Getting Started with Maxwell:
Transient Problem
The information contained in this document is subject to change without notice. ANSYS, Inc. makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. ANSYS, Inc. shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2010 SAS IP, Inc. All rights reserved.

Ansoft, Maxwell, and Optimetrics are registered trademarks or trademarks of SAS IP, Inc. All other trademarks are the property of their respective owners.

Unigraphics and Parasolid geometry translators incorporated in this software are used under license from Siemens PLM © 2006. All rights reserved.

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.

<table>
<thead>
<tr>
<th>Edition</th>
<th>Date</th>
<th>Software Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Feb 2008</td>
<td>Maxwell 12</td>
</tr>
<tr>
<td>2</td>
<td>April 2010</td>
<td>Maxwell 13</td>
</tr>
<tr>
<td>3</td>
<td>November 2010</td>
<td>Maxwell 14.0</td>
</tr>
</tbody>
</table>
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**
   - Maxwell Solution Types ........................................ 1-3
   - The Maxwell Desktop ........................................... 1-4
   - General Procedure for Setting Up Maxwell Designs 1-5
   - About the Example Design ................................. 1-6

2. **Setting Up the Design**
   - Open Maxwell and Save a New Project ................. 2-2
   - Specify a Solution Type ....................................... 2-3
   - Set the Drawing Units ........................................ 2-4

3. **Importing the Geometric Model**
   - Open the Magnetostatic Project ....................... 3-2
   - Copy and Paste Objects between Projects ........... 3-3

4. **Defining the Design Properties**
   - Verify Material Properties .............................. 4-2
   - Assign Excitations .......................................... 4-3
     - Define the Currents ...................................... 4-3
     - Add a Winding ............................................. 4-4
     - Add a Winding Terminal ................................. 4-4
   - Boundary Conditions ....................................... 4-5
Set Up the External Circuit .......................... 4-5
Set Up Mesh Operations ......................... 4-11
Specify the Eddy Effect Calculation .............. 4-11
Specifying Torque Calculation .................... 4-12

5. Setting Up and Running the Analysis
Set Up the Analysis ................................. 5-2
Run the Analysis .................................... 5-2

6. Post Processing the Results
Plot the Magnetic Flux Density Vector .......... 6-2
  Create an Object List ............................. 6-2
  Plot the Quantity ................................ 6-2
  Set the Solution Context ......................... 6-3
  Adjust the Plot Parameters ...................... 6-4
Plot the Current Density Distribution ........... 6-5
Plot Torque and Current .......................... 6-7
  Create a Torque vs. Time Plot .................. 6-7
  Create a Current vs. Time Plot ................. 6-8
  Close the Plot .................................. 6-9

7. Including Motion in the Simulation
Add Motion to the Design .......................... 7-2
  Add a Band Object to the Design ............... 7-2
  Assign Motion to the Band Object ............... 7-3
  Apply Meshing to the Band Object ............... 7-4
Set Up the Transient Analysis .................... 7-4
Run the Transient Analysis ....................... 7-5
Post Process the Transient Results ............... 7-6
  Create a Position vs. Time Plot ................. 7-6
  Current vs. Time Plot with Motion ............... 7-7
  Torque vs. Time Plot with Motion ............... 7-8
  Create a Power Loss vs. Time Plot ............... 7-9
  Close the Project and Exit Maxwell ............... 7-10
1 Introduction

This Getting Started Guide is written for Maxwell beginners and experienced users who would like to quickly re-familiarize 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>Maxwell 14.0>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
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

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.

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.

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

3-2 Importing the Geometric Model
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 objects 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.
For the transient problem, you want to use a pulse excitation to drive the coils. In order to accomplish this, you will assign an 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.

   ![History Tree Diagram]

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 in 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).
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.

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.
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:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>V1</td>
<td>0</td>
<td>Initial voltage</td>
</tr>
<tr>
<td>V2</td>
<td>5.97</td>
<td>Peak voltage</td>
</tr>
<tr>
<td>Td</td>
<td>0</td>
<td>Initial delay time</td>
</tr>
<tr>
<td>Tr</td>
<td>0.001</td>
<td>Rise time</td>
</tr>
<tr>
<td>Tf</td>
<td>0.001</td>
<td>Fall time</td>
</tr>
<tr>
<td>Pw</td>
<td>1</td>
<td>Pulse width</td>
</tr>
<tr>
<td>Period</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

   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 positive 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.
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.
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.
6-10 Post Processing the Results
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**.
   g. The **Properties** window appears.
   h. Click the **Attribute** tab.
   i. Change the **Name** to **band**.
Verify that **band** is assigned the material property of **vacuum** (which should be the default).

Click **OK**.

An polyhedron object named **band** is drawn.

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

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

2. 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**.

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

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

5. Click the **Attribute** tab.

6. Change the **Name** to **air_rotor**.

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

8. 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.
Getting Started with Maxwell: A Transient Problem

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

Numbers
3D Modeler window 1-4

A
adding a band object 7-2
adding a solution setup 5-2, 7-4
adding a trace 6-8
adding a winding 4-4
adding a winding terminal 4-4
adding motion to the design 7-2
analysis setup 5-2, 7-4
Analyze command 5-2, 7-5
analyzing the solution 5-2
analyzing the transient solution 7-5
applying a mesh 7-4
assigning excitations 4-3
assigning motion to the band 7-3
attributes
  plot properties 6-2

B
B vector 6-2

background
  boundary condition 4-5
band object
  adding 7-2
  applying meshing 7-4
  assigning motion 7-3
band object for motion 7-2

C
calculating the eddy effect 4-11
closing a plot 6-9
closing Maxwell 7-10
closing the project 7-10
coil terminal 4-3
context-sensitive help 1-iii
conventions used in guide 1-i
coordinate system 7-2
copyright notice 1-2
Create Field Plot dialog box 6-5
Create Report dialog box 6-8
creating a report 7-6
creating an object list 6-2
current
  assigning excitations 4-3
  plotting 6-7
current density distribution 6-5
current vs. time plot 6-8, 7-7
### Index-2

**D**
- desktop 1-4
  - Maxwell 1-4
- drawing units 2-4
- driving circuit for winding 4-5

**E**
- eddy current effect 4-11
- editing an external circuit 4-10
- electric 3D field solutions 1-3
- example Maxwell design 1-6
- excitations 4-3
  - winding terminal 4-4
- exiting Maxwell 7-10
- exporting a netlist 4-10
- external circuit for winding 4-5

**F**
- field overlays 6-2

**G**
- general design procedure 1-5

**H**
- harmonic steady-state solutions 1-3
- help
  - Ansoft technical support 1-iii
  - context-sensitive 1-iii
  - on dialog boxes 1-iii
  - on menu commands 1-iii
- history tree 1-4

**I**
- including motion in the simulation 7-2

**L**
- length-based mesh refinement 4-11

**M**
- Mag J
  - plotting the current density distribution 6-5
- magnetic flux density vector 6-2
- magnetostatic solutions 1-3
- Maxwell
  - closing 7-10
  - design diagram for example problem 1-6
  - exiting 7-10
  - renaming a design 2-2
  - renaming a project 2-2
  - saving project 7-10
- Maxwell Circuit Editor 4-5
- Maxwell desktop 1-4
- Maxwell solution types 1-3
- mechanical quantities 7-6
- menu bar 1-4
- mesh
  - band object 7-4
- mesh operations 4-11
- Message window 1-4
- modifying plot attributes 6-2
- motion
  - adding to design 7-2
  - band object 7-3

**N**
- netlist
  - exporting 4-10

**O**
- object list
Getting Started with Maxwell: A Transient Problem

creating 6-2
opening a project 2-2

P
plot
   closing 6-9
plot attributes 6-2
plotting
   current density distribution 6-5
   current vs. time 6-8, 7-7
   magnetic flux density vector 6-2
   object list 6-2
   position vs. time 7-6
   power loss vs. time 7-9
   preparation 6-2
   torque vs. time 6-7, 7-8
plotting current 6-7
plotting torque 6-7
position vs. time plot 7-6
post processing 6-2, 7-6
power loss vs. time plot 7-9
preparing for a plot 6-2
Progress window 1-4
project
   closing 7-10
Project Manager window 1-4
project tree 1-4
projects
   opening and saving 2-2
properties
   plot attributes 6-2

Q
quantity
   plotting 6-2

R
refining the mesh 4-11
renaming

a design 2-2
a project 2-2
report type 6-8, 7-6
reverting to the initial mesh 7-4
rotational actuator example 1-6
running for transient case 7-5
running the analysis 5-2
running the transient analysis 7-5

S
saving a project 2-2
saving project 7-10
Set Solution Context command 6-3
Set View Context dialog box 6-3
setting a working coordinate system 7-2
setting drawing units 2-4
setting eddy effects 4-11
setting solution context 6-3
setting the time step 6-3
setting up an external circuit 4-5
setting up Maxwell designs 1-5
setting up mesh operations 4-11
setting up the analysis 5-2, 7-4
simulation
   running 5-2
   transient 7-5
sinusoidal solutions 1-3
solution
   transient 7-5
solution context 6-3
solution setup 5-2, 7-4
solution simulation 5-2
solution type 1-3, 2-3
sources
   assigning 4-3
   specifying the eddy effect calculation 4-11

T
TEAM workshop problem 24 1-6
terminal
Getting Started with Maxwell: A Transient Problem

winding 4-4

time domain solution 1-3

time step
  setting solution context 6-3

toolbars 1-4

torque vs. time plot 6-7, 7-8
	trademark notice 1-2

transient design 7-2

transient electric fields 1-3

transient feature 6-7

transient results
  post processing 7-6

transient simulation 7-5

transient solution type 2-3

transient solutions 1-3

V

varying global quantities as a function of
  time 6-7

verify materials 4-2

view context 6-3

W

winding
  adding 4-4
    external driving circuit 4-5

winding terminal
  adding 4-4

windows in Maxwell desktop 1-4

working coordinate system (CS) 7-2