Maxwell® 3D
Field Simulator

Getting Started:

A 3D Magnetic Force Problem

February 2002

Notice

The information contained in this document is subject to change without notice.

Ansoft 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. Ansoft 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.

Ansoft Corporation
Four Station Square
Suite 200
Pittsburgh, PA 15219
(412) 261 - 3200

UNIX® is a registered trademark of UNIX Systems Laboratories, Inc.

Windows™ is a trademark of Microsoft® Corporation.

Printing History

New editions of this manual include material updated since the previous edition. The manual printing date, indicating the manual's current edition, changes when a new edition is printed. Minor corrections and updates 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 which are rearranged because of changes on a previous page are not considered to be revised.

Edition	Date	Software Revision
1	February 1991	1.0
2	April 1992	1.2
3	December 1993	3.0
4	October 1996	4.1
5	September 1999	5.0
6	December 2000	6.0
7	February 2002	9.0

Typeface Conventions

Computer	Computer type is used for on-screen prompts and messages, for field names, and for keyboard entries that must be typed in their entirety exactly as shown. For example, the instruction "**copy file1**" means to type the word **copy**, to type a space, and then to type **file1**.
Menu/Command	Computer type is also used to display the commands that are needed to perform a specific task. Menu levels are separated by forward slashes (/). For example, the instruction "**Choose File/Open**" means to choose the **Open** command under the **File** menu.
Italics	Italic type is used for emphasis and for the titles of manuals and other publications. Italic type is also used for keyboard entries when a name or a variable must be typed in place of the words in italics. For example, the instruction "**copy** *filename*" means to type the word **copy**, to type a space, and then to type the name of a file, such as **file1**.
Keys	Helvetica type is used for labeled keys on the computer keyboard. For example, the instruction "Press **Return**" means to press the key on the computer that is labeled **Return**.

Installation

Before you use Maxwell 3D, you must:

1. Set up your system's graphical windowing system.
2. Install the Maxwell software, using the directions in the Ansoft *PC* or *UNIX Installation Guide.*

If you have not yet done these steps, refer to the Ansoft *Installation* guides and the documentation that came with your computer system, or ask your system administrator for help.

Using a Graphical User Interface

If you are familiar with the concepts of using a mouse, menus, and other graphical user interface (GUI) tools, skip to Chapter 1, "Introduction."

If you have not used GUI systems before, this section will help you understand some of the terminology used in this guide. Since GUIs are basically visual, the best way to learn to use them is by practicing on your system.

Most GUI systems use a mouse as a pointing device, with which you can select areas on the screen for command execution and moving from one program to another. Your mouse may have 2 or 3 buttons; Maxwell 3D ignores the middle button on 3-button models, since Ansoft products do not use this button. You can program mouse buttons to work in non-standard ways, as you might want to if you are left-handed. For simplicity, the left-hand button (under your forefinger if you are right-handed) is called the left button, and the one on far right is the right mouse button. You will probably find the terms intuitive once you use these buttons a few times.

Point and Click; Right Click

To choose an item with the mouse, first move it on your desk until the *arrow* cursor is on that item; you are now “pointing” at the item. Next, press and release the left button; this is called “clicking.” Point-and-click is the most common action you will make with your mouse. Generally, “click” refers to a *left* mouse button click.

You can sometimes use your *right* mouse button to access or enter commands. In the 3D Modeler for instance, a right mouse button click causes a short menu of commands to appear at the mouse cursor. Generally, “right click” refers to a right mouse button click.

Double-Click

Occasionally you may want to select all of the text in a box, or perform a special task (such as indicating the end of drawing a line) while you are using Maxwell 3D. You can do this efficiently by quickly clicking twice with your left mouse button — a “double-click”.

Dragging Objects; Click and Hold

When you are drawing in the 3D Modeler, you can often use your mouse to enter objects and move around the screen. Frequently, you will click the mouse button and *hold it down* until the next part of the command is reached (the object is moved, the next point is entered, and so forth). If you click and hold on the edge of a window, you can position, or *drag*, the window on your screen. You can often drag objects in Maxwell; experiment to see what will move.

Tool Bars

Tool bars are shortcut methods for entering commands. There is a tool bar in the 3D Modeler and most of the setup modules for several commands. To use a tool bar, click the mouse cursor on the button you want to use. Here is an example of a tool bar:

Menus

Within each screen of Maxwell 3D are areas which list subsets, or menus, of commands. You can access a menu by clicking your mouse on the word or button that indicates the menu. The menu is pulled down, listing the commands available on that menu. (For example, the **New** command is available on the **File** menu.) Usually, the menu will remain displayed until you choose a command, or click on the desktop to exit. If the menu does not remain displayed, click and *hold* the mouse button, then release the button to make your choice.

An arrow on the right side of a command indicates that there is a submenu for that command. An ellipsis (. . .) indicates that a pop-up window appears after choosing this command.

When you are asked to use a menu command, each level is separated by a "/". Thus, to zoom in on a drawing, you would choose the **View/ Zoom In** menu command. To open a new file, you would choose **File/ New**.

There are also pop-up menus, which appear when you right-click on a Maxwell modeler window. Choose commands from these menus in the same way as from menu bars.

For more information on using GUIs, refer to "User Interface" in the Maxwell Control Panel's online help system.

Other References

For detailed information on Maxwell 3D commands, refer to the online documentation for the Maxwell 3D Field Simulator.

To start Maxwell 3D, you must first access the Maxwell Control Panel. For more detailed information on the Maxwell Control Panel commands, refer to the Maxwell Control Panel's online help system.

Table of Contents

1

Introduction

The Maxwell 3D Field Simulator is an interactive software package that uses finite element analysis to solve three-dimensional electrostatic, magnetostatic, and eddy current problems.

Maxwell 3D allows you to compute:

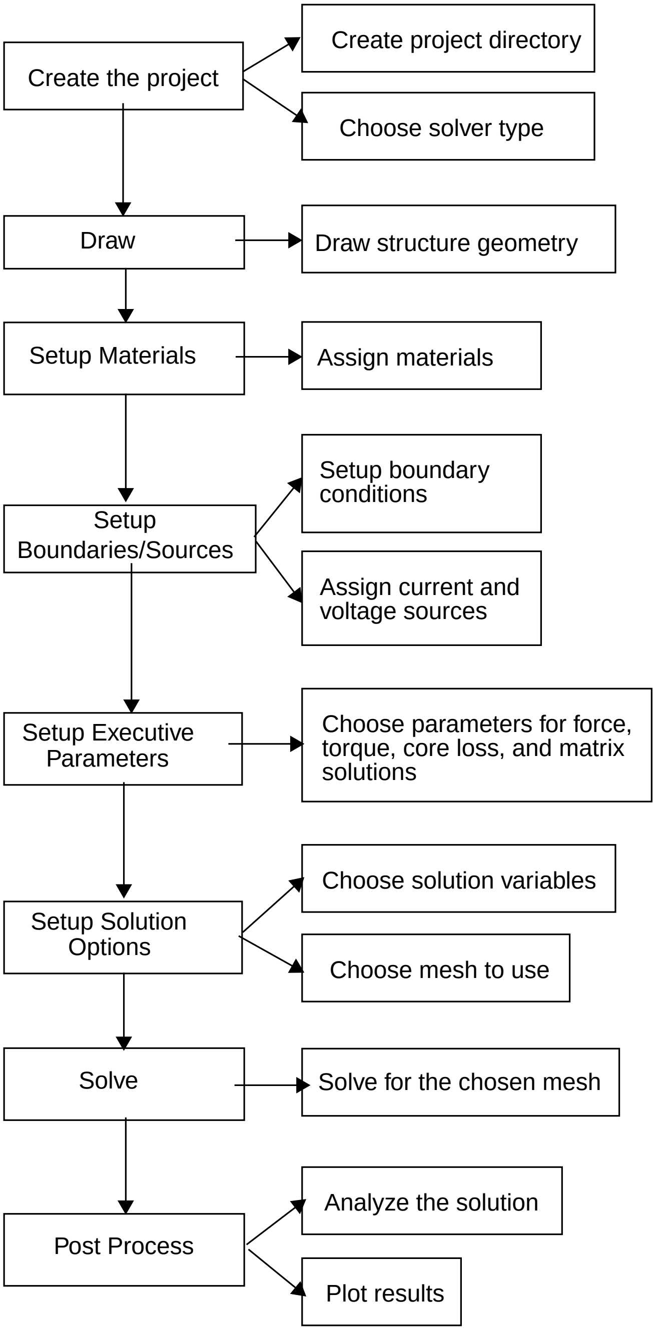
- Static electric fields, forces, torques, and capacitances caused by voltage distributions and charges.
- Static magnetic fields, forces, torques, and inductances caused by DC currents, static external magnetic fields, and permanent magnets. Fields can be simulated in structures that contain linear and nonlinear materials.
- Time-varying magnetic fields, forces, torques, and impedances caused by AC currents and oscillating external magnetic fields.
- Thermal quantities, such as temperature and heat flow.

In this guide you will use Maxwell 3D to solve a problem that introduces you to the basic functions of the software. If you have not used finite element analysis before, do not worry. Most of the details associated with the finite element method are transparent, allowing you to concentrate on design analysis.

General Procedure

The general procedure for creating and analyzing a project is summarized in the following list and flowchart:

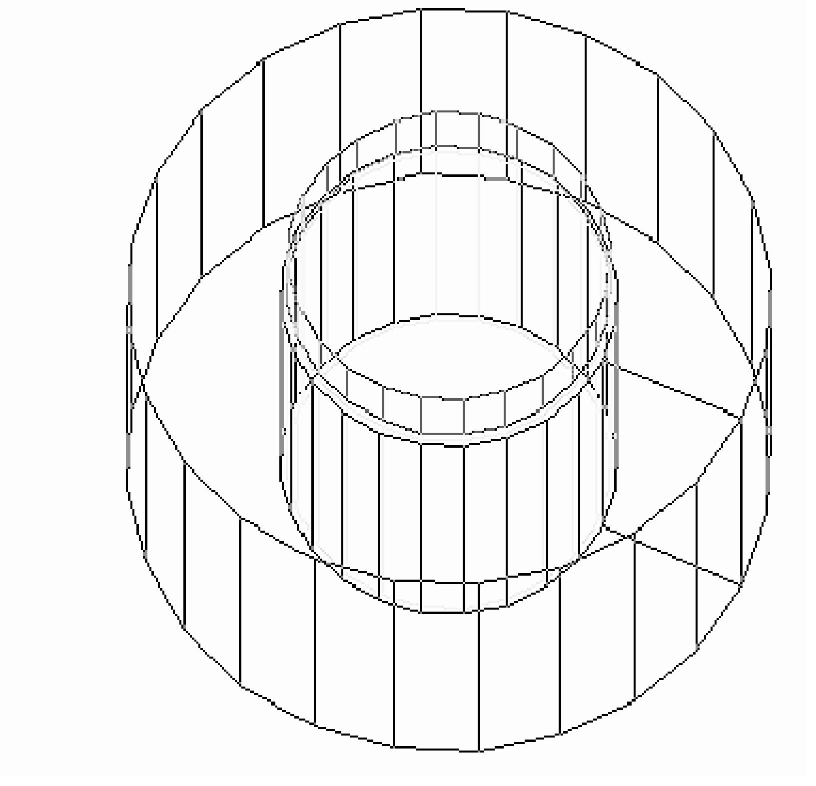
1. Create a project for Maxwell 3D.
2. Draw the model — in this case a simple magnet assembly.
3. Set up the problem:
   a. Assign materials to objects.
   b. Set up boundary conditions.
   c. Assign voltage and current sources.
4. Define problem parameters: forces, torques, or related values.
5. Setup the solution:
   a. Choose meshing variables and the mesh for which you want to solve.
   b. Specify how accurately (or how quickly) you want your problem to be solved.
6. Solve the problem. You can view solution information during and after the solution process.
7. Use the Post Processor to analyze and plot the solution results.


Create the project
Create project directory
Choose solver type
Draw
Draw structure geometry
Setup Materials
Assign materials
Setup Boundaries/Sources
Setup boundary conditions
Assign current and voltage sources
Setup Executive Parameters
Choose parameters for force, torque, core loss, and matrix solutions
Setup Solution Options
Choose solution variables
Choose mesh to use
Solve
Solve for the chosen mesh
Post Process
Analyze the solution
Plot results

The Sample Problem

The model is a permanent magnet placed next to an electromagnet. The permanent magnet is composed of a simple cylinder which rests just above the electromagnet.

The electromagnet is composed of a copper coil and steel core. A terminal resides within the electromagnet. You will send a current of 8000 amperes through the terminal and observe the virtual forces on each magnet.



Meshes

Maxwell 3D divides the problem space into many tetrahedra-shaped regions. Dividing a structure into thousands of smaller regions (elements) allows the software to compute a field solution separately in each element. The smaller the elements, the more accurate the final solution. (Of course, the more elements used, the longer the solution takes.)

To analyze the problem, Maxwell 3D generates a finite element mesh for each model using the geometry and material attributes. The field in each element is represented with a separate polynomial. If the field solution for an element in the mesh is not within the percentage of error that you specified, the software divides that element into smaller elements and generates a field solution for the smaller elements. It repeatedly adapts the mesh until the solution is at the desired accuracy.

Time: The estimated total time needed to complete this manual is approximately three hours.

2

Create a Project

The typical problem solved in this guide will help familiarize you with various aspects of Maxwell 3D.

Warning:

Unless you have a specific reason for doing so, avoid running Maxwell 3D when you are logged in as **root**. In general, running any software when you are logged in as **root** on a UNIX system could destroy your system.

The goals for this chapter are to:

- Access the Maxwell Control Panel.
- Start the Project Manager.
- Create a project directory.
- Create a new project.

Time:

The total time needed to complete this chapter is approximately 5 minutes.

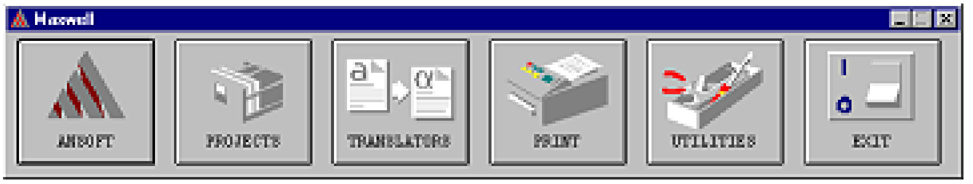
Access the Maxwell Control Panel

The Maxwell Control Panel allows you to create and open projects. You can also directly access those program modules shared by all Ansoft products. You must start the Control Panel in order to start Maxwell 3D.

➤ To start the Maxwell Control Panel:

- UNIX: Enter **maxwell &** in a console window. The Maxwell Control Panel appears on the screen.
- PC: Use the **Start** menu, or double-click on the Maxwell icon to bring up the Control Panel.

The Maxwell Control Panel appears.



Use the Control Panel to access any Maxwell products you have purchased, including those using the Maxwell 3D Field Simulator.

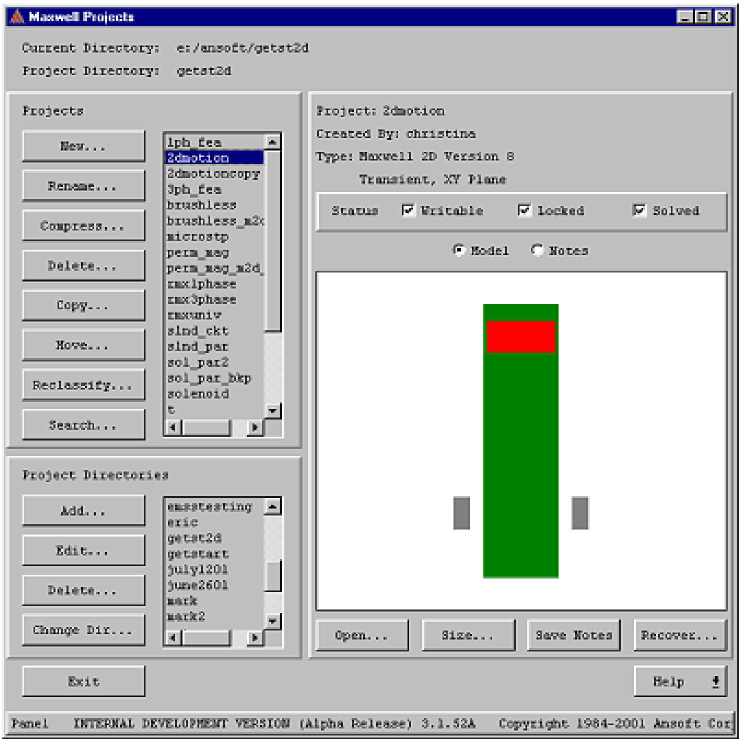
Start the Project Manager

You can use the Project Manager to create, rename, or delete project files. The Project Manager also allows you to access projects created with other Ansoft products.

➤ To display the Project Manager:

- Choose **Projects** in the Maxwell Control Panel.

The Project Manager appears.



Note: From now on, when you are asked to “choose” a button or command, click the left mouse button on it.

Create a Project Directory

The first step in using Maxwell 3D to solve a problem is to create a project directory and a project in which to save all the data associated with the problem.

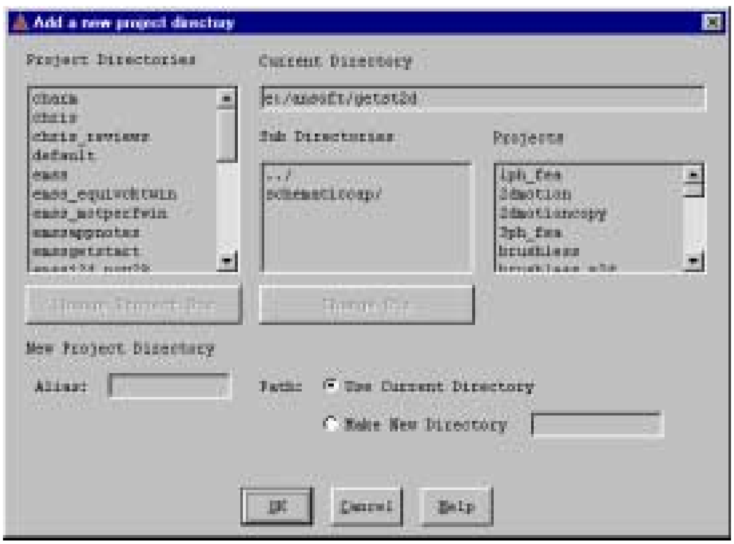
A project directory contains a specific set of projects created with the Ansoft software. You can use project directories to categorize projects any number of ways. For example, you might want to store all projects related to a particular facility or application in one project directory. You will now create a project directory.

The **Project Manager** should still be on the screen. Add the **getstart** directory that will contain the Maxwell 3D project you create using this *Getting Started* guide.

Note: If you've already created a project directory while working through one of the other Ansoft *Getting Started* guides, skip to the "Create a New Project" section on page 2-6.

➤ To create a project directory:

1. Choose **Add** from the Project Manager Window. The **Add a new project directory** window appears, listing the directories and subdirectories.



2. Double-click on the sub-directory names until the current directory (at the top of the window) is the one where you want to locate the project.
3. Enter **getstart** in the **Alias** field. An alias is a project directory

name that refers to the current directory. You can use aliases to refer to project directories located in different computer directories, or across network drive locations.

4. Select **Make New Directory**.
5. Choose **OK**. The new project directory appears in the project directory list.

Note: ☞

Maxwell 3D projects can only be created in directories which have aliases — that is, directories that have been identified as project directories using the **Add** command.

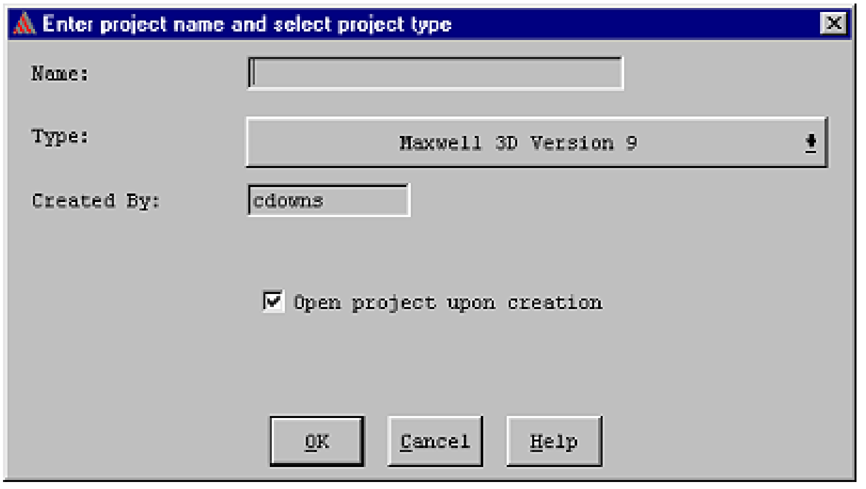
➤ Do the following if you want to change directories to look at another's contents:
   1. Choose **Change Dir** from the Project Directories box.
   2. Double-click the left mouse button on the directory you wish to view.
   3. Choose **Done** when you are have selected the desired directory.

Create a New Project

Now that you've created the project directory, create a project within that directory.

➤ To create the new project:

1. Select **getstart** by clicking the left mouse button on its name. The project directory name is highlighted.
2. Choose **New**. The **Enter project name and select project type** window appears.



3. Enter **Magnet** in the **Name** field.
4. Select the project type you want to use from the pull-down menu bar. For this project, choose **Maxwell 3D Version 9**. The rest of the menu lists the licenses for other Ansoft products; some of these are part of the Maxwell 3D package.
5. Enter your name or user ID in the **Created By** field:
   - If you are using UNIX or Microsoft NT, your login name is automatically entered here.
   - If you are using Windows, you may enter your user name or leave the field as it is.
6. Clear the **Open project upon creation** check box, allowing you to create the new project without immediately launching the software.
7. Choose **OK** or press **Return** to create the new project.
   The project name appears in the list of projects in the **getstart** project directory.

Enter Notes

It is generally a good idea to save the notes about the new project so that the next time you use Maxwell 3D, you can view information about a project without opening it.

➤ To enter a description for this project:

1. Click in the **Notes** area (its border is highlighted). Enter your notes on the project, such as the following:

   ```
   This sample problem was created using Maxwell
   3D and the 3D Getting Started Guide.
   ```

   As you begin typing text, the **Save Notes** button below the Notes area is no longer grayed out, indicating that it is enabled.
2. When you are done typing the description, choose **Save Notes**. The notes are saved, and the **Save Notes** button changes becomes disabled.

Note: ☞ Grayed-out text on commands or buttons means that the command or button is temporarily disabled.

3

Draw the Model

Now that you have created the empty project, the next step is to open the project and create the object in the model.

The goals for this chapter are to:

- Open a project.
- Define a problem region for the project.
- Create the magnetostatic model.
- Save the project.

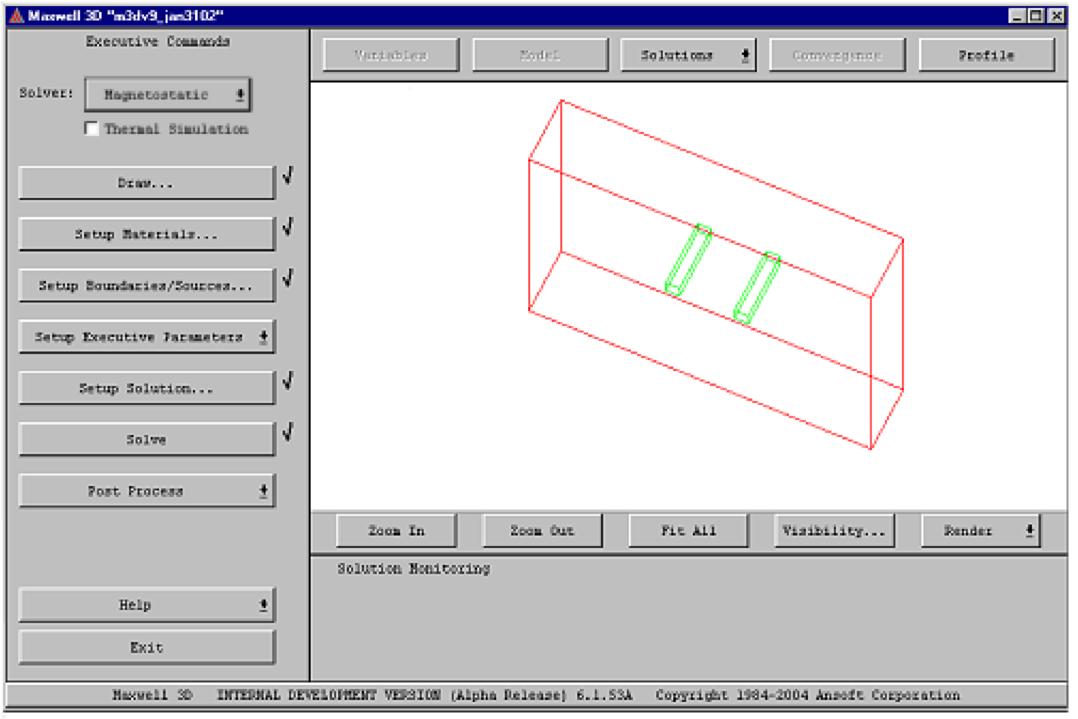
Time:

The total time needed to complete this chapter is approximately one hour.

Open the Project

➤ To open the new project:

1. The project name should still be selected. If not, select the project name in the **Maxwell Projects** window. The name is highlighted.
2. Choose **Open** under the **Notes** area. The Executive Commands window appears. A column of buttons on the left (the Executive Commands menu) gives you access to modules in the order you are expected to follow.



By default, **Magnetostatic** appears as the **Solver** type. Choosing this button allows you to change between Electrostatic, Magnetostatic, and Eddy Current solvers. If you have purchased additional modules, Thermal and Transient solvers may also be available.

Since you will be creating a magnetostatic model, leave this setting unchanged at Magnetostatic.

Start the 3D Modeler

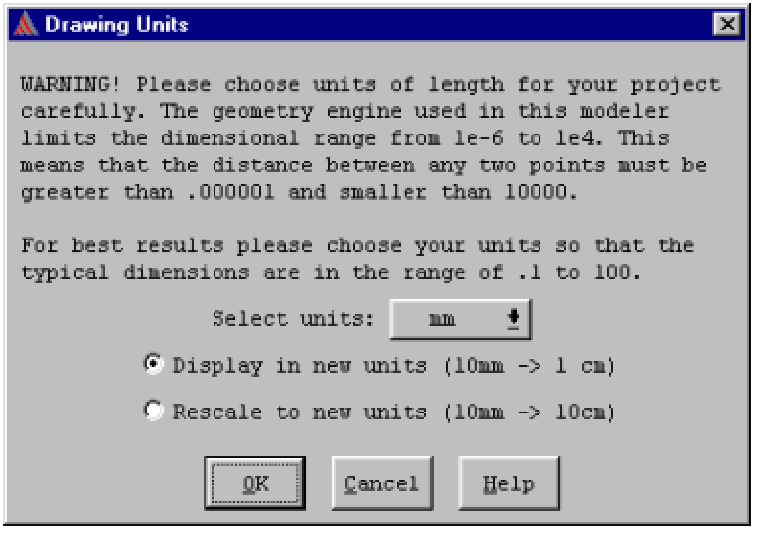
To draw the geometric model, use the 3D Modeler, which is the portion of Maxwell 3D that allows you to create objects.

When you start the 3D Modeler, four distinct windows, called view windows, appear. Three of these windows show two-dimensional views of the model you are creating, while the fourth window displays a full three-dimensional view.

A dot in each window shows the position of the modeling cursor; this dot is blinking in the active window. To draw in a window, activate it by clicking in that window. To determine which coordinate system is used by a window, move the cursor over that window and look at the letters next to the cursor.

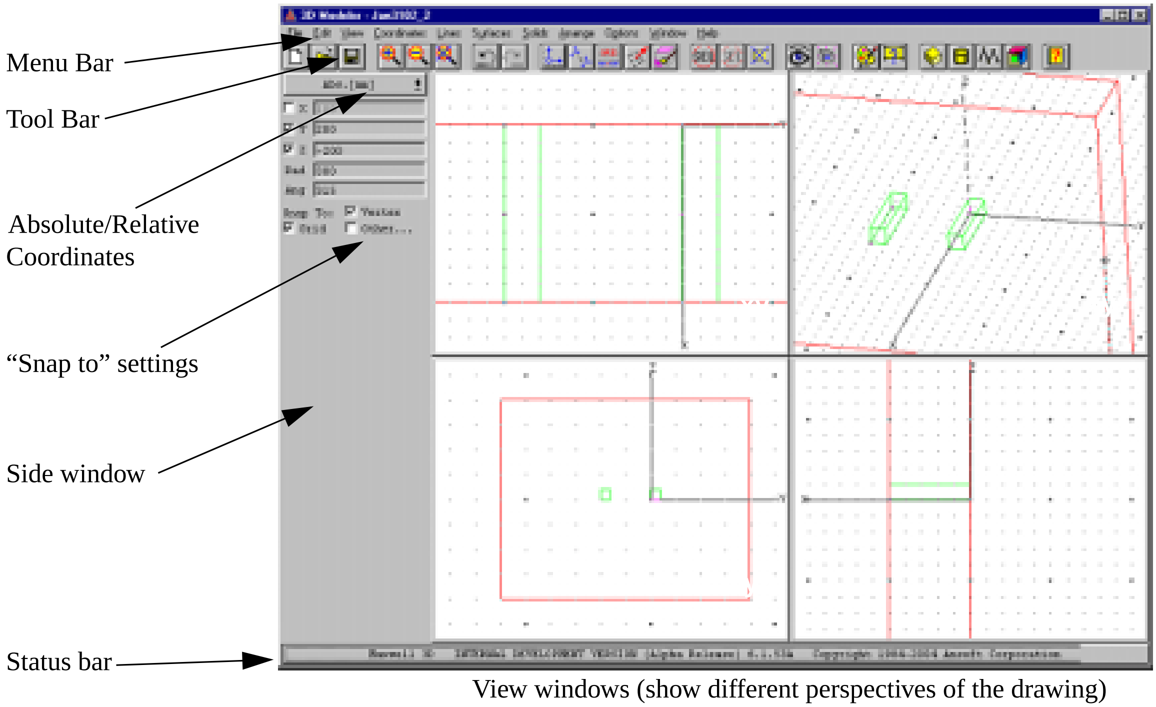
➤ To start the 3D Modeler:

1. Choose **Draw** from the top of the Executive Commands menu. The 3D Modeler window appears. As the software opens the 3D Modeler window, you may be prompted to choose the units of length you want to use in the following window:



2. If you are prompted, choose **OK** to accept the default of millimeters **(mm)**. The units chosen are shown at the Absolute / Relative coordinates menu.

The 3D Modeler window is divided into several parts:



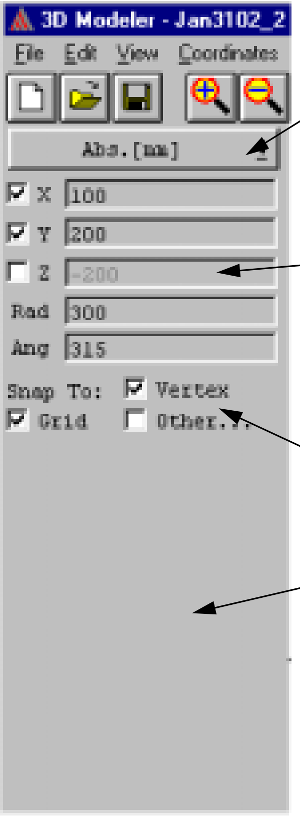
Note:

If you require further information on any topic in Maxwell 3D, such as the 3D Modeler commands or windows, there are several options for displaying help:

- Choose **Help** in a window.
- Press **F1**, see the cursor change to **?**, then click on the item on which you need help.
- Use the **Contents** or **Index** commands from the **Help** menu.

Side Window

The area on the left of the 3D Modeler (or on the right, depending on your preference settings) is referred to as the *side window*. The side window is where you can change the coordinates or set the snaps of the model to something other than the default. This window is also where many command-specific fields appear.



Use this toggle menu to select the type of coordinate system to use. You may select from an absolute coordinate system or a relative coordinate system. The selected coordinate system appears as the name of the menu.

Use these fields to enter the X-, Y-, or Z-coordinates and the radius, distance, or angle. The check box next to the coordinate fields must be selected to enable the coordinate field. These coordinate fields are used to enter the coordinates for a variety of commands.

Use these checkbox to select the type of “snap-to” you want to employ when selecting objects or object artifacts (vertices, lines, faces, and so forth). When you select the **Other** checkbox, a window appears allowing you to select from a variety of “snap-to” options.

Use the blank area under the coordinate section for entering information for many commands. Fields appear in this area allow you to enter information specific to the command you just selected.

Snaps

The **Grid** and **Vertex** snaps are set by default and already active.

➤ To select the snap-to behavior:

1. Choose the **Other** check box under **Snap To** on the left side. A window appears below the coordinates fields.
2. Select the type of **Edge Snaps** you prefer, from the following:

Grid inters.	Allows you to set the snaps at the points where the grid intersects an axis.
Edge center	Allows you to set the snaps at the central points of the edges.
Arc center	Allows you to set the snaps at the arc center.

3. Select the type of **Face Snaps** that you prefer, from the following:

Axis inters.	Allows you to set the snaps at the points where an axis crosses the face of an object.
Face center	Allows you to set the snaps at the center of the object face.

4. Choose **OK** to accept the snap-to behavior.

Define the Problem Region: Coordinates and Units

The area containing the model is called the *problem region*; the four view windows provide you with different perspectives of the problem region, which is initially empty except for the coordinate axis.

Note: Because the window can be customized, it may differ slightly from those shown in this guide. For instance, if someone has used the 3D Modeler before you, they may have changed the default position of the tool bar. Do not worry if this happens; simply use the windows as they appear in the program. This guide reminds you to check when default settings are needed for your project.

In order to draw the model, you must activate a window in which to create it.

➤ To activate the window in which you will draw the model:

- Click in the top left (xy) window to activate it. The cursor becomes tagged with the current coordinate system of the window it is placed in. This window is where you will begin to draw your model.

Absolute and Relative Coordinates

In this guide, you will be working in absolute coordinates. If relative coordinates are set, the coordinate system measures from an origin defined by you, which you can change. If absolute coordinates are set, the coordinate system measures from a system-defined origin.

Look at the **Absolute/Relative** coordinates menu in the upper left part of the 3D Modeler window. Make sure absolute coordinates are chosen, immediately under the left-hand end of the tool bar.

If absolute coordinates are not selected, do the following:

- Select **Absolute** from the **Rel. [mm]** menu in the side window.

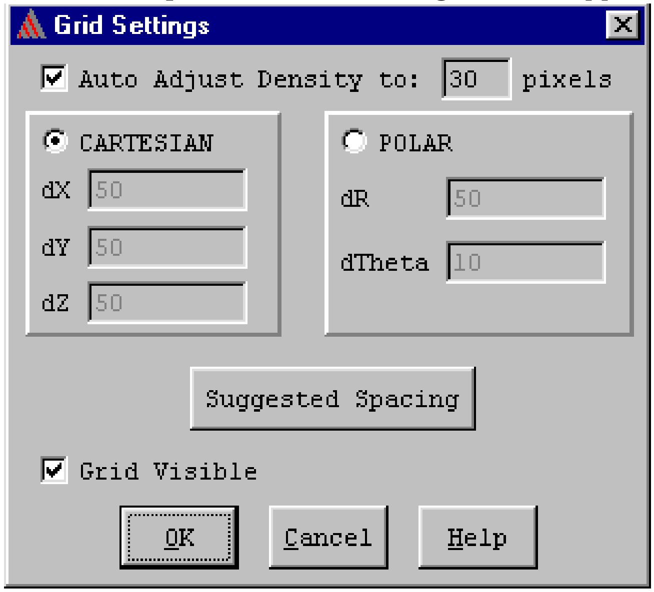
Grids

The 3D Modeler uses grid settings to provide a visual guide as you draw objects. There are no *particular* unit types associated with the grid points. However, you can set up the grid so that each grid point is displayed at a given number of units. The default settings keep the grid points at about 30 pixels apart, even if you zoom in and out, and start with each grid point displayed at 20 mm.

Set the grid points to five units apart, so the display will change as you zoom in and out.

➤ To set the grid points to five units apart:

1. If it is not already selected, click on **Grid** from the **Snap To** settings in the 3D Modeler window so you can use the grid for placing objects. This should be selected by default.
2. Choose **View/Setup Grid**. The **Grid Settings** window appears.



Note: **CTRL+G** is listed to the right of the **Setup Grid** menu option. Any time the Executive Commands window is displayed, you can press the **CTRL** key concurrently with **G** to display the Grid Settings window. Whenever a hotkey like this is displayed with a menu item, you can use it to access the command.

3. Deselect **AutoAdjust Density**. This allows the density of the grid points to change as you zoom in or out.
4. Make sure **Cartesian** coordinates are activated.
5. Enter **5** in the **dX** field.
6. Enter **5** in the **dY** field.
7. Enter **5** in the **dZ** field.
8. Leave **Grid Visible** activated. If it is not, choose the button to activate it.
9. Choose **OK** or press **Return**. The cartesian grid changes to the new values. The grid disappears because the active window now displays a region more narrow than the grid points are spaced.
10. Set the grid for the remaining windows in the same manner.

The grids in these windows are removed from the screen.

Note: When you specify the grid settings for the view window, no grid will be visible, though the model in that window will adhere to the grid settings.

Note that the grid only changes in the active (XY) window. Since grid settings are display settings, they only change the display of the active window. You must change the display setting in each of the view windows in which you wish to change the grid density. Changing the display will make it easier to draw the objects in the model.

Zooming In and Out of the View Window

The displayed grid points in the xy window are still too close together to be very useful. Because you changed the display from 20 mm to 5 mm, there are now four points to every original one. You may need to zoom in to make the display more usable.

➢ To zoom in to the problem area:

1. Hold the right mouse button down in the XY view window.
   A menu appears.
2. Choose **Zoom** from the menu. The cursor changes to a magnifying glass with a “+”.
3. Hold the left mouse button down in the middle of the window.
4. Move the cursor toward the bottom of the window. The display zooms out from the origin. (Moving in the opposite direction zooms in.)
5. Repeat the zoom (two or three times) until the grid points are spaced reasonably.
6. Click the right mouse button to exit the zoom command.

When the grid appears, you are ready to start drawing.

Create the Electromagnet

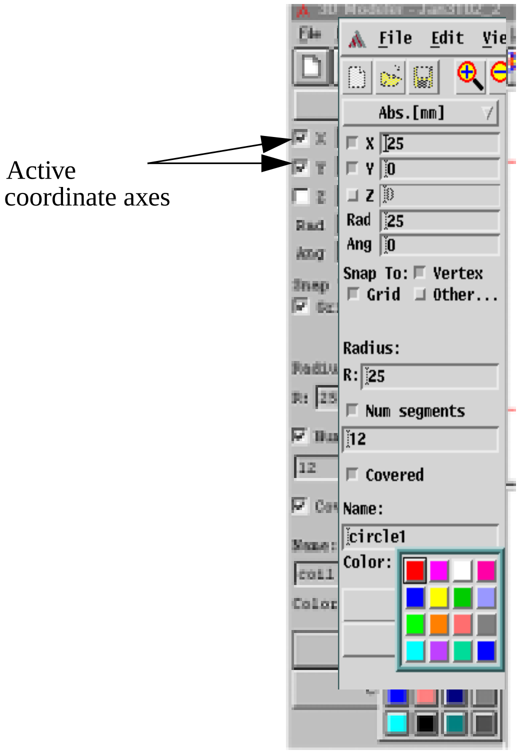
The first object you need to draw is the electromagnet, which consists of both a coil and a core. This is modeled as a hollow cylinder, which will eventually represent a number of wires wrapped around a central core.

Draw the Coil

In this model, the coil is formed by creating a circle, then sweeping the circle along a path to create a cylinder.

Draw the Circle

The first step is to create the circle which you will later sweep to form a cylinder.



➤ Create the circle:

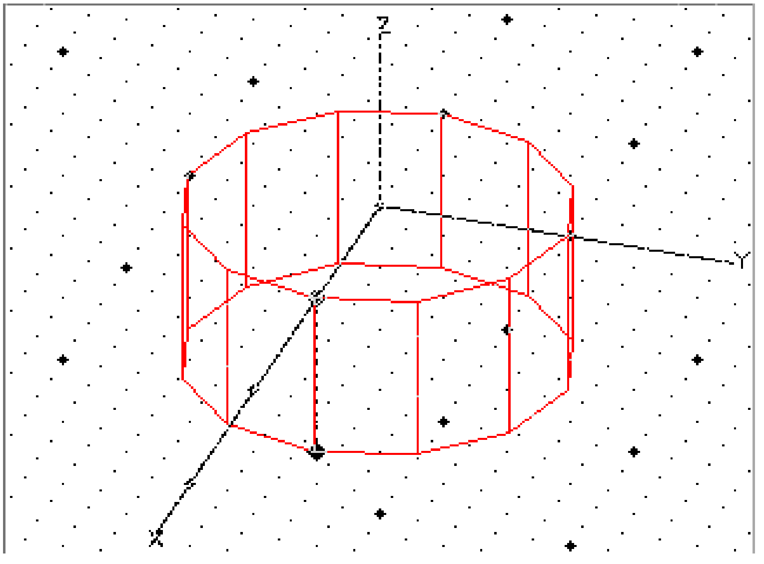
1. Choose **Lines/Circle**. The radius and name fields appear in the side window.
2. Click in the xy window to make it active.
3. Click on the origin in the xy window. The coordinates in the side window should all be zero.
4. Select **Z** as the **Circle axis** if it is not already selected.
5. Choose **Enter**. The modeler accepts the center point, and fields for entering circle information appear in the side window.
6. Enter **25** in the **X** field. The **Radius** field changes to 25, and an outline of the circle appears. You may only enter coordinates for active axes.
7. Leave **Num segments** set to its default value of 12 segments for the circle.
8. Leave **Covered** selected. This makes the circle a solid 2D disk, not an open circle.
9. Double-click on the **Name** field to modify the default name. Enter **coil** to name the circle.
10. Click on the colored box next to **Color**. A color palette appears. Select red from the color palette.
11. Choose **Enter**. The circle appears in all the windows.

Create the Cylinder

➤ To sweep the circle and create the cylinder:

1. Double-click on the origin in the xy window to make sure X, Y, and Z are all zero. This is the vector start point.
2. Choose **Solids/Sweep/Along Vector** from the menu. A message may appear, warning you that the 2D object you are going to sweep will be deleted after you sweep it.
3. If the message appears, choose **OK** to acknowledge the warning. A list of defined two-dimensional objects appears in the side window. Since you have drawn only one Z-dimensional object, **coil** is the only name listed.
4. Click on **coil** to select it, then choose **OK**. The circle is highlighted in a different color, and the fields used for entering the vector appear in the side window.
5. Enter **-25** in the **Z** field under **Enter vector**. A vector line appears in the drawing windows in the negative z direction, and the vector length field changes to **25**.
6. Choose **Enter** to sweep the circle into an cylinder.
7. Click in the 3D view window to activate it.
8. To see the entire model, choose **View/Fit All/All Views**, or click the following toolbar icon: .
9. Repeat steps 7 and 8 for the rest of the windows.

The 3D Modeler assigns the cylinder the same name as the original circle, **coil**.



Create the Hole for the Core

Since the **coil** represents wire wrapped around the core, you need to create space for the core in the cylinder which represents the coil. To do this, you must remove just enough of the coil to allow the core to fit, with a small air separation so that the 3D Modeler can treat the objects separately. The basic procedure is to:

- Create a “dummy” object to use as a place marker.
- Subtract the volume of the dummy object from the coil.

➤ To create the dummy cylinder:

1. Activate the xy view window by clicking in it.
2. Choose **Solids/Cylinder**. The side window prompts you for the center of the cylinder base.
3. Double-click on the **X** field and enter **0**.
4. Enter **0** for the **Y** and **Z** values. If either of these fields are inactive, activate them by clicking on the check box to the left of the field.
5. Select **Z** as the **Cylinder axis**.
6. Choose **Enter** to accept the center point of (0, 0, 0). The side window prompts you for the rest of the cylinder attributes.
7. Enter **13** in the radius (**R**) field. This causes the new cylinder to be slightly larger than the core that follows. Press **TAB** to move to the height field.
8. Enter **-25** in the height (**H**) field.
9. Accept **12** as the number of segments for the base circle of the cylinder in the **Num segments** field.
10. Accept **cyl1** for the name of the new cylinder.
11. Select the color square, and choose blue for the cylinder.
12. Choose **Enter** to display the new object. The dummy cylinder is now inserted into the primary cylinder.

➤ Subtract the volume of the dummy object:

1. Choose **Solids/Subtract** from the menu. The status bar prompts you to choose from the list of displayed solids for an object to subtract *from*.
2. Select **coil**. The coil object is highlighted.
3. Choose **OK**. The status bar now prompts you to select an object to subtract.
4. Select **cyl1**, the dummy object, from the new list of solids. Its volume will be subtracted from **coil**. **cyl1** is highlighted.
5. Choose **OK**. **cyl1** is removed from the model. The **coil** is now a hollow cylinder.

Draw the Core

Now, you need to create the central core. The following procedure shows you an alternate method of creating a cylinder.

➤ To create the core:

1. Choose the cylinder tool bar icon . The side window prompts you for the center of the cylinder base.
2. Double-click on any axis fields that are not zero, and change them to **0**. Activate any inactive fields you need to use.
3. Select **Z** as the **Cylinder axis**.
4. Choose **Enter** to accept the center point of (0, 0, 0). The side window prompts you for the rest of the cylinder attributes.
5. Enter **12.5** in the **R** radius field.

Note: If the grid were set up to do so, you could select the point by clicking on it, as you did while creating the last cylinder. However, this radius would fall between grid points. In general, it is more accurate to enter the points and distances into their respective fields than to try drawing them.

6. Enter **-25** in the **H** height field.
7. Accept **12** as the number of segments for the base circle of the cylinder in the **Num segments** field.
8. Enter **core** for the **Name** of the new cylinder.
9. Choose yellow for the core.
10. Choose **Enter** to accept the core cylinder. The electromagnet is now complete.

Saving Your Project

While you are working on a model, it is a good idea to periodically save the geometry. Maxwell 3D does not automatically save your work. If a problem occurs that causes the simulator to close unexpectedly, saving the project regularly can keep you from having to redraw the model.

➤ To save your project:

- Choose **File/Save** from the menu bar. A progress bar appears, indicating how much of the file is verified. The bar disappears when the file is saved.

Note: ☞ When saving, the 3D Modeler checks to see if your background encompasses the entire model. If it does not, a window appears, asking if you want to expand the background. Since the problem will not be set up correctly unless the background encompasses all the objects, choose **Expand Background**. The 3D Modeler displays a background box around the objects.

Create the Magnet

The next step is to create the permanent magnet above the core of the electromagnet. You will need to create another cylinder to represent the permanent magnet, then move it into position to create an air gap between the electromagnet and the permanent magnet.

Draw the Magnet

➤ To create the magnet:

1. Click on the cylinder tool bar icon.
2. Click at (0, 0, 0) in the xy window. If any coordinate is *not* set to 0, activate it in the side window and set it to 0.
3. Select **Z** as the **Cylinder axis**.
4. Choose **Enter**.
5. Enter **12.5** in the **R** field (radius).
6. Enter **5** in the **H** field (height).
7. Accept **12** as the number of segments for the base circle of the cylinder in the **Num segments** field.
8. Enter **magnet** for the object name.
9. Choose green for the magnet. This color contrasts distinctly with the other objects in the model and allows you to view each object in the model more easily.
10. Choose **Enter** to accept the cylinder attributes.

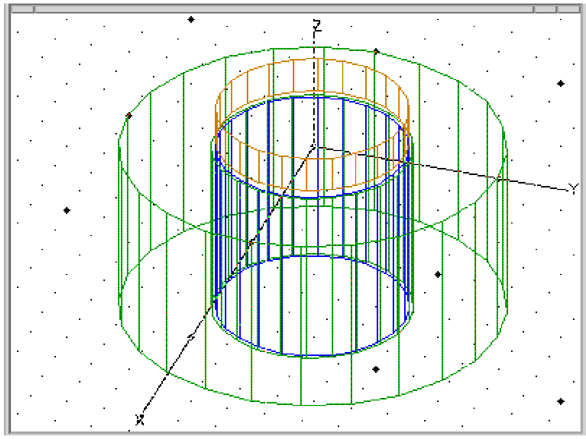
Move the Magnet

At this point, the permanent magnet rests on the electromagnet. To observe the fields between the two magnets, you must create a tiny air gap between them.

➤ To move the magnet into position:

1. Choose **Edit/Select** from the menu bar. A list of objects appears.
2. Select **magnet** from the list of objects.
3. Choose **OK**. The magnet is highlighted.
4. Choose **Arrange/Move** from the menu. The side window displays a menu for entering the motion vector.
5. Make sure the X and Y vectors are set at their defaults of **0** in the **Enter vector** fields.
6. Enter **1** for the Z vector; the vector length changes to 1 in the z direction.
7. Choose **Enter** to accept the vector. The magnet moves up

along the z-axis one millimeter.



8. Choose **Edit/Deselect All** to deselect any selected objects.

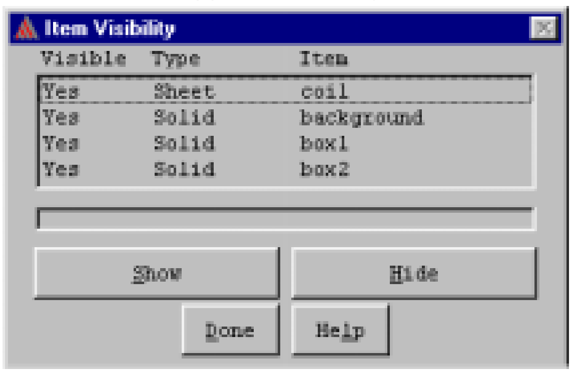
Toggle Off the Background

The background box is essentially an imaginary box that contains the problem region. Though this box is helpful in displaying the boundaries of the problem region, it is not essential in creating your model and does not affect the project and is often hidden by default.

If it is not already hidden, turn off the background object.

➤ To turn off the background box:

1. Choose **Edit/Visibility/By Item**. The **Item Visibility** window appears.
2. Select **background** from the window.
3. Choose **Hide** to toggle its visibility off:



4. Choose **Done** to accept the listed settings.

The project background is turned off (although it still exists). The background remains off until you toggle it back on.

Create a Terminal

The last object you need to create for the model is a terminal. The terminal is an exact cross-section of the conduction path of the coil which provides the current source in the model. The terminal itself does not have any physical representation — that is, it does not exist in a physical sense. It is a simulated object used in a model when you need to compute the current density.

The general steps for creating a terminal are the following:

- Create a rectangular object.
- Define 3D attributes for this object by making it into a *sheet*.

A sheet is a two dimensional object with three dimensional attributes; this permits Maxwell 3D to manipulate it as a three dimensional object.

Note: **Polyline** is an important command to know; it can be used to create arcs and splines, in addition to straight lines. Because of this, you will use **Polyline** in this tutorial, although the **Lines/Rectangle** command is generally the one you want to use to create rectangles.

➤ To prepare a window to create the terminal:

1. Click in the YZ window to activate it. This window displays the plane in which you will create the terminal.
2. Choose the **Zoom In** tool bar icon . The cursor changes to a crosshair.
3. Click in the window above and to the left of the model. A box appears, sized by the current position of the cursor and anchored by the original click point.
4. Click just below and to the right of the model, so the model is enclosed by the zoom box. The window display enlarges to fill the box. Repeat the zoom until the model fills the window. Now the objects will be much easier to work with.

Create the Coil Terminal

Now that the window has been zoomed in toward the model, you are ready to create the coil terminal.

➤ To create the terminal object:

1. Choose **Lines/Polyline**, or click on the polyline toolbar icon . The side window lists any polylines or sheets in your project; right now none are listed. At the bottom of the side window is a field for entering the name of the line you want to create or edit.

2. Double-click on the default name, and enter **terminal**.
3. Choose **OK** to accept the name and continue with the terminal creation. The side window displays polyline creation choices, and the status bar prompts you for the first point.
4. Choose light blue for the terminal.
5. Make certain **Add Vert** mode is activated, so you can add vertices to the object.
6. Make certain **Straight** is the type of polyline you are entering. This menu also allows you to enter arcs and curves (smooth polylines).
7. Click on the (0, 25, 0) point in the yz window.
8. Choose **Enter** to place the first point. An **X** appears at the point when it is accepted. If you place the point at a location other than (0, 25, 0), choose **Cancel** to cancel the action and repeat the polyline command from step one.

Warning: Wait until the point is accepted before entering another point, or the 3D Modeler will accept the next point instead.

9. Enter **13** in the y-coordinate field, then choose **Enter** to accept this point. Because the snap settings are spaced at five units apart, you can’t click on this point unless you change the **Snap To** settings.
10. Enter **-25** in the z-coordinate field. Leave the y-coordinate at **13**. Choose **Enter** to accept this point.
11. Click on the (0, 25, -25) point. You should still be working in the yz plane, since you deactivated the x-coordinate field. Choose **Enter** to accept this point.
12. Choose **Close** to return to the original point and close the rectangle.
13. Deselect **Covered** to make the polyline an open object.
14. Choose **Done**. A two dimensional object is entered into the project with vertices at:

    (0,25,0) (0,13,0) (0,13,-25) (0,25,-25)

➤ Make the terminal a sheet object:

1. To view the type, choose **Edit/Visibility/By Item**, or click the tool bar icon  . The **Item Visibility** window appears. The type for **terminal** is Polyline; the other objects are solids. Choose **Done**, since you are not changing any object visibility.
2. Choose **Surfaces/Cover Lines**. All 2D objects (in this case **terminal**) are listed in the side window.
3. Select **terminal**. The terminal is highlighted.
4. Choose **OK**.
5. Choose **Edit/Visibility/By Item** to view the objects again. The terminal is now of type sheet. Maxwell 3D can now manipulate it like a 3D object, although you will be unable to assign materials to it.

Define the Problem Region

When Maxwell 3D solves a problem, it takes the background as the default problem region, and solves over that volume. In this problem, the background will not be large enough to provide accurate results; the background “object” itself will interfere with the problem physics because much of the model exists outside the problem region.

The problem region should generally be three to five times the size of your model to ensure that the model exists completely within the region.

The general steps for defining the problem region are the following:

- Define an object for the outer limits of your model.
- Define the problem region around those outer limits.

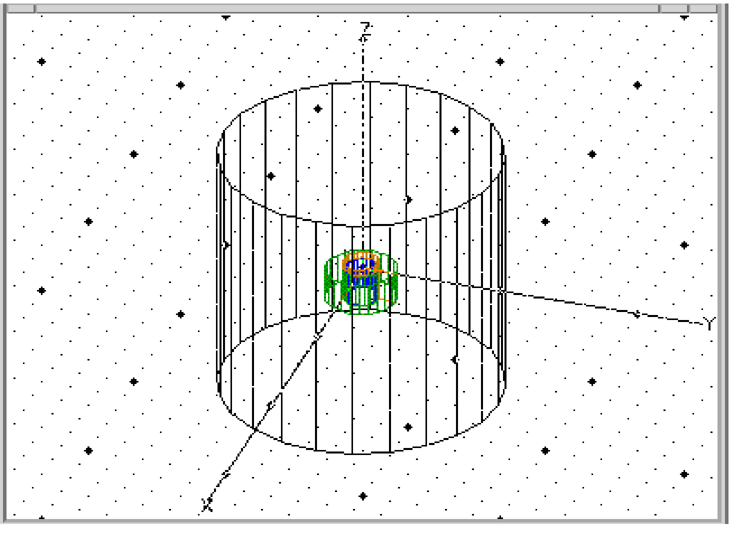
➤ To draw an object for the limits of the model:

1. Activate the 3D view window by clicking in it.
2. Click on the (0, 0, 0) point in the 3D view window.
3. Click on the cylinder tool bar icon.
4. Enter the coordinates (**0, 0, -100**) for the center of the cylinder circle.
5. Leave **Z** as the *Cylinder axis*.
6. Choose **Enter** to accept this point.
7. Enter **100** for the radius.
8. Enter **200** for the height.
9. Enter **region** for the object name.
10. Choose grey for the problem region.
11. Choose **Enter** to accept the cylinder attributes.

Define the Problem Region

➤ To define the size of the problem region:

1. Choose **Options/Region/Fit All**. New fields appear in the side window.
2. Enter **0** in the **Padding Percent** field. This will fit the problem region as closely as possible to the largest object (in this case the **region** cylinder).
3. Leave **Round off** deselected. This is used when fitting a large padding percent to the problem space.
4. Choose **Enter** to accept the problem region. The problem region is represented as a cube around the model, which Maxwell 3D uses for solving the problem.
5. Click in each view window.
6. To view the problem region in every window, choose **View/ Fit All/All Views**, or click the toolbar icon.



Shading and Rendering

Now that the model is complete, you can observe the colors of your objects. Shading the objects in the model allows you to visualize the solid objects more easily.

➤ To shade the objects:

1. Choose a view window to make it active.
2. Choose **View/Render/Smooth Shaded** from the menu bar. The objects in the model fill in with colors. Notice that the region blocks out all of the model in the 3D view window.
3. Choose **View/Render/Wireframe** to see all of the objects in wire frame mode.

Exit the 3D Modeler

Now that you have created the geometry for the model, return to the Executive Commands window. Save the project as you exit.

➤ To exit the 3D Modeler:

1. Choose **File/Exit** from the menu. A window prompts you to save the changes.
2. Verify your model before saving it. If there are any inconsistencies in the drawing, the 3D Modeler warns you before your project is saved, and gives you a chance to correct any problems.
3. Choose **Yes** to exit the Modeler. A progress bar shows you the project being verified.

After you exit the 3D Modeler, you return to the Executive Commands window.

Note: ☞

- The **Draw** button in the Executive Commands menu has a check mark next to it.
- The **Setup Materials** button is now active (it was grayed out before you started the 3D Modeler).

Exit the 3D Modeler

4

Define The Problem

You now need to assign materials to the objects. Maxwell 3D uses a database of predefined attributes for commonly used materials. These definitions are locked so that you cannot accidentally change or delete them.

In this section, you will assign the following materials to the objects:

- Assign the core as steel.
- Assign the coil as copper.
- Assign the magnet as NdFe35.

You also need to assign boundary conditions and sources to your model so that the following two conditions are met:

- The current in the coil is defined.
- The terminal in the coil is the current source.

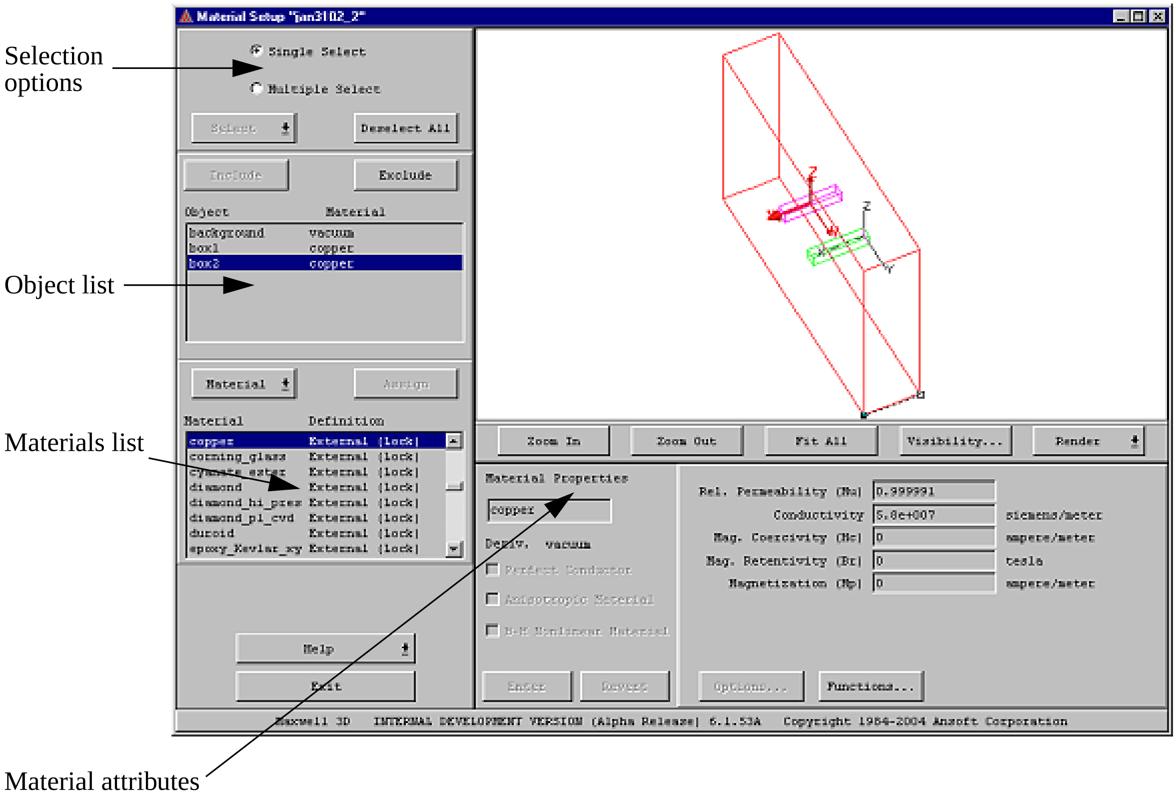
Time:

The total time needed to complete this chapter is approximately 15 minutes.

Access the Material Manager

➤ Choose **Setup Materials** to display the **Material Setup** window.

This window lists the 3D objects in your project, so you can select and assign materials to them. Below the object list is a list of materials in the material database. Ansoft includes definitions for approximately 80 materials, which are shown as **External(Lock)**.



Exclude the Background

A **background** object is automatically created when the model is saved and occupies any part of the problem region not occupied by created objects. Because the **background** is not a part of the physical model involved in the solution, exclude it from the model.

➤ To exclude the background:

1. Select **background**.
2. Choose **Exclude** (above the object list) so that it will not be considered an object while the problem is being solved.
3. Deselect **background** by choosing it from the list.

Assign Steel to the Core

➤ To assign steel to the core:

1. Choose **Single Select** to select one object at a time.
2. Select **core** from the object list.
3. Select **steel_1008** from the materials list. Use the scroll bar to locate this material if it does not already appear.
4. Choose **Assign**. The core is now assigned the properties of steel_1008.

Assign Copper to the Coil

➤ To assign copper to the coil:

1. Select **coil** from the object list.
2. Select **copper** from the materials list.
3. Choose **Assign**. The coil is assigned the material properties of copper.

Assign Vacuum to the Problem Region

➤ To assign vacuum to the problem region:

1. Select **region** from the object list.
2. Select **vacuum** from the materials list.
3. Choose **Assign**. The problem region is assigned the properties of a vacuum.

Assign NdFe35 to the Magnet

A magnetic material must be assigned to the object that is to be a permanent magnet.

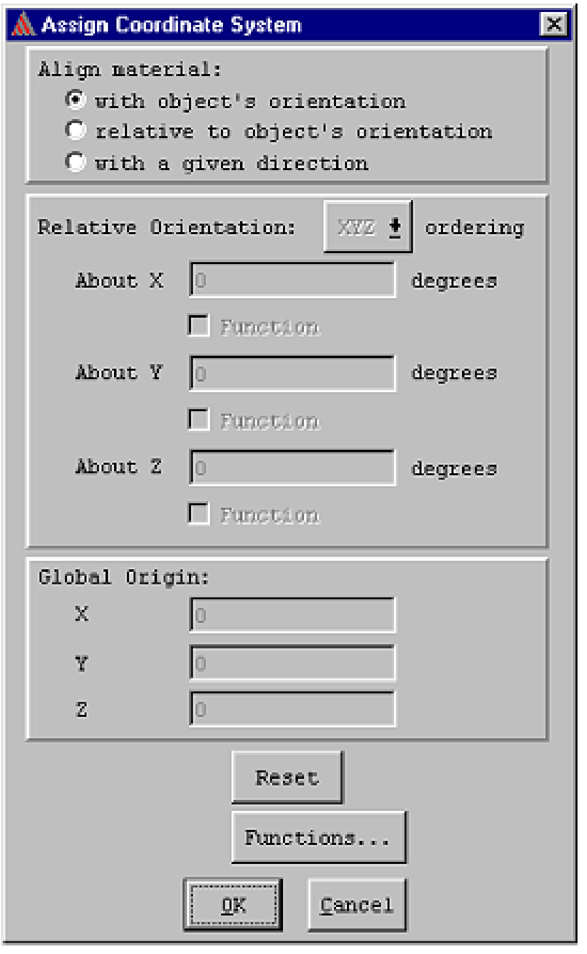
When you assign a magnetic material to an object, you must also define a direction of magnetization for the material. By default, the direction of magnetization is along the x-axis. However, in this problem, the direction of the magnetization in the core points along the negative Z-axis.

To model this, you must change the direction of magnetization to act at a 90 degree angle from the default. This aligns the direction of magnetization of the material along the negative z-axis, moving it from the X-axis.

➤ To assign NdFe35 to the magnet:

1. Select **magnet** in the object list.
2. Scroll to **NdFe35** in the material list, then highlight it and choose **Assign**. (Uppercase materials are listed before

lowercase.) The **Assign Coordinate System** window appears.



3. Select **relative to object's orientation** if it is not already selected.
4. Enter **90** in the **About Y** field to cause the magnetization direction to be aligned along the z-axis.
5. Leave the other parameters set to their defaults.
6. Choose **OK** to accept the alignment.

Now that you have completed assigning the materials to the objects in the model, you are ready to specify boundary conditions.

➤ To exit the Material Manager:

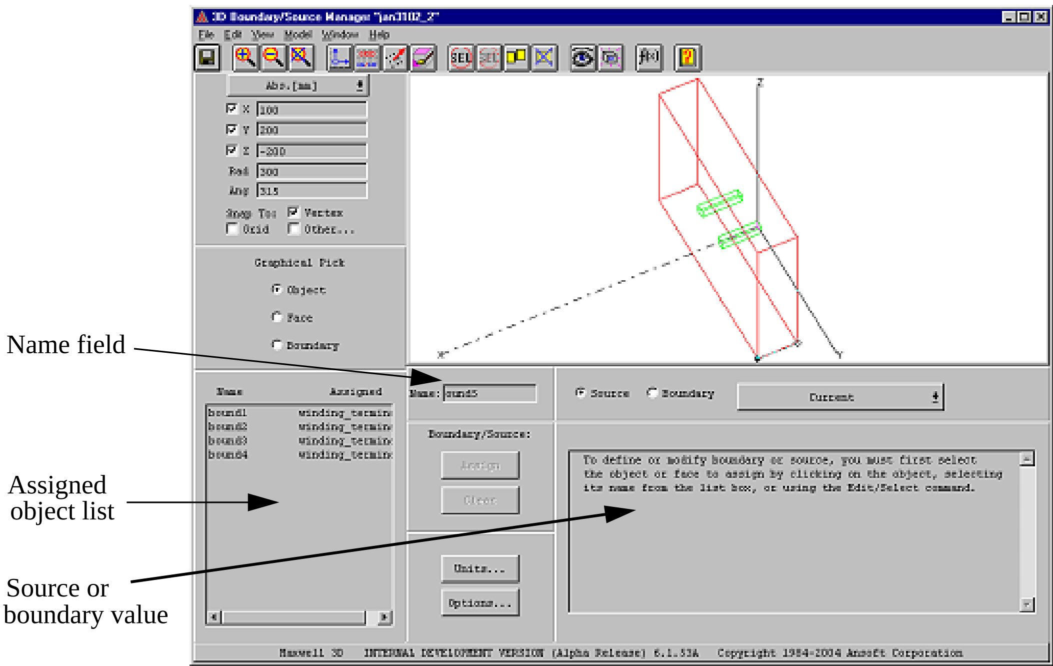
- Choose **Exit** to leave the Material Manager. Be sure to save the changes as you exit.

Access the 3D Boundary/Source Manager

Now that you have assigned materials to the objects, you need to set the conditions for your problem. Notice that after you exited from the Material Setup, both the **Setup Materials** and **Setup Boundaries/ Sources** buttons are checked. Maxwell 3D automatically assigns boundaries and sources to the objects in the model based on the material properties assigned to the model. However, in this problem, you will change the value of the current being applied to the model. The 3D Boundary / Source Manager allows you to assign boundary conditions and source values to the objects in the model.

- *Boundaries* specify the behavior of the electric or magnetic field at the edges of objects, including the edge of the entire problem region (the edge of the background). This limits the problem to the area defined by the boundaries, and prevents the problem from becoming unmanageable.
- *Sources* are used to set the electric or magnetic potential on a surface to a specific value (or function).

➤ To start the 3D Boundary / Source Manager, choose **Setup Boundaries/Sources**. The **3D Boundary/Source Manager** window appears.



Assign a Current to the Coil

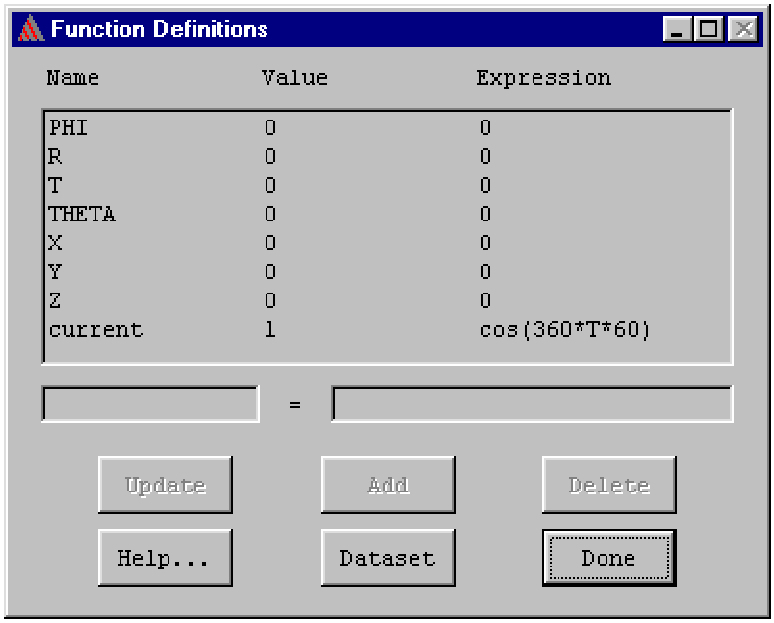
Now you must define the source that provides the current flowing through the coil. To do this, use the terminal you created as the source. When you assign a source value to the terminal, the direction of the current flow in the coil is defined.

In this model, you will define the source value as a function. Though the source itself is a constant value, by defining it as a function, you can later modify the functional source value without the need to directly redefine the terminal in the Boundary Manager.

Define the Source

➤ To define the source:

1. Choose the **Model/Functions** menu option. The **Function Definitions** window appears. You use this window to define the nominal (default) function value.



2. Enter **Source** in the first field of the variable equation.
3. Enter **8000** in the value field on the right.
4. Choose **Add**. This adds the value to the symbol table.
5. Choose **Done**. **Source** has now been defined as 8000, and can be assigned to the terminal.

Assigning the Source to the Terminal

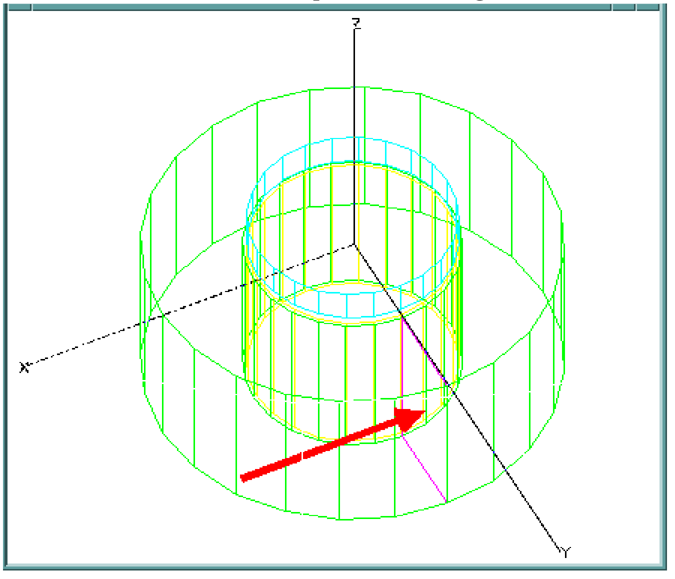
➤ To assign **Source** to **terminal**:

1. Choose **Edit/Select/By Name** from the menu. The **Select By Name** window appears.
2. Select **Object** if it is not already selected. The list of objects appears.
3. Select **terminal**. Make certain it is the only object highlighted.
4. Choose **Done**.
5. Enter **CurrentIn** in the **Name** field to define the source you are applying to the terminal.
6. Enter **Source** in the **Value** field to assign the value to the **terminal**. Check that the units are amperes (A).
7. If the units are not amperes:
   - choose **Units**,
   - select **A** to highlight it, and
   - choose **OK** to specify the units.
8. Choose **Assign**. **CurrentIn** now has a value of 8000 ampere-turns.

Check the Direction of the Current

➤ To check the direction of current:

1. Select **CurrentIn**. An arrow indicating the current direction appears.
2. Check that the arrow showing the current direction points toward the YZ plane. If not, choose **Swap direction**, choose **Assign**, and then to point the current in the correct direction. In this model, the arrow should point to the right.



Exit the 3D Boundary/Source Manager

Now that you have assigned the current source to the model, you are ready to define the values you wish to solve for.

➤ To exit the 3D Boundary / Source Manager:

- Choose **File/Exit** to exit the module, making sure you save your project as you exit.

5

Generate a Solution

Now that you've defined your problem, you need to tell Maxwell 3D what information you are looking for, and how accurate you want the solution to be. For this chapter, you will:

- Use **Setup Executive Parameters** to solve for magnetic forces on the coil, core, and the magnet. You can then look at the solution to see that these forces add up to zero to check the accuracy of the results.
- Use **Setup Solution** to define the how accurate you want the solution to be, and at what point you want the solution process to end.

Time:

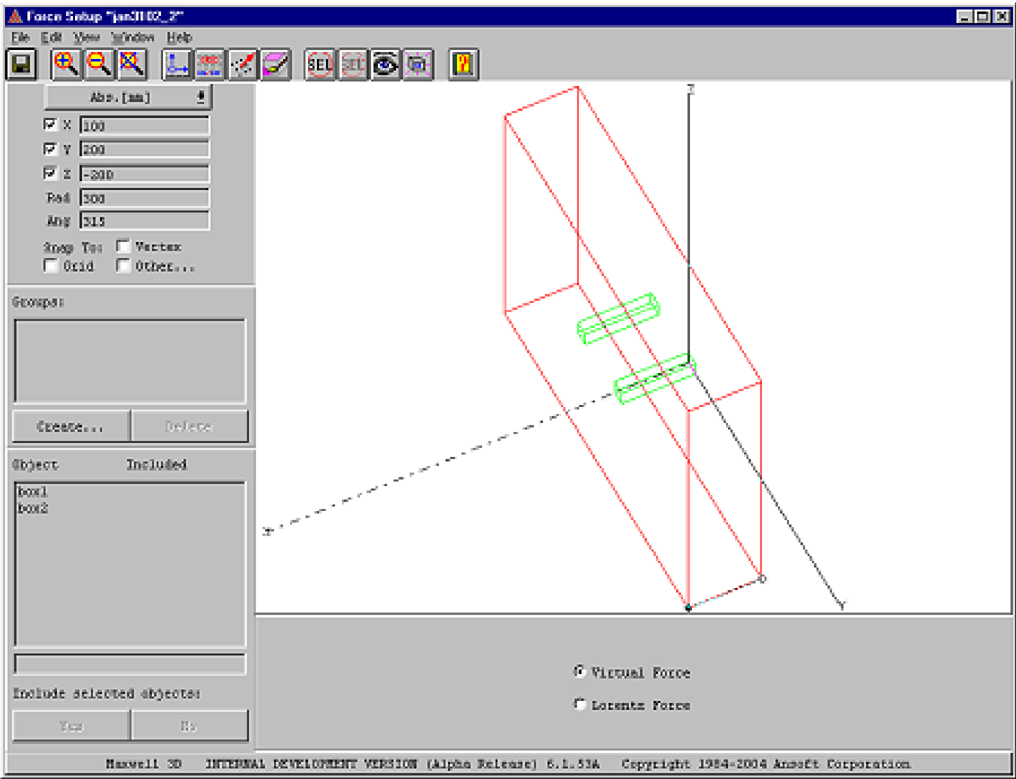
The total time needed to complete this chapter is approximately 35 minutes.

Setup Executive Parameters

The next step for this problem is to specify the Executive Parameters for the objects in the model. In this problem, you are solving for the virtual forces on the magnet, the coil assembly, and the entire model.

➤ To choose the executive parameters:

1. Click and hold the **Setup Executive Parameters** button in the Executive Commands menu. You have the option of specifying the force, torque, or matrix parameters the system will solve for. The matrix parameters can be capacitance, inductance, or impedance, depending on the problem. Refer to the online documentation for further discussion of the executive parameters.
2. Choose **Setup Executive Parameters/Force**. The **Force Setup** window appears.



3. Check that **Virtual Force** is chosen as the type of force. The virtual force is the force on the object due to its material properties. You would activate **Lorentz Force** if you were calculating forces caused by a magnetic field.

Create a Force Setup for the Magnet

Force is calculated on groups of objects. A group is a collection of objects on which the total virtual force will be solved. When the force solutions are generated, the virtual force on each group will vary because each group will contain a different set of objects.

➤ To create the force setup for the magnet:

1. Choose **Create** to create a new group. A pop-up window appears.
2. Enter **MagnetAssy** for the new name.
3. Choose **OK**. **MagnetAssy** is added to the **Groups** list.
4. Select **magnet** in the **Object** list.
5. Choose **Yes** to include the magnet in the **MagnetAssy** group.

Create a Force Setup for the Coil and Core

➤ To create the force setup for the coil:

1. Choose **Create** to create a new force group.
2. Enter **CoilAssy** for the new name.
3. Choose **OK**.
4. Select **coil** and **core** from the object list. If you are using a PC, hold the **Shift** key or the **Ctrl** key while you click on the object names, to choose more than one object at a time.
5. Choose **Yes** to include the coil and core in the **CoilAssy** group.

Create a Force Setup for the Entire Model

➤ To create the force setup for the model:

1. Choose **Create** to create a new force group.
2. Enter **Model** for the new name.
3. Choose **OK**.
4. Select **magnet**, **coil**, and **core** in the object list.
5. Choose **Yes** to include the objects in the **Model** force group.

Exiting the Executive Parameters Module

Now that you have specified the executive parameters for the problem, you are ready to specify the solution criteria.

➤ To exit the Executive Parameters module for force:

- Choose **File/Exit**, saving the changes.

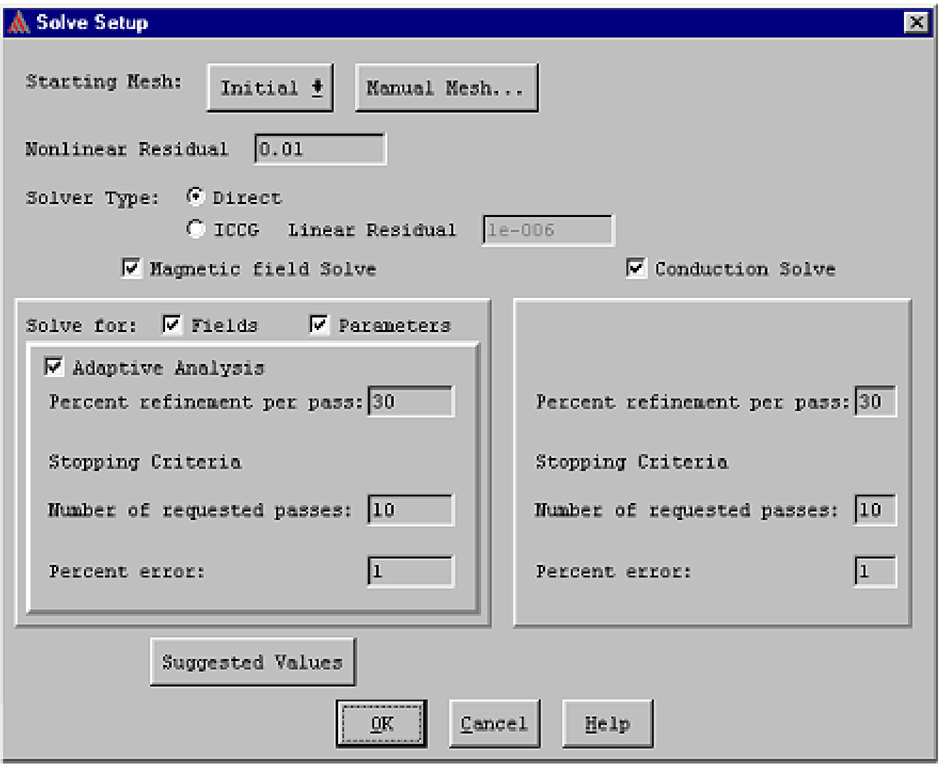
Specifying Solution Criteria

Once you have set up your model, the next step is to specify the solution options. This means you must define the following:

- The range and increments of any data input to the solver.
- The precision of the results.
- The number of times you want to repeat the solution process.

➤ To access the **Solve Setup** window:

- Choose **Setup Solution/Options** from the Executive Commands window. The **Solve Setup** window appears.



Solver Type

Leave the **Solver Type** set to **Direct**. The **Direct** solver will always converge. Generally use this choice for smaller meshes (less than 1500 elements), since the small size matrix normally corresponds to the initial coarse mesh. The **ICCG** (incomplete conjugate gradient) solver is faster for large matrices, but occasionally fails to converge (usually on magnetic problems with high permeabilites and small air-gaps). In this problem, the **Direct** solver is sufficient.

Magnetic Field Solve

Leave **Magnetic Field Solve** selected. This instructs the software to generate a nominal field solution.

Residual

The residual values specify how close each solution must come to satisfying the equations that are used to compute the magnetic field. Accept the default values for the solver residual.

Solve for Field and Parameters

➤ To solve for fields and parameters:

- Leave the **Solve for:** options, **Fields** and **Parameters**, and the **Conduction Solve** buttons selected. These options tell the system what types of solutions to generate.

When this option is selected...	*This occurs...*
Fields	A field solution is generated.
Parameters	Any quantities that you set up using the **Setup Executive Parameters** command are computed. In this case, the forces on the model are computed.
Conductions Solve	Any DC conduction solutions are generated. In this case, the conduction solutions on the coil are generated.

Adaptive Analysis

The initial mesh is the set of tetrahedra that are created by default and represents the points in the model at which the solution will be calculated. Adaptive analysis refers to the process of breaking down the tetrahedra in the mesh to produce more accurate results in the final solution.

The starting mesh above the **Residual** field is set to **Initial**.

➤ To specify the solution criteria for the adaptive analysis:

1. Leave **Adaptive Analysis** selected. This instructs the system to solve the problem iteratively, refining the regions of the mesh in which the largest error exists. Refining the mesh makes it more dense in the area of highest error, resulting in a more accurate field solution.
2. Enter **50** for the **Percent refinement per pass**.
3. Enter **8** for the **Number of requested passes**. After each iteration, the system calculates the total energy of the system and the percentage of this energy that is caused by the solution error. It then checks to see if the number of requested passes has been completed, or if the percent error and the change in percent error between the last two passes match the requested values. If either of these criteria are met, the solution process is complete.
4. Leave **2** as the **Percent Error**.

Conduction Solution Options

The DC conduction solver generates solutions only for the DC conduction paths in the magnetostatic problem. These paths show the direction of the flow of the field.

➤ To generate the DC conduction solution:

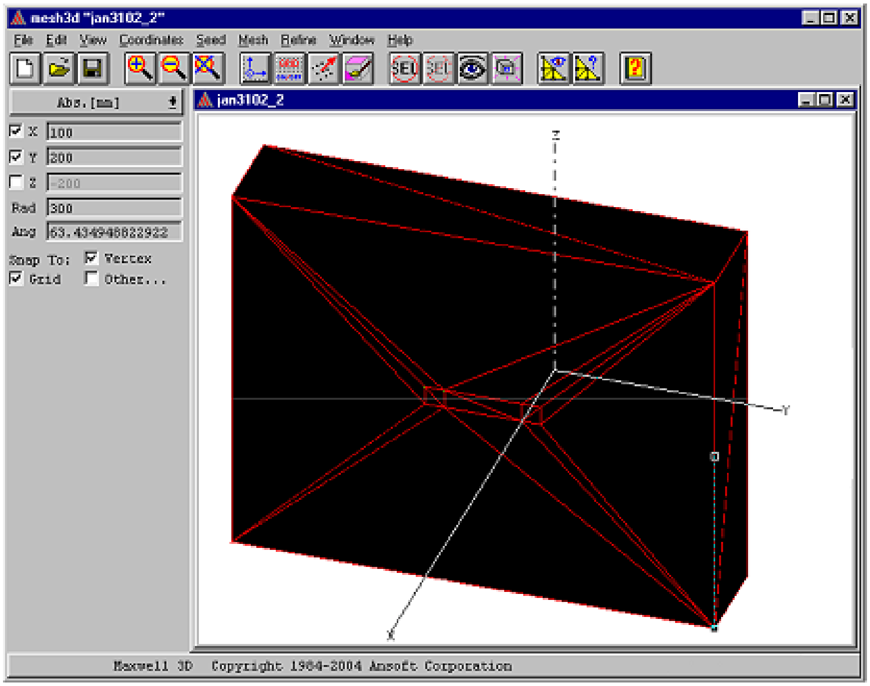
1. Leave **Conduction Solve** selected. Doing so instructs the system to compute the conduction solution iteratively.
2. Enter **50** for the **Percent refinement** per pass.
3. Enter **8** for the **Number of requested passes**.
4. Leave **2** as the **Percent Error**.

Viewing the Mesh

If you want to see the mesh that Maxwell will use to solve your project, you can now create and display it. The following procedure allows you to optionally display the mesh.

➤ To view the mesh:

1. Choose **Manual Mesh**. The Meshmaker appears.



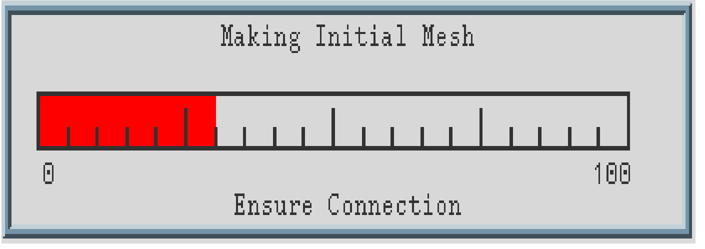
   This is the command you would use to change the mesh from the one assigned as a default. No mesh will be displayed until Maxwell generates one for this project for the first time.
2. Choose **Mesh/Make** from the menu. A progress bar appears, showing you the status of the mesh creation.
3. Choose **Mesh/Show Mesh** to display the mesh you have created.
4. Choose **File/Exit** to exit the Meshmaker.
5. Choose **Yes** when asked to save the changes to the mesh. If you do not save the mesh, the solver will recalculate the **Initial Mesh** in the solution. You return to the **Solve Setup** window.
6. Choose **OK** to accept the setup options and exit to the Executive Commands window.

Solve

Now that the problem is set up, you are ready to generate a solution for it.

➤ To solve for the forces in your problem:

- Choose **Solve/Nominal Problem** from the Executive Commands menu. A progress bar appears, telling you what is being solved for. **Solving Fields With Force** and **Refining Mesh** will be repeated several times.



Time:

This problem solved in approximately 16 minutes on a Hewlett-Packard 9000 workstation with 98 megabytes of RAM. Depending on the computing resources you have available, this solution time may vary.

Monitoring the Solution

While the solution is generating, you can observe the progress of the solution as it converges to meet the solution criteria. You can monitor the convergence of the force solution from the Executive Commands window.

Viewing Convergence Data

As the solver starts to generate the solution, you can observe the convergence of the solution.

➢ To view the convergence:

- Choose **Convergence**. The number of completed and remaining passes appear along with the convergence criteria. A **Magnetostatic** pull-down menu also appears. This menu allows you to display either **Magnetostatic** or **DC Conduction** solutions. Because you are concentrating on the magnetostatic solution and not the DC conduction paths in the problem, leave the default **Magnetostatic** selected.

Completing the Solution Process

When the solution is complete, a message appears telling you the solution process is complete.

Choose **OK** to return to the Executive Commands window.

Viewing the Final Convergence Data

Now that the solution has converged, return to the convergence window and observe the final solutions for the model. The energy error decreased on each successive pass until it converged to the percent error value specified as the solution criteria. At the same time that this error decreased, the number of tetrahedra in the mesh increased.

Plot the Number of Tetrahedra

You can now observe the convergence of the solution in a graphical format.

➤ To view the graphs:

- Choose **Convergence Display / Plot Tetrahedra** from the **Convergence Display** menu in the Executive Commands window. A plot of the tetrahedra versus pass appears. Notice how the number of tetrahedra increased during each pass. This shows how the tetrahedra were broken down and recalculated as the mesh was refined.

Plot the Percent Energy Error

Now observe the percent energy error. This is the target value at which the solution converges.

➤ To plot the percent energy error:

- Choose **Convergence Display/Plot Percent Energy Error**. A plot of the percent energy error versus pass appears. Notice how the error decreased with each pass until it intersected the dotted line at the bottom of the plot. The line represents the convergence value, which you specified when you set up the solution. The point of intersection shows the value at which the solution converged.

Viewing the DC Conduction Convergence

In the **Solve Setup** window, you specified the solution criteria for the DC conduction solution. During the solution process, the conduction solution was computed.

➤ To view the DC conduction convergence:

1. Choose **DC Conduction** from the **Convergence Data** pull-down menu. The convergence for the conduction solution appears.
2. Choose **Convergence Display/Table**.

The DC conduction solution appears.

Viewing the Statistics of the Solution

You can now view the profile of the solution to observe the computing resources that were used during the solution process.

➤ To view the profile of the data:

- Choose **Profile** from the Executive Commands window.

The **Profile Table** appears, displaying the **Real Time**, **CPU Time**, **Memory Size**, and **Num Elements** per pass.

- **Real Time** represents the taken by the solver to complete the pass in real time.
- **CPU Time** represents the time taken by the solver to actually compute the solution for the pass.
- **Memory Size** represents the amount of memory required to complete the pass.
- **Num Elements** represents the number o f tetrahedra required to compute the solution in that pass.

6

Analyze the Solution

Now that you have generated a solution for the forces on the model, you can use the Post Processor to analyze and plot the results.

The goals for this chapter are to:

- Examine the computed force values.
- Plot the magnetic fields in the model.
- Plot and modify a cutplane to display the magnetic fields in the model.
- Modify the display of the plot.

Time:

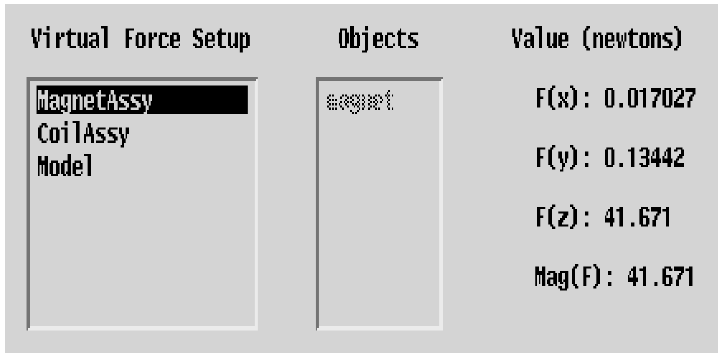
The total time needed to complete this chapter is approximately 45 minutes.

View Forces

Once you have a solution, you can use the **Solutions/Force** command in the Executive Commands window to examine the virtual forces.

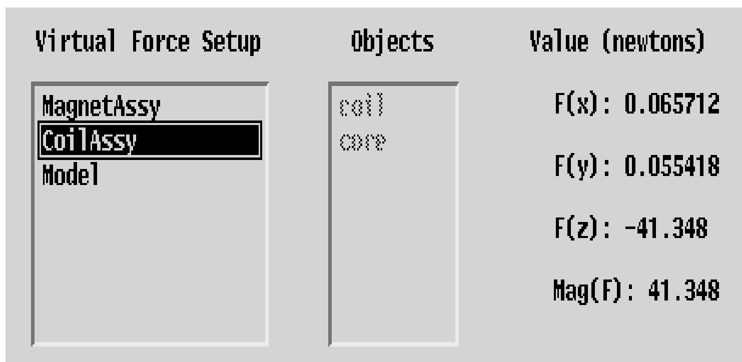
➤ To view the resulting virtual X, Y, and Z forces on the objects in the model:

1. Choose **Solutions/Force** at the top of the Executive Commands window. The following window appears for virtual force:



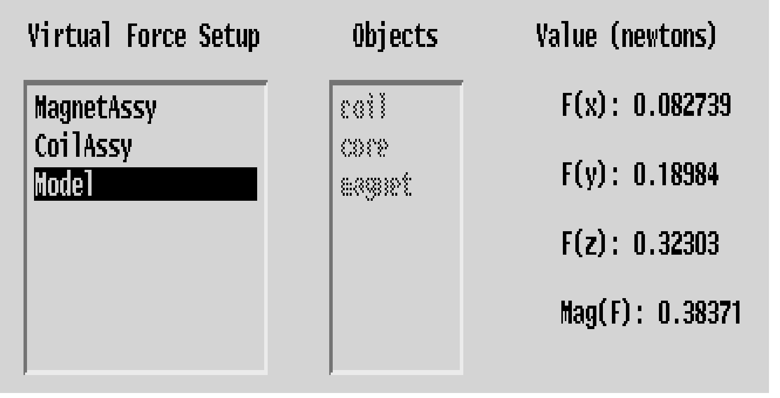
   The forces on each of the setups you defined in the Boundary Manager are displayed to the right of the **Objects** list. In this problem, the forces you are particularly interested in are those along the Z-axis. In this example, the force along the z-axis is approximately 41.6N. Depending on your machine's configurations, your results may differ slightly.

2. Now select **CoilAssy**. The following values appear:



   The value along the Z-axis is approximately -41.34 N. As you switch between setups, note that the **F(z)** values for **MagnetAssy** and **CoilAssy** are nearly equal and opposite.

3. Now select **Model** from the **Virtual Force Setup** list. The following values appear:



The forces along the Z-axis for the **Model** setup (which includes all objects in the model) add up to approximately zero. This indicates that the forces in the system are in equilibrium and that the solution has converged to a reasonably accurate result.

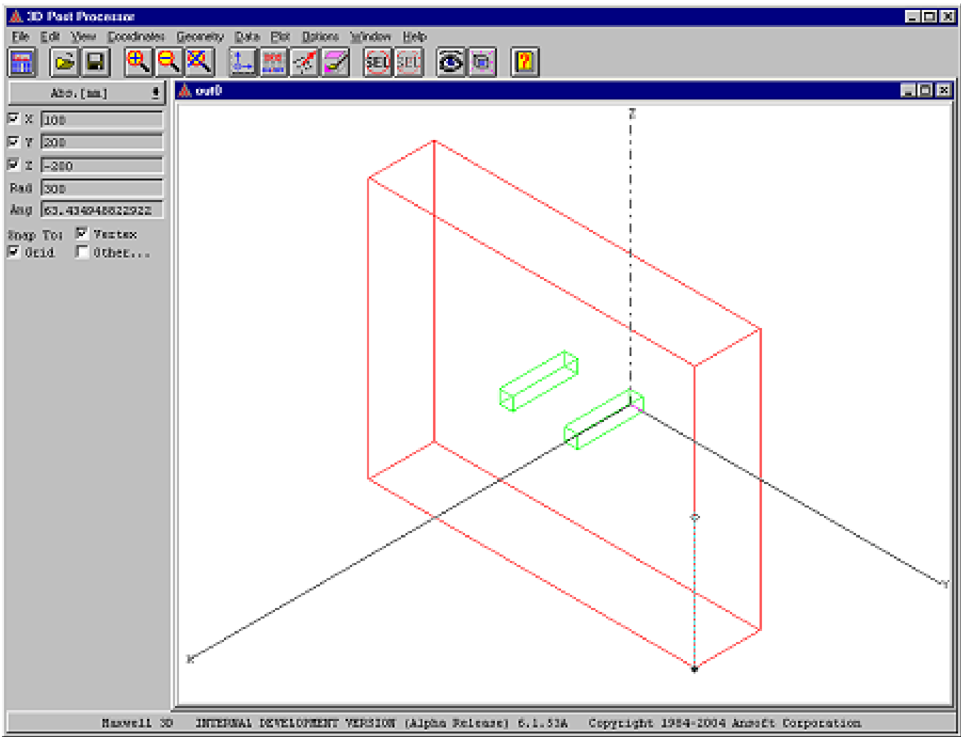
Access the Post Processor

The Post Processor allows you to plot the fields for which you have generated solutions. For this problem, you will use the Post Processor to create an arrow plot for the B-field and a magnitude plot of the field in two planes.

You can use the Post Processor to plot the results over the model.

➤ To start the Post Processor:

- Choose **Post Process/Nominal Problem** from the Executive Commands menu. The 3D Post Processor appears.



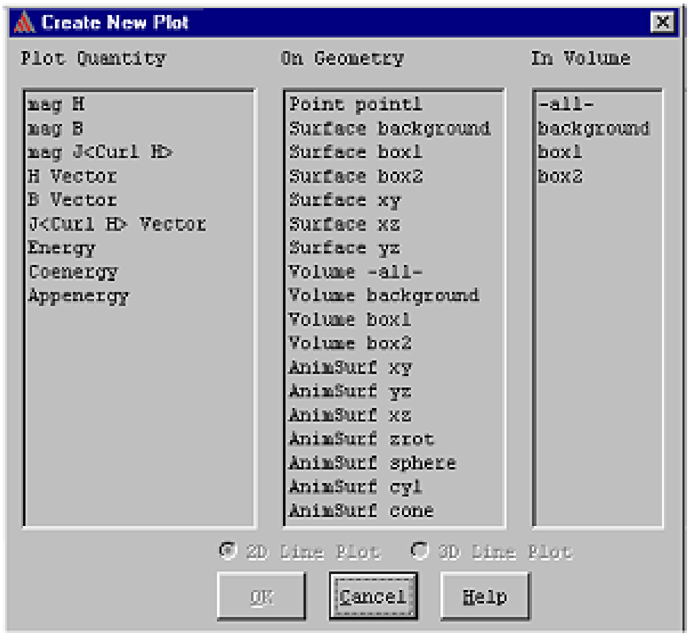
Create an Arrow Plot of the B-Field

Since you are interested in how the magnetic field changes as you move vertically in the model, you will want to look at the solution over the YZ plane.

Arrow plots indicate the path of the B-field in the model. In this case, you will create an arrow plot to show the direction of the B-field on the entire surface of the yz plane.

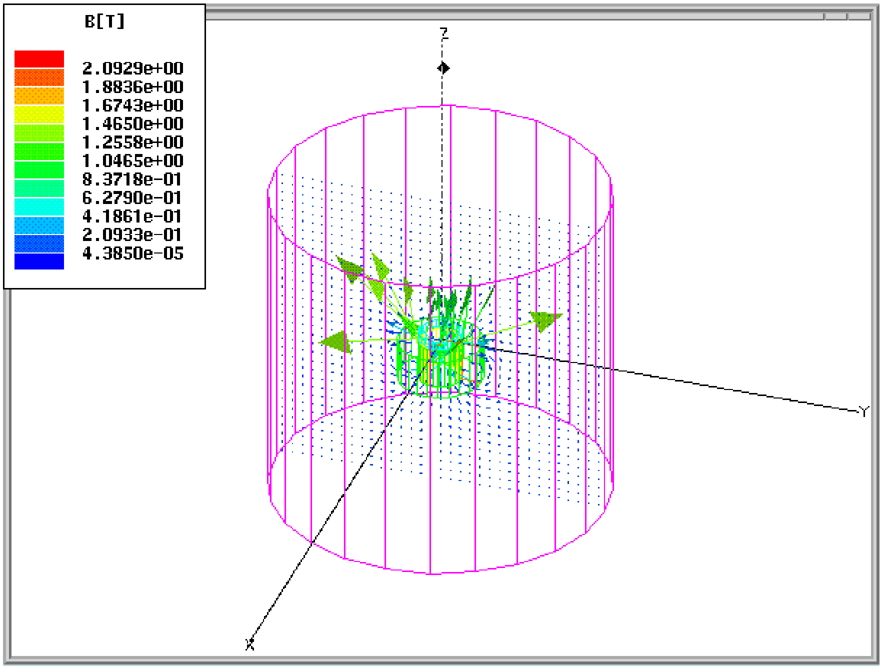
➤ To plot the field through the center of the model:

1. Choose **Plot/Field** from the menu. The **Create New Plot** window appears, prompting you to enter plot input information.



2. Choose **B Vector** from the **Plot Quantity** list. This tells the Post Processor what quantity to plot.
3. Choose **Surface yz** from the **On Geometry** list. **Surface yz** is the geometry on which you wish to plot the quantity.
4. Choose **-all-** from the **In Volume** list. This plots the B vectors on the yz plane, through all the objects in the model.
5. Choose **OK**. The input to the plot is accepted, and the **Vector Surface Plot** window appears, prompting you to enter the vector display information.
6. Under **Arrow Options**, select **2D** as the **Type** if it is not already selected.

7. Enter **6** in the **Spacing** field. This makes the arrows appear closer together in the plot.
8. Under **Plot Scale**, make sure the **Divisions** field is set to **11**. If it is not, enter **11** in the **Divisions** field. This determines how many divisions you wish to specify in the color key.
9. Leave all other values set to their defaults.
10. Choose **OK** to accept the values. The Post Processor calculates and displays the vector plot. The following vector plot appears, showing the direction of the flux lines:



Using the Post Processor, you can display several plots at once, comparing values in the same plane. For instance, you can display an arrow plot and a magnitude plot at the same time.

Change the View of the Plot

Now that the arrow plot is complete, you can change the view of the plot to observe the B-field more easily. To do this, zoom in on the model to see the vector field more easily, then return to the normal view using commands from the menu bar.

➢ To zoom in on the plot:

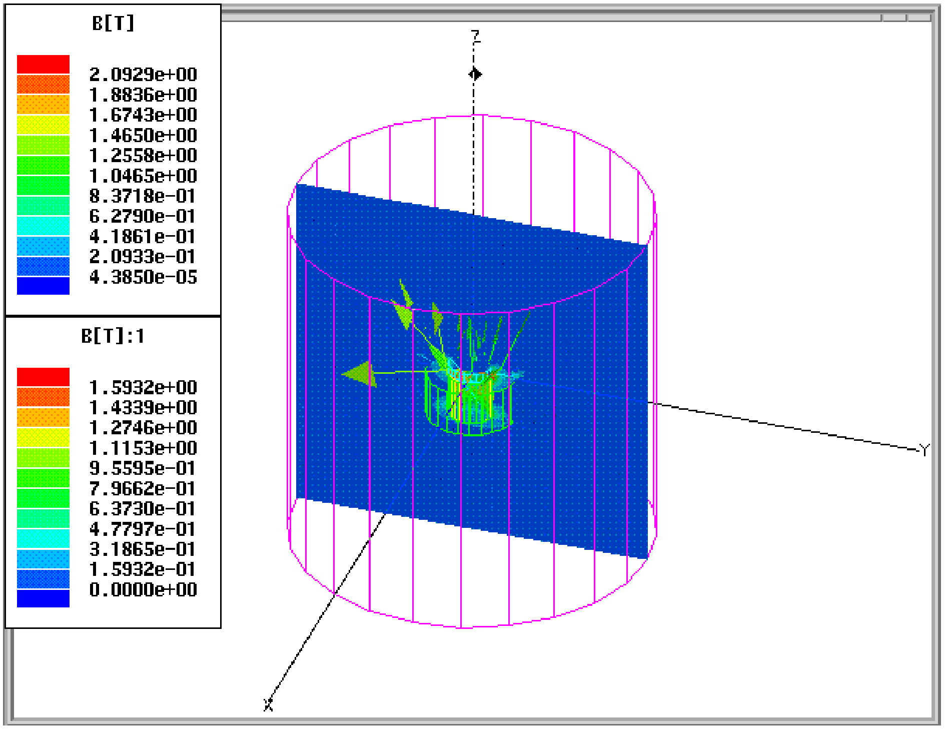
1. Choose **View/Zoom In** from the menu bar. The cursor changes to a set of crosshairs.
2. Click on a point in the upper-left corner of the region where the arrows of the B-field are visible. This marks the first vertex of the region that you will zoom in towards.
3. Click on a point in the lower-right corner of the region where the arrows of the B-field are visible. This marks the opposite vertex of the region that you will zoom in toward. When you have selected this point, the window will resize the arrows in the plot, enlarging the view.
4. Choose **View/Fit All/All Views** to return the model to its default view.

Plot the Magnitude of the B-Field

The arrow plot is still visible on the screen. What you will now do is overlay the arrow plot of the B-field with a plot of its magnitude. Thus, you will be able to see the direction of the B-field as it relates to its magnitude.

➤ To display the magnitude of the B-field:

1. Choose **Plot/Field**.
2. Select **mag B** from the **Plot Quantity** list. This time, instead of plotting the vectors of **B**, the Post Processor plots the magnitude of **B**.
3. Select **Surface yz** from the **On Geometry** list.
4. Select **-all-** from the **In Volume** list.
5. Choose **OK**. The **Scalar Surface Plot** window appears.
6. Ensure that the **Divisions** field is still **11**, and then choose **OK**. The Post Processor calculates and displays the new plot.
7. Click and drag the **B[T]:1** plot key window to the lower right corner of the window. Because two plots are superimposed over each other, you will need to change the placement of the plot keys so you can see them both.



8. Release the mouse button.

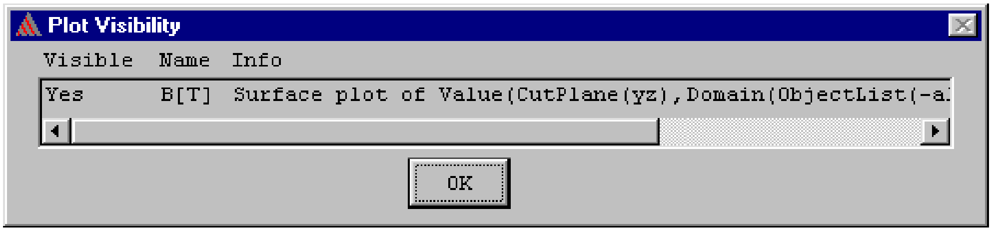
Above, the shaded area is the intensity of the B-field. This shows the strength of the field in the YZ plane.

Turning Off Displayed Plots

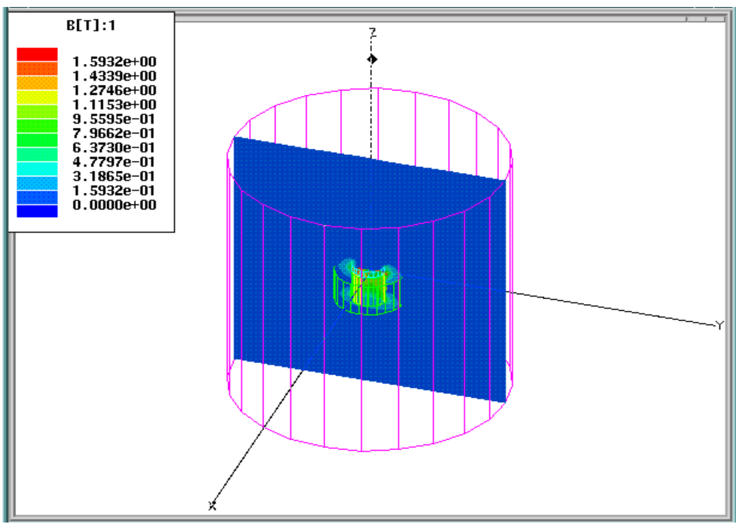
You may decide that you do not want more than one plot displayed at a time. For instance, you might want to emphasize the magnitude plot without being distracted by the vector arrows. Now, you can turn the plot visibilities on and off without losing plot information.

➤ To turn off the vector plot:

1. Choose **Plot/Visibility**. The **Plot Visibility** window appears.



2. Select **B[T]** (the vector plot, which you created first) to toggle the visibility to **No**.
3. Choose **OK** to set the visibility. Only the shaded plot of the B magnitudes is displayed. You can zoom in to view the plot in more detail around the magnet.
4. Move the color key into the upper left corner of the window.



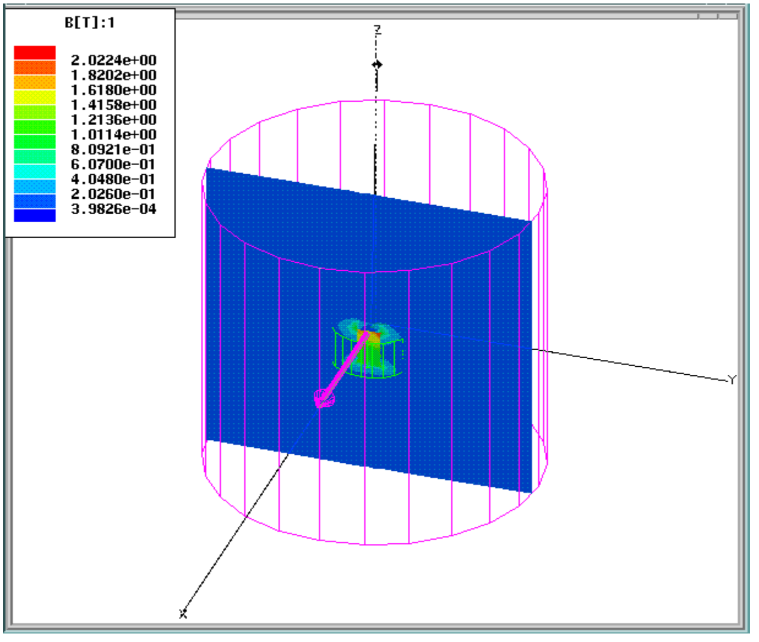
You can also plot values other than those in the planes that intersect the origin; such planes are special instances of cutplanes. You can define new cutplanes using the Post Processor.

Redisplay the Shaded Plot on a Cutplane

You can observe the magnitude of B on a plane other than the original surface.

➤ To examine the magnitude of B in a different plane:

1. Choose **Geometry/Modify/Cutplane** to define the cutplane. The cutplane menu appears in the side window.
2. Choose **yz** to modify the vertical plot. The displayed cutplane changes orientation.
3. Click on **>** to change the position of the cutplane. The cutplane moves forward along the X-axis. You can now display values along this plane.
4. Choose **Recalculate** to calculate the values for the new cutplane. After recalculating the values, the Post Processor changes the display.



5. Choose **Done** to exit the cutplane window.

Exit the Post Processor

Your project is complete. You are now ready to exit the Post Processor.

➤ To exit the Post Processor:

- Choose **File/Exit**. Remember to save any changes you want to keep in your project.

Exit Maxwell 3D

Close your project and exit Maxwell 3D.

➤ To exit Maxwell 3D:

1. Choose **Exit** from the Executive commands screen. You return to the Projects Manager.
2. Choose **Exit**. You return to the Maxwell Control Panel.
3. Choose **Exit** from the Control Panel to exit the software.

Index

Numerics

A

B

C

D

E

F

T

U

V

W

Z