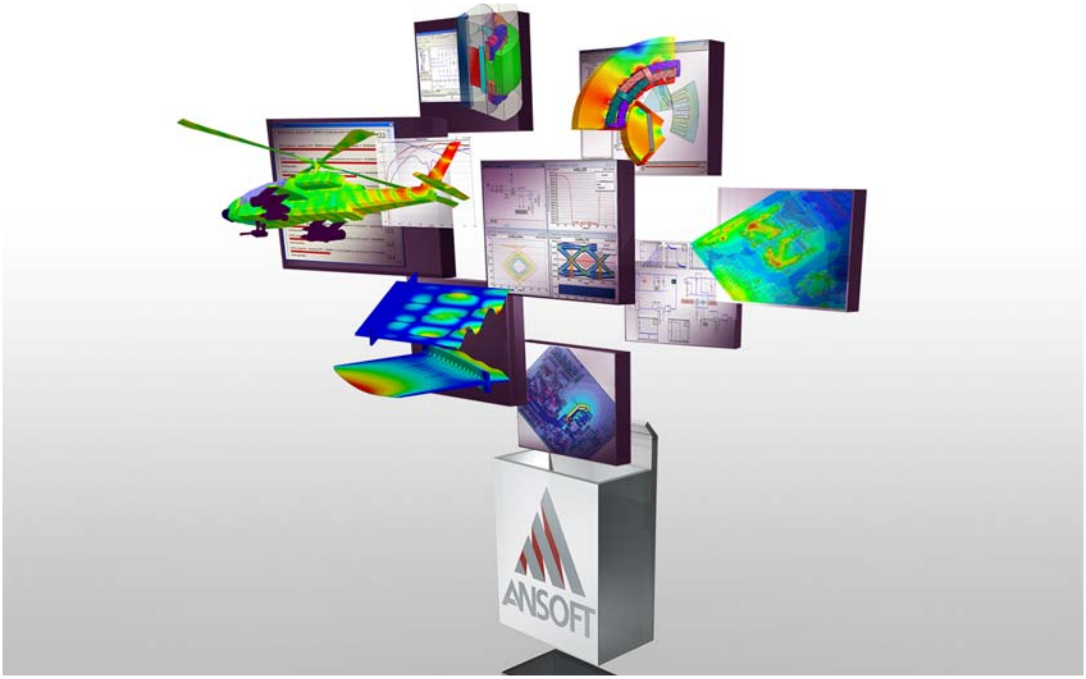


Getting Started with Maxwell: Transient Problem

ANSOFT
ANSYS Product Suite

ANSYS, Inc.
275 Technology Drive
Canonsburg, PA 15317
Tel: (+1) 724-746-3304
Fax: (+1) 724-514-9494
General Information: AnsoftInfo@ansys.com
Technical Support: AnsoftTechSupport@ansys.com

November 2010
Inventory: 002890

New editions of this manual incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates that are incorporated at reprint do not cause the date to change.

Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

Edition	Date	Software Version
1	Feb 2008	Maxwell 12
2	April 2010	Maxwell 13
3	November 2010	Maxwell 14.0

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this guide.

- The project tree is the main project area of the **Project Manager** window. These two terms (project tree and **Project Manager** window) may be used interchangeably in this guide.
- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:

  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.

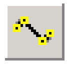
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, click **Maxwell>Excitations>Assign>Voltage**.

  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Menu commands are often separated by the ">" symbol. For example, "Click **File>Exit**".
- Italic type is used for the following:

  - Emphasis.

  - The titles of publications.

  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *file name*" means to type the word **copy**, to type a space, and then to type a file name.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press **Shift+F1**" means to press the **Shift** key and the **F1** key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,

“Click **Draw>Line**  ” means that you can also click the **Draw Line** toolbar button to execute the **Line** command.

Getting Help

Ansoft Technical Support

To contact the Ansoft technical support staff in your geographical area, please go to the Ansoft website, http://www.ansoft.com, click the **Contact** button, and then click **Support**. Phone numbers and e-mail addresses are listed for the technical support staff. You can also contact your Ansoft account manager to obtain this information.

All Ansoft software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is helpful to include specific information about what steps were taken or what stages the simulation reached. This promotes more rapid and effective debugging.

Context-Sensitive Help

To access online help from the Maxwell user interface, do one of the following:

- To open a help topic about a specific Maxwell menu command, press **Shift+F1**, and then click the command or toolbar icon.
- To open a help topic about a specific Maxwell dialog box, open the dialog box, and then press **F1**.

Table of Contents

1 Introduction

This Getting Started Guide is written for Maxwell beginners and experienced users who would like to quickly refamiliarize themselves with the capabilities of Maxwell. This guide leads you step-by-step through solving and analyzing the results of a rotational actuator magnetostatic problem with motion.

By following the steps in this guide, you will learn how to perform the following tasks:

- Modify a model's design parameters.
- Assign variables to a model's design parameters.
- Specify solution settings for a design.
- Validate a design's setup.
- Run a Maxwell simulation.
- Plot the magnetic flux density vector.
- Include motion in the simulation.

Note This guide assumes that you have already completed the magnetostatic example in *Getting Started with Maxwell: Designing a Rotational Actuator*. If you have not, you may use the project in the example directory; however, it is strongly recommended that you complete the magnetostatic example.

Estimated time to complete this guide: 1 hour 30 minutes.

Maxwell Solution Types

Maxwell® is an interactive software package that uses finite element analysis (FEA) to simulate (solve) electromagnetic field problems. Maxwell integrates with other Ansoft software packages to perform complex tasks while remaining simple to use. Maxwell® incorporates both a set of 2D solvers and 3D solvers in an integrated user interface. This guide will focus on 3D capabilities. 2D problems examples are cover in separate 2D Getting Started Guides.

The following six types of stand-alone solutions are supported by Maxwell 3D:

- **Magnetostatic linear and nonlinear 3D fields** caused by a user-specified distribution of DC current density and permanent or externally applied magnetic fields. Materials can be non-linear and anisotropic. Additional quantities that can be computed include torque, force, and self and mutual inductances.
- **Harmonic (sinusoidal variation in time) steady-state magnetic fields** with pulsation-induced eddy currents in massive solid conductors caused by one of the following:

  a. A user-specified distribution of AC currents (all with the same frequency but with possibly different initial phase angles).

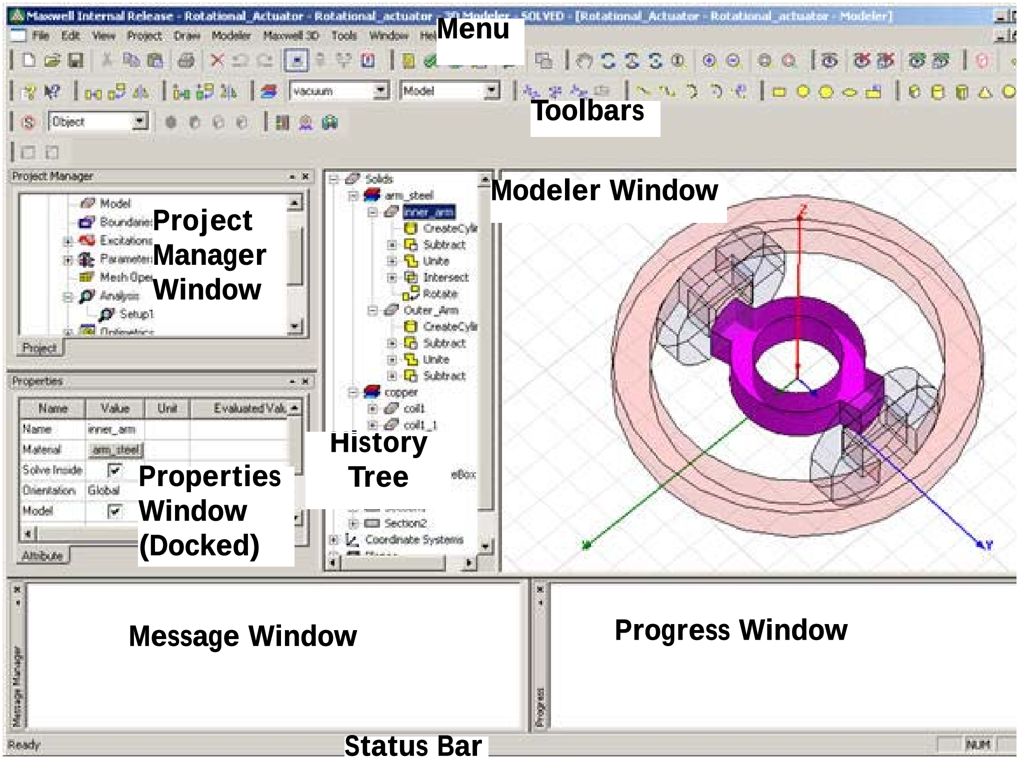
  b. Externally applied magnetic fields.

  This solution includes displacement currents for calculating near field electromagnetic wave radiation.
- **Transient (time domain) magnetic fields** caused by permanent magnets, conductors, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time. Rotational or translational motion effects can be included in the simulation.
- **Electrostatic 3D fields** caused by a user-specified distribution of voltages and charges in non-conducting regions. Additional quantities that can be computed include torque, force, and capacitances.
- **Electric DC Conduction 3D fields** in conductors characterized by a spatial distribution of voltage, electric field, and current density. Power loss can also be computed. In addition, optional simulation of fields in insulating materials is supported.
- **Transient (time domain) 3D Electric fields** caused by time dependent voltage, current and charge distributions. All sources are arbitrary functions of time.

In addition, Maxwell may be coupled with other simulators to provide a greater range of solution capability. Couplings to ePhysics for thermal and stress analysis, HFSS for ferrite analysis, and Simplorer for Finite Element/Circuit co-simulation are all supported.

The Maxwell Desktop

The following graphic shows the different sections of the Maxwell desktop:



General Procedure for Setting Up Maxwell Designs

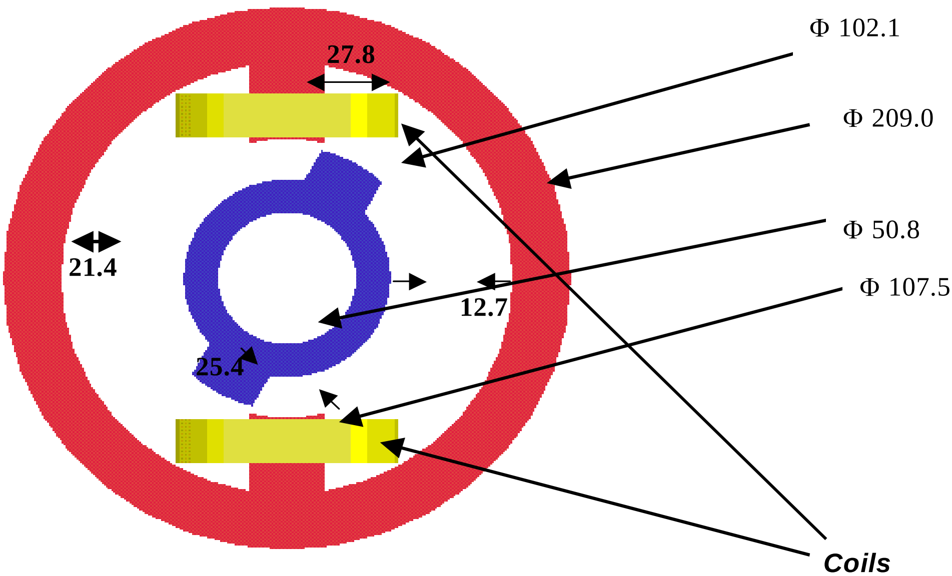
You are not required to follow a specific order when setting up your Maxwell design (as was required for Maxwell Version 10 and earlier). However, the following order is recommended, particularly for new users:

1. Open Maxwell by double-clicking the desktop icon or by clicking **Start>Programs>Ansoft>Maxwell14.0>Maxwell 14.0** from the Windows taskbar.
2. Save a new project.
3. Draw the geometry of the model.
4. Optionally, modify the model's design parameters.
5. Assign variables to design parameters.
6. Assign excitations and boundary conditions.
7. Specify solution settings.
8. Run a Maxwell simulation.
9. Create post-processing plots.
10. Create a parametric analysis.
11. Create a field animation of the parametric analysis results.
12. Include motion in the transient design.

About the Example Design

The application described in this Getting Started guide is an extension of the *TEAM Workshop Problem 24* rotational actuator design. The geometry is shown below:

Axial length (rotor and stator) = 25.4



The outer part is a ferromagnetic nonlinear armature carrying two coils. The inner part is made of the same nonlinear material and can rotate around an axis. The inner and outer parts of the device are co-axial.

The field distribution will likely cause the flux density to concentrate in the two steel armatures in the regions where the distance between them is minimal. The expected edge effect will then further increase the field concentration.

In this example, we will compute the torque acting on the inner armature and the flux linkage of the two coils. Simulation results show a 3D electromagnetic time-transient problem with the effects of large motion included. Both the rotor and stator are made of solid ferromagnetic steel, creating significant eddy current effects. A nonlinear B-H curve is considered for the stator and rotor steel. The solution includes the estimated mechanical rotor inertia. For a presentation of the results and the corresponding FEM code, see the IEEE Transactions on Magnetics, Vol 38, No. 2, March 2002, pp 609-612.

2 Setting Up the Design

In this chapter you will complete the following tasks:

- Open and save a new project.
- Insert a new Maxwell design into the project.
- Select a solution type for the project.
- Set the drawing units for the design.

Open Maxwell and Save a New Project

*To save your project frequently, click **File>Save**.*

A project is a collection of one or more designs that is saved in a single *.mxwl file. A new project is automatically created when Maxwell is launched.

To open Maxwell and save a project:

1 Double-click the **Maxwell 14.0** icon on your desktop to launch Maxwell.

- You can also start Maxwell by clicking **Start>Programs>Ansoft>Maxwell 14.0>Maxwell 14.0** from Windows.

*If you want a new Maxwell design to be automatically inserted every time you open Maxwell, do the following: Click **Tools> Options>General Options**, and click the **Project Options** tab. Select the **Insert a design of type** radio button, and select **Maxwell** from the pull-down list. Click **OK**.*

2 Click **Project**>**Insert Maxwell 3D Design**.

The new design is listed in the project tree. By default, it is named **Maxwell3DDesign1**. The **Modeler** window appears to the right of the Project Manager.

3 Click **File>Save As**.

The **Save As** dialog box appears.

4 Locate and select the folder in which you want to save the project.

5 Type **Rotational_Act_TR** in the **File name** box, and click **Save**.

The project is saved in the specified folder under the name **Rotational_Act_TR.mxwl.**

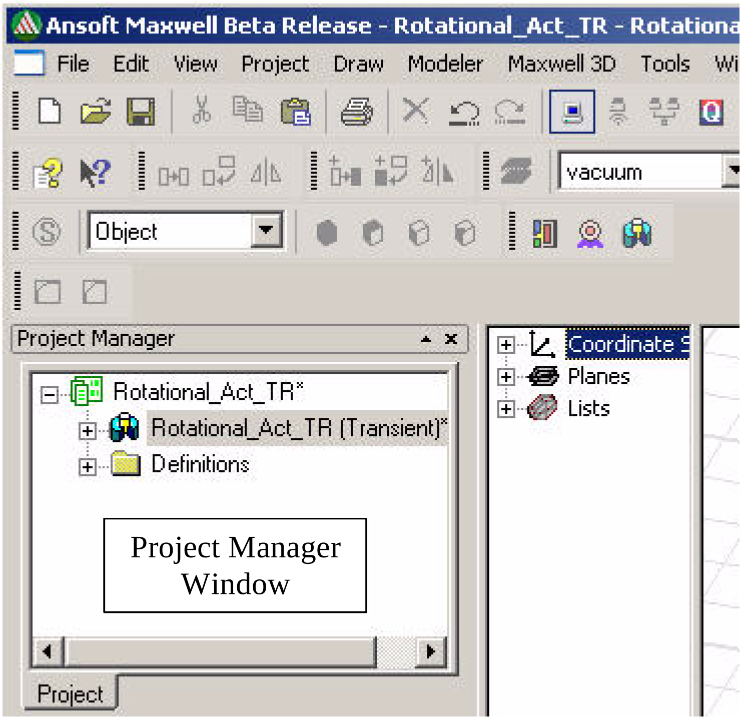
6 Rename the design:

a. Right-click **Maxwell3DDesign1**.
A shortcut menu appears.

b. Select **Rename**.
The design name becomes highlighted and editable.

c. Type **Rotational_Act_TR** as the name for the design, and press **Enter**.

The project and design are now both named **Rotational_Act_TR**.



Specify a Solution Type

As mentioned in the introduction, multiple solution types are available, depending on the specific application. For this design, choose a **Transient** solution.

To specify the solution type:

1. Click **Maxwell3D>Solution Type**.

   The **Solution Type** dialog box appears.

2. Select the **Transient** radio button.
3. Click **OK**.

Set the Drawing Units

To set the drawing units:

1. Click **Modeler>Units**.

   The **Set Model Units** dialog box appears.
2. Select **mm** from the **Select units** pull-down menu.
3. Click **OK**.

3 Importing the Geometric Model

In this chapter you will open the Magnetostatic Getting Started project, copy the objects definitions and material properties, and paste the objects and materials into the **Rotational_Act_TR** transient project.

If you have not completed the magnetostatic guide *Getting Started with Maxwell: Designing a Rotational Actuator*, it is strongly recommended that you do so to gain necessary knowledge of modeling and material assignment principles. You may, however, use the project in the example directory.

In this chapter you will complete the following tasks:

- Open the Magnetostatic Getting Started example.
- Copy and paste geometry and materials to the current project.

Open the Magnetostatic Project

The geometry and materials used in this project are identical to the magnetostatic guide **Rotational_Actuator** project.

To open the magnetostatic project:

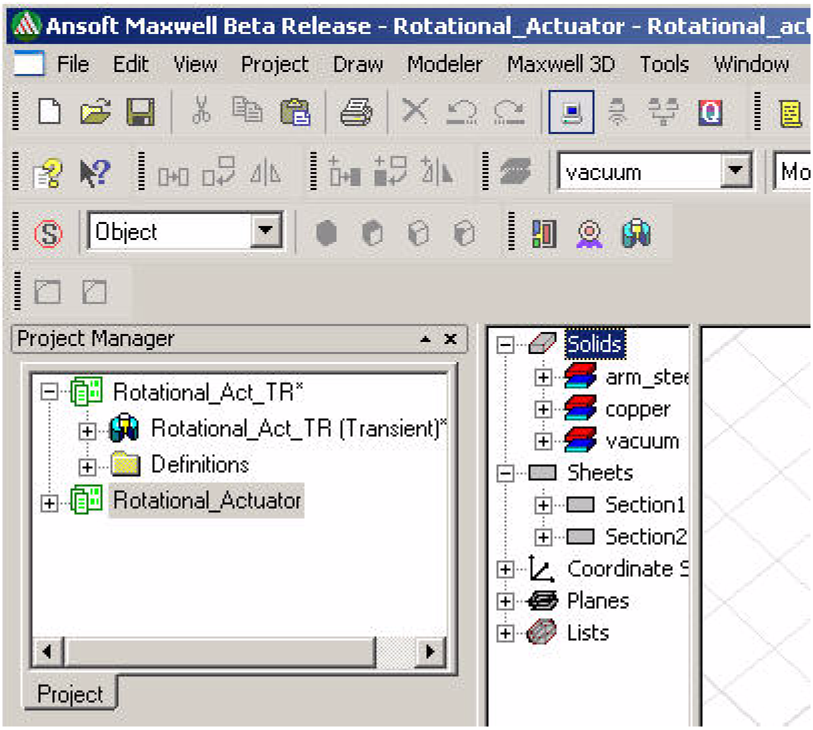
1 Click **File>Open**.

The Windows file browser opens.

2 Locate the folder containing the **Rotational_actuator** project from the magnetostatic guide, *Getting Started with Maxwell: Designing a Rotational Actuator*.

3 Select the file **Rotational_actuator.mxwl** and click **Open**.

The project is opened and is now listed in the Project Manager Window as shown.



Copy and Paste Objects between Projects

The ability to copy and paste objects and their associated material assignments is a useful and time saving function of the Maxwell software.

In order to copy objects, all objects must be selected.

1 With the **Rotational_actuator** project in the modeler window, click in the modeler window and then click **Edit>Select All** to select all object regardless of their visibility status.

   The History Tree will expand and highlight all objects in the design.

2 Click **Edit>Copy** to copy the object and material definitions to the clipboard.

3 Click on the **Rotational_Act_TR(Transient)** design in the **Project Manager** window.

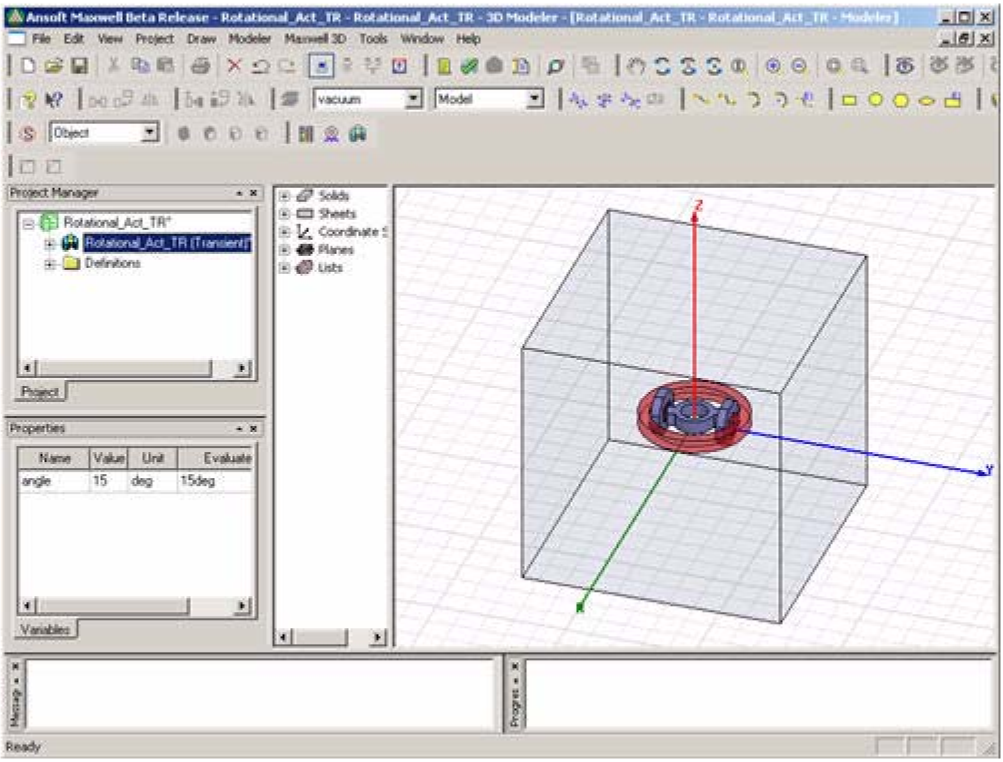
4 Click **Maxwell3D>3D Model Editor** in the menu to switch the **Modeler** window to the transient project.

5 Click **Edit>Paste** to paste all objects and material definitions into the transient project.

6 Click **Edit>View>Fit All>All Views** to fit the objects to the window. You may also use the keyboard shortcut **Ctrl-D**.

7 In the **Project Manager** window, select the magnetostatic project **Rotational_Actuator** and click **File>Close**.

Your screen should look approximately like the one below.



8 Click **File>Save** to save the model before moving on to the next chapter.

4 Defining the Design Properties

For the transient problem, you want to use a pulse excitation to drive the coils. In order to accomplish this, you will assign a external current winding excitation to the coils and use the Maxwell Circuit Editor to create the external driving circuit including a pulse source. You will also allow the software to calculate the eddy currents in the solid metal objects in the model.

In this chapter you will complete the following tasks:

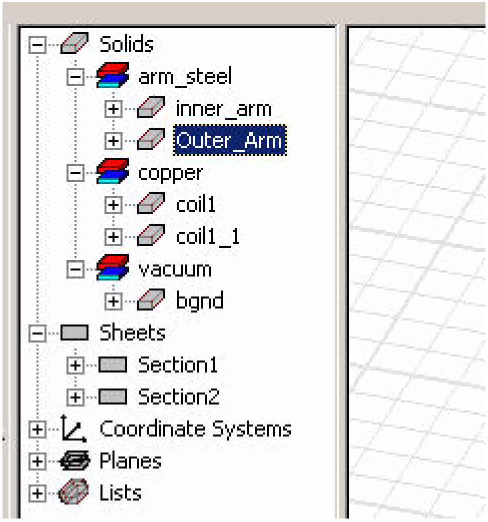
- ✓ Verify material properties.
- ✓ Assign excitations.
- ✓ Set up an external circuit for the current winding
- ✓ Set up the mesh operations.
- ✓ Specify the eddy effect.

Verify Material Properties

Material properties are automatically transferred when you copied the geometry objects. You can view these properties by viewing the **Attribute** tab of the **Properties** window.

To verify the nonlinear material for the armatures:

1 Expand the **History Tree** as shown.



2 Double-click the **Outer_arm** object in the history tree.

The **Properties** window appears.

3 In the **Material** row, click the button in the **Value** column labeled **arm_steel**, then click **Edit**.

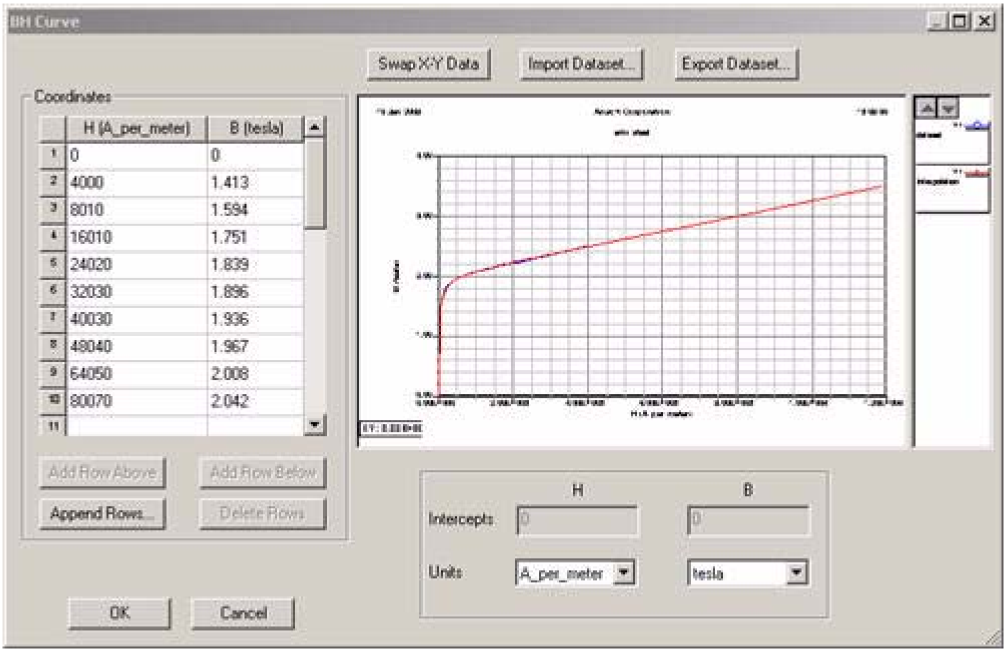
The **Select Definition** dialog box appears.

4 Click the **View/Edit Material** button.

The **View/Edit Material** dialog box appears.

5 In the **Relative Permeability** row, click the **B-H Curve** button.

The **BH Curve** dialog box appears as shown.



6 Click **Cancel** in the **BH Curve** dialog.
The **BH Curve** dialog box closes.

7 Continue dismissing dialogs until you have returned to the **Modeler** window.

Assign Excitations

Currents need to be defined and assigned as excitations for the two coil terminals.

Define the Currents

To define the currents:

1 Select **Section1** and **Section 2** in the history tree under **Sheets**.

2 Click **Maxwell3D>Excitations>Assign>Coil Terminal**.
The **Coil Terminal Excitation** dialog box appears.

3 Type **350** n the **Number of Conductors** box.

4 Click **OK**.

Add a Winding

To add a winding for the excitation:

1. Click **Maxwell3D>Excitations>Add Winding**.

   The **Winding** dialog box appears.
2. Type **currentwinding** in the **Name** box (the default is **Winding1**).
3. Set the **Type** to **External**.
4. Select the **Stranded** radio button.
5. Leave the **Initial Current** set to the default value of **0** (zero).

Note We are using an external circuit to supply the excitation to the coil. For this example, we also could have used a voltage type of excitation.

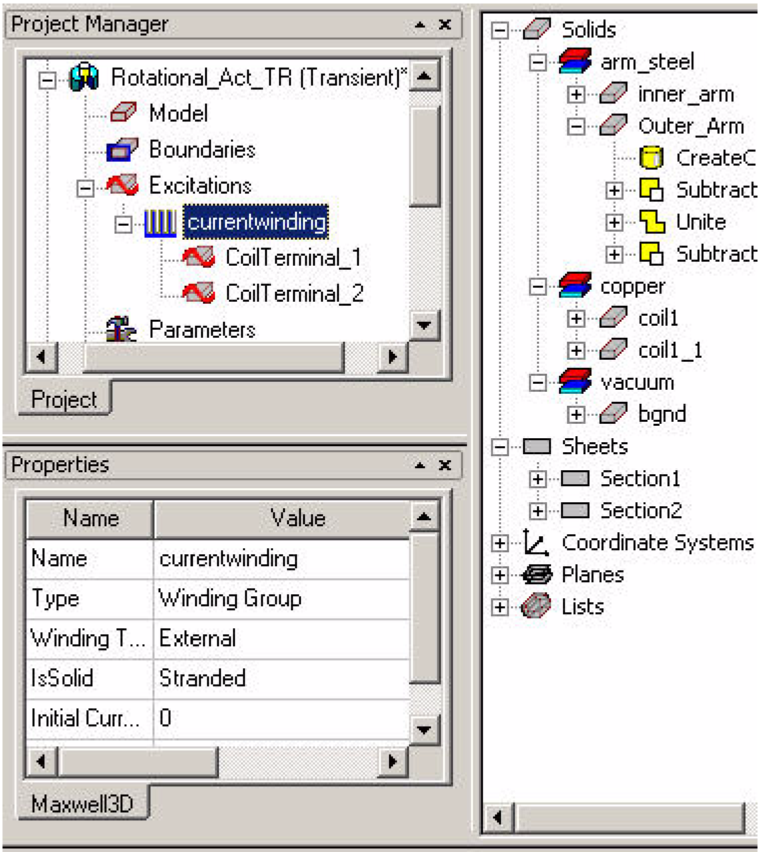
6. Click **OK**.

Add a Winding Terminal

To add a terminal for the winding:

1. In the project tree, under **Excitations**, right-click **currentwinding**.

   A shortcut menu appears.
2. Select **Add Terminals** from the shortcut menu.
   The **Add Terminals** dialog box appears.
3. In the list, select **CoilTerminal_1**, press and hold down the **Shift** key, and select **CoilTerminal_2**.
4. Click **OK**.
   In the project tree, the two terminals are moved beneath the winding as shown below.



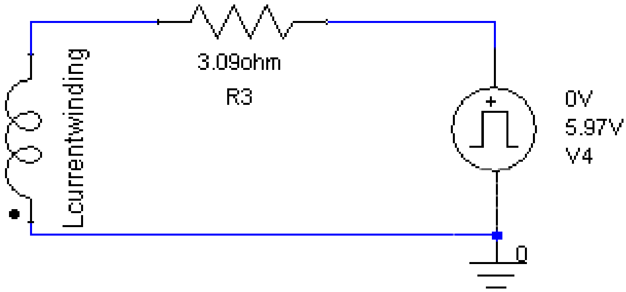
Boundary Conditions

The region box (**bgnd**) by default has all faces assigned with magnetic flux tangent boundary conditions. Thus, for this problem no additional boundary conditions are needed.

Set Up the External Circuit

The driving circuit for the winding in this design consists of a voltage source in series with a resistor and with the winding.

When complete, the circuit should look similar to the figure below.



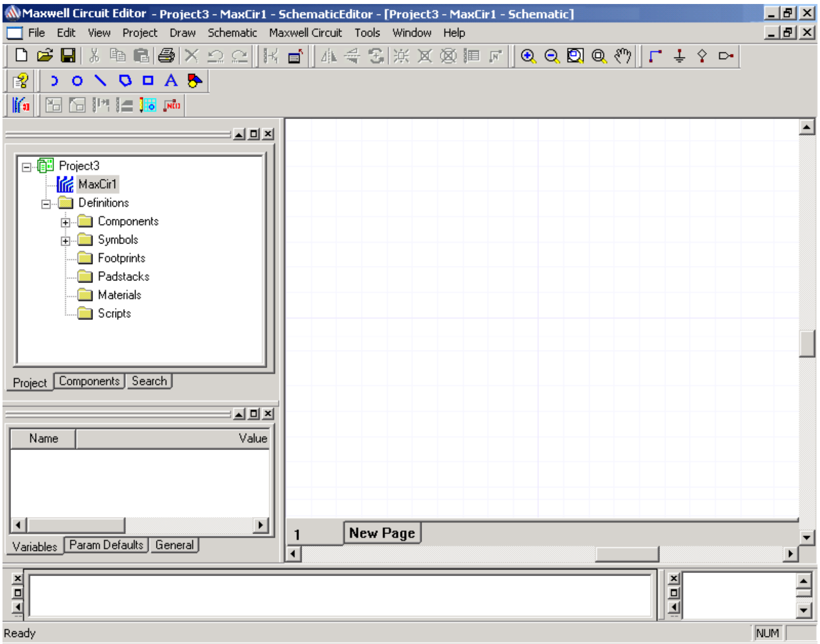
Add the Circuit Elements

To add the circuit elements in Maxwell Circuit Editor:

1 Open the Maxwell Circuit Editor:

Click **Start>Programs>Ansoft>Maxwell 14.0>Maxwell Circuit Editor**.

The Maxwell Circuit Editor program opens with a default circuit sheet as shown below.



2 Click the **Components** tab on the project tree window.

3 Place the winding circuit element on the sheet:

a. In the project tree, under **Maxwell Circuit Elements/Dedicated Elements**, select the **Winding** element.

b. Click on, and Drag the **Winding** element onto the sheet.

c. Right-click in the Schematic window, and select **Finish** to exit component placement mode.

d. To view the properties, double-click the component in the Schematic window.
The **Properties** window appears.

e. Change the **Name** to **currentwinding**, the same name you used when defining the winding in the Maxwell design.

f. Click **OK**.

g. Click **Draw>Rotate**, and position the winding vertically.

4 Place a resistor on the sheet:

a. In the project tree, under **Passive Elements**, select **Res:Resistor**.

b. Drag the resistor onto the sheet.

c. Right-click, and select **Finish** to exit placement mode.

d. Double-click the symbol of the resistor, change the value of the resistor, **R**, to **3.09**, keep the **Unit** value set to **ohm**, and click **OK**.

5 Place a voltage pulse on the sheet:

a. In the project tree, under **Sources** select a **VPulse** element (Pulse Voltage Source).

b. Drag it to the sheet, and then right-click and select **Finish**.

c. Double-click the source element symbol on the sheet, and then specify the following source characteristics:

Parameter	Value	Description
V1	0	Initial voltage
V2	5.97	Peak voltage
Td	0	Initial delay time
Tr	0.001	Rise time
Tf	0.001	Fall time
Pw	1	Pulse width
Period	2	

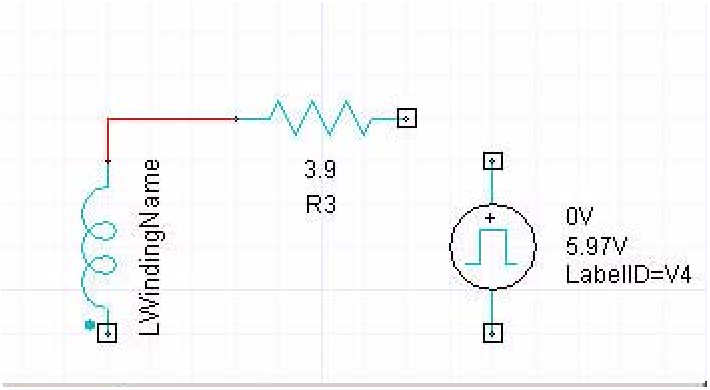
d. Leave the other fields set to the default values, and click **OK**.

Connect the Circuit Elements in Series

To connect the circuit elements in series:

1 From within the Maxwell Circuit Editor, click **Draw>Wire**.

2 Click one terminal of the inductor and draw the wire to one resistor terminal as shown.



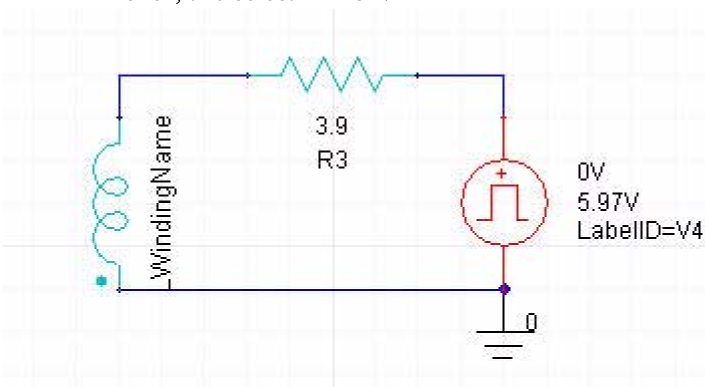
3 Repeat until a closed circuit is formed.

Note For the current winding, the “dotted” terminal is positive and current is postive when it flows from the “dotted” terminal to the “undotted” terminal. Connecting the winding as shown results in negative current in the plot in the **Create a Current vs Time Plot** in Chapter 6. You may reverse the winding orientation with no change to the problem except for the plot which would show positive current in that case.

4 When done, place the Ground symbol:

a. Click **Draw>Ground** (or click the **Ground** symbol on the toolbar),

b. Place the **Ground** symbol on the sheet as shown below, right-click, and select **Finish**.



Export the Netlist

To export the netlist:

1. From within the Maxwell Circuit Editor, click **Maxwell Circuit>Export Netlist**.

   The **Netlist Export** dialog box appears.
2. Select the folder where you want to save the external circuit file.
3. Type **trans_circ** in the **File name** box.
4. Click **Save**.
   The **Netlist Export** dialog box closes and the Maxwell Circuit Editor reappears.

Save the Maxwell Circuit Editor Project

To save the project and exit Maxwell Circuit Editor:

1. Click **File>Save**, type **ExternalCircuit** as the name, and click **Save** to save the Maxwell Circuit Editor project.
2. Click **File>Exit** to close the Maxwell Circuit Editor program.

Assign the External Circuit

To assign the circuit in Maxwell (which should still be open):

1. Click **Maxwell3D>Excitations>External Circuit>Edit External Circuit**.

   The **Edit External Circuit** dialog box appears.
2. Click **Import Circuit**.

   The **Select File** dialog box appears.
3. Select **Designer Net List Files (*.sph)** from the **Files of type** pull-down list.
4. Browse to the location where you saved the circuit, select **trans_circ.sph**, and click **Open** to import it.
5. Click **OK** to close the **Edit External Circuit** dialog box.

Set Up Mesh Operations

This example involves transient magnetic fields in the presence of massive (solid) conductors, creating eddy currents. To catch the effects with reasonable accuracy, a finer mesh is required in those objects because skin effects are part of an accurate transient solution.

To seed the mesh to the desired density in the **Outer_arm** and **Inner_arm** objects:

1. Select **Outer_arm** from the history tree, press and hold down **Ctrl**, and then select **Inner_arm**.
2. Click **Maxwell3D>Mesh Operations>Assign>Inside Selection>Length Based**.

   The **Element Length Based Refinement** dialog box appears.
3. Type **10** in the **Maximum Length of Elements** box, and select **mm** as the units.
4. Leave the **Restrict the Number of Elements** check box unchecked.
5. Click **OK**.

This operation refines the mesh at run-time before the transient problem solution begins. This mesh will be used for all time steps; therefore, the mesh density should be appropriate for the anticipated field behavior for the entire transient analysis.

Specify the Eddy Effect Calculation

Eddy effects can be calculated in objects with non-zero electric conductivity.

To calculate eddy effects:

1. Click **Maxwell3D>Excitations>Set Eddy Effects**.

   The **Set Eddy Effect** dialog box appears.
2. Select the check boxes for the **Inner_arm** and **Outer_arm** objects.
3. Click **OK**.

Specifying Torque Calculation

To set up the torque calculation:

1. Select the **Inner_arm** object by clicking its name in the history tree window.
2. In the project tree, right-click **Parameters** row.
   A shortcut menu appears.
3. Select **Assign>Torque** from the shortcut menu.
   The **Torque** dialog box appears.
4. Select **Global:Z** from the **Axis** pull-down list.
5. Select the **Positive** radio button for the axis orientation.
6. Click **OK**.

5

Setting Up and Running the Analysis

In this chapter you will complete the following tasks:

- Set up the analysis.
- Run and solve the analysis.

Set Up the Analysis

To set up the analysis:

1. Right-click **Analysis** in the project tree.

   A shortcut menu appears.
2. Select **Add Solution Setup**.

   The **Solve Setup** dialog box appears.
3. Click the **General** tab.
4. Type **0.04** in the **Stop time** box, and select **s** as the unit.
5. Type **0.005** in the **Time step** box, and select **s** as the unit.
6. To save fields during the solution:
   a. Click the **Save Fields** tab.
   b. Select **Linear Step** from the **Type** pull-down list to identify the type of sweep.
   c. Type **0.0** in the **Start** box.
   d. Type **0.04** in the **Stop** box.
   e. Type **0.005** in the **Step Size** box.
   f. Click **Add to List**.
7. Click **OK**.
8. Click **File>Save** to save the model before running the analysis.

Run the Analysis

To run the analysis:

- Right-click **Analysis** in the **Project Manager** window, and select **Analyze** in the shortcut menu. The time required to complete the analysis depends upon the speed and memory capability of your machine and other applications that may be using machine resources. The status of the simulation is reported in the **Progress** bar.

6

Post Processing the Results

In this chapter you will complete the following tasks:

- Plot the magnetic flux density vector.
- Plot the current density distribution.
- Plot the torque versus time.
- Plot the current versus time.

Plot the Magnetic Flux Density Vector

Plot the flux density vector on the mid-vertical symmetry plane of the device. You previously set up a relative coordinate system (CS1) containing the desired plot plane.

Create an Object List

Since we want to plot the results only in the two armatures, create a list of these two objects to prepare for the plot.

To create the list of objects:

1. Select the **Outer_arm** and **Inner_arm** objects.
2. Click **Modeler>List>Create>Object List**.

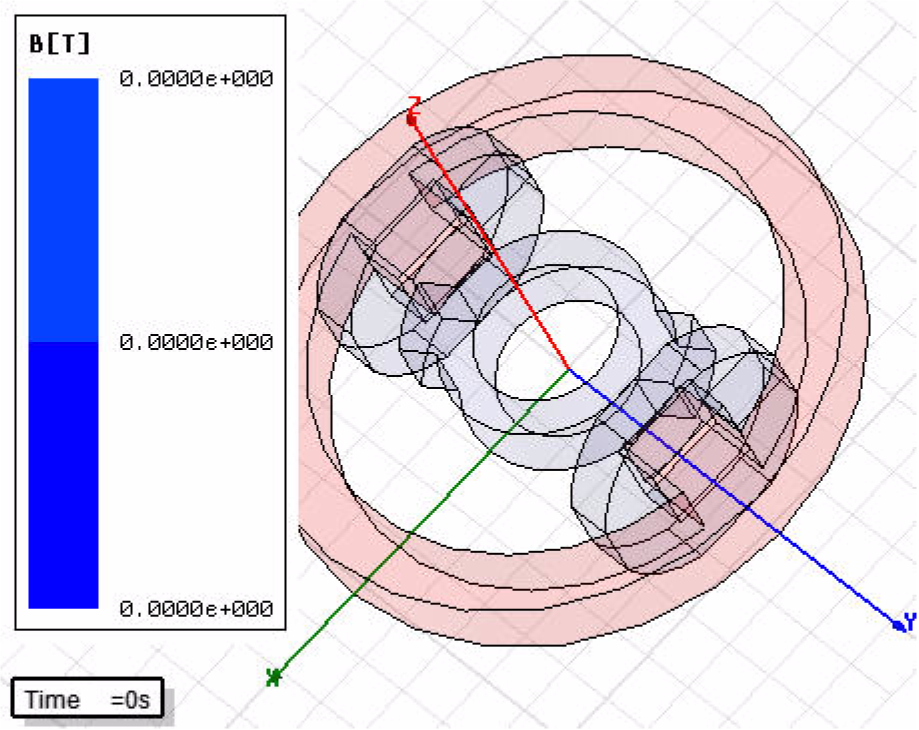
   The list of selected objects (Objectlist1) is added under **Lists** in the history tree window.

Plot the Quantity

To create the plot:

1. Change the rendering of both **Outer_arm** and **Inner_arm** to wireframe by clicking **View>Render>Wire Frame**.
2. In the history tree, select the **RelativeCS1:XY** plane under **Planes**.
3. In the project tree, right-click **Field Overlays**, and select **Fields>B>B_Vector**.

   The **Create Field Plot** dialog box appears.
4. Make sure **B_Vector** is selected in the **Quantity** list.
5. Select **Objectlist1** in the **In Volume** list.
6. Click **Done**.
7. The **B_Vector** plot is displayed as shown below. The vector values are zero at **0sec** as a result of the pulse source used in the winding setup having zero initial voltage.



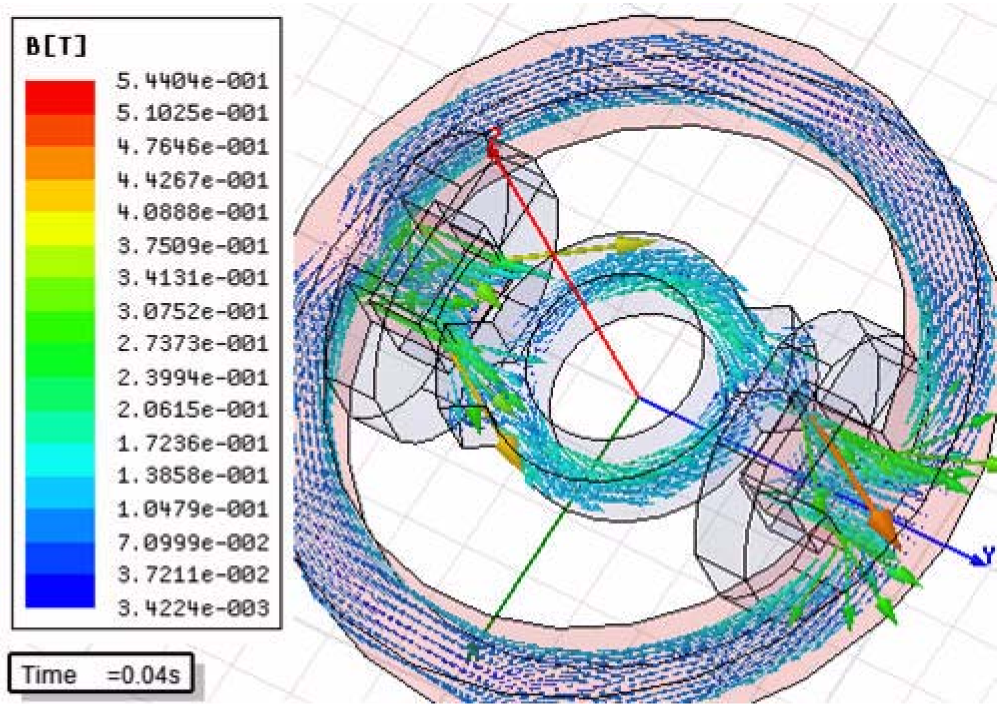
Set the Solution Context

To change the time step for post processing:

1. Click **View>Set Solution Context** or double click on the time context display in the modeler window.

   The **Set View Context** dialog box appears.

2. Select the **Setup1** from the **Solution Name** pull-down list.
3. Set the time step from the **Time** pull-down to **0.04s**.
4. Click **OK**. The plot automatically updates to the new time as shown.



Adjust the Plot Parameters

1 Click **Maxwell3D>Fields>Modify Plot Attributes**.
The **Select Plot Folder** dialog box appears.

2 Select **B** from the list, and click **OK**.
A dialog box appears where you can modify the plot attributes.

Note You may Right-Click in the legend of any displayed plot to display a shortcut menu allowing you to hide or modify the plot.

3 Click the **Scale** tab.

4 Select the **Use Limits** check box.

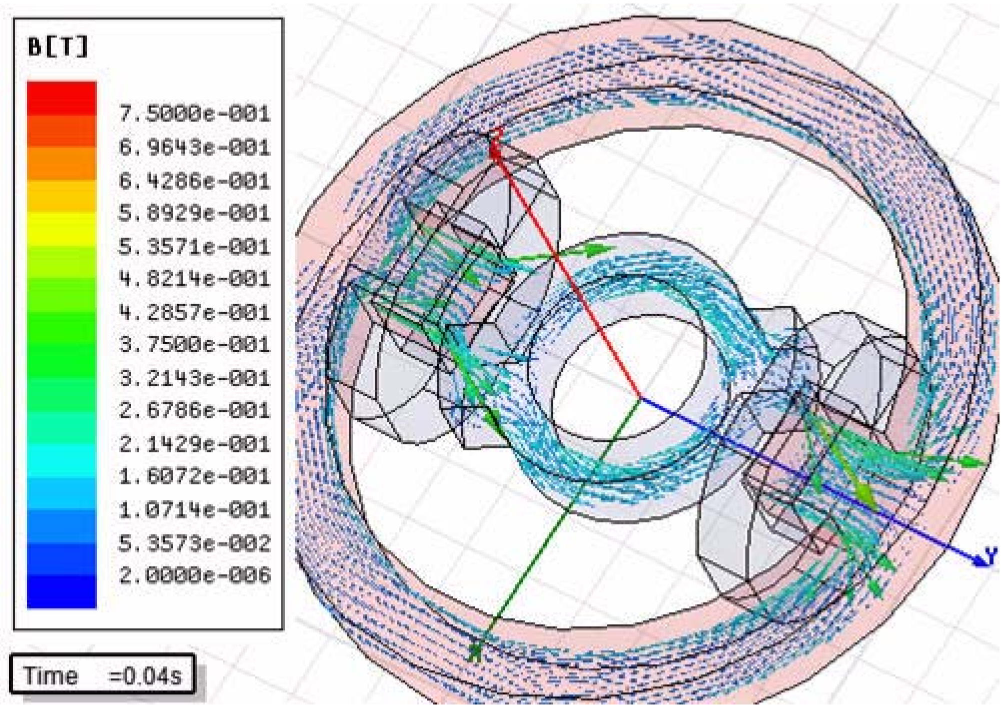
5 Type **2.0E-006** in the **Min** box, and type **7.5E-001** in the **Max** box.

6 Click to **Plots** tab.

7 In the Vector Plot section, set **Min** to **1** and **Max** to **5**.

8 Click **Apply**, and then click **Close**.

The plot should look similar to the figure below:



9 Right-click on **B_Vector1** in the Project Manager window and select **Plot Visibility** in the shortcut menu to turn off the plot.

10 Set the **Solution Context** back to **0s**

Plot the Current Density Distribution

To plot the current density distribution on the same XY plane of CS1 (**RelativeCS1:XY**):

1 Select the plane, and right-click in the modeler window.

A shortcut menu appears.

2 Select **Fields>J>Mag_J** from the shortcut menu.

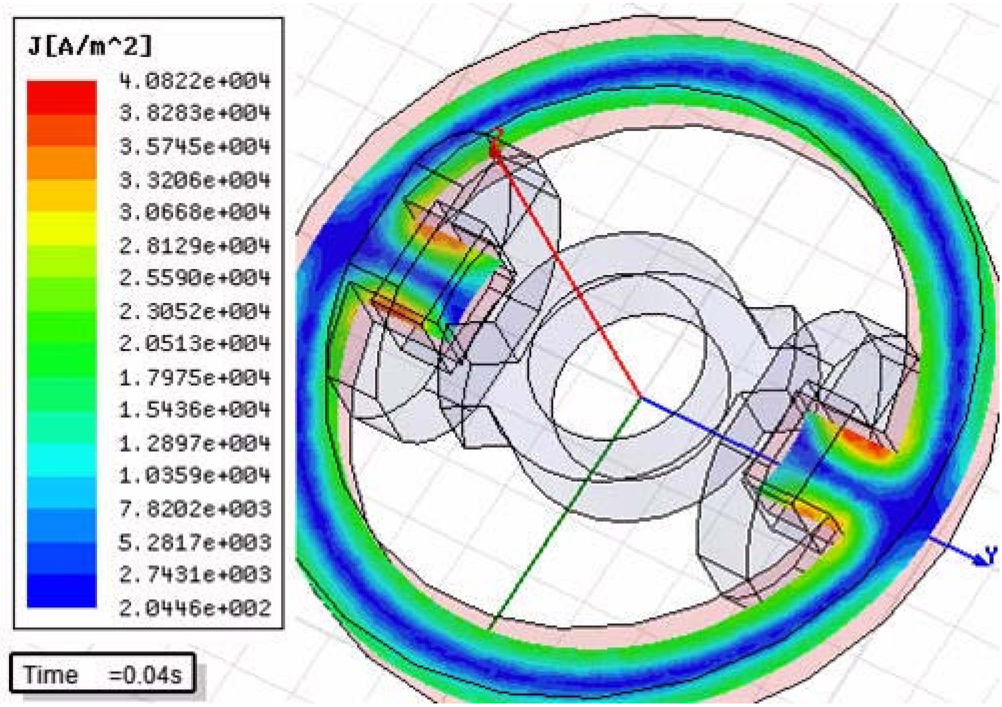
The **Create Field Plot** dialog box appears.

3 Select **Outer_arm** from the **In Volume** List.

4 Click **Done** to plot.

5 Set the **Solution Context** to **0.04s**.

The field partially penetrates the stator, and the transient distribution of the current density shows significant skin effects.



Plot Torque and Current

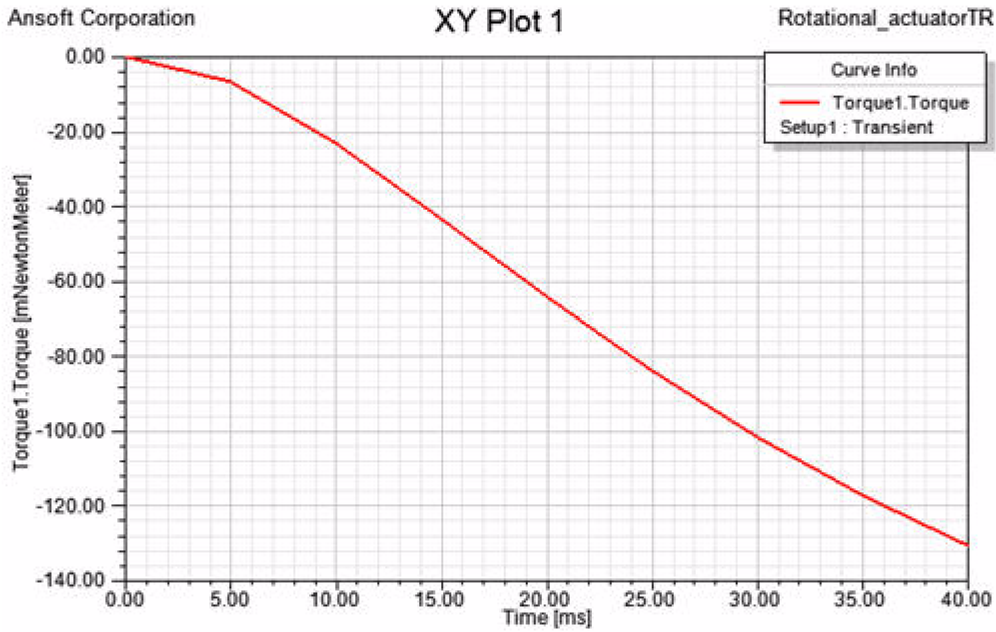
An important transient analysis feature is the ability to vary global quantities as a function of time. Examples of such quantities include currents and voltages, power loss, torque/force, flux linkage of windings, and induced voltages.

Create a Torque vs. Time Plot

To create a plot of the torque as a function of time:

1. Right-click **Results** in the project tree, and select **Create Transient Report>Rectangular Plot**.

   The **Traces** dialog box appears.
2. From the **Solution** pull-down list, select the solution setup (**Setup1**).
3. From the **Category** list, select **Torque**.
4. Click the **New Report** button.
5. Click **Close** to dismiss the dialog.

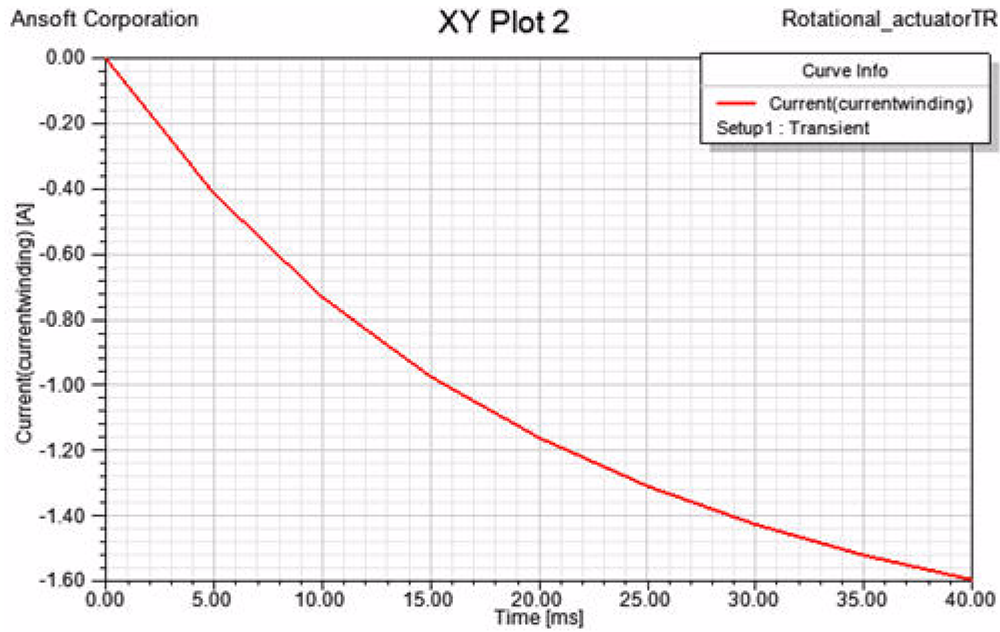


Create a Current vs. Time Plot

To create a plot of the current as a function of time:

1. Right-click **Results** in the project tree, and select **Create Transient Report>Rectangular Plot**.

   The **Traces** dialog box appears.
2. From the **Solution** pull-down list, select the solution setup (**Setup1**).
3. From the **Category** list, select **Winding**.
4. From the **Quantity** list, select **Current**.
5. Click the **New Report** button.
6. Click **Close** to display the dismiss the dialog.



Close the Plot

To close the open plot:

- Click the **X** in the upper right corner of the plot window.

Note After you close a plot, it is still available to view later, listed under **Field Overlays** in the project tree.

7 Including Motion in the Simulation

In order to include the effects of motion of the **Inner_arm** the object must be isolated from the rest of the model using a mesh band. In order to create this mesh band, you will add two objects, between the **Inner_arm** and **Outer_arm** objects.

In general, any moving object must be isolated from the stationary model using a mesh band. More information is available in the online help under the topic *Maxwell 3D Technical Notes:Meshing and Band Setting Recommendations for 3D Transient Applications with Motion*.

In this chapter you will complete the following tasks:

- Add large motion to the simulation.
- Analyze the transient solution with motion.
- Post process the transient results.
- Close the project and exit Maxwell.

Add Motion to the Design

Before adding motion to the design, save the “without motion” (or non-transient) design and create a copy.

To save and copy the design:

1. Click **File>Save** to save the design.
2. In the project tree, right-click the **Rotational_Act_TR** design listed under the project, and select **Copy**.
3. In the project tree, right-click the name of the project (also **Rotational_Act_TR**), and select **Paste**.
   A second copy of the same design appears under the single project; by default, the name of the new design is **Rotational_Act_TR1**.
4. Double-click **Rotational_Act_TR1** to make it active.

Add a Band Object to the Design

The band object is a regular polyhedron positioned so that it contains all rotating objects inside it.

To add the band object:

1. Set the working coordinate system to CS1:

   Click **Modeler>Coordinate System>Set Working CS**, select **RelativeCS1**, and click **Select**.
2. Create a regular polyhedron around the Z axis named **band**:
   a. Click **Draw>Regular Polyhedron**.
   b. Type **(0, 0 -121)** in the **(X, Y, Z)** boxes, for the origin, and then press **Enter**.
   c. Type **(52.5, 0, 0)** in the **(dX, dY, dZ)** boxes, for the radius, and press **Enter**.
   d. Type **(0, 0, 242)** in the **(dX, dY, dZ)** boxes, for the height, and press **Enter**.
      The **Segment number** dialog box appears.
   e. Type **24** in the **Number of segments** text box.
   f. Click **OK**.
      The **Properties** window appears.
   g. Click the **Attribute** tab.
   h. Change the **Name** to **band**.

i. Verify that **band** is assigned the material property of **vacuum** (which should be the default).

j. Click **OK**.
An polyhedron object named **band** is drawn.

3 Create a cylinder named **air_rotor** with the following properties

a. Click **Draw>Cylinder**.
The cursor changes to a small black box, indicating that you are in **Drawing** mode.

b. Enter the center of the cylinder base by typing **(0,0,-120)** in the **(X, Y, Z)** boxes at the bottom of the screen, and press **Enter**.

c. Type **51.05** for the radius in the **dX** box at the bottom of the screen, and press **Enter**.

d. Type **240** for the height in the **dZ** box, and press **Enter**.
The **Properties** window appears.

e. Click the **Attribute** tab.

f. Change the **Name** to **air_rotor**.

g. Verify that **air_rotor** is assigned the material property of **vacuum** (which should be the default).

h. Click **OK**.
A cylinder named **air_rotor** is drawn.

Assign Motion to the Band Object

The circumference of the **band** object falls between the inner armature and the outer armature and contains inside it the **air_rotor** and **Inner_arm** objects.

To set the band object:

1 Select **band** from the history tree.

2 Right-click **Model** in the project tree under **Rotational_Act_TR1**, and then select **Motion Setup>Assign Band**.
The **Motion Setup** dialog box appears.

3 Click the **Type** tab.

4 Select **Rotation** as the **Motion Type**.

5 Set the **Rotation Axis** to **Global:Z** and select the **negative** radio button.

6 Click the **Mechanical** tab.

7 Select the **Consider Mechanical Transient** check box.

8. Type **0.0024** in the **Moment of Inertia** text box and **0.015** in the **Damping** text box.
9. Click **OK**.

Apply Meshing to the Band Object

To apply the appropriate mesh operation to the band object:

1. Select the **band** object in the history tree.
2. Click **Maxwell3D>Mesh Operations>Assign>Inside Selection>Length Based**.
   The **Element Length Based Refinement** dialog box appears.
3. Type **20** in the **Maximum Length of Elements** box, and select **mm** as the units.
4. Click **OK**.
5. To initialize the problem, right-click **Setup1** in the project tree, and select **Revert to Initial Mesh**.

Now you are ready to start the analysis with the effect of large motion included.

Set Up the Transient Analysis

To set up a second analysis:

1. Right-click **Analysis** in the project tree.
   A shortcut menu appears.
2. Select **Add Solution Setup**.
   The **Solve Setup** dialog box appears.
3. Click the **General** tab.
4. Type **0.9** in the **Stop time** box, and select **s** as the unit.
5. Type **0.005** in the **Time step** box, and select **s** as the unit.
6. Add a sweep:
   a. Click the **Save Fields** tab.
   b. Select **Linear Step** from the **Type** pull-down list to identify the type of sweep.
   c. Type **0.0** in the **Start** box.
   d. Type **0.9** in the **Stop** box.
   e. Type **0.005** in the **Step Size** box.
   f. Click **Add to List**.

7 Click **OK**.

Run the Transient Analysis

To run the analysis:

- Under **Analysis** in the project tree, right-click **Setup2**, and select
  Analyze.

Post Process the Transient Results

For the transient case, additional mechanical quantities are available (to represent as 2D plots as functions of time), which were not available before adding motion to the design. When you create a report for a solution that is set to **Transient**, these new quantities can be added as traces.

Create a Position vs. Time Plot

To create a plot of the position as a function of time:

1. Right-click **Results** in the project tree, and select **Create Transient Report>Rectangular Plot**.

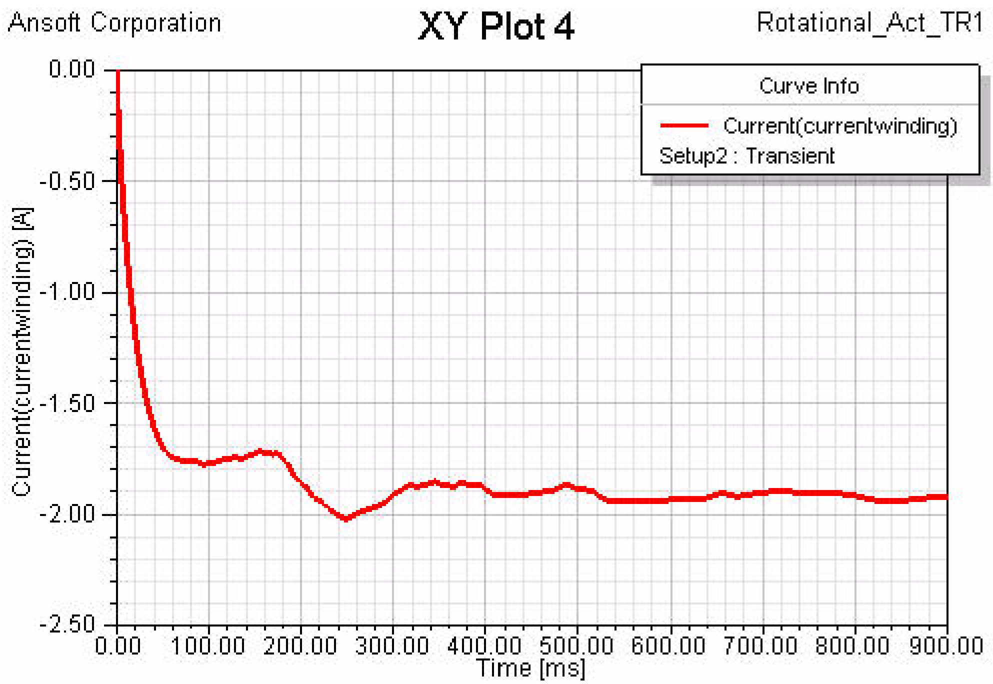
   The **Traces** dialog box appears.
2. From the **Solution** pull-down list, select the solution setup (**Setup2**).
3. From the **Category** list, select **Position**.
4. Click the **New Report** button.
5. Click **Close** to dismiss the dialog.

Current vs. Time Plot with Motion

To create a plot of the current as a function of time:

1. Right-click **Results** in the project tree, and select **Create Transient Report>Rectangular Plot**.

   The **Traces** dialog box appears.
2. From the **Solution** pull-down list, select the solution setup (**Setup2**).
3. From the **Category** list, select **Winding**.
4. From the **Quantity** list, select **Current**.
5. Click the **New Report** button.
6. Click **Close** to dismiss the dialog.

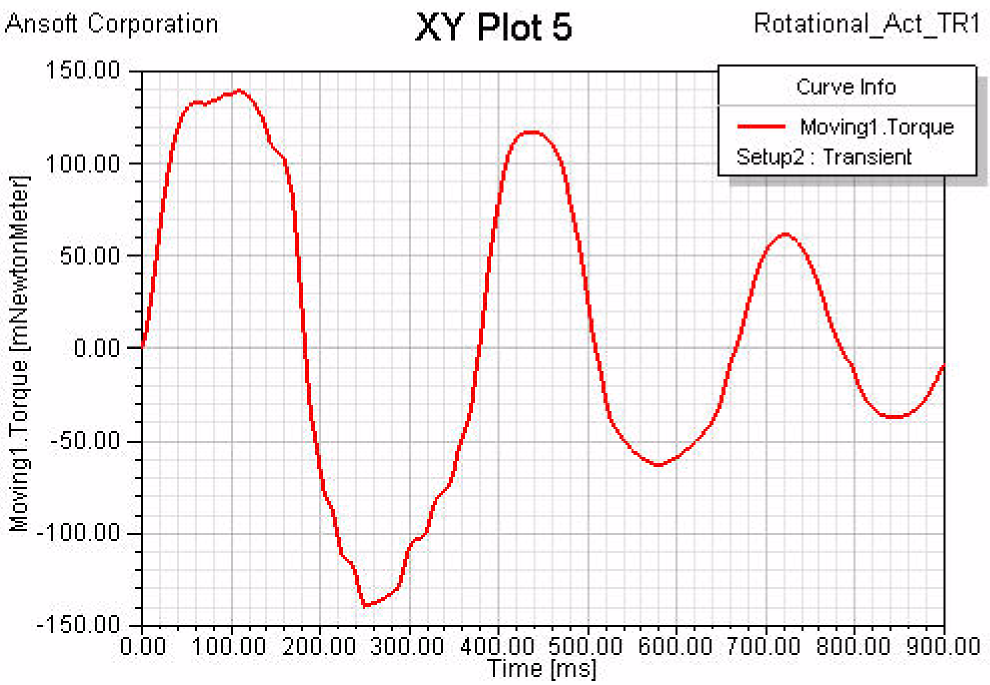


Torque vs. Time Plot with Motion

To create a plot of the torque as a function of time:

1. Right-click **Results** in the project tree, and select **Create Report>Transient>Rectangular Plot**.

   The **Traces** dialog box appears.
2. From the **Solution** pull-down list, select the solution setup (**Setup2**).
3. From the **Category** list, select **Torque**.
4. From the **Quantity** list, select **Moving1.Torque**.
5. Click the **New Report** button.
6. Click **Close** to dismiss the dialog.



Create a Power Loss vs. Time Plot

To create a plot of the eddy current power loss in the **Inner_Arm** and **Outer_Arm** as a function of time:

1 Right-click **Results** in the project tree, and select **Create Transient Report>Rectangular Plot**.

The **Traces** dialog box appears.

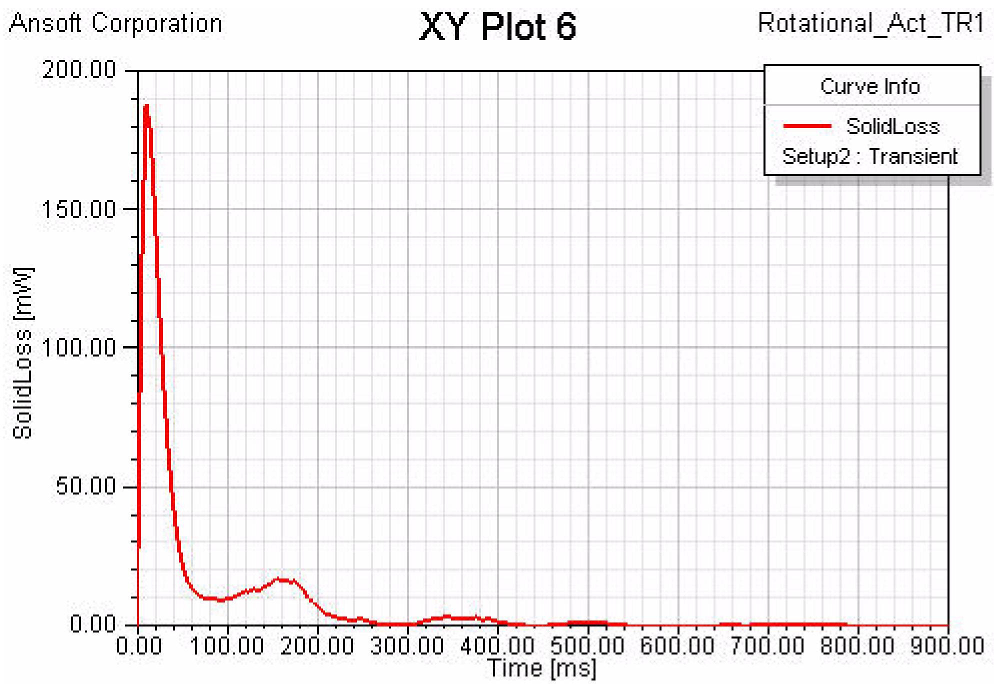
2 From the **Solution** pull-down list, select the solution setup (**Setup2**).

3 From the **Category** list, select **Loss**.

4 From the **Quantity** list, select **Solid Loss**.

5 Click the **New Report** button.

6 Click **Close** to dismiss the dialog.



Close the Project and Exit Maxwell

Congratulations! You have successfully completed his *Getting Started with Maxwell: Transient Problem*! You may close the project and exit the Maxwell software.

1. Click **File>Save** to save the project.
2. Click **File>Close**.
3. Click **File>Exit** to exit Maxwell.

Index

D

E

F

G

H

I

L

M

N

O

V

W