to a primary resistor, $R_p$ and the AC voltage source, $V_{\mathrm{ac}}$ while the secondary winding is connected to the secondary load resistor, $R_s$ as shown in Figure 2.

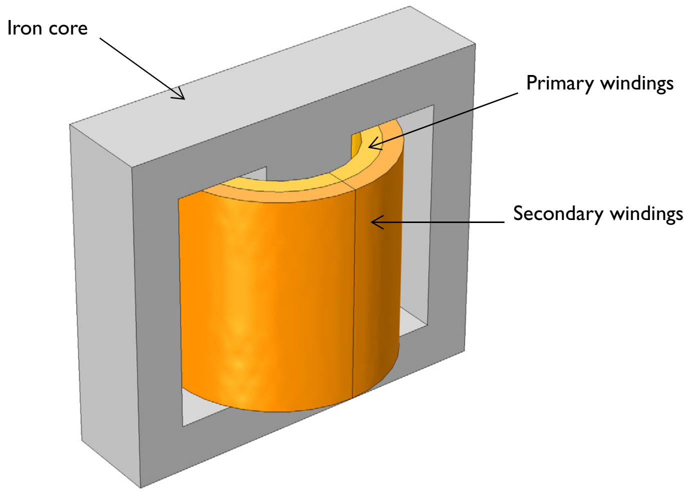



*Figure 1: Model illustration of an E-core transformer.*

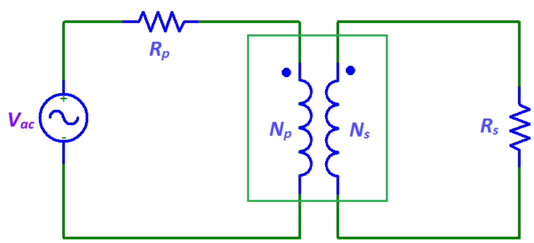



*Figure 2: A transformer connected to an external circuit with voltage source and resistors.*

The model is solved in time domain for a line frequency of 50 Hz. Several important design parameters such as the magnitude of the input voltage, the line frequency, the number of turns in the coils, and the coil resistance are parameterized and can therefore easily be changed.

A transformer works by the principle of Faraday's law of induction which states that the induced voltage ($V_{\mathrm{in}}$) in a coil is proportional to the rate of change of magnetic flux ( $\phi$ ) and the number of turns ($N$) in a coil as shown in Equation 1.

$$V_{\mathrm{in}} = -N\frac{d\phi}{dt} \tag{1}$$

If two coils are coupled, Equation 1 can be used to deduce that the induced voltage in the secondary coil ($V_s$) is proportional to the induced voltage in the primary coil ($V_p$):

$$\frac{V_s}{V_p} = \frac{N_s}{N_p} \tag{2}$$

Here $N_s$ and $N_p$ are the number of turns in the secondary and primary coils, respectively. $N_p/N_s$ is known as the turn ratio.

## *Results and Discussion*

Figure 3 shows the surface plot of the magnetic flux density norm distribution and the arrow plots of the currents in the windings at $t = 50$ ms.

Figure 4 shows the slice and the arrow plot of the magnetic flux density norm in the core at $t = 50$ ms.

Figure 5 and Figure 6 display the induced voltage in the primary and secondary windings respectively. Since the number of turns on each winding is equal, the induced voltage in both windings is same as given by Equation 2.

The currents flowing through the primary and secondary windings are shown in Figure 7 and Figure 8, respectively.

Figure 9 displays the voltage induced in the primary winding for a step-down transformer with a turn ratio of $N_p/N_s = 1000$ and supply voltage of 25 kV.

Finally, the induced voltage in the secondary winding for a step-down transformer is displayed in Figure 10. This induced voltage is 1000 times smaller compared to the voltage in the primary winding of Figure 9.

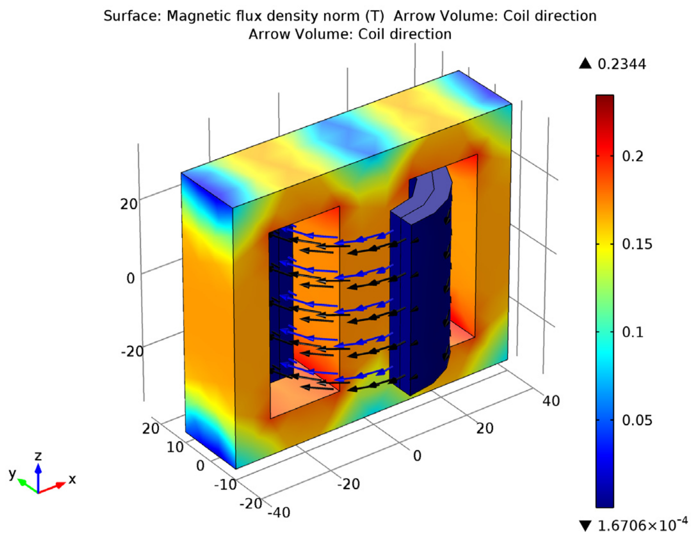



*Figure 3: Magnetic flux density norm and the currents in the windings at t = 50ms.*

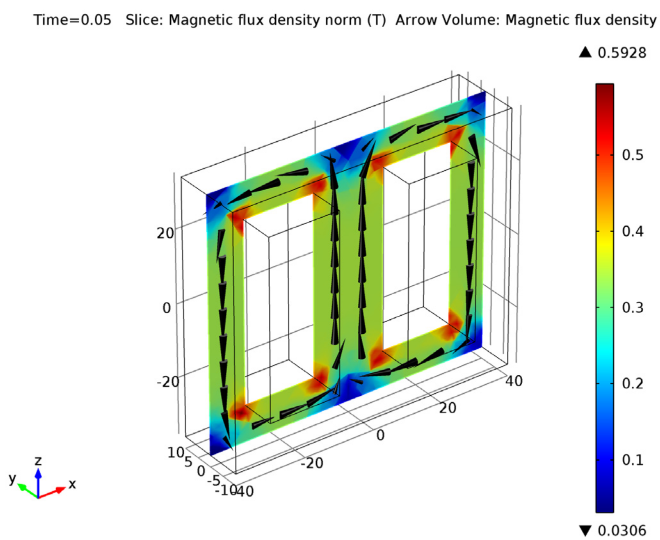



*Figure 4: Magnetic flux density inside the transformer core at t = 50ms.*

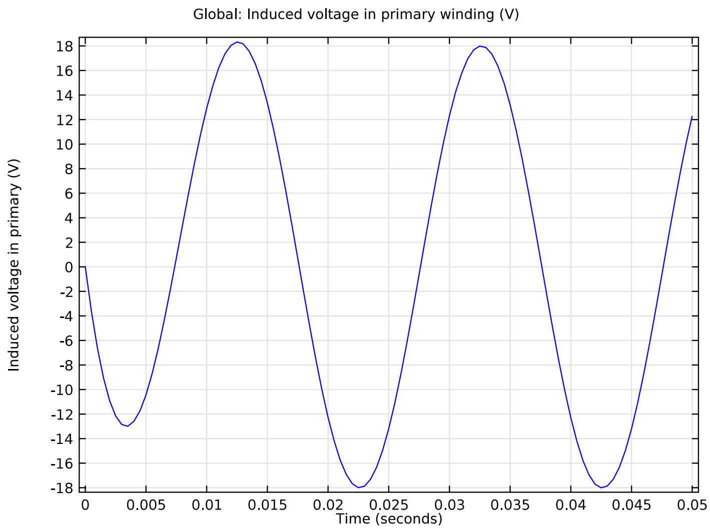



*Figure 5: The induced voltage in the primary winding versus time.*

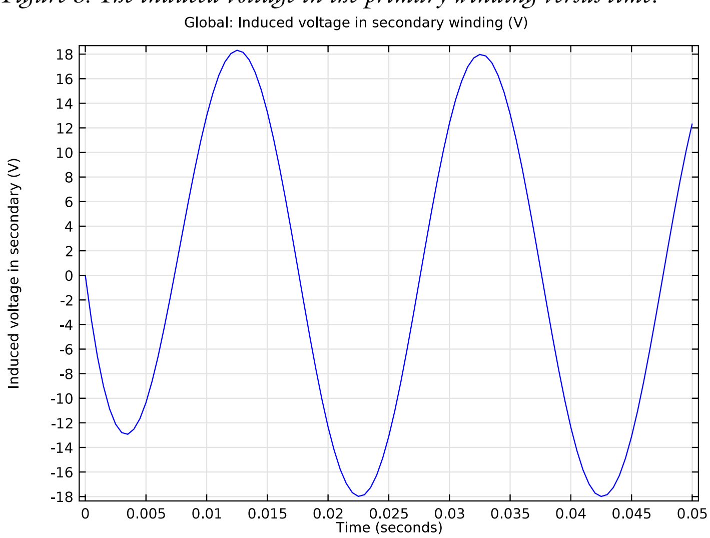



*Figure 6: Induced voltage in the secondary windings versus time.*

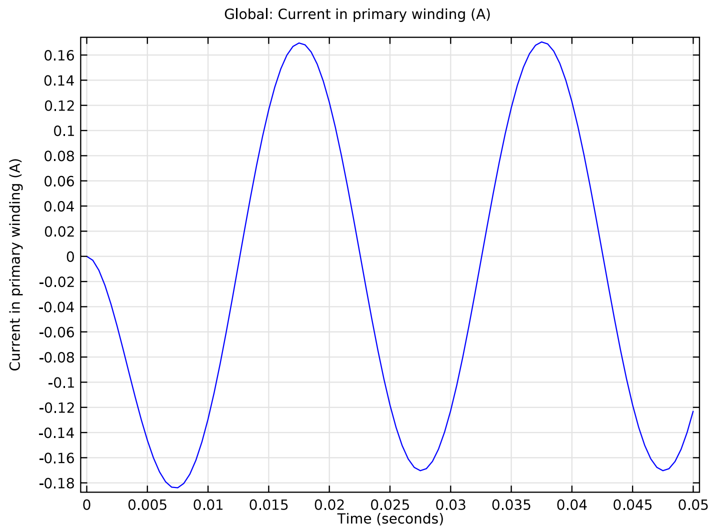



*Figure 7: Current in the primary winding versus time.*

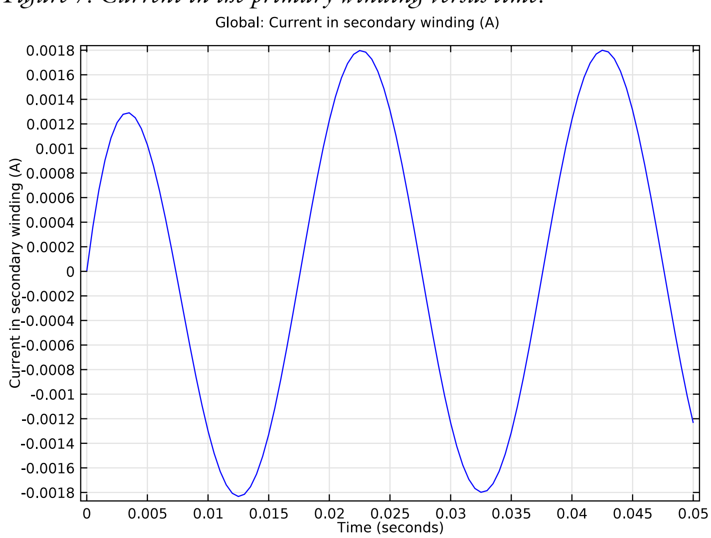



*Figure 8: Current in the secondary winding versus time.*

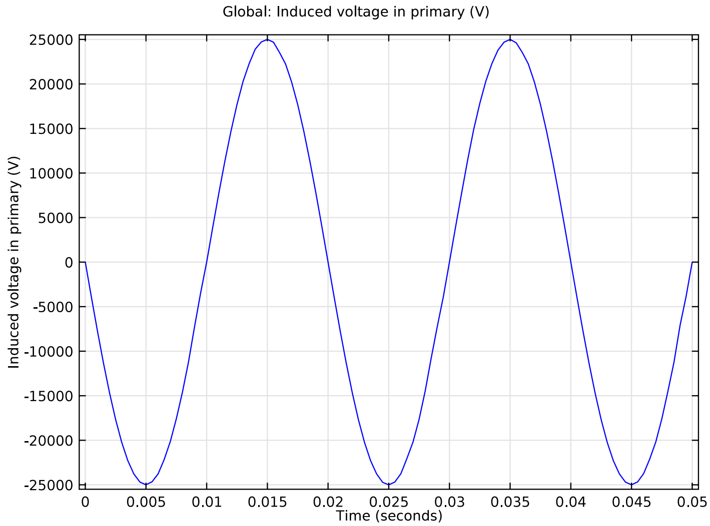



*Figure 9: Induced voltage in the primary winding versus time for a step-down transformer.*

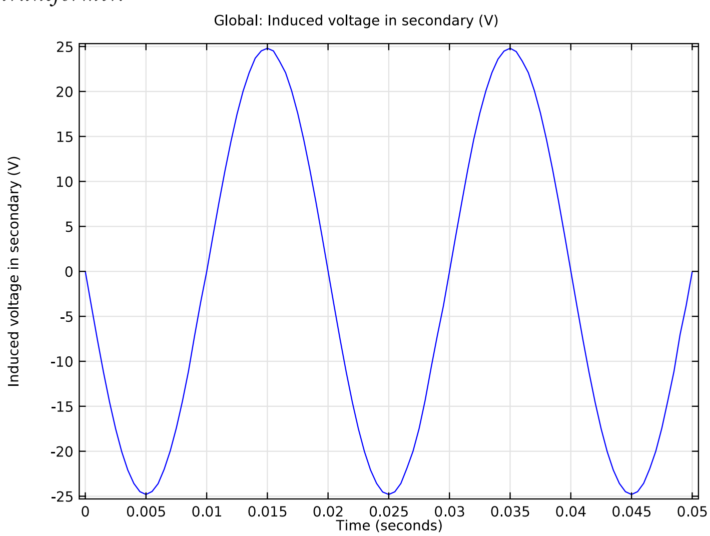



*Figure 10: Induced voltage in the secondary winding versus time for a step-down transformer.*

## *Notes About the COMSOL Implementation*

Use the Magnetic Fields physics interface to model the magnetic fields of the transformer. Model the primary and secondary windings with Multi-Turn Coil Domain features. Connect the primary and secondary windings to an external circuit with the AC voltage source and resistors using an Electrical Circuit physics interface. Add Coil Current Calculation study steps to calculate the current in the coil domains. Perform a Time Dependent study to determine the voltage and currents in both the primary and secondary windings.

**Model Library path:** `ACDC_Module/Other_Industrial_Applications/ecore_transformer`

## *Modeling Instructions*

### MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **AC/DC>Magnetic Fields (mf)**.

4 Click **Add Selected**.

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

6 Click **Add Selected**.

7 Click **Next**.

8 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

9 Click **Finish**.

### GLOBAL DEFINITIONS

Define all the required parameters.

#### *Parameters*

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Description |
|---|---|---|
| Rp | 100[ohm] | Primary side resistance |
| Rs | 10[kohm] | Secondary side resistance |
| Np | 300 | Number of turns in primary winding |
| Ns | 300 | Number of turns in secondary winding |
| nu | 50[Hz] | Frequency of supply voltage |
| Vac | 25[V] | Supply voltage |

## GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

Use the following instructions to construct the model geometry. First, create the transformer core.

### *Work Plane 1*

1 Right-click **Model 1>Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 In the **y-coordinate** edit field, type 10.

### *Rectangle 1*

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 80.

4 In the **Height** edit field, type 70.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Click the **Build Selected** button.

### *Rectangle 2*

1 Right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 60.

4 In the **Height** edit field, type 50.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Click the **Build Selected** button.

*Rectangle 3*

1 Right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 20.

4 In the **Height** edit field, type 50.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Click the **Build Selected** button.

*Compose 1*

1 Right-click **Plane Geometry** and choose **Boolean Operations>Compose**.

2 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

3 In the **Compose** settings window, locate the **Compose** section.

4 In the **Set formula** edit field, type `r1-r2+r3`.

5 Clear the **Keep interior boundaries** check box.

6 Click the **Build Selected** button.

*Extrude 1*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 Right-click **Extrude 1** and choose **Build Selected**.

3 In the **Extrude** settings window, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

| Distances (mm) |
| --- |
| 20 |

Create the geometry for the primary and the secondary windings.

*Work Plane 2*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Plane Definition** section.

3 In the **z-coordinate** edit field, type -23.

4 Click the **Build Selected** button.

*Circle 1*

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 2** right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** edit field, type 24.

5 Click to expand the **Layers** section. In the table, enter the following settings:

| Thickness (mm) |
|---|
| 4 |
| 4 |

6 Click the **Build Selected** button.

*Extrude 2*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 Right-click **Extrude 2** and choose **Build Selected**.

3 In the **Extrude** settings window, locate the **Distances from Plane** section.

4 In the table, enter the following settings:

| Distances (mm) |
|---|
| 46 |

Finish the geometry by creating the outer boundary box.

*Block 1*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type 150.

4 In the **Depth** edit field, type 130.

5 In the **Height** edit field, type 150.

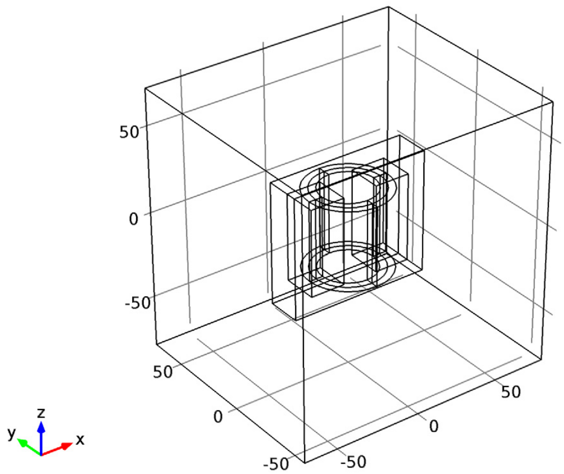
6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Click the **Build Selected** button.

8 Click the **Zoom Extents** button on the Graphics toolbar.

Choose wireframe rendering to get a better view of the interior parts.

**9** Click the **Wireframe Rendering** button on the Graphics toolbar.




The geometry looks as shown in the figure above.

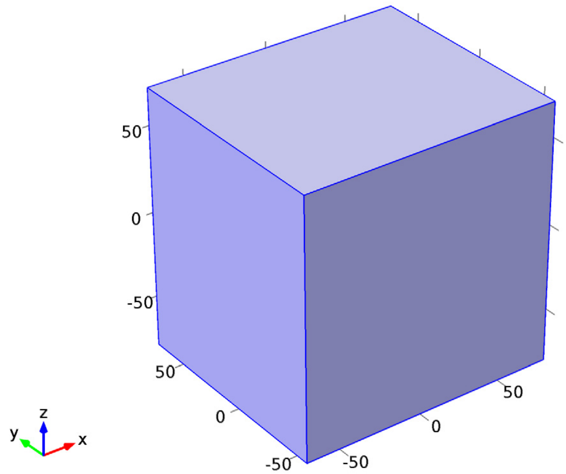
## DEFINITIONS

Hide the outer geometry boundaries to visualize the results only in the transformer core and windings.

### *View 1*

1. In the **Model Builder** window, expand the **Model 1>Definitions** node.
2. Right-click **View 1** and choose **Hide Geometric Entities**.
3. In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.
4. From the **Geometric entity level** list, choose **Boundary**.

**5** Select Boundaries 1–5 and 56 only.




Define domain and boundary selections for the core and the windings. First, create a selection for the core domain.

*Explicit 1*

**1** In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

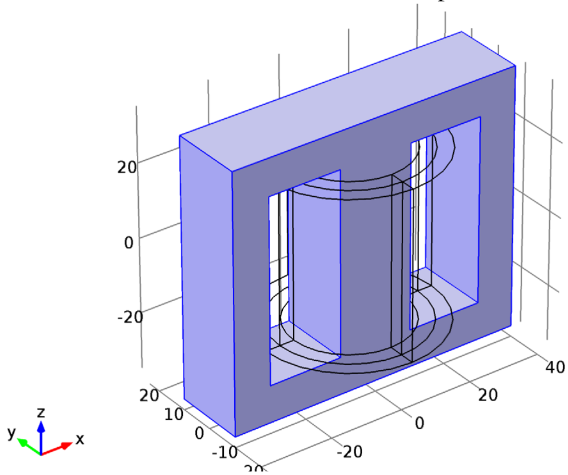
**2** Right-click **Explicit 1** and choose **Rename**.

**3** Go to the **Rename Explicit** dialog box and type `Core` in the **New name** edit field.

**4** Click **OK**.

**5** Select Domain 2 only.

**6** Click the **Zoom Extents** button on the Graphics toolbar.




Specify a selection for the primary winding.

*Explicit 2*

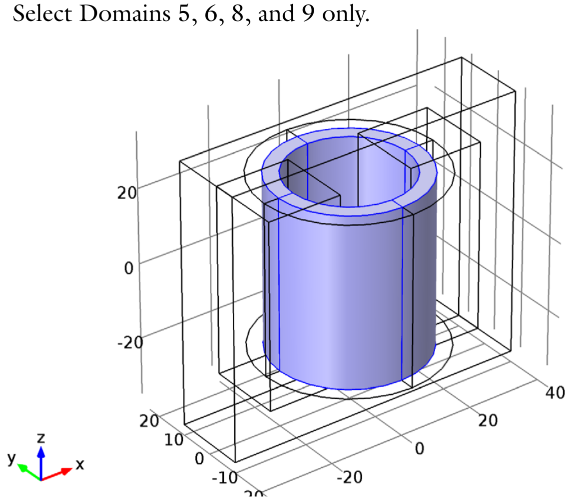
**1** In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

**2** Right-click **Explicit 2** and choose **Rename**.

**3** Go to the **Rename Explicit** dialog box and type `Primary Winding` in the **New name** edit field.

**4** Click **OK**.

**5** Select Domains 5, 6, 8, and 9 only.



Add a selection for the secondary winding.

*Explicit 3*

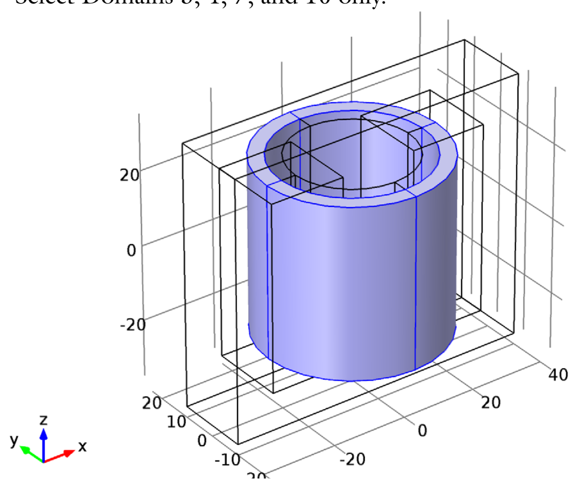
1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 3** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Secondary Winding` in the **New name** edit field.

4 Click **OK**.

5 Select Domains 3, 4, 7, and 10 only.




*Explicit 4*

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 4** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Windings` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 Click **Paste Selection**.

7 Go to the **Paste Selection** dialog box.

8 In the **Selection** edit field, type `3-10`.

9 Click the **OK** button.

Select only the outer surface boundaries of the primary winding.

*Explicit 5*

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 5** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Primary Winding Insulation` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Click **Paste Selection**.

8 Go to the **Paste Selection** dialog box.

9 In the **Selection** edit field, type `22-29,36-39,41-43,45`.

10 Click the **OK** button.

Select only the outer surface boundaries of the secondary winding.

*Explicit 6*

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 6** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Secondary Winding Insulation` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Click **Paste Selection**.

8 Go to the **Paste Selection** dialog box.

9 In the **Selection** edit field, type `15-20,22,23,32-34,36,45-48`.

10 Click the **OK** button.

Now set up the physics for the magnetic field. Choose the linear vector elements to discretize the magnetic vector potential. This will make the model computationally efficient. Typically, the default quadratic elements are recommended.

### MAGNETIC FIELDS

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

2 In the **Magnetic Fields** settings window, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

Apply Ampere's law in the core and the air domain.

*Ampère's Law 2*

1 In the **Model Builder** window, right-click **Magnetic Fields** and choose **Ampère's Law**.

2 In the **Ampère's Law** settings window, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Core**.

4 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **HB curve**.

Add Multi-Turn Coil nodes to model the primary and the secondary windings.

*Multi-Turn Coil 1*

1 Right-click **Magnetic Fields** and choose **Multi-Turn Coil**.

2 In the **Multi-Turn Coil** settings window, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Primary Winding**.

4 Locate the **Coil Type** section. From the list, choose **Numeric**.

5 Locate the **Multi-Turn Coil** section. In the $N$ edit field, type `Np`.

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

*Automatic Current Calculation 1*

Right-click **Model 1>Magnetic Fields>Multi-Turn Coil 1** and choose **Automatic Current Calculation**.

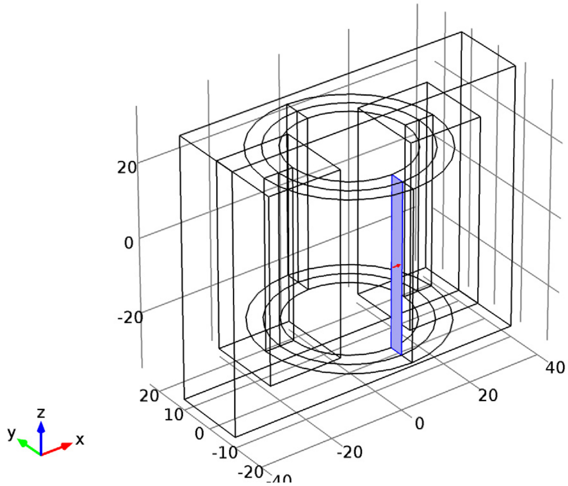
*Input 1*

1 In the **Model Builder** window, under **Model 1>Magnetic Fields>Multi-Turn Coil 1** right-click **Automatic Current Calculation 1** and choose **Input**.

2 In the **Input** settings window, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

**4** Select Boundary 35 only.




### *Electric Insulation 1*

1. Right-click **Automatic Current Calculation 1** and choose **Electric Insulation**.
2. In the **Electric Insulation** settings window, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Primary Winding Insulation**.

### *Multi-Turn Coil 2*

1. In the **Model Builder** window, right-click **Magnetic Fields** and choose **Multi-Turn Coil**.
2. In the **Multi-Turn Coil** settings window, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Secondary Winding**.
4. Locate the **Coil Type** section. From the list, choose **Numeric**.
5. Locate the **Multi-Turn Coil** section. In the $N$ edit field, type `Ns`.
6. From the **Coil excitation** list, choose **Circuit (current)**.

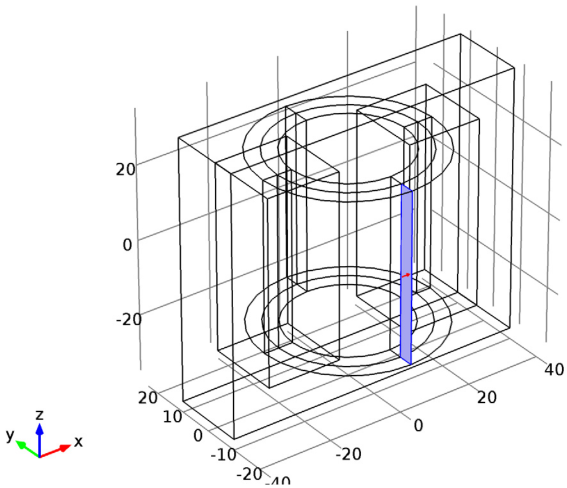
### *Automatic Current Calculation 1*

Right-click **Model 1>Magnetic Fields>Multi-Turn Coil 2** and choose **Automatic Current Calculation**.

### *Input 1*

1. In the **Model Builder** window, under **Model 1>Magnetic Fields>Multi-Turn Coil 2** right-click **Automatic Current Calculation 1** and choose **Input**.
2. In the **Input** settings window, locate the **Boundary Selection** section.
3. Click **Clear Selection**.

**4** Select Boundary 31 only.




*Electric Insulation 1*

**1** Right-click **Automatic Current Calculation 1** and choose **Electric Insulation**.

**2** In the **Electric Insulation** settings window, locate the **Boundary Selection** section.

**3** From the **Selection** list, choose **Secondary Winding Insulation**.

Add gauge fixing to numerically stabilize the model, which uses numeric coils in the Multi-Turn Coil nodes. This consumes more computational memory but helps to make the model convergence.

*Gauge Fixing for A-Field 1*

**1** In the **Model Builder** window, right-click **Magnetic Fields** and choose **Gauge Fixing for A-Field**.

Gauge fixing must be applied on all domains in which the Magnetic Fields physics interface is active.

**2** In the **Gauge Fixing for A-Field** settings window, locate the **Domain Selection** section.

**3** From the **Selection** list, choose **All domains**.

#### ELECTRICAL CIRCUIT

Add the external circuit elements to the primary and the secondary side of the transformer as shown in Figure 2.

*Voltage Source 1*

1 In the **Model Builder** window, under **Model 1** right-click **Electrical Circuit** and choose **Voltage Source**.

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

3 From the **Source type** list, choose **Sine source**.

4 In the $V_{src}$ edit field, type `Vac`.

5 In the $f$ edit field, type `nu`.

*External I Vs. U 1*

1 In the **Model Builder** window, right-click **Electrical Circuit** and choose **External I Vs. U**.

2 In the **External I Vs. U** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p | 2 |
| n | 1 |

4 Locate the **External Device** section. From the $V$ list, choose **Coil voltage (mf/mtcd1)**.

*Resistor 1*

1 Right-click **Electrical Circuit** and choose **Resistor**.

2 In the **Resistor** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p | 0 |
| n | 2 |

4 Locate the **Device Parameters** section. In the $R$ edit field, type `Rp`.

*External I Vs. U 2*

1 Right-click **Electrical Circuit** and choose **External I Vs. U**.

2 In the **External I Vs. U** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p | 3 |

4 Locate the **External Device** section. From the $V$ list, choose **Coil voltage (mf/mtcd2)**.

*Resistor 2*

1 Right-click **Electrical Circuit** and choose **Resistor**.

2 In the **Resistor** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p | 0 |
| n | 3 |

4 Locate the **Device Parameters** section. In the $R$ edit field, type `Rs`.

Assign materials for the model. Begin by specifying air for all domains.

## MATERIALS

*Material Browser*

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 In the **Material_browser** window, click **Add Material to Model**.

Change the conductivity value to 10 S/m.This small value for conductivity helps to improve the stability of the solver.

*Air*

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

| Property | Name | Value |
|---|---|---|
| Electrical conductivity | sigma | `10[S/m]` |

Next, assign the soft iron (without loss) material for the core. Modify the conductivity value to 10 S/m.

*Material Browser*

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **AC/DC>Soft Iron (without losses)**.

3 In the **Material_browser** window, click **Add Material to Model**.

*Soft Iron (without losses)*

1 In the **Model Builder** window, under **Model 1>Materials** click **Soft Iron (without losses)**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Core**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

| Property | Name | Value |
|---|---|---|
| Electrical conductivity | sigma | 10[S/m] |

Use the following steps to visualize the nonlinear B-H curve of the soft iron.

1 In the **Model Builder** window, expand the **Soft Iron (without losses)** node.

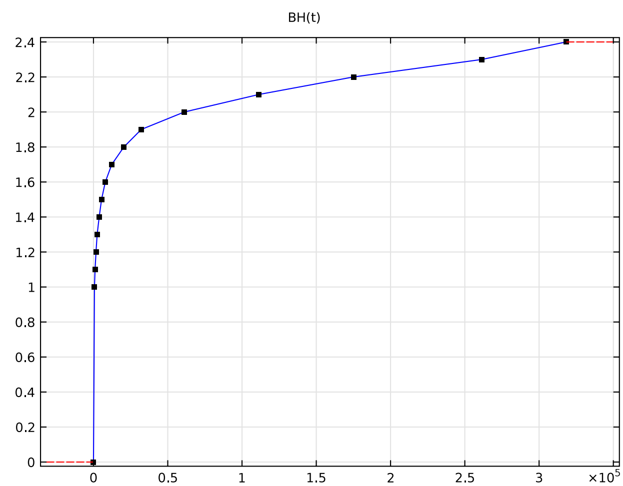
*Interpolation*

1 In the **Model Builder** window, expand the **Model 1>Materials>Soft Iron (without losses)>BH curve** node, then click **Interpolation**.

2 In the **Interpolation** settings window, locate the **Interpolation and Extrapolation** section.

3 From the **Extrapolation** list, choose **Constant**.

4 Click the **Plot** button.




## MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra coarse**.

*Free Tetrahedral 1*

1 Right-click **Model 1>Mesh 1** and choose **Free Tetrahedral**.

2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

*Size 1*

1 Right-click **Model 1>Mesh 1>Free Tetrahedral 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 Go to the **Paste Selection** dialog box.

6 In the **Selection** edit field, type `2-10`.

7 Click the **OK** button.

8 In the **Size** settings window, locate the **Element Size** section.

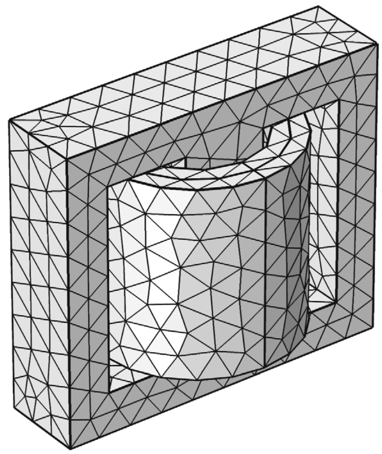
9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

11 In the associated edit field, type `8`.

12 Click the **Build All** button.

After adjusting the hiding settings by clicking on the **View Unhidden Only** button, the mesh should look like that shown in the figure below. A coarse mesh is used here to solve the model quickly. For real simulations, you are recommended to use a finer mesh.



### STUDY 1

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

2 In the **Study** settings window, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

Add the Coil Current Calculation study steps for the automatic current calculation in the primary and the secondary windings.

*Step 2: Coil Current Calculation*

1 Right-click **Study 1** and choose **Coil Current Calculation**.

2 In the **Model Builder** window, under **Study 1** right-click **Step 2: Coil Current Calculation** and choose **Move Up**.

*Step 3: Coil Current Calculation 2*

1 Right-click **Study 1** and choose **Study Steps>Coil Current Calculation**.

2 In the **Model Builder** window, under **Study 1** right-click **Step 3: Coil Current Calculation 2** and choose **Move Up**.

3 In the **Coil Current Calculation** settings window, locate the **Study Settings** section.

4 In the **Coil name** edit field, type 2.

*Step 3: Time Dependent*

Solve the problem in time domain from 0 to 50 milliseconds.

1 In the **Model Builder** window, under **Study 1** click **Step 3: Time Dependent**.
2 In the **Time Dependent** settings window, locate the **Study Settings** section.
3 In the **Times** edit field, type `range(0,5e-4,0.05)`.
4 Select the **Relative tolerance** check box.
5 In the associated edit field, type `0.001`.

Follow the steps given below to tune the solver. Such tuning is necessary in order to successfully use a realistic nonlinear B-H curve in a large time-dependent model. As this model is nonlinear in space and time, the solver needs to be robust enough to handle such nonlinearities.

### *Solver 1*

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
2 Expand the **Solver 1** node.
3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** node.
4 Right-click **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** and choose **Fully Coupled**.
5 Right-click **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1>Direct** and choose **Enable**.
6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** click **Fully Coupled 1**.
7 In the **Fully Coupled** settings window, click to expand the **Method and Termination** section.
8 From the **Jacobian update** list, choose **On every iteration**.
9 In the **Maximum number of iterations** edit field, type `25`.
10 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

## RESULTS

### *Data Sets*

Create separate data sets for the windings and the core domain. This is useful to visualize the results in different domain independently.

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Solution 1** and choose **Duplicate**.
2 Right-click **Results>Data Sets>Solution 4** and choose **Add Selection**.

3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **Windings**.

6 Select the **Propagate to lower dimensions** check box.

7 Right-click **Results>Data Sets>Solution 4>Selection** and choose **Rename**.

8 Go to the **Rename Selection** dialog box and type `Windings` in the **New name** edit field.

9 Click **OK**.

10 Right-click **Results>Data Sets>Solution 1** and choose **Duplicate**.

11 Right-click **Results>Data Sets>Solution 5** and choose **Add Selection**.

12 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

13 From the **Geometric entity level** list, choose **Domain**.

14 From the **Selection** list, choose **Core**.

15 Select the **Propagate to lower dimensions** check box.

16 Right-click **Results>Data Sets>Solution 5>Selection** and choose **Rename**.

17 Go to the **Rename Selection** dialog box and type `Core` in the **New name** edit field.

18 Click **OK**.

19 Right-click **Results>Data Sets>Solution 1** and choose **Add Selection**.

20 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

21 From the **Geometric entity level** list, choose **Domain**.

22 Click **Paste Selection**.

23 Go to the **Paste Selection** dialog box.

24 In the **Selection** edit field, type `2,4,6-10`.

25 Click the **OK** button.

Use the following steps to generate a surface plot of the magnetic flux density norm and arrow plot of the currents in the windings. The figure should look like that shown in Figure 3.

### *3D Plot Group 1*

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 1** and choose **Surface**.

3 Right-click **3D Plot Group 1** and choose **Arrow Volume**.

4 In the **Arrow Volume** settings window, locate the **Data** section.

5 From the **Data set** list, choose **Solution 4**.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil direction (mf.mtcd1.eCoilx,...,mf.mtcd1.eCoilz)**.

7 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** edit field, type 10.

8 Find the **y grid points** subsection. In the **Points** edit field, type 10.

9 Find the **z grid points** subsection. In the **Points** edit field, type 5.

10 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

11 In the associated edit field, type 8.

12 From the **Color** list, choose **Blue**.

13 Right-click **Results>3D Plot Group 1>Arrow Volume 1** and choose **Duplicate**.

14 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil direction (mf.mtcd2.eCoilx,...,mf.mtcd2.eCoilz)**.

15 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

16 Click the **Plot** button.

Follow the steps below to reproduce the magnetic flux density norm plot as shown in Figure 4.

*3D Plot Group 2*

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 5**.

4 Right-click **Results>3D Plot Group 2** and choose **Slice**.

5 In the **Slice** settings window, locate the **Plane Data** section.

6 From the **Plane** list, choose **zx-planes**.

7 In the **Planes** edit field, type 1.

8 In the **Model Builder** window, right-click **3D Plot Group 2** and choose **Arrow Volume**.

9 In the **Arrow Volume** settings window, locate the **Arrow Positioning** section.

10 Find the **x grid points** subsection. In the **Points** edit field, type 10.

11 Find the **y grid points** subsection. In the **Points** edit field, type 1.

12 Find the **z grid points** subsection. In the **Points** edit field, type 10.

13 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

14 Select the **Scale factor** check box.

15 In the associated edit field, type `30`.

16 From the **Color** list, choose **Black**.

17 Click the **Plot** button.

Next, plot the induced voltage in the primary winding.

### *1D Plot Group 3*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 3** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Induced voltage in primary` in the **New name** edit field.

4 Click **OK**.

5 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated edit field, type `Time (seconds)`.

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

9 In the associated edit field, type `Induced voltage in primary (V)`.

### *Induced voltage in primary*

1 Right-click **Results>1D Plot Group 3** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil voltage (mf.VCoil_1)**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
| --- |
| `Induced voltage in primary winding` |

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 Click the **Plot** button.

6 Compare the figure with that shown in Figure 5.

Plot the induced voltage in the secondary winding. The plot is as shown in Figure 6.

### *1D Plot Group 4*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 4** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Induced voltage in secondary` in the **New name** edit field.

4 Click **OK**.

5 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated edit field, type `Time (seconds)`.

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

9 In the associated edit field, type `Induced voltage in secondary (V)`.

*Induced voltage in secondary*

1 Right-click **Results>1D Plot Group 4** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil voltage (mf.VCoil_2)**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
| --- |
| `Induced voltage in secondary winding` |

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 Click the **Plot** button.

Plot the current in the primary winding and compare the plot with Figure 7.

*1D Plot Group 5*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 5** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Current in primary winding` in the **New name** edit field.

4 Click **OK**.

5 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated edit field, type `Time (seconds)`.

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

9 In the associated edit field, type `Current in primary winding (A)`.

*Current in primary winding*

1 Right-click **Results>1D Plot Group 5** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil current (mf.ICoil_1)**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
|---|
| `Current in primary winding` |

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 Click the **Plot** button.

Next, plot the current in the secondary winding.

*1D Plot Group 6*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 6** and choose **Rename**.

3 Go to the **Rename 1D Plot Group** dialog box and type `Current in secondary winding` in the **New name** edit field.

4 Click **OK**.

5 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated edit field, type `Time (seconds)`.

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

9 In the associated edit field, type `Current in secondary winding (A)`.

*Current in secondary winding*

1 Right-click **Results>1D Plot Group 6** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil current (mf.ICoil_2)**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
|---|
| `Current in secondary winding` |

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 Click the **Plot** button.

6 Compare the plot with Figure 8.

Modify the model to simulate a step down transformer with a turn ratio of 1000 and $R_p = R_s$. In addition, change the supply voltage to $V_{\mathrm{ac}} = 25\mathrm{kV}$.

## GLOBAL DEFINITIONS

### *Parameters*

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Description |
|---|---|---|
| Rs | 100[ohm] | Secondary side resistance |
| Np | 3e5 | Number of turns in primary winding |
| Vac | 25[kV] | Supply voltage |

## ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

## MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

3 Click **Finish**.

## STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Study** settings window, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

### *Step 2: Coil Current Calculation*

1 Right-click **Study 2** and choose **Coil Current Calculation**.

2 In the **Model Builder** window, under **Study 2** right-click **Step 2: Coil Current Calculation** and choose **Move Up**.

*Step 3: Coil Current Calculation 2*

1 Right-click **Study 2** and choose **Study Steps>Coil Current Calculation**.

2 In the **Model Builder** window, under **Study 2** right-click **Step 3: Coil Current Calculation 2** and choose **Move Up**.

3 In the **Coil Current Calculation** settings window, locate the **Study Settings** section.

4 In the **Coil name** edit field, type `2`.

*Step 3: Time Dependent*

1 In the **Model Builder** window, under **Study 2** click **Step 3: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,5e-4,0.05)`.

4 Select the **Relative tolerance** check box.

5 In the associated edit field, type `0.001`.

*Solver 4*

1 In the **Model Builder** window, right-click **Study 2** and choose **Show Default Solver**.

2 Expand the **Solver 4** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solver 4>Time-Dependent Solver 1** node.

4 Right-click **Study 2>Solver Configurations>Solver 4>Time-Dependent Solver 1** and choose **Fully Coupled**.

5 Right-click **Study 2>Solver Configurations>Solver 4>Time-Dependent Solver 1>Direct** and choose **Enable**.

6 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 4>Time-Dependent Solver 1** click **Fully Coupled 1**.

7 In the **Fully Coupled** settings window, click to expand the **Method and Termination** section.

8 From the **Jacobian update** list, choose **On every iteration**.

9 In the **Maximum number of iterations** edit field, type `25`.

10 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

### RESULTS

Plot the induced voltage in the primary winding for a step down transformer.

*1D Plot Group 7*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 7** and choose **Rename**.
3 Go to the **Rename 1D Plot Group** dialog box and type `Induced voltage in primary-II` in the **New name** edit field.
4 Click **OK**.
5 In the **1D Plot Group** settings window, locate the **Data** section.
6 From the **Data set** list, choose **Solution 6**.
7 Locate the **Plot Settings** section. Select the **x-axis label** check box.
8 In the associated edit field, type `Time (seconds)`.
9 Select the **y-axis label** check box.
10 In the associated edit field, type `Induced voltage in primary (V)`.

### *Induced voltage in primary-II*

1 Right-click **Results>1D Plot Group 7** and choose **Global**.
2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil voltage (mf.VCoil_1)**.
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
|---|
| `Induced voltage in primary` |

4 Locate the **Legends** section. Clear the **Show legends** check box.
5 Click the **Plot** button.
6 Compare the plot with Figure 9.

Finally, plot the induced voltage in the secondary winding for a step down transformer. The plot should look like that shown in Figure 10.

### *1D Plot Group 8*

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **1D Plot Group 8** and choose **Rename**.
3 Go to the **Rename 1D Plot Group** dialog box and type `Induced voltage in secondary-II` in the **New name** edit field.
4 Click **OK**.
5 In the **1D Plot Group** settings window, locate the **Data** section.
6 From the **Data set** list, choose **Solution 6**.

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

8 In the associated edit field, type `Time (seconds)`.

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

10 In the associated edit field, type `Induced voltage in secondary (V)`.

*Induced voltage in secondary-II*

1 Right-click **Results>1D Plot Group 8** and choose **Global**.

2 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Coil parameters>Coil voltage (mf.VCoil_2)**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| **Description** |
|---|
| `Induced voltage in secondary` |

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 Click the **Plot** button.