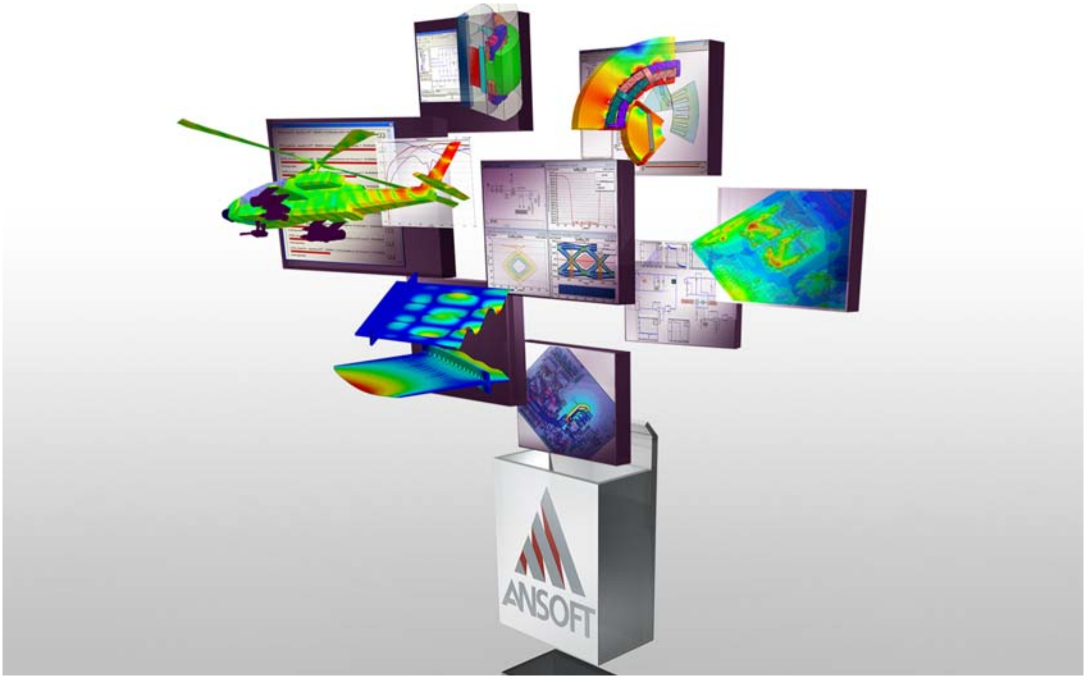


Getting Started with Maxwell: Designing a Rotational Actuator

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: 002889

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.

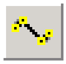
Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Pages that are rearranged due to changes on a previous page are not considered to be revised.

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 re-familiarize themselves with the capabilities of Maxwell. This guide leads you step-by-step through creating, solving, and analyzing the results of solving a rotational actuator magnetostatic problem.

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

- Draw a geometric model.
- 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.
- Run a parametric analysis.
- Create an animation using saved parametric field data.

Estimated time to complete this guide: 60minutes.

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.

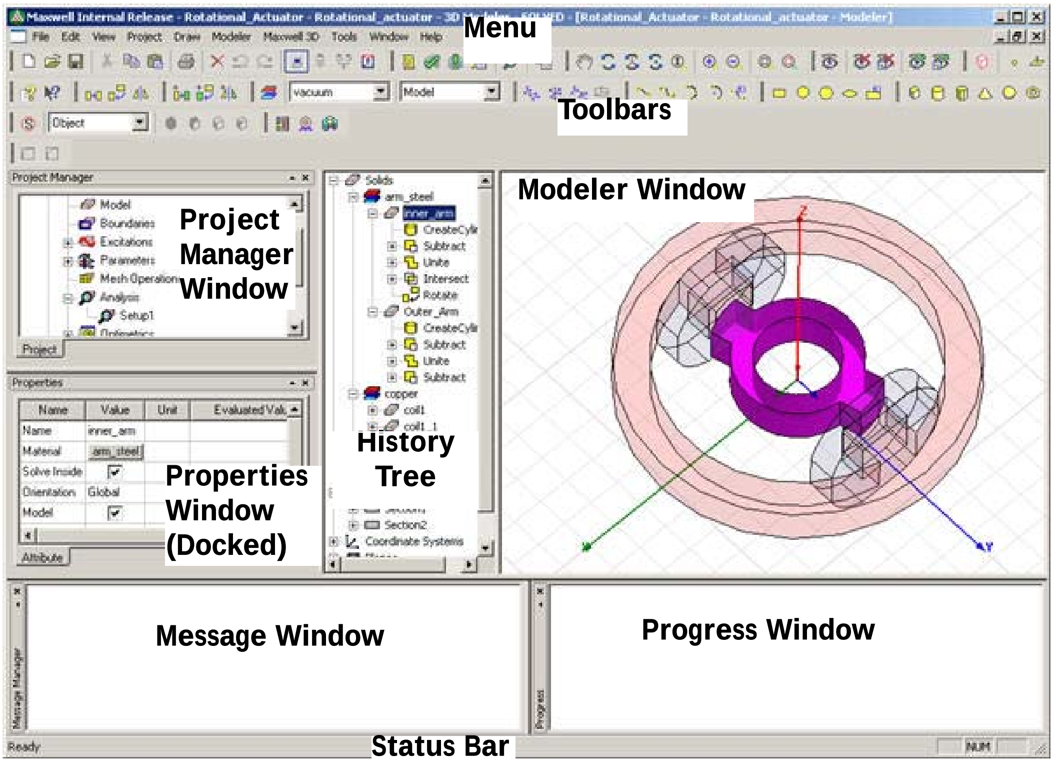
  This solution includes displacement currents for calculating near field electromagnetic wave radiation.
- **Transient (time domain) magnetic fields** caused by permanent magnets, conductors, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time. Rotational 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 artibrary 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:



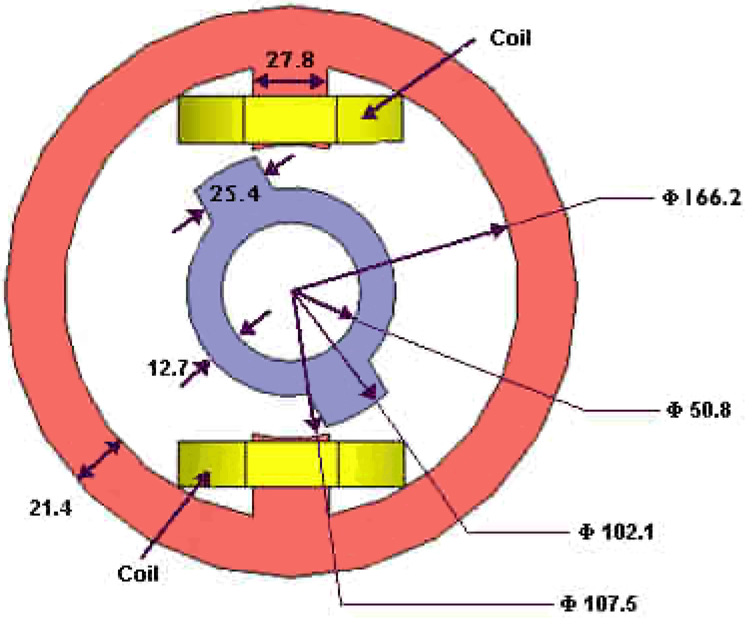
General Procedure for Setting Up Maxwell Designs

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.

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 Depth (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. 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.

Open Maxwell, add a new design, and save the default project with a new name.

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* ***Maxwell3D*** *from the pull-down list. Click* ***OK****.*

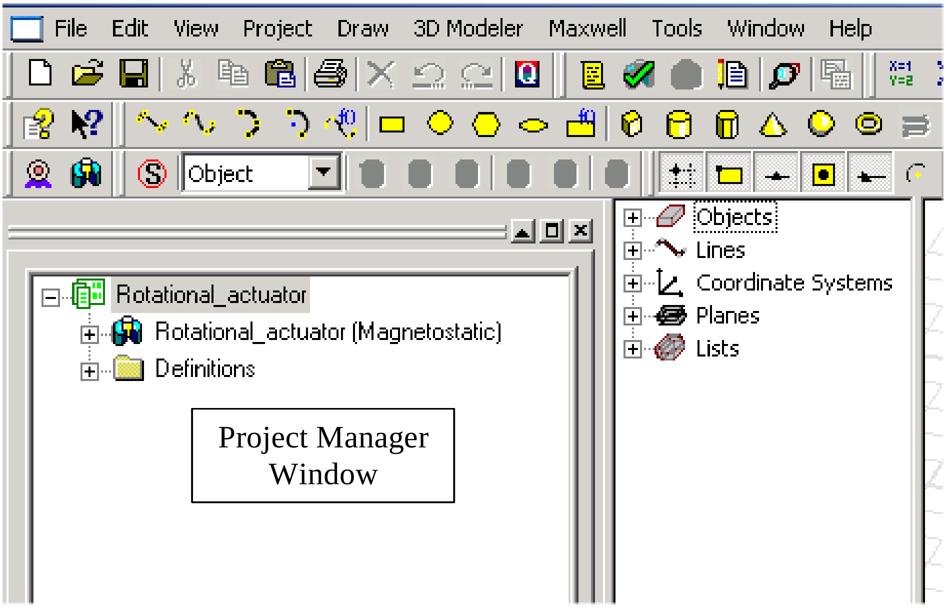
To open Maxwell and save a project:

1. Double-click the **Maxwell 14.0** icon on your desktop to launch Maxwell.
   - You can also start Maxwell by clicking **Start>Programs>Ansoft>Maxwell14.0>Maxwell14.0** 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 (another name for the project tree).
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_actuator** in the **File name** box, and click **Save**.
6. The project is saved in the specified folder under the name **Rotational_actuator.mxwl**.Rename the design:
   a. Right-click **Maxwell3DDesign1**.
      A shortcut menu appears.
   b. Select **Rename**.
      The design name becomes highlighted and editable.
   c. Type a **Rotational_actuator** as the name for the design, and press **Enter**.
      The project and design are now both named **Rotational_actuator**.

You can add additional designs to the project by repeating step 2 above, or by clicking the corresponding icon on the toolbar. For the current simulation task, we do not need to create additional designs.

For More Information see the Online Help Topic:

General Options: Project Options Tab



Specify a Solution Type

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

1 Click **Maxwell3D>Solution Type** from the menus.

The **Solution Type** dialog box appears.

2 Select the **Magnetostatic** radio button.

3 Click **OK**.

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

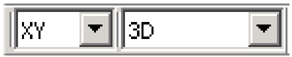
Creating the Geometric Model

In this chapter you will complete the following tasks:

- Set the drawing plane and movement mode.
- Create the outer armature of the actuator by subtracting and uniting objects.
- Create the inner armature of the actuator.
- Create the coils.
- Create the coil terminals.
- Create the background object.
- Finalize the geometry by rotating the inner arm.

Set the Drawing Plane and Movement Mode

Before creating the geometry, make sure the **XY** drawing plane is selected and **3D** is selected as the movement mode.These options are on the **3D Modeler** toolbar.



To set the drawing plane:

- Select **XY** from the **Drawing plane** pull-down list on the **3D Modeler Draw** toolbar.

To set the movement mode:

- Select **3D** from the **Movement mode** pull-down list on the **3D Modeler Draw** toolbar.

Create the Outer Armature Object

The outer armature consists of two cylinders (for an outer and inner radius) that are subtracted to leave the armature. Magnetic poles are then added to the armature object.

Draw the Outer Cylinder

Create the outer radius of the outer armature object.

To create the outer cylinder:

1 Click **Draw>Cylinder**.

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

2 Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system, and press the **Tab** key to jump to the manual entry area in the Status Bar at the bottom of the screen.

3 Notice the Status Bar is prompting for Radius of the cylinder. Type **104.5** for the radius in the **dX** box, and ensure that **dY** and **dZ** are set to **zero**. Press **Enter**.

4 The Status Bar is now prompting for height of the cylinder. Type **25.4** for the height in the **dZ** box, and press **Enter**. The cylinder is created and the default properties appear in the **Properties Window**.

Note Optionally, you may use the pop-up Properties Window by configuring user options.

5 In the **Attribute** tab, change the **Name** (currently **Cylinder1**) to **Outer_arm**.

6 Change the color of the cylinder to **red**:

a. In the **Color** row, click the **Edit** button.
The **Color** palette dialog box appears.

b. Select any of the red shades from the **Basic colors** group, and click **OK** to return to the **Properties** window.

7 Set the transparency to **0.6**:

a. Click the button for the **Transparent** property.
The **Set Transparency** dialog box appears.

b. Type **0.6** in the text box, and click **OK** to return to the **Properties** window.

8 Optionally, click the **Command** tab to view and edit the geometric data. For this example, we do not need to edit the geometric data.

Note You can also view the **Command** tab by double-clicking the CreateCylinder entry in the history tree window.

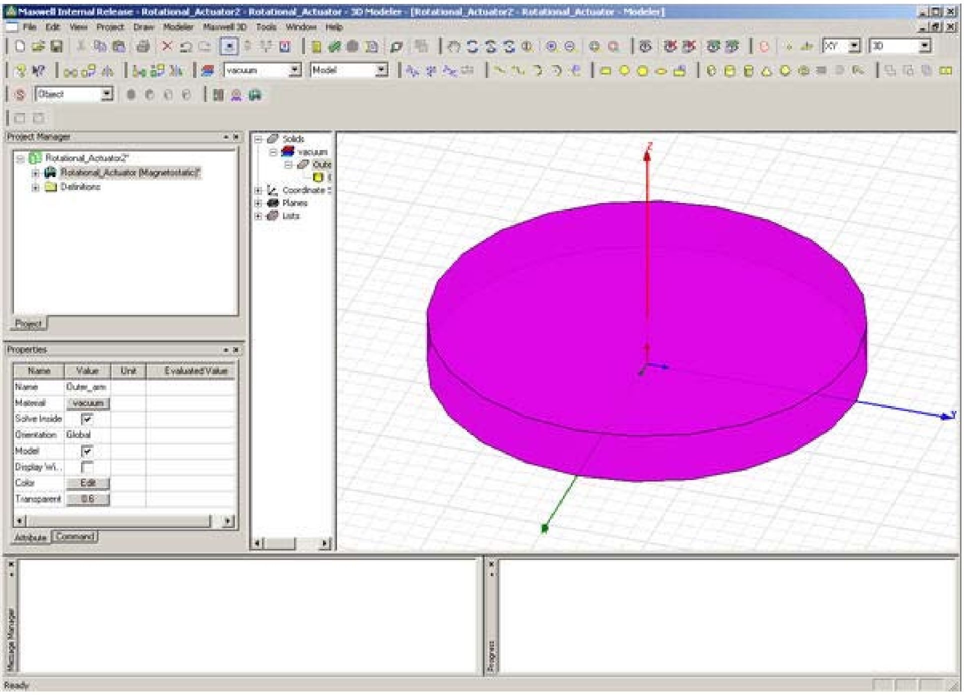
9 Optionally, when using the pop-up **Properties** dialog, click **OK** to close the **Properties** window. A cylinder named **Outer_arm** is drawn.

View the Entire Cylinder

It is easier to view the model you are drawing if you set it to fit the screen.

To fit the entire model on the screen:

- Click **View>Fit All>All Views** or use the keyboard shortcut **Ctrl+D**.



Draw the Inner Cylinder

Add the inner radius for the outer armature object.

To create the inner cylinder:

1. Click **Draw>Cylinder**.

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

2. Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system. Press **Tab** to navigate to the keyboard entry area on the Status Bar.

3 Type **83.1** for the radius in the **dX** box at the bottom of the screen, and press **Enter**.

Note Use the Tab key to navigate between value fields (from X to Y to Z and from dX to dY to dZ).

4 Type **25.4** for the height in the **dZ** box, and press **Enter**.
The **Properties** window appears.
5 Click the **Attribute** tab.
6 Change the **Name** to **Cylinder_tool**.
7 Click **OK** when you are done making any desired edits.
The **Properties** window closes.

Subtract the Cylinders

Subtract **Cylinder_tool** from **Outer_arm**.

To subtract the second cylinder from the first:

1 In the history tree, select the **Outer_arm** cylinder, press and hold down **Ctrl**, and select the **Cylinder_tool** cylinder.

Note The first object selected appears under Blank Parts, and the second object selected appears under Tool Parts. Tool parts are removed during a Boolean operation (unless cloned) and the final part takes on the name and other characteristics of the Blank Part.

2 Click **Modeler>Boolean>Subtract**.
The **Subtract** dialog box appears.
3 If necessary, move the **Outer_arm** object to the **Blank Parts** list and the **Cylinder_tool** object to the **Tool Parts** list. To move an object from one list to another, select it, and click the appropriate arrow.
4 Click **OK**.

Add the Poles to the Outer Armature

Add two magnetic poles to the outer armature. To do so, you need to create a box, move the box into the position for the poles, and use the **Mirror** command to create a duplicate of the box. Then unite the three model objects, and subtract a newly created cylinder to arrive at the final shape.

To create the box for the outer armature magnetic poles:

1. Click **Draw>Box**.
2. Type the box position **(-13.9, 0, 0)** in the **X**, **Y**, and **Z** fields at the bottom of the screen, and then press **Enter**.
3. Type the box size **(27.8, -40, 25.4)** in the **dX**, **dY**, **dZ** fields, and then press **Enter**.
   The **Properties** window appears.
4. Click **OK**.
   A box named **Box1** is drawn.

Move the Box into a Pole Position

To move the box into the correct position for one of the magnetic poles:

1. Select **Box1** from the history tree.
2. Click **Edit>Arrange>Move**.
3. Type **(0, 0, 0)** in the **(X, Y, Z)** fields as the origin of the move vector, and press **Enter**.
4. Type **(0, -45, 0)** in the **(dX, dY, dZ)** fields as the target point of the move vector, and press **Enter**.

Create a Duplicate of the Pole Box

To create a duplicate of the box using mirroring:

1. Select **Box1** from the history tree.
2. Click **Edit>Duplicate>Mirror**.
3. Type **(0, 0, 0)** in the **(X, Y, Z)** fields as coordinates for the anchor point on the mirror plane, and press **Enter**.
4. Type **(0, 1, 0)** in the **(dX, dY, dZ)** fields as coordinates of target point of the vector normal to the mirror plane, and press **Enter**.

A second box, named **Box1_1**, is drawn.

Unite the Outer Armature and Magnetic Pole Boxes

To unite the three objects in the model:

1. In the history tree window, select **Outer_arm**, hold down the **Ctrl** key, and then select **Box1** and **Box1_1**.
2. Select **Modeler>Boolean>Unite**.
   The first selected object was **Outer_arm**; therefore, the default name for the final object is **Outer_arm**.

Finalize the Outer Armature Magnetic Pole Faces

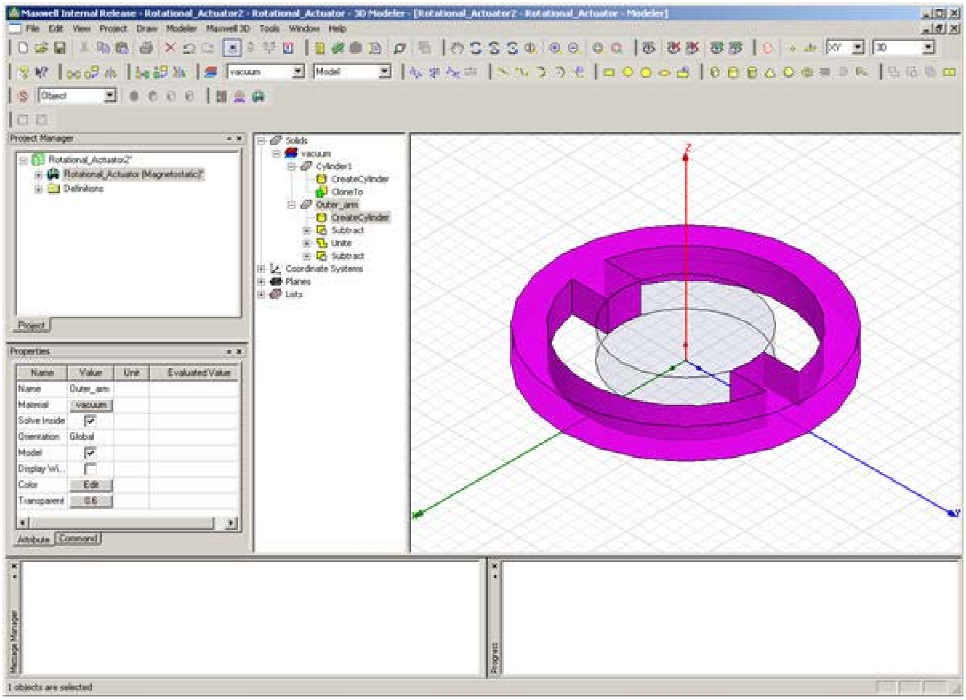
To provide the final shape for the magnetic pole faces:

1. Create a cylinder with the center at **(0, 0,0)**, a radius of **53.75**, and a height of **25.4**:
   a. Click **Draw>Cylinder**.
      The cursor changes to a small black box, indicating that you are in **Drawing** mode.
   b. Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system. **Tab** to the keyboard entry area.
   c. Type **53.75** for the radius in the **dX** box at the bottom of the screen, and press **Enter**.
   d. Type **25.4** for the height in the **dZ** box, and press **Enter**.
      The **Properties** window contains the properties of the new cylinder, **cylinder1**.
2. Since we will be using a similar cylinder in the next section, we will make a copy of **cylinder1** for later use.
   a. Select **cylinder1**.
   b. Click **Edit>Copy** to create a copy of the object on the clipboard.
   c. Next select **Edit>Paste** to paste a new copy, named **cylinder2** in the history tree, into the project at the same location as the original.
3. Subtract **cylinder1** from **Outer_arm** to achieve the curved edges on the poles:
   a. Select the **Outer_arm** object, press and hold the **Ctrl** key, and then select **cylinder1**.
   b. Select **Modeler>Boolean>Subtract**.
      The **Subtract** dialog box appears.

Note Pressing **F1** with any dialog on the screen will open the Context-Sensitive Help system to the appropriate page for that dialog.

c. Make sure the **Outer_arm** object is in the **Blank Parts** list and the **cylinder1** object is in the **Tool Parts** list.

d. Click **OK**.
The **Outer_arm** object should look as shown in the following graphic:



4 Click **File>Save** to save all of the operations up to this point.

Create the Inner Armature Object

The inner armature consists of two cylinders (for an outer and inner radius) that are subtracted to leave the armature. Magnetic poles are then added to the armature object.

Draw the Inner Armature Cylinders

To draw the inner armature:

1 Create a cylinder called **shaft** with the following properties.

Property	Value
Center	(0, 0, 0)
Radius(dX)	25.4mm
Height(dZ)	25.4mm

Step by Step instructions if needed:

a. Click **Draw>Cylinder**.

b. Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system. **Tab** to the keyboard entry area on the Status Bar.

c. Type **25.4** for the radius in the **dX** box at the bottom of the screen, and press **Enter**.

d. Type **25.4** for the height in the **dZ** box, and click **Enter**.
The **Properties** window appears.

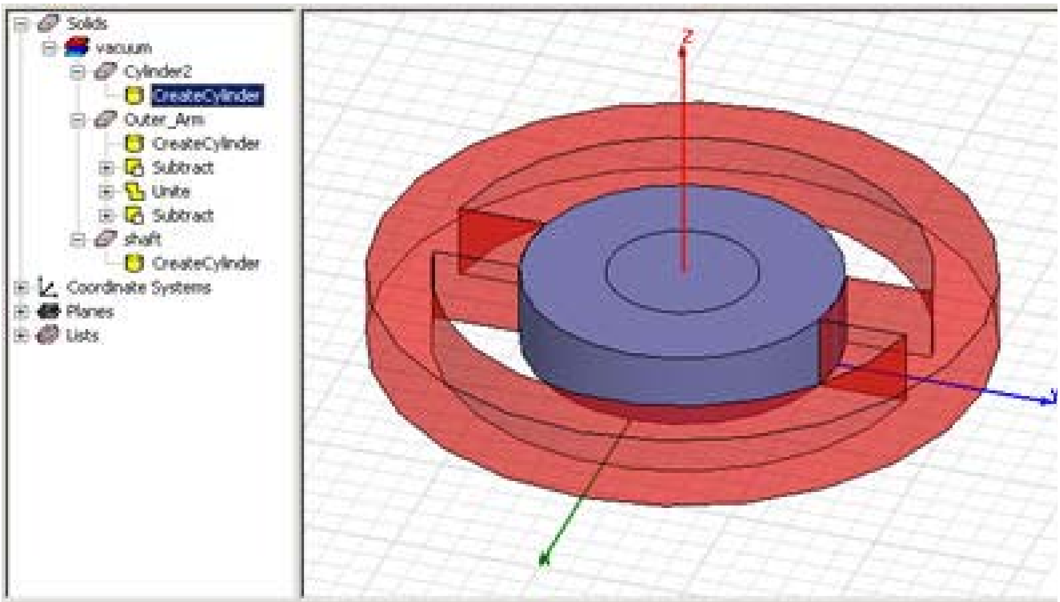
e. Click the **Attribute** tab.

f. Change the **Name** to **shaft**.

g. Click **OK**.
A cylinder named **shaft** is drawn.

2 The properties of **cylinder2** can be modified to create the outer radius of the inner armature to eliminate the need to draw another object.

a. Select the CreateCylinder command in the history tree under the **cylinder2** object.



b. In the **Properties Window**, select the **Command** tab.

c. Select the value field containing **53.75** corresponding to **Radius.** The field becomes highlighted and editable.

d. Change the value to **38.1** and press **Enter**. The radius of **cylinder2** is changed in the modeler window.

e. In the history tree, select **cylinder2**. The **Attribute** tab is now visible in the **Properties Window**.

f. Change the name of the object by selecting **cylinder2** in the value field corresponding to **Name** and entering **Inner_arm**. Press **Enter**.

g. Subtract **shaft** from **Inner_arm**:

h. Select **Inner_arm**, press and hold down the **Ctrl** key, and select **shaft**.

i. Select **Modeler>Boolean>Subtract** from the menu bar. The **Subtract** dialog box appears.

j. Make sure **Inner_arm** is in the **Blank Parts** list and **shaft** is in the **Tool Parts** list.

k. Click **OK**.

Add the Poles to the Inner Armature

To create two magnetic poles for the inner armature, you need to create another box, move the box into the correct position, and use the **Mirror** command to create a duplicate of the box. Unite the inner armature with the two boxes, and intersect it with a cylinder to arrive at the final shape.

To create a box for the inner armature magnetic poles:

1. Click **Draw>Box**.
2. Type the box position **(-12.7, 0, 0)** in the **X**, **Y**, and **Z** fields at the bottom of the screen, and then press **Enter**.
3. Type the box size **(25.4, -20, 25.4)** in the **dX**, **dY**, **dZ** fields, and then press **Enter**.
   The **Properties** window appears.
4. Click **OK**.
   A box named **Box2** is drawn.

Move the Box into Position

To move the box into position:

1. Select **Box2** from the history tree.
2. Select **Edit>Arrange>Move**.
3. Type **(0, 0, 0)** in the **(X, Y, Z)** fields as the origin of the move vector, and press **Enter**.
4. Type **(0, -35, 0)** in the **(dX, dY, dZ)** fields as the target point of the move vector, and press **Enter**.
   The **Properties** window appears.
5. Click **OK**.

Create a Duplicate of the Box

To create a duplicate of the box using mirroring:

1. Select **Box2** in the history tree.
2. Select **Edit>Duplicate>Mirror**.
3. Type **(0, 0, 0)** in the **(X, Y, Z)** fields as coordinates for the anchor point on the mirror plane, and press **Enter**.
4. Type **(0, 1, 0)** in the **(dX, dY, dZ)** fields as coordinates of target point of the vector normal to the mirror plane, and press **Enter**.

The **Properties** window appears.

5 Click **OK**.
A box named **Box2_1** is drawn.

Unite the Inner Armature and Magnetic Pole Boxes

To unite the inner armature object with the two boxes:

1 In the history tree, select **Inner_arm**, press and hold down the **Ctrl** key, and then select **Box2** and **Box2_1**.

2 Select **Modeler>Boolean>Unite**.
Because the first selected object was **Inner_arm**, the final object name is **Inner_arm**. The name of the objects can be changed in the **Properties** window on the **Attribute** tab.

Finalize the Inner Armature Magnetic Pole Faces

To provide the final shape for the magnetic pole faces:

1 Create a cylinder with the center at **(0, 0,0)**, a radius of **51.05**, and a height of **25.4**:

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

b. Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system.

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

d. Type **25.4** for the height in the **dZ** box, and press **Enter**.
The **Properties** window appears.

e. Click the **Attribute** tab.

f. Change the **Name** to **finalpole2**.

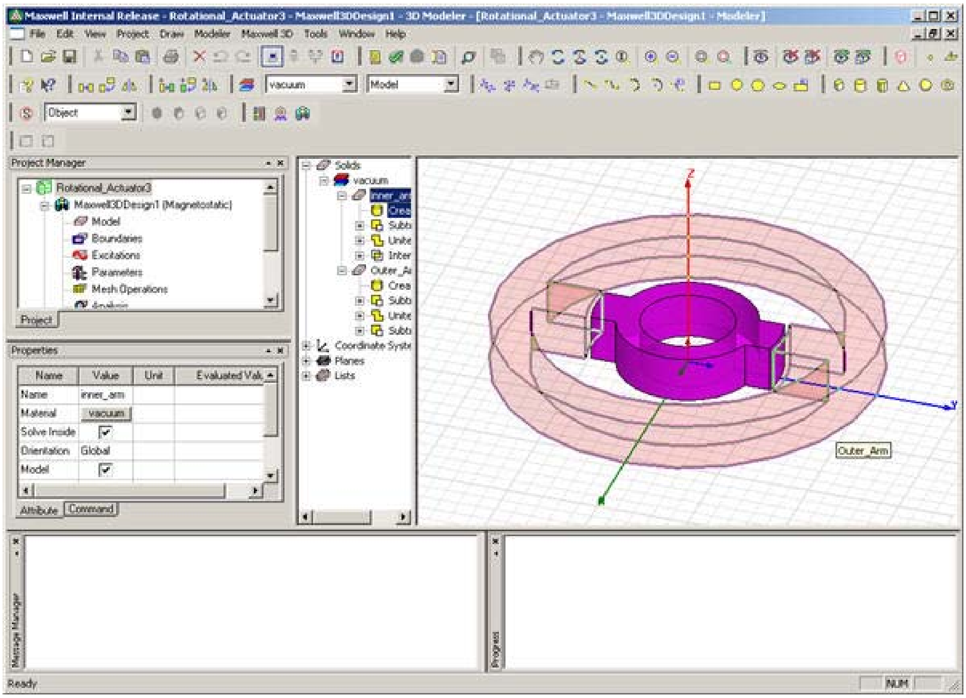
g. Click **OK**.
A cylinder called **finalpole2** is drawn.

2 Intersect **Inner_arm** and the new **finalpole2** object:

a. In the history tree, select the **Inner_arm** object, press and hold down the **Ctrl** key, and then select **finalpole2**.

b. Select **Modeler>Boolean>Intersect**.

The final armatures should look like the following:



3 Click **File>Save** to save all of the operations up to this point.

Create the Coils

First create a new coordinate system (RelativeCS1) so that in the new coordinate system the XY plane becomes a median plane of the model. Then create a cross-section of the coils and sweep it across a path that will create the 3D coils. Then intersect the coil shape with a cylinder and create a duplicate, to achieve the final coil shape.

Create the New Coordinate System

To create the new coordinate system:

1. Select **Modeler>Coordinate System>Create>Relative CS>Offset**.
2. Press the **Tab** key to switch to the keyboard entry area of the Status Bar. Type the new origin **(0, 0, 12.7)** in the **(X, Y, Z)** boxes, and then press **Enter**.
   The new coordinate system is created and named **RelativeCS1**.

Sweep a Cross-Section Across a New Path

The coil(s) are created by sweeping the coil cross-section along a path.

Set the Drawing Plane to ZX

To set the drawing plane:

- Select **ZX** from the **Drawing plane** pull-down list on the **3D Modeler Draw** toolbar.

Draw the Sweep Path

To create the path you want to use as the sweep path:

1. Click **Draw>Rectangle**.
2. **Tab** to the keyboard entry area and type **(-17, 0, -15.5)** in the **(X, Y, Z)** boxes, for the rectangle position, and then press **Enter**.
3. Type **(34, 0, 31)** in the **(dX, dY, dZ)** boxes, for the rectangle dimensions, and press **Enter**.
   The **Properties** window appears.
4. Click the **Attribute** tab.
5. Change the **Name** to **path**.

6 Click **OK**.
A rectangle named **path** is drawn.

7 Uncover the faces:

a. Click **Edit>Select>Faces**, and select **path** by clicking on it in the **Modeler** window.

b. Click **Modeler>Surface>Uncover Faces**.

Set the Drawing Plane to YZ

To set the drawing plane:

- Select **YZ** from the **Drawing plane** pull-down list on the **3D Modeler Draw** toolbar.

Draw the Cross-Section of the Coil

To draw the cross-section of the coil:

1 Click **Draw>Rectangle** to draw the cross-section of the coil.

2 Type **(0, 0, 15.5)** in the **(X, Y, Z)** boxes, for the rectangle position, and then press **Enter**.

3 Type **(0, 17, 24)** in the **(dX, dY, dZ)** boxes, for the rectangle dimensions, and press **Enter**.
The **Properties** window appears.

4 Click the **Attribute** tab.

5 Change the **Name** to **coil1**.

6 Click **OK**.
A rectangle named **coil1** is drawn.

Sweep the Cross-Section Along the Path

To sweep the cross-section (**coil1**) along the path (**path**) to create the coil:

1 In the history tree, select **path**, press and hold down the **Ctrl** key, and then select **coil1**.

2 Click **Draw>Sweep>Along Path**.
The **Sweep along path** dialog box appears.

3 Click **OK** to accept the defaults.
The **Properties** window appears.

4 Click **OK** to create the coil. The coil retains the name **coil1** from the cross section used to create it.

Set the Drawing Plane Back to ZX

To set the drawing plane:

- Select **ZX** from the **Drawing plane** pull-down list on the **3D Modeler Draw** toolbar.

Intersect the Coil Shape with a Cylinder

The final coil shape has rounded outside corners.

Using the new coordinate system, **RelativeCS1**, do the following to achieve the final shape:

1. Create a cylinder at the origin with a radius (**dZ**) of **43** mm and a height (**dY**) of **17** mm:
   a. Click **Draw>Cylinder**.
      The cursor changes to a small black box, indicating that you are in **Drawing** mode.
   b. Select the center of the cylinder by clicking at the **(0,0,0)** location, which is the origin for the coordinate system.
   c. Type **43** for the radius in the **dZ** box at the bottom of the screen, and press **Enter**.
   d. Type **17** for the height in the **dY** box, and press **Enter**.
      The **Properties** window appears.
   e. Click the **Attribute** tab.
   f. Change the **Name** to **round**.
   g. Click **OK**.
      A cylinder named **round** is drawn.
2. Intersect the coil and the new cylinder:
   a. In the history tree, select **coil1**, press and hold down the **Ctrl** key, and then select **round**.
   b. Click **Modeler>Boolean>Intersect**.
      The intersected object is named **coil1**.

Move the Coil into the Final Position

To move the coil into its final position.

1. Select **coil1** in the history tree window.
2. Select **Edit>Arrange>Move**.
3. Type **(0, 0, 0)** in the **(X, Y, Z)** fields, for the origin of the new location, and then press **Enter**.
4. Type **(0, 54.5, 0)** in the **(dX, dY, dZ)** fields as the target point of new dimensions, and press **Enter**.
   The **Properties** window appears.
5. Click **OK**.

Create a Mirror Duplicate of the Coil

To create a second coil by mirroring the first:

1 Select **coil1** in the history tree window.

2 Select **Edit>Duplicate>Mirror**.

3 Type **(0, 0, 0)** in the **(X, Y, Z)** fields as the coordinates of the anchor point on the mirror plane, and then press **Enter**.

4 Type **(0, 1, 0)** in the **(dX, dY, dZ)** fields as coordinates of the target point of the vector normal to the mirror plane, and press **Enter**.
The **Properties** window appears.

5 Click **OK**.
A second coil is created and named **coil1_1**.

Create the Coil Terminals

To create the terminals for the coils:

1 In the history tree, select **coil1**, press and hold down the **Ctrl** key, and select **coil1_1**.

2 Click **Modeler>Surface>Section**.

3 Select **XY** as the **Section Plane**.

4 Click **OK**.

5 Click **Modeler>Boolean>Separate Bodies**. This separates the interlinked sheet objects created when the intersection of the XY plane created two terminals in each coil. The resulting four objects are automatically named

- **Coil1_Section1**
- **Coil1_Section1_Separate1**
- **Coil1_1_Section1**
- **Coil1_1_Section1_Separate1**

6 Delete the two redundant terminals:

a. In the history tree window, select **Coil1_Section1_Separate1**, press and hold down the **Ctrl** key, and select **Coil1_1_Section1_Separate1**.

b. Press **Delete**.

7 Double-click on **Coil1_Section1** in the **Sheets** section of the history tree.

The **Properties** window is displayed.

8 Change the name of **Coil1_Section1** to **Section1** and click **OK**.

9 Repeat steps 8 and 9 to change the name of **Coil1_1_Section1** to **Section2**.

Create the Background (Region)

Define a background region box with the origin at (-250, -250, -250) and the dimensions of (500, 500, 500).

To create the background region box:

1 Click **Draw>Box**.

2 Type the box position **(-250, -250, -250)** in the **X**, **Y**, and **Z** fields at the bottom of the screen, and then press **Enter**.

3 Type the box size **(500, 500, 500)** in the **dX**, **dY**, **dZ** fields, and then press **Enter**.
The **Properties** window appears.

4 Click the **Attribute** tab.

5 Change the **Name** (currently **Box3**) to **bgnd**.

6 Set the transparency to **0.9**:

a. Click the button for the **Transparent** property.
The **Set Transparency** dialog box appears.

b. Type **0.9** in the text box, and click **OK** to return to the **Properties** window.

7 Click **OK**.
A box named **bgnd** is drawn.

Note Alternatively, the **Draw>Region** command may be used to create the background object.

8 Press **Ctrl-D** to fit the drawing in the window.

Finalize the Geometry

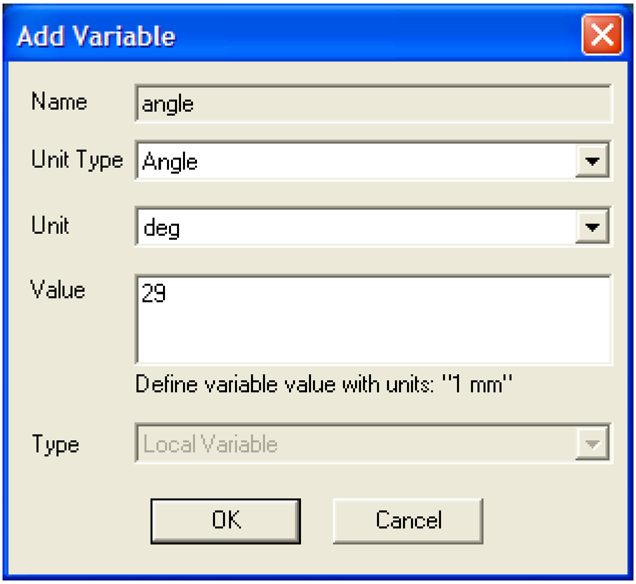
To finalize the geometry:

1 Select the **Inner_arm** object in the history tree.

2 Click **Edit>Arrange>Rotate**.
The **Rotate** dialog box appears.

3 Select the **Z** radio button for the **Axis**.

4 Thinking ahead, we will want to evaluate the device over a range of armature angles. Therefore, enter "**angle**" into the value field.
The **Add Variable** dialog appears as shown to specify the value for the variable.

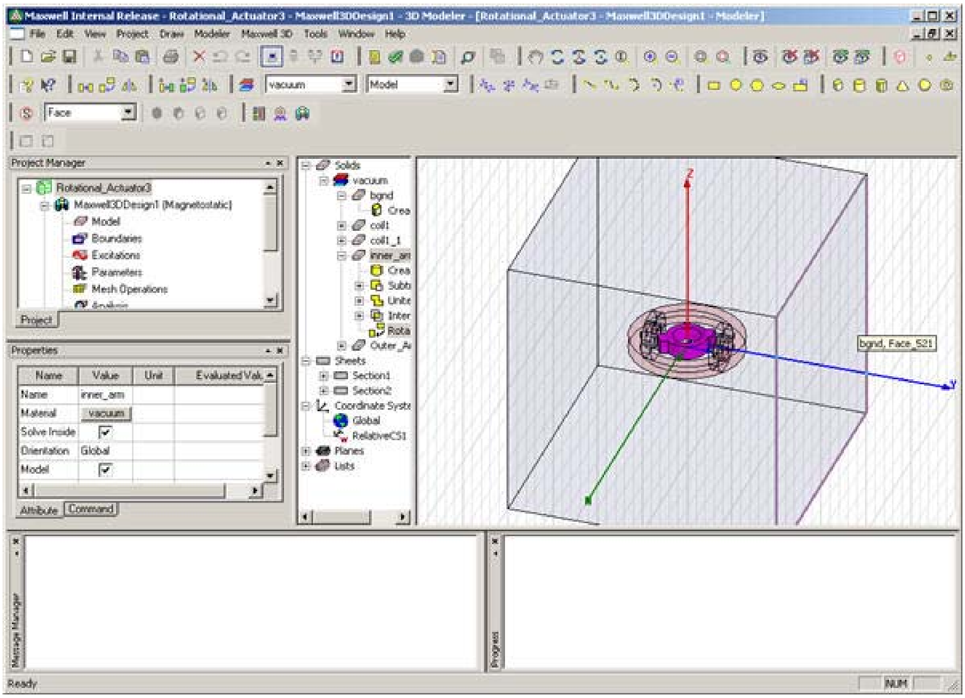


5 Ensure that the Unit Type is set to Angle, and the Unit is set to deg. Type **29** in the **Value** text box.

6 Click **OK**.

Note Most numeric entry fields allow entry of a variable name for use in parametric or optimization.

The final geometry should look similar to the following:



7 Click **File>Save** to save the final version of the model before moving on to defining materials.

4

Defining Material Properties

In this chapter you will complete the following tasks:

- ✓ Define material properties.
- ✓ Assign excitations.
- ✓ Set up torque and inductance parameter calculations.
- ✓ Set up a matrix.

Define Material Properties

Default material properties are automatically assigned when you create the geometry objects. You can view these properties by viewing the **Attribute** tab of the **Properties** window. The default material for all objects is **vacuum**, which can be changed as soon as you draw an object. For this example, you will set the material definitions for all objects at the same time after the entire geometry has been created.

Define the Nonlinear Material for the Two Armatures

To define the nonlinear material for the armatures:

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

   The **Properties** window appears.
2. In the **Material** row, click the value in the **Value** column.

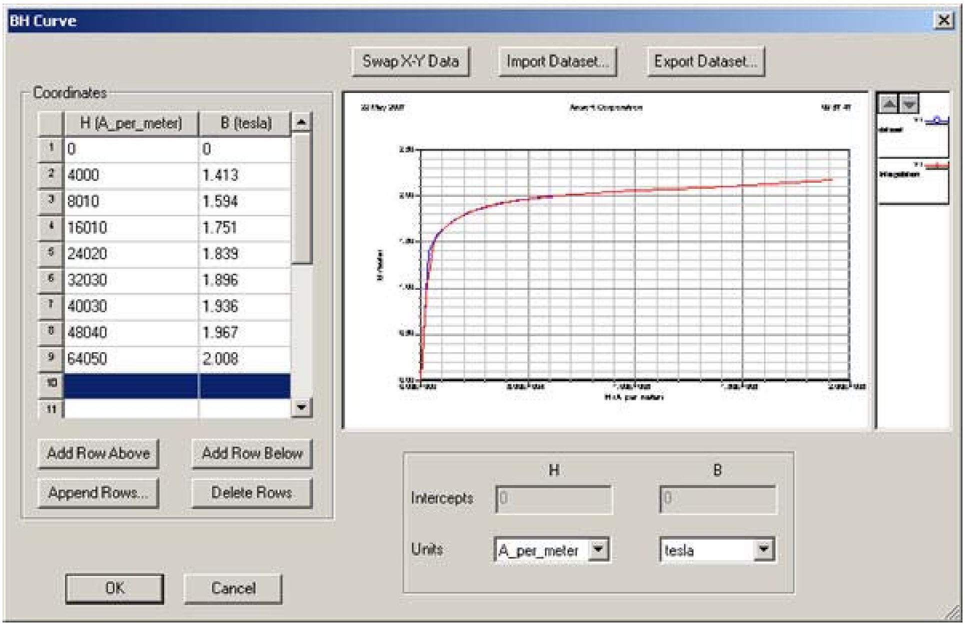
   A drop-down list appears.
3. Select the **Edit...** item in the list.

   The **Select Definition** dialog box appears.
4. On the **Materials** tab, click the **Add Material** button.

   The **View / Edit Material** dialog box appears.
5. Type **arm_steel** in the **Material Name** box.
6. Do the following in the **Properties of the Material** section:

   a. In the **Type** column of the **Relative Permeability** row, select **Nonlinear** from the pull-down list.

   b. In the **Value** column of the **Relative Permeability** row, click the **BH Curve** button.
   The **BH Curve** dialog box appears. By default, 10 rows are available to enter data points, but this example requires 20.



c. In the **Coordinates** section, append 10 additional rows to the table to reach a total of 20 data rows:

- Click the **Append Rows** button.
- Type **10** in the **Number of rows** text box, and click **OK**. The **BH Curve** dialog box reappears, with 20 rows now available in the table.

d. Type the following **H** and **B** data in the **Coordinates** section:

Note When entering data into the Coordinates section, the Tab key can be used to sequentially move to the next entry position.

	H (A/m)	B (T)
1.	0	0
2.	4000	1.413
3.	8010	1.594
4.	16010	1.751
5.	24020	1.839

6.	32030	1.896
7.	40030	1.936
8.	48040	1.967
9.	64050	2.008
10.	80070	2.042
11.	96080	2.073
12.	112100	2.101
13.	128110	2.127
14.	144120	2.151
15.	176150	2.197
16.	208180	2.24
17.	272230	2.325
18.	304260	2.37
19.	336290	2.42
20.	396000	2.5

e. Click **OK**.
The **BH Curve** dialog box closes, and the **View / Edit Material** dialog box reappears.

f. In the **Value** column of the **Bulk Conductivity** row, enter **2e6**.

g. Click **Validate Material**.
A green check mark appears if the material is valid.

7 Click **OK** to close the **View / Edit Material** dialog box.
The **Select Definition** dialog box reappears.

8 Click **OK** to close the **Select Definition** dialog box.
The **Properties** window reappears.

9 Click **OK** to close the **Properties** window.

Assign Material Properties

Select the **Inner_arm** object, and assign the newly defined **arm_steel** property. Select the two coils, and assign **copper** as the material property:

To assign the material properties to the model objects:

1 Double-click the **Inner_arm** object in the history tree.
The **Properties** window appears.

2 In the **Material** row, click the field in the **Value** column.
A drop-down list of available materials appears.

3 Select **arm_steel** from the list of materials.

4 Click **OK** to close the **Properties** window.

5 Repeat steps 1 through 4 to assign **copper** to **coil1**.

6 Repeat steps 1 through 4 to assign **copper** to **coil1_1**.

7 Leave the material assignment for the **background** object unchanged.

Assign Excitations

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

To define the currents:

1 Select **Section1** and **Section2** in the history tree.

2 Click **Maxwell3D>Excitations>Assign>Current** from the menu.
The **Current Excitation** dialog box appears.

Note Excitations may also be assigned using the shortcut menu. Right-click on **Excitations** in the Project tree. In the Shortcut menu, select **Assign>Current**.

3 Type **675.5** in the **Value** text box, and select **A** as the units.

4 Select **Stranded** as the **Type**.

5 Click **OK**.

By default, all faces of the region box (**background**) are assigned with magnetic flux tangent boundary conditions. Therefore, no additional boundary conditions are required for this example problem.

Set Up Parameter Calculations

In this example, you will calculate the torque and inductance matrix parameters.

Set Up the 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. Leave the **Type** set to **Virtual**.
5. Select **Global:Z** from the **Axis** pull-down list.
6. Select the **Positive** radio button for the axis orientation.
7. Click **OK**.

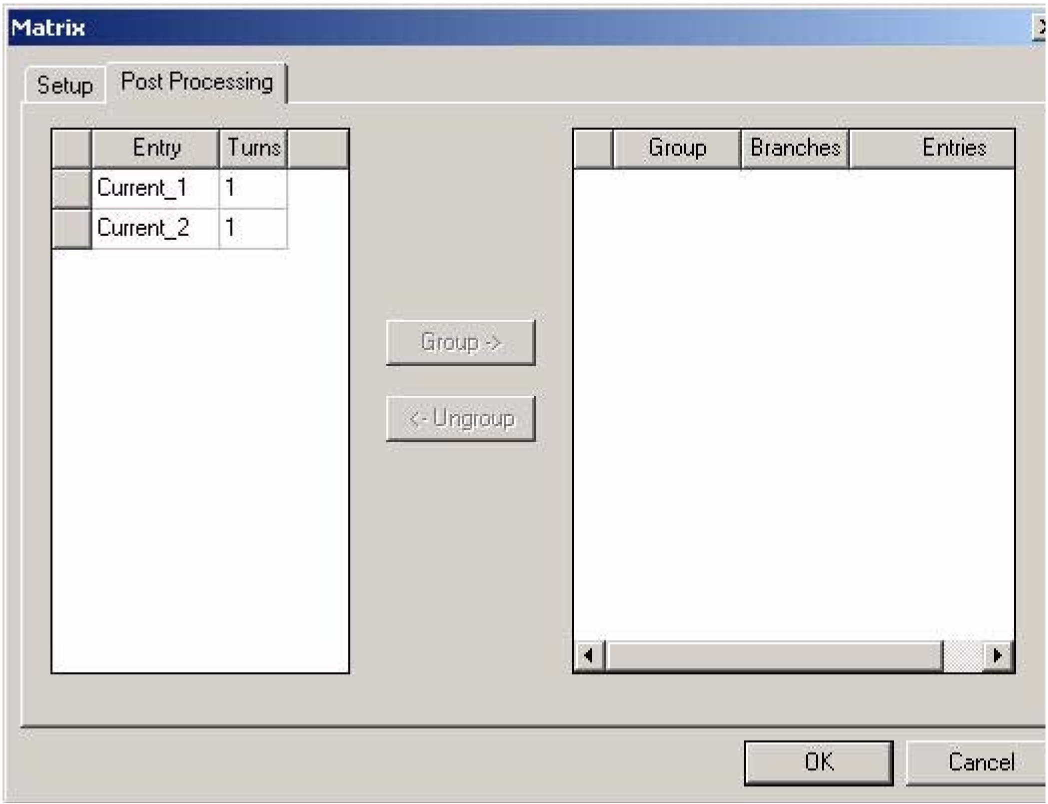
Set Up the Inductance Matrix Calculation

To set up the inductance matrix calculation:

1. Right-click the **Parameters** field in the project tree.
   A shortcut menu appears.
2. Select **Assign>Matrix** from the shortcut menu.
   The **Matrix** dialog box appears.
3. Click the **Setup** tab.
4. Select **Include** for both **coil1** and **coil1_1** for the inductance calculations to be performed. The self inductances and mutual inductances will be calculated.
5. Click **OK**.

Exploring the Matrix Setup Grouping Functionality

The matrix setup provides a grouping functionality on the **Post Processing** tab:



Thus, in addition to inductance matrix entries calculation, Maxwell can perform a grouping calculation.

To perform a grouping calculation:

1. Select the matrix entry and specify the corresponding number of turns.
2. Select all matrix entries to be involved in grouping and click the Group button.
3. Specify the number of branches.

The operations performed by the grouping function can be one of the three cases:

- Series connection if the number of branches is set to 1.
- Parallel connection if number of branches is equal with the total number of coils (matrix entries).
- Series/parallel if the number of branches is different from the two above.

Example 1: Series/Parallel

Assume a situation with 4 coils, each with 18 turns, all 4 selected to be grouped with the number of branches set to 2. In this case, 9 turns from each of the 4 coils (18 / 2 branches = 9) are connected in series and paralleled, with the other 9 turns of the same coils also connected in series.

Example 2: Series/Parallel

Assume a situation with 5 coils, each with 15 turns and number of branches set to 3. In this case, the equivalent S/P corresponds to taking the first 5 turns from each of the 5 coil and connecting them in series, taking the next 5 turns from the same coils and connecting them in series, taking the final 5 turns from the coils and connecting them in series, and then finally connect the emerging 3 subgroups in parallel.

Note For the grouping in the S/P case to correspond to a physical situation, the number of turns must be an integer multiple of the number of branches.

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 the **Analysis** field 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. Accept the default values (**Maximum number of passes** = **10** and **Percent Erro**r = **1**).
   These settings instruct the solver to solve up to 10 passes as the automatic adaptive mesh refinement refines the mesh and improves the accuracy of the solution at run time.
5. Click **OK**.

Run the Analysis

To run the analysis:

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

   A shortcut menu appears.
2. Select **Analyze All**.
3. To visualize the progress of the solution:
   a. Right-click the **Setup1** field (located under the **Analysis** field), and select **Convergence**.
   b. Make sure the **Convergence** tab is selected.
   c. Select a tabular or graphical format for how to visualize the information about the energy, number of finite elements, torque, etc.
   d. Click **OK**.
4. When the solution is complete (it should take between 3 and 5 minutes on most PCs), look at the value of the torque from **LastAdaptive**.

   The value should be about -0.173 N m.

6 Post Processing the Results

In this chapter you will complete the following task:

- Plot the magnetic flux density vector.
- Plot the magnetic flux density magnitude.

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 (RelativeCS1) 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**.
   ObjectList1 is created under the **Lists** section of the History Tree.

Plot the Quantity

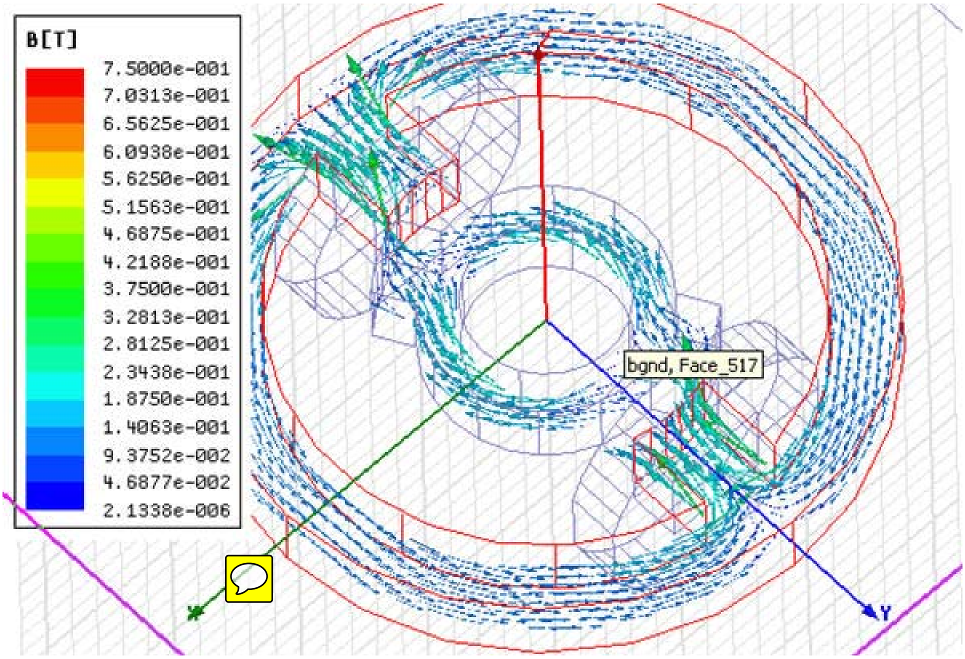
To create the plot:

1. Change the rendering of objects in the model to wireframe by clicking **View>Render>Wire Frame**.
2. In the history tree, select the **RelativeCS1:XY** plane under **Planes**. This will be the plane to create plots on.
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. Click **Maxwell3D>Fields>Modify Plot Attributes**.
   The **Select Plot Folder** dialog box appears.
8. Select **B** from the list, and click **OK**.
   A dialog box appears where you can modify the plot attributes.
9. Click the **Scale** tab.
10. Select the **Use Limits** radio button.
11. Type **2.1338E-006** in the **Min** box, and type **7.5E-001** in the **Max** box.
12. Click the **Plots** tab.

13 Under the **Vector Plot** parameters slide the **Spacing** rule all the way to the left and enter 4 in the **min** box.

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

The plot should look similar to the figure below:



Plot the Magnetic Flux Density Magnitude

To avoid overlapping with the previous plot, right-click the name of the previous plot**(B_Vector)** under **Field Overlays**, and un-check the **Plot Visibility** box.

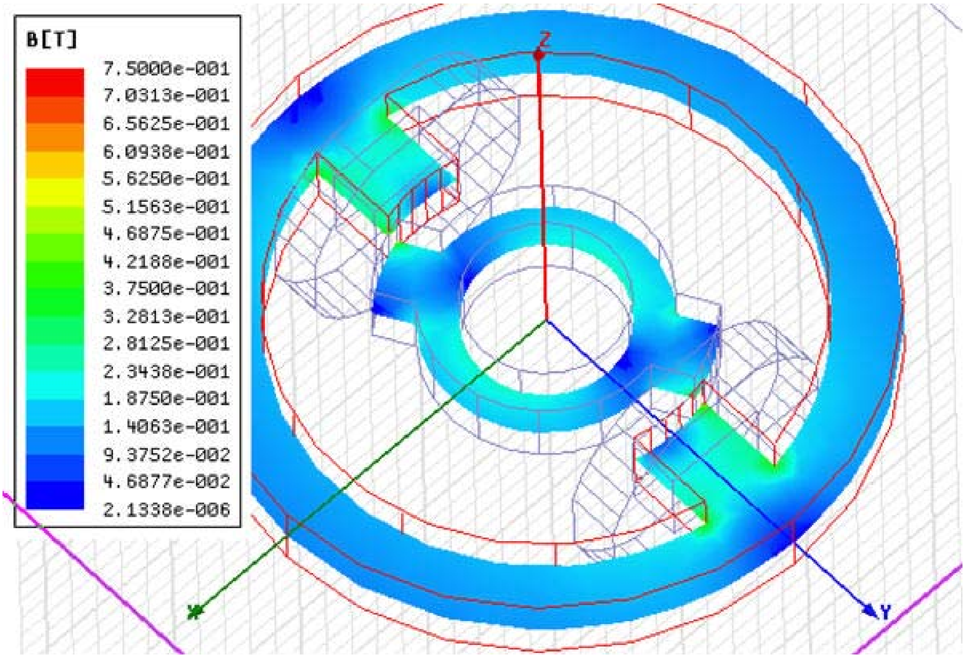
Follow the procedure used to create the previous plot to plot the magnetic flux density magnitude(**B_Mag**) on the same XY plane of the RelativeCS1 coordinate system.

1 Once the plot is displayed, right-click in the color key and select **Modify**.

2 In the **Scale** tab, set the **Number of divisions** to **100** to get a smooth plot.

3 Click **Close**.

The plot should look similar to the figure below:



7 Running a Parametric Analysis

In this chapter you will complete the following tasks:

- Run a parametric analysis.
- Create an analysis report.
- Close the project and exit Maxwell.

Parameterization and Parametric Analysis

*View the design variations that will be solved in table format under the **Table** tab. This enables you to visualize the design variations that will be solved and manually adjust sweep points if necessary.*

Parameters can be easily added to the setup for the purpose of changing assigned values (making "what if" type of analysis easy to perform) or for the purpose of setting up and running parametric analysis. The quantities that can be parameterized include geometric attributes, material properties, excitations, etc.

In this application we define the rotation angle of the **Inner_arm** object as a parameter and then perform a parametric analysis aimed at obtaining the self and mutual inductance of the two coils as well as the torque for a whole range of rotation angles. The rotation angle was specified as a variable during the creation of the geometric model.

To define the angle variable as a design parameter:

1. Right-click the **Optimetrics** row in the project tree.

   A shortcut menu appears.

*You can also use the **Design Properties** command to add new variables.*

2. Select **Add>Parametric** from the shortcut menu.
3. In the **Setup Sweep Analysis** dialog, click **Add** to add a variable to the sweep.
4. Select the variable **angle** from the drop-down list.
5. Select **Linear Step** as the type of sweep.
6. Enter **0 deg** for the **start** value, **30 deg** for the **stop** value, ans **5 deg** for the **step** value.
7. Click the **Add** button, and then **OK**.
8. Select the **Calculations** tab.
9. Click the **Setup Calculations** button.
10. In the **Add/Edit Calculation** window,select **Magnetostatic** for **Report Type**, **Setup1:LastAdaptive** for **Solution**, and **None** for **Parameter**.
11. Click on the **Trace** tab.
12. Under **Category** select the following variables:
    a. Select **Torque>Torque1.Torque** and click **Add Calculation**.
    b. Select **L>Matrix1.L(Current_1, Current_1)** and click **Add Calculation**.
    c. Select **L>Matrix1.L(Current_1, Current_2)** and click **Add Calculation**.

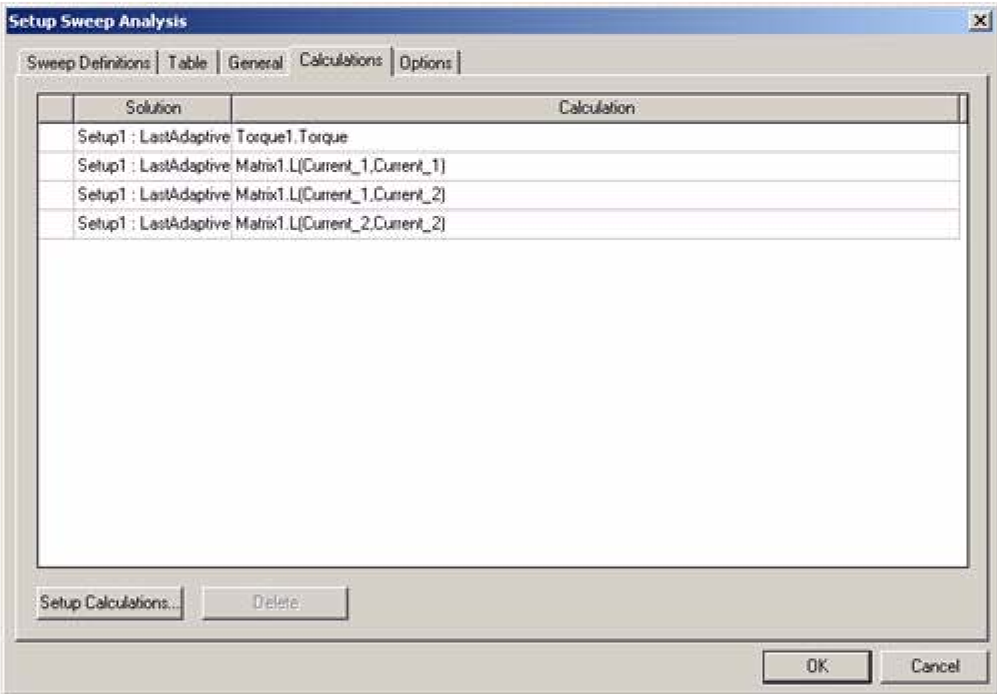
d. Select **L>Matrix1.L(Current_2, Current_2)** and click **Add Calculation**.

13 Click **Done**.

14 Back in the **Setup Sweep Analysis** window, select the **Options** tab.

15 Select the **Save Fields And Mesh** check box.

16 Click **OK** when complete.



A **ParametricSetup***n* entry is now shown under Optimetrics in the project tree.

To start the parametric analysis:

1 Right-click the **ParametricSetup1** row in the project tree.

A shortcut menu appears.

2 Select **Analyze**.

Note Because you are solving multiple geometric problems, the solution time required will be proportionately longer than solving the non-parametric solution in the previous chapter.

Create a Parametric Analysis Report

Once the parametric analysis in complete, create analysis report:

To create a report:

1 Right-click the **Results** row in the project tree.

A shortcut menu appears.

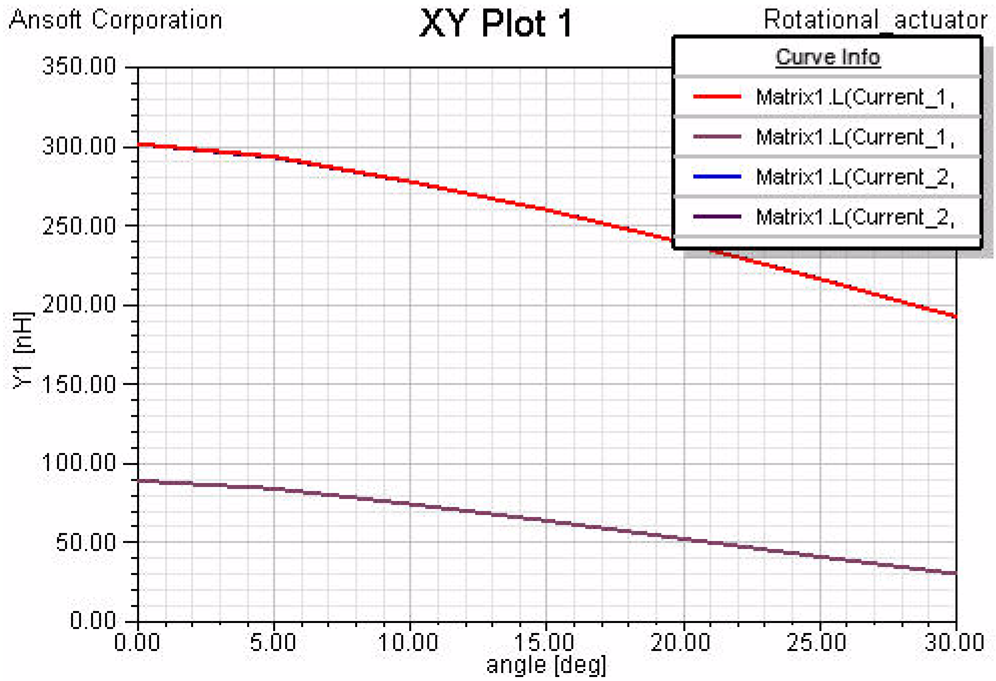
2 Select **Create Magnetostatic Report>Rectangular Plot**.

3 In the **New Report** window set the **Solution** to **Setup1:LastAdaptive** and **Parameter** to **None**.

4 Under **Category** select **L** and hold down the **ctrl** key and select **Matrix1.L(Current_1,Current_1)**, **Matrix1.L(Current_1,Current_2)**, and **Matrix1.L(Current_2,Current_2)**, and click **New Report**.

5 Click **Close** when finished.

The plot should look similar to the figure below.



The L11 and L22 traces almost overlap. This is normal since the respective inductances should be identical.

Create an Animation Using Saved Parametric Field Data

You can perform an animation using the saved data.

To perform an animation:

1. In the project tree, under **Field Overlays**, select the desired field plot that is to be animated (such as the **B vector** field plot created in chapter 6).
2. Right-click the plot, and select **Animate**.
   The **Setup Animation** dialog box appears.
3. Click the **Swept Variable** tab.
4. Select the desired variable from the **Swept variable** pull-down menu (**angle** in this case).
5. Select the desired variations from the **Select values** section.
6. Click **OK** to start the animation process.
   The **Animation** play panel appears, allowing you to pause and otherwise control the animation.
7. Click **Close** to stop the animation display.

Note You can export the animation as an animated **.gif** or **.avi** file by clicking **Export** in the **Animation** play panel.

Note Maxwell can take advantage of computing resources on various computers that can be accessed on a local network. Using the **Distributed Processing** option (licensed separately), you can solve parametric designs in parallel (simultaneously) on multiple user-selected computers available on the local network. Please contact your Ansoft account manager for details.

Close the Project and Exit Maxwell

Congratulations! You have successfully completed *Getting Started with Maxwell: Designing a Rotational Actuator*! 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

A

B

C

X