



**Maxwell[®] 2D
Student Version**

Getting Started:

A 2D Magnetostatic Problem

November 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.

This document contains proprietary information which is protected by copyright. All rights are reserved.

Ansoft Corporation

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

Microsoft® Windows® is a registered trademark of Microsoft Corporation.

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

Maxwell® is a registered trademark of Ansoft Corporation.

Printing History

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 which are incorporated at reprint do not cause the date to change.

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

Edition	Date	Software Revision
1	May 2001	8.1
2	November 2002	9.0

Installation

Before you use Maxwell SV, you must:

1. Set up your system's graphical windowing system.
2. Install the Ansoft software, using the directions in the *Ansoft PC 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 assistance.

Getting Started

If you are using Maxwell SV for the first time, the following two guides are available for the Student Version of Maxwell 2D:

- *Getting Started: A 2D Electrostatic Problem*
- *Getting Started: A 2D Magnetostatic Problem*

Additional Getting Started guides are available for the standard version of Maxwell 2D.

These short tutorials guide you through the process of setting up and solving simple problems in Maxwell SV, providing you with an overview of how to use the software.

Other References

To start Maxwell SV, you must first access the Maxwell Control Panel.

For information on all Maxwell Control Panel and Maxwell SV commands, refer to the following online documentation:

- *Maxwell Control Panel online help*. The Control panel online help contains a detailed description of all of the commands in the Maxwell Control Panel and in the Utilities Panel. The Maxwell Control Panel allows you to create and open projects, print screens, and translate files. The Utilities Panel is accessible through the Maxwell Control Panel and enables you to view licensing information, adjust colors, open and create 2D models, open and create plots using parametric equations, and evaluate mathematical expressions.
- *Maxwell 2D online help*. The Maxwell 2D online help contains a detailed description of the Maxwell 2D software and the Parametric Analysis module. Maxwell SV does not provide parametric capabilities.

Typeface Conventions

Field Names	Bold type is used for on-screen prompts, field names, and messages.
Keyboard Entries	<p>Bold type is used for entries that must be entered as specified.</p> <p>Example: Enter 0.005 in the Nonlinear Residual field.</p>
Menu Commands	<p>Bold type is used to display menu commands selected to perform a specific task. Menu levels are separated by a carat.</p> <p>Example 1: The instruction “Click File>Open” means to select the Open command on the File menu within an application.</p> <p>Example 2: The instruction “Click Define Model>Draw Model” means to select the Draw Model command from the Define Model button on the Maxwell SV Executive Commands menu.</p>
<i>Variable Names</i>	<p>Italic type is used for keyboard entries when a name or variable must be typed in place of the words in italics.</p> <p>Example: The instruction “copy <i>filename</i>” means to type the word copy, to type a space, and then to type the name of a file, such as file1.</p>
<i>Emphasis and Titles</i>	Italic type is used for emphasis and for the titles of manuals and other publications.
Keyboard Keys	<p>Bold type, in Arial font, is used for labeled keys on the computer keyboard.</p> <p>Example: The instruction “Press Return” means to press the key on the computer that is labeled Return.</p>

Getting Started: A 2D Magnetostatic Problem

Table of Contents

1.	Introduction	1-1
	The Sample Problem	1-2
	General Procedure for Creating and Solving a 2D Model	1-3
	Goals	1-5
2.	Creating the Project	2-1
	Access the Control Panel	2-2
	Start the Project Manager	2-2
	Create a Project Directory	2-3
	Create a New Project	2-4
	Access the Project Directory	2-4
	Add the New Project	2-4
	Save Project Notes	2-5
	Open the New Project and Run Maxwell SV	2-6
	Overview of the Executive Commands Window	2-7
	Executive Commands Menu	2-7
	Display Area	2-7
3.	Creating the Model	3-1
	Define Model Type	3-2
	Select the Solver Type	3-2
	Define the Drawing Plane	3-2
	Access the 2D Modeler	3-3
	Screen Layout	3-3
	The Solenoid Model	3-5

Getting Started: A 2D Magnetostatic Problem

Set Up the Drawing Region	3-6
Change the Drawing Size	3-7
Change Grid Spacing	3-8
Create the Geometry	3-9
Draw the Plugnut	3-9
Create a Rectangle	3-9
Change the Plugnut's Name and Color	3-10
Keyboard Entry	3-10
Draw the Core	3-11
Draw the Coil	3-12
Save the Geometry	3-13
Draw the Yoke	3-13
Draw the Bonnet	3-14
Completed Geometry	3-14
Exit the 2D Modeler	3-15
4. Setting Up the Model	4-1
Assign Materials to Objects	4-2
Access Material Manager	4-2
Assign Copper to the Coil	4-3
Assign Cold Rolled Steel to the Bonnet and Yoke	4-3
Assign Neo35 to the Core	4-7
Assign SS430 to the Plugnut	4-10
Accept Default Material for Background	4-12
Exit the Material Manager	4-12
Set Up Boundaries and Current Sources	4-12
Access the 2D Boundary/Source Manager	4-13
Boundary Manager Screen Layout	4-13
Types of Boundary Conditions and Sources	4-14
Set Source Current on the Coil	4-14
Assign Balloon Boundary to the Background	4-15
Exit the Boundary Manager	4-16
Set Up Force Computation	4-16
5. Generating a Solution	5-1
View Solution Criteria	5-2
Starting Mesh	5-2
Manual Mesh	5-2
Solver Choice	5-3
Solver Residual	5-3

Solve for Fields and Parameters	5-3
Adaptive Analysis	5-3
Exit Setup Solution Options	5-3
Start the Solution	5-4
Monitoring the Solution	5-4
Viewing Convergence Data	5-5
Solution Criteria	5-5
Completed Solutions	5-6
Completing the Solution Process	5-6
Viewing Final Convergence Data	5-7
Plotting Convergence Data	5-8
Viewing Statistics	5-9
6. Analyzing the Solution	6-1
View Force Solution	6-2
Access the Post Processor	6-2
Post Processor Screen Layout	6-3
General Areas	6-3
Plot the Magnetic Field	6-3
Redisplay the Magnetic Field Plot	6-5
Identify Saturated Materials	6-6
Plot Magnetic Flux Density	6-6
Plot the B-H Curve	6-8
Exit the Post Processor	6-9
Exit Maxwell SV	6-9
Exit the Maxwell Software	6-9

Introduction

This guide is a tutorial for setting up a magnetostatic problem using version 9.0 of Maxwell 2D Student Version (SV), a software package for analyzing electromagnetic fields in cross-sections of structures. Maxwell SV uses finite element analysis (FEA) to solve two-dimensional (2D) electromagnetic problems.

To analyze a problem, you need to specify the appropriate geometry, material properties, and excitations for a device or system of devices. The Maxwell software then does the following:

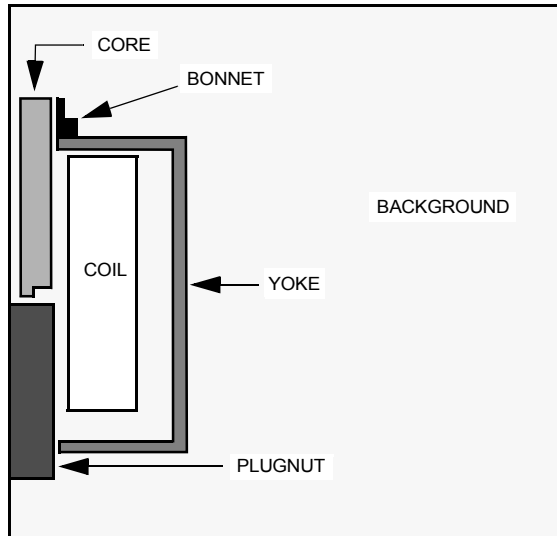
- Automatically creates the required finite element mesh.
- Calculates the desired electric or magnetic field solution and special quantities of interest, such as force, torque, inductance, capacitance, or power loss. You can select any of the following solution types: Electrostatic, Magnetostatic, Electrostatic, Eddy Current, DC Conduction, AC Conduction, Eddy Axial. The student version does not contain thermal, transient, or parametric capabilities.
- Allows you to analyze, manipulate, and display field solutions.

Many models are actually a collection of three-dimensional (3D) objects. Maxwell SV analyzes a 2D cross-section of the model, then generates a solution for that cross-section, using FEA to solve the problem. Dividing a structure into many smaller regions (finite elements) allows the system to compute the field solution separately in each element. The smaller the elements, the more accurate the final solution.

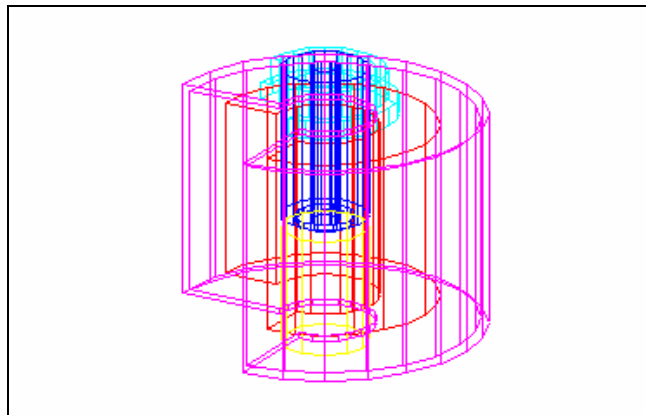
The Sample Problem

The sample problem, shown below, is a solenoid that consists of the following objects:

- Core
- Bonnet
- Coil
- Yoke
- Plugnut



The 2D diagram above actually represents a 3D structure that has been revolved around an axis of symmetry, as shown in the figure below. In this figure, part of the 3D model has been cut away so that you can see the interior of the solenoid.



Since the cross-section of the solenoid is constant, it can be modeled as an axisymmetric model in Maxwell SV.

General Procedure for Creating and Solving a 2D Model

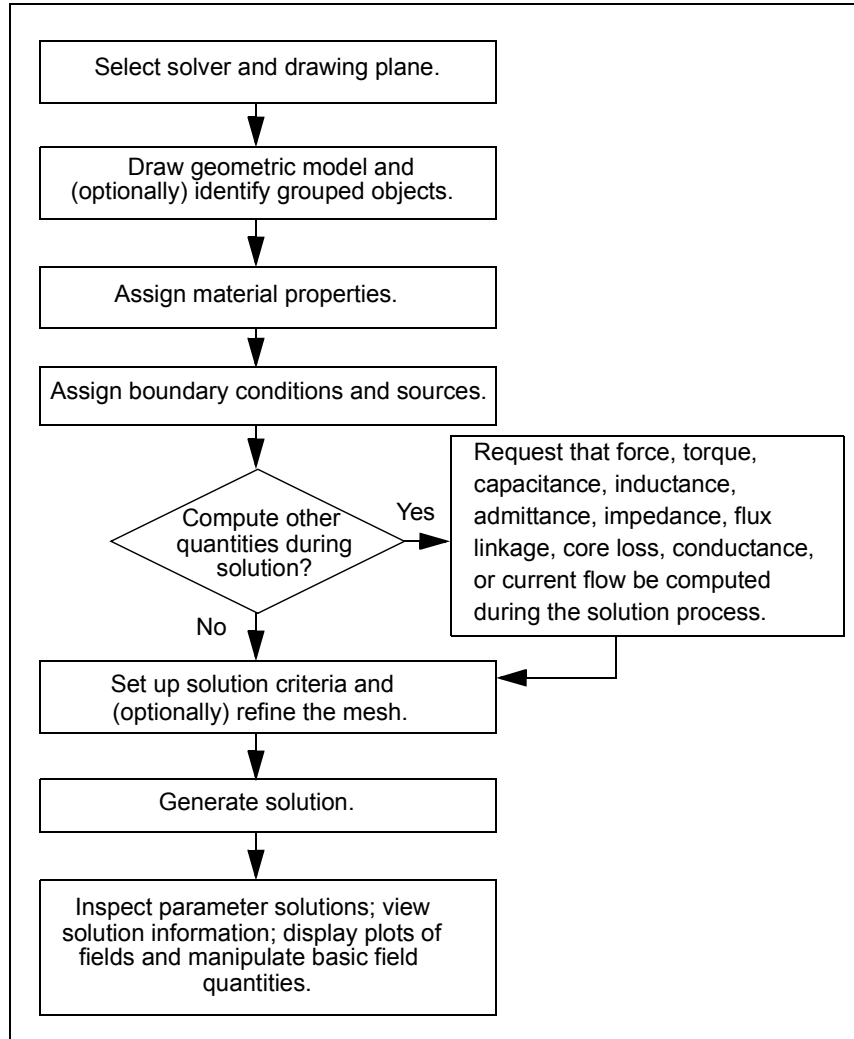
The general procedure to follow when using the software to create and solve a 2D problem is given below:

1. Use the **Solver** command to specify which of the following electric or magnetic field quantities to compute:
 - **Electrostatic**
 - **Magnetostatic**
 - **Eddy Current**
 - **DC Conduction**
 - **AC Conduction**
 - **Eddy Axial**
2. Use the **Drawing** command to select one of the following model planes:
 - XY Plane** Cartesian models appear to sweep perpendicularly to the cross-section.
 - RZ Plane** Axisymmetric models appear to revolve around an axis of symmetry in the cross-section.
3. Use the **Define Model** commands to create the geometric model. When you click **Define Model**, the following menu appears:
 - Draw Model** Allows you to access the 2D Modeler and build the objects that make up the geometric model.
 - Group Objects** Allows you to group discrete objects that are actually one electrical object. For instance, two terminations of a conductor that are drawn as separate objects in the cross-section can be grouped to represent one conductor.
4. Use the **Setup Materials** command to assign materials to all objects in the geometric model.
5. Use the **Setup Boundaries/Sources** command to define the boundaries and sources for the problem. This determines the electromagnetic excitations and field behavior for the model.
6. Use the **Setup Executive Parameters** command to instruct the simulator to compute one or more of the following special quantities during the solution process:
 - Matrix (capacitance, inductance, admittance, impedance, or conductance matrix, depending on the selected solver).
 - Force
 - Torque
 - Flux lines
 - Post Processor macros
 - Core loss
 - Current flow
7. Use the **Setup Solution Options** command to enter parameters that affect how the solution is computed.

Getting Started: A 2D Magnetostatic Problem

8. Use the **Solve** command to solve for the appropriate field quantities.
9. Use the **Post Process** command to analyze the solution.

These commands must be chosen in the sequence in which they appear. You must first create a geometric model using the **Define Model** command before you specify material characteristics for objects using the **Setup Materials** command. A check mark appears on the menu next to each step which has been completed.



Goals

The goals in *Getting Started: A 2D Magnetostatic Problem* are as follows:

- Determine the force on the core due to the source current in the coil.
- Determine whether any of the nonlinear materials reach their saturation point.

Do the following to accomplish these goals:

1. Draw the pluggnut, core, coil, yoke, and bonnet using the **Draw Model** command under **Define Model**.
2. Define materials using the **Setup Materials** command.
3. Define boundary conditions and current sources using the **Setup Boundaries/Sources** command.
4. Request that the force on the core be computed, using the **Setup Executive Parameters** command.
5. Specify solution criteria and generate a solution using the **Setup Solution Parameters** and **Solve** commands. You will compute both a magnetostatic field solution and the force on the core.
6. View the results of the force computation, using the **Solutions>Force** command.
7. Plot saturation levels and contours of equal magnetic potential via the **Post Process** command.

This simple problem illustrates the most commonly used features of Maxwell SV.

Time This guide should take approximately 3 hours to work through.

Creating the Project

This guide assumes that Maxwell SV has already been installed as described in the Ansoft *PC Installation Guide*.

The goals for this chapter are to:

- Create a project directory in which to save sample problems.
- Create a new project in that directory in which to save the solenoid problem.
- Open the project and run Maxwell SV.

This chapter also includes a description of the sample problem and your goals for solving it.

Time This chapter should take approximately 15 minutes to work through.

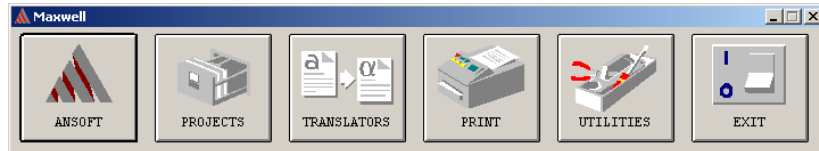
Access the Control Panel

The Maxwell Control Panel allows you to create and open projects and directly access program modules shared by Ansoft products. You must start the Control Panel in order to start Maxwell SV.

To start the Maxwell Control Panel, do one of the following:

- Use the **Start** menu to select **Programs>Ansoft>Maxwell SV**.
- Double-click the **Maxwell SV** icon.

The Maxwell Control Panel appears.

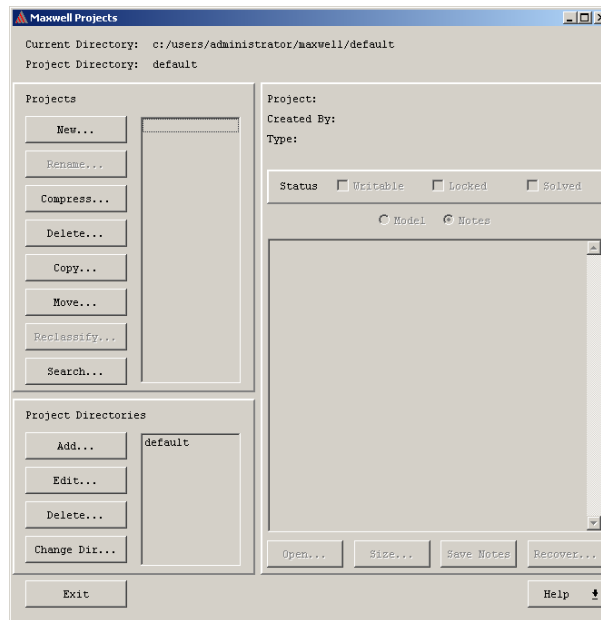


If the Maxwell Control Panel does not appear, see the Ansoft installation guides for possible reasons. See the online help on the Maxwell Control Panel for a detailed description of other control panel options.

Start the Project Manager

Use the Project Manager to create, rename, or delete project files and to access projects created with other Ansoft products.

To display the Project Manager, click **PROJECTS** in the Maxwell Control Panel. The Project Manager appears.



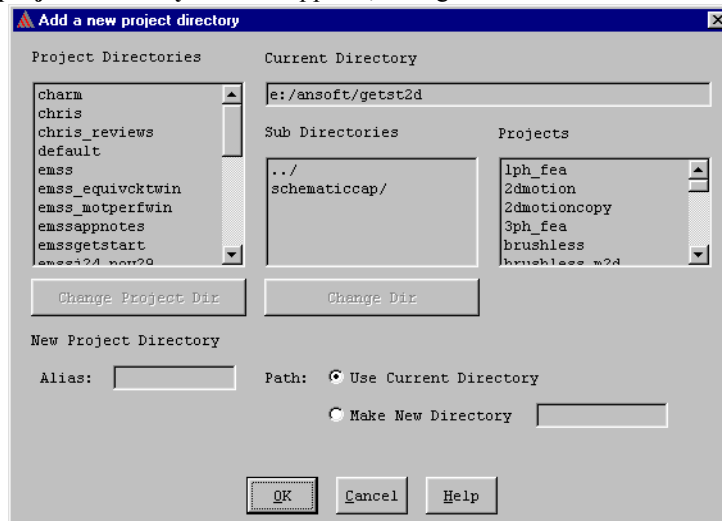
Create a Project Directory

A project directory contains a specific set of projects created with Ansoft software. You can use project directories to categorize projects. For example, you might want to store all projects related to a particular application in one project directory. You will add the **getting_started_SV** directory for the Maxwell project you create using this *Getting Started* guide.

Note If you have already created a project directory while working through another *Getting Started* guide, such as *Getting Started: An Electrostatic Problem*, use the same directory and skip to the “Create the Solenoid Project” section.

To add a project directory, do the following in the Project Manager:

1. Click **Add** from the **Project Directories** area at the lower-left corner of the window. The **Add a new project directory** window appears, listing the directories and subdirectories.



2. Type the following in the **Alias** field:

getting_started_SV

Maxwell SV projects are usually created in directories which have aliases — that is, directories that have been identified as project directories using the **Add** command.

3. Select **Make New Directory** near the bottom of the window. The name you entered as the alias automatically appears in the field beside this option.
4. Click **OK**.

You return to the **Project Manager** menu. The **getting_started_SV** directory is created under the current default project directory, and **getting_started_SV** now appears in the **Project Directories** list.

Create a New Project

You are now ready to create a new project named **solenoid** in the **getting_started_SV** project directory.

Access the Project Directory

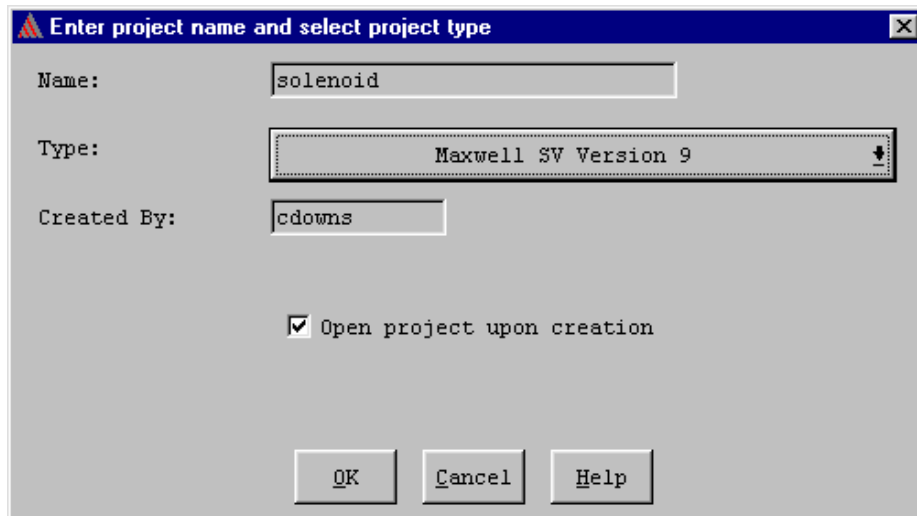
Before you create the new project, select **getting_started_SV** from the **Project Directories** list at the bottom left of the menu.

The current directory displayed at the top of the **Project Manager** menu changes to show the path name of the directory associated with the **getting_started_SV** alias. If you have previously created the microstrip model, it will be listed in the **Projects** box. Otherwise, the **Projects** box is empty — no projects have been created yet in the **getting_started_SV** project directory.

Add the New Project

To add the new project:

1. Click **New** in the **Projects** box at the top left of the menu. The **Enter project name and select project type** window appears.



2. Enter **solenoid** in the **Name** field.
3. Select the type of project to be created.
 - a. To display all available project types, click the button next to the **Type** field. A menu appears, listing all Ansoft software packages that have been installed.
 - b. Click **Maxwell SV Version 9**.
4. Optionally, enter your name in the **Created By** field. The name of the person who logged onto the system appears by default.
5. Deselect **Open Project Upon Creation**. You do not want to automatically open the project at this point, so that you may enter project notes first.
6. Click **OK**. The information you just entered appears in the corresponding fields in the **Selected Project** box. Because you created the project, **Writeable** is selected, showing that you have access to the project.

Save Project Notes

It is a good idea to save a description of your new project so that next time you can view information about the project without opening it.

To enter a description of the project:

1. Leave the **Notes** radio button selected; this is the default.
2. Click in the **Notes** area. The cursor changes to an I-beam in the upper-left corner of the **Notes** box, indicating that you can begin typing text.
3. Enter a description of the project.

For example, enter the following text:

```
This sample magnetostatic problem was created using Maxwell  
SV and the magnetostatic Getting Started guide.
```

After you begin entering text, the text of the **Save Notes** button that appears below the **Notes** box now becomes black, indicating that it is enabled. Before you began typing in the **Notes** box, **Save Notes** was grayed out, or disabled.

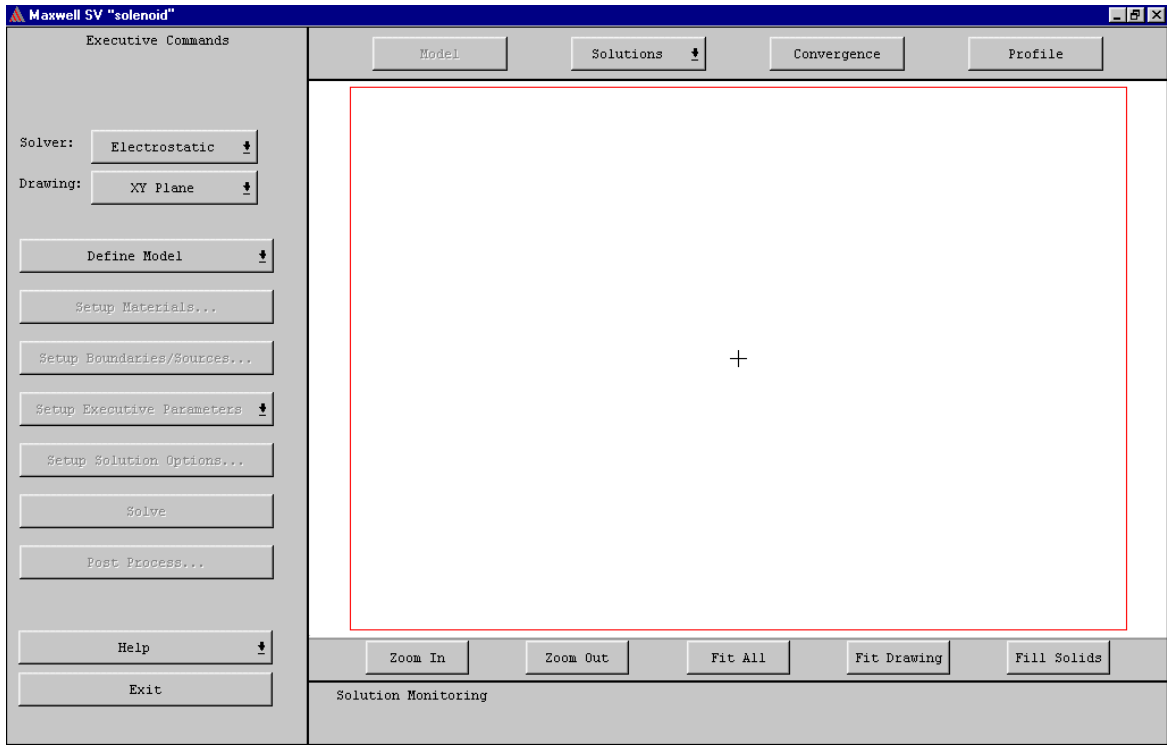
4. When you are done entering the description, click **Save Notes**. After you do, the **Save Notes** button is disabled again.

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

Open the New Project and Run Maxwell SV

The newly created project **solenoid** should still be selected in the **Projects** list. (If it is not, simply select it.) Now you are ready to run Maxwell SV.

To start Maxwell SV, click **Open** in the **Project** window. The **Executive Commands** menu of Maxwell SV appears.



Overview of the Executive Commands Window

The **Executive Commands** window acts as a path to each step of creating and solving the model problem. You select each module through the **Executive Commands** menu, and the software brings you back to this menu when you are finished. You also view the solution process through this menu.

The **Executive Commands** window is divided into two major sections: the **Executive Commands** menu and a Display area.

Executive Commands Menu

The **Executive Commands** menu on the left side of the screen let you define the type of problem you are solving and then call up the various modules you will need to create and solve the problem. The function of each button is explained under “General Procedure for Creating and Solving a 2D Model” on page 1-3.

Display Area

The display area shows the project’s geometry. Since you have not yet drawn the model, this area is blank. The commands along the bottom of the window allow you to change your view of it as follows:

Zoom In	Zooms in on an area of the window, magnifying the view.
Zoom Out	Zooms out from an area, shrinking the view.
Fit All	Changes the view to display all items in the window. Items will appear as large as possible without extending beyond the window.
Fit Drawing	Displays the entire drawing space.
Fill Solids	Displays objects as solids rather than outlined objects. Toggles with Wire Frame .
Wire Frame	Displays objects as wire-frame outlines. Toggles with Fill Solids .

This window can also display solution profile and convergence information while the problem is being solved, as detailed in Chapter 6, “Analyzing the Solution.”

Creating the Model

In the last chapter, you created and opened the **solenoid** project and reviewed the general procedure for drawing, setting up, and solving 2D problems. Now you are ready to use Maxwell SV to solve the solenoid problem. This chapter shows you how to select the type of field solution that's computed and create the geometry for the solenoid problem that was described in Chapter 1, "Introduction."

The goals for this chapter are to:

- Define the type of model — which includes selecting the solver type and the drawing plane.
- Set up the problem region.
- Draw the objects that make up the solenoid model.
- Save the solenoid model.

You are now ready to start using the simulator.

Time This chapter should take approximately 30 minutes to work through.

Define Model Type

Before you start drawing the model, you need to specify which field quantities to compute and whether the model will be cartesian (XY plane) or axisymmetric (RZ plane).

Select the Solver Type

Maxwell SV can compute the following types of field quantities:

- Electrostatic** Static electric fields.
- Magnetostatic** Static magnetic fields.
- Eddy Current** Time-varying magnetic fields and eddy currents.
- DC Conduction** Conduction currents caused by DC voltage differentials.
- AC Conduction** Conduction currents caused by AC voltage differentials.
- Eddy Axial** Eddy currents induced by time-varying magnetic fields.

For the solenoid problem, you will compute the static magnetic fields in the device and find the force on the core due to these fields. Therefore, select **Magnetostatic** as the field solver.

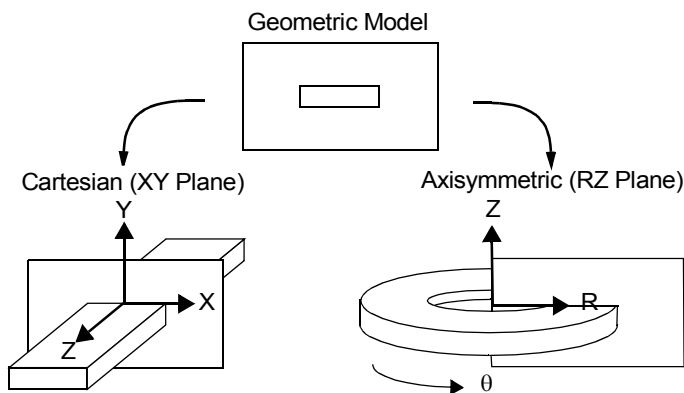
To select the **Magnetostatic** field solver, click **Solver>Magnetostatic** on the **Executive Commands** menu.

Define the Drawing Plane

In Maxwell 2D and SV, you can choose from two different types of geometric models:

- **Cartesian.** Visualize a cartesian (XY plane) geometry as a rectangle extending perpendicular to the plane being modeled.
- **Axisymmetric.** Visualize an axisymmetric (RZ plane) geometry as a rectangle revolving around an axis of symmetry, Z.

Both types of geometry are illustrated below.



Since it represents a device that is revolved around an axis of symmetry, the solenoid will be drawn in the RZ plane.

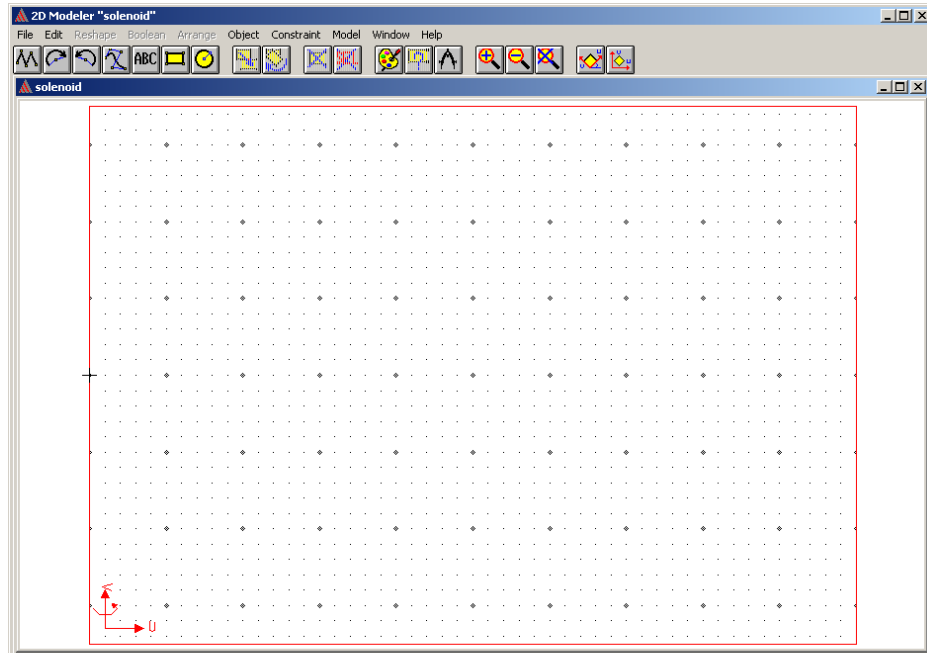
To define the drawing plane, click **Drawing>RZ Plane**.

Access the 2D Modeler

Use the 2D Modeler to create the objects in the model.

To access the 2D Modeler:

- Click **Define Model>Draw Model** from the **Executive Commands** window. The 2D Modeler appears.



Screen Layout

The following sections provide a brief overview of the 2D Modeler.

Menu Bar

The 2D Modeler's menu bar appears at the top of the window. Each item in the menu bar has a menu of commands associated with it. If a command name has an arrow next to it, that command also has a menu of commands associated with it. If a command name has an ellipsis next to it, that command has a window or panel associated with it.

To display the menu associated with a command in the menu bar, do one of the following:

- Click on it.
- Press and hold down the **Alt**, and then press the key of the underlined letter on the command name.

For example, to display the **Window** command's menu, do one of the following:

- Click on it.
- Press the **Alt** key, and type a **W**.

After you click **Window**, the **Window** menu appears. Click outside the **Window** menu to make it disappear.

Project Window

The main **2D Modeler** window contains the Drawing Region, the grid-covered area where you draw the objects that make up your model. This main window in the 2D Modeler is called the project window. A project window contains the geometry for a specific project and displays the project's name in its title bar. By default, one subwindow is contained within the project window. Optionally, you can open additional projects in the 2D Modeler using the **File>Open** command. Opening several projects at once is useful if you want to copy objects between geometries. See the Maxwell 2D online help for more details.

Subwindows

Subwindows are the windows in which you create the objects that make up the geometric model. By default, this window:

- Has points specified in relation to a local uv coordinate system. The u-axis is horizontal, the v-axis is vertical, and the origin is marked by a cross in the middle of the screen.
- Uses millimeters as the default drawing unit.
- Has grid points 2 millimeters apart. The default window size is 100 millimeters by 70 millimeters.

Note If a geometry is complex, you may want to open additional subwindows for the same project so that you can alter your view of the geometry from one subwindow to the next. To do so, use the **Window>New** command as described in the Maxwell 2D online help. For this geometry, however, a single subwindow should be sufficient.

Status Bar

The status bar, which appears at the bottom of the 2D Modeler screen, displays the following:

U and V Displays the coordinates of the mouse's position on the screen and allows you to enter coordinates using the keyboard.

UNITS Drawing units in which the geometry is entered.

SNAPTO Which point — grid point or object vertex — is selected when you choose points using the mouse. By default, both grid and object points are selected so that the mouse snaps to whichever one is closest.

Message Bar

A message bar, which appears above the status bar, displays the functions of the left and right mouse buttons. When selecting or deselecting objects, the message bar displays the number of items that are currently selected. When you change the view in a subwindow, it displays the current magnification level.

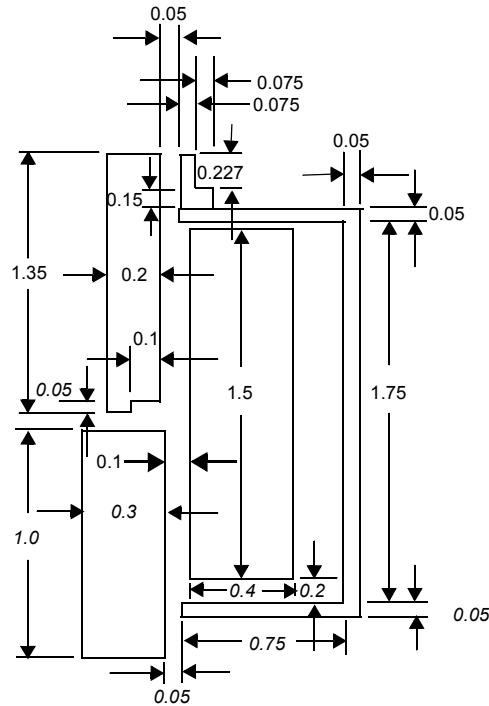
Toolbar

The toolbar is the vertical stack of icons that appears on the left side of the 2D Modeler screen. Icons give you easy access to the most frequently used commands that allow you to draw objects, change your view of the problem region, deselect objects, and so on. Click an icon and hold the button down to display a brief description of the command in the message bar at the bottom of the screen.

For more information on these areas of the 2D Modeler, refer to the Maxwell 2D online help.

The Solenoid Model

The dimensions of the solenoid that you will be modeling are shown below. Use these dimensions, which are given in inches, to create the geometric model.



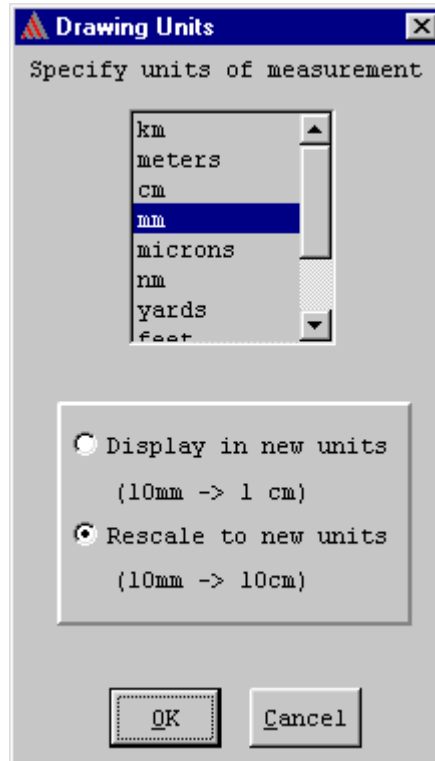
The following pages step you through drawing the above model.

Set Up the Drawing Region

The first step in using the 2D Modeler is to specify the drawing units to use in creating the model. In this case, you must change the units from millimeters to inches.

To set up the drawing region:

1. Click **Model>Drawing Units** from the menu bar. The **Drawing Units** window appears.



2. Select **Inches**.
3. Click the **Rescale to new units** radio button so the drawing area size will be converted to inches.
4. Click **OK**.

The units are changed from millimeters to inches, and **inches** now appears next to **UNITS** in the status bar.

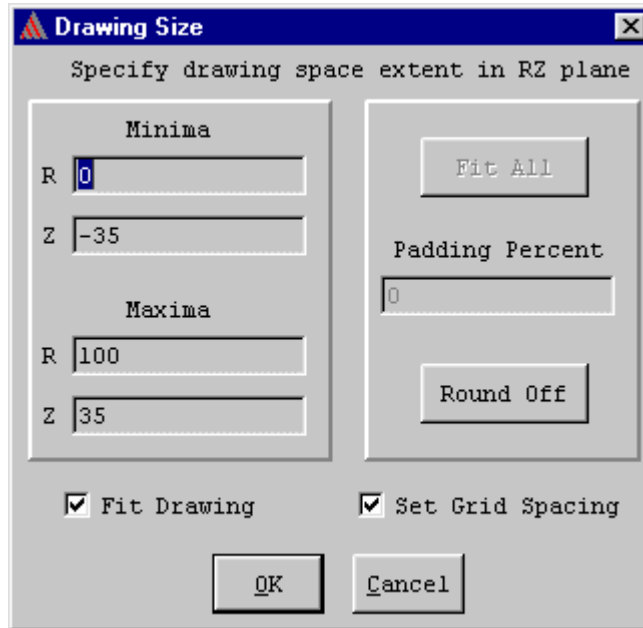
Note Throughout the rest of this guide, all commands will be given as **Menu>Command**. For example, the command you just used to change the drawing units would be given as: Click **Model>Drawing Units**.

Change the Drawing Size

Because the size of the solenoid model is relatively small compared to the area now represented on the screen, you will need to reduce the size of the drawing space.

To change the drawing size:

1. Click **Model>Drawing Size**. The **Drawing Size** window appears.



2. The **Minima** fields set the coordinates for the lower-left corner of the drawing. Change the **Minima** coordinates to **(0,−4.5)** by doing the following:
 - a. Leave the **Minima R** value set to **0**.

Note Because the simulator assumes that the Z-axis (the left edge of the drawing region) represents the axis of rotational symmetry in an axisymmetric model, you cannot enter a value for R that is less than zero.

- b. Double-click the left mouse button in the **Minima Z** field.
 - c. Type **−4.5**.
3. The **Maxima** fields set the coordinates for the upper-right corner of the drawing. Change the **Maxima** coordinates, using the procedure you just used to change the **Minima** coordinates:
 - a. Change the **Maxima R** value to **3**.
 - b. Change the **Maxima Z** value to **4.5**.
 4. Click **OK**.

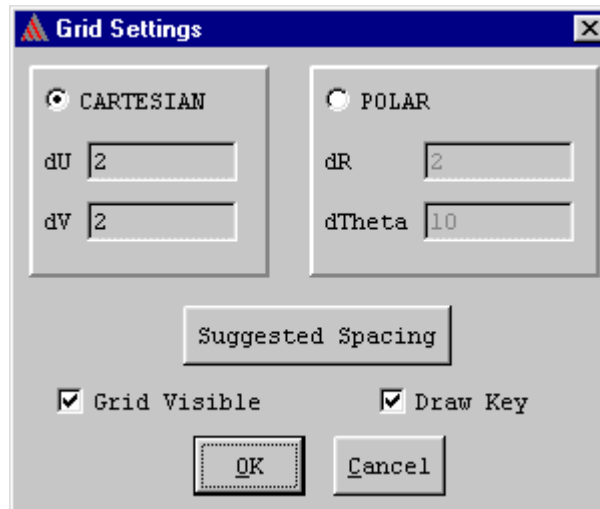
Change Grid Spacing

The final adjustment that needs to be made in the problem region is setting the spacing between the grid of points to which the cursor “snaps.” This also sets the visible grid in the subwindow. Because **snap-to-grids** is selected, the mouse snaps to the visible grid.

When you changed the drawing units, the 2D Modeler set the grid spacing to **0.5**. This means that there are 0.5 inches between each grid point. To simplify drawing the solenoid’s geometry, change the spacing between grid points to 0.05 inches.

To change the grid spacing:

1. Click **Window>Grid**. The following window appears:



2. Leave **CARTESIAN** selected. You want a rectangular grid displayed in the window.
3. Change the grid spacing in the **dU** field to **0.1**. This changes the grid spacing in the U direction.
4. Change the grid spacing in the **dV** field to **0.1**. This changes the grid spacing in the V direction.
5. Leave the **Grid Visible** and **Draw Key** check boxes selected. These display the u- and v-axes in the lower-left corner of the region and cause the grid to be displayed in the subwindow.
6. Click **OK**.

Now you are ready to create the geometry.

Create the Geometry

The solenoid is made up of five objects: a plugnut, core, coil, yoke, and bonnet. All objects are created using the **Object** commands as described in the following sections.

A sixth object — the “background” object — contains all areas of the model that are not occupied by other objects and does not have to be explicitly drawn. The background object is used when defining material properties and boundary conditions in Chapter 4, “Setting Up the Model.”

Draw the Plugnut

First, draw the plugnut, and specify its name and color.

Create a Rectangle

To create the plugnut:

1. Click **Object>Rectangle**. The cursor turns into crosshairs and the following text appears in the message bar:
MOUSE LEFT: Select first corner point of rectangle
MOUSE RIGHT: Abort command
2. Select the first corner of the plugnut, the upper-left corner, as follows:
 - a. Move the mouse to the point on the grid where the u- and v-coordinates are **(0, -0.2)**. Remember that the coordinates of the cursor’s current location are displayed in the **U** and **V** fields in the status bar.
 - b. Click the left mouse button to select the point. A new message appears in the message bar:
MOUSE LEFT: Select second corner point of rectangle
MOUSE RIGHT: Abort command
3. Select the second corner, the lower right corner, as follows:
 - a. Move the mouse to the point on the grid where the u- and v-coordinates are **(0.3, -1.2)**.
 - b. Click the left mouse button to select the point.

The **New Object** window appears.



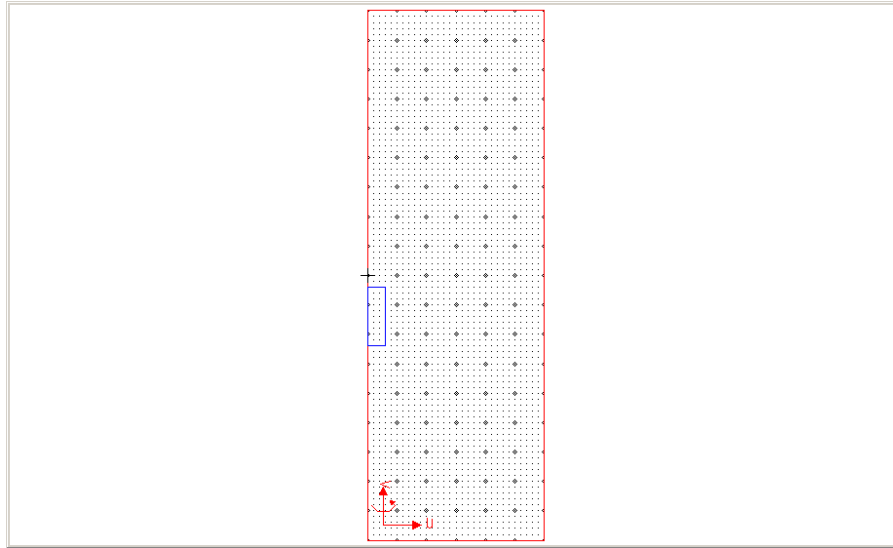
Change the Plugnut's Name and Color

By default, the object you are creating is assigned the name **object1** and the color red. The field **Name** is selected by default.

To change the name and color:

1. Type **plugnut** in the **Name** field.
2. Click the left mouse button on the red box. A palette of 16 colors appears.
3. Click on the blue box in the palette.
4. Click **OK**.

The object now appears as shown below. It is blue and has the name **plugnut**.



Keyboard Entry

In the “Change the Drawing Size” section, you changed the drawing units from millimeters to inches and left the default grid spacing set to 0.1 inches. However, several of the points that you will select as you draw the objects in the solenoid model lie *between* grid points. You can select those in-between points in one of two ways:

- Use keyboard entry, that is, enter the coordinates directly into the **U** and **V** fields in the status bar.
- Change the grid spacing to produce a tighter grid. If you do this, the screen may become cluttered with tightly-spaced grid points and make point selection difficult.

Use keyboard entry to enter several of the dimensions of the solenoid’s geometry.

Note As you enter points using the keyboard, be careful that you do not move the mouse. If the cursor leaves the status area, the coordinates will change to reflect the cursor’s new position.

Draw the Core

Use the **Object>Polyline** command to draw the irregular solenoid core.

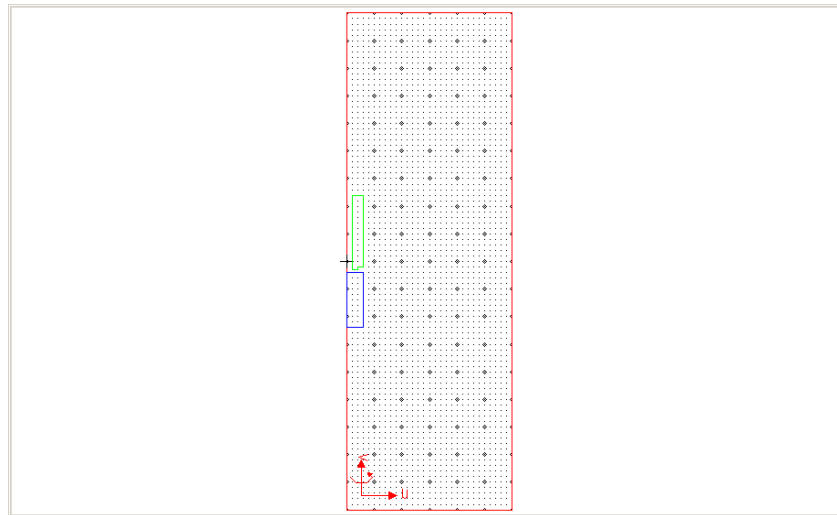
To draw the core of the solenoid:

1. Click **Object>Polyline**. The following text appears in the message bar:
 MOUSE LEFT: Click to enter point, click at previous point to finish
 MOUSE Right: Back up to previous point
2. Click the left mouse button at the u-and v-coordinates **(0.1, 1.2)** to select the first point of the core.
3. Click the left mouse button at the u-and v-coordinates **(0.1, -0.15)** to select the next point. A line connects the two points.
4. Select the following points in the same manner:

U	V
0.2	-0.15
0.2	-0.1
0.3	-0.1
0.3	1.2

5. Double-click at **(0.1, 1.2)** to complete the object, or press **Enter** twice. The **New Object** window appears, prompting you to change the object's name.
6. Change the **Name** to **core**, its **Color** to light green, and click **OK**.

The completed core should appear as shown below:



Draw the Coil

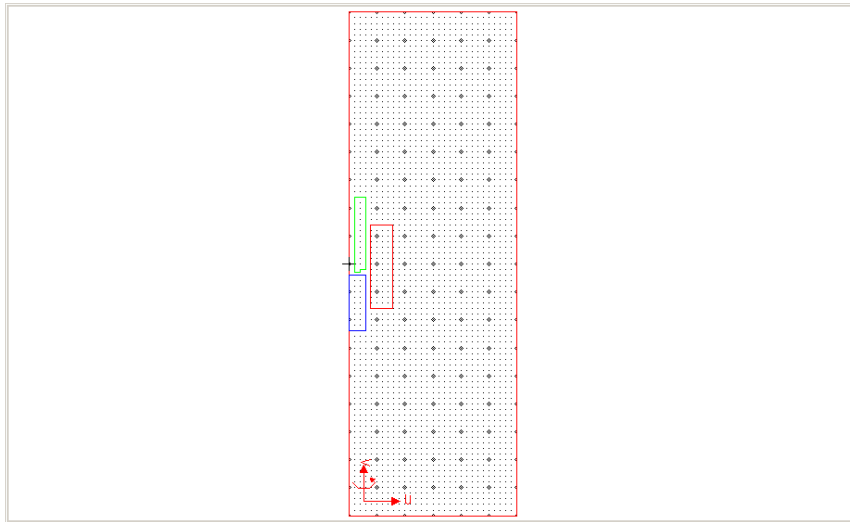
To draw the coil using keyboard entry:

1. Click **Object>Rectangle**.
2. To select the first corner of the coil, enter its coordinates on the keyboard. The u-coordinate of the upper left corner, 0.375, lies between grid points. Select this corner as follows:
 - a. Double-click in the **U** field in the status bar.
 - b. Type **0.375**.
 - c. Press the **Tab** key to move to the **V** field in the status bar.
 - d. Type **0.7**.
 - e. Do one of the following to accept the point:
 - Press **Return**.
 - Click **Enter** from the status bar.
3. Select the lower right corner of the coil (0.775, -0.8) in the same way:
 - a. Enter **0.775** in the **U** field in the status bar.
 - b. Enter **-0.8** in the **V** field in the status bar.
 - c. Press **Return** or click **Enter** from the status bar to accept the point.

The **New Object** window appears, prompting you to change the object's color and name.

4. Change the object's **Name** to **coil** and its **Color** to red, and then click **OK**.

The coil should appear as shown below:



Note From this point on, help text that is displayed in the message bar will generally not be included in this manual. Therefore, be sure to look at the message bar if you need to know what effect clicking a mouse button has for a command.

Save the Geometry

Maxwell SV does not automatically save your work while you are drawing. It is a good idea to periodically save the geometry while you are working on it.

To save the solenoid model, click **File>Save**. The geometry is saved in a file called **solenoid.sm2** in the **solenoid.pjt** directory (in which the files for the **solenoid** project are stored). This directory is located under the **getting_started_SV** project directory.

Draw the Yoke

After you draw the coil and save the geometric model, draw the yoke for the solenoid.

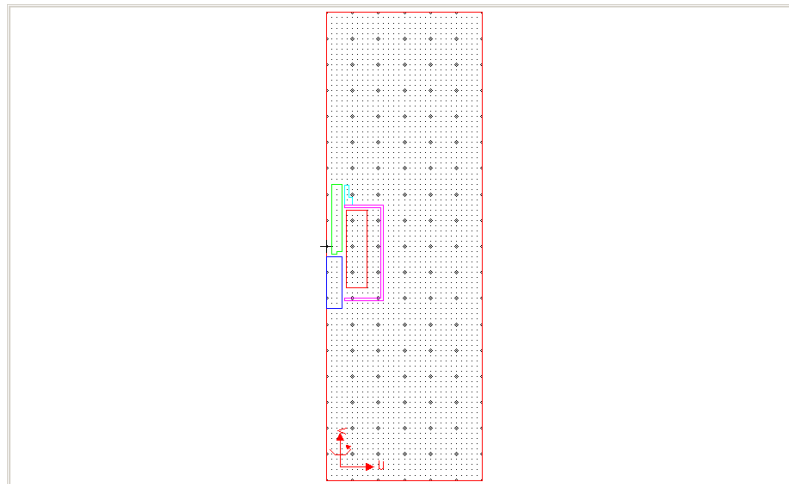
To draw the solenoid's yoke:

1. Click **Object>Polyline**, and then click on the point **(0.35, -1.05)** to select the first corner of the yoke. Click **OK** after clicking on the point for the first corner.
2. Select the remaining points for the yoke, using the following coordinates:

U	V
1.1	-1.05
1.1	0.8
0.35	0.8
0.35	0.75
1.05	0.75
1.05	-1.0
0.35	-1.0
0.35	-1.05

After you enter the final coordinate **(0.35, -1.05)**, press **Enter** twice. The **New Object** window appears.

3. Change the object's **Name** to **yoke** and its **Color** to purple, and click **OK**.



Draw the Bonnet

The last object you need to draw is the bonnet. Use keyboard entry to specify coordinates that lie between grid points.

To draw the bonnet:

1. Click **Object>Polyline**.
2. To select the first corner of the bonnet, use the keyboard to enter the following coordinates:

U	0.35	Press Tab .
V	0.8	Press Return .
3. After selecting the point, press the **Tab** key to highlight the **U** field again.
4. To select the remaining points, use the keyboard to enter the following coordinates:

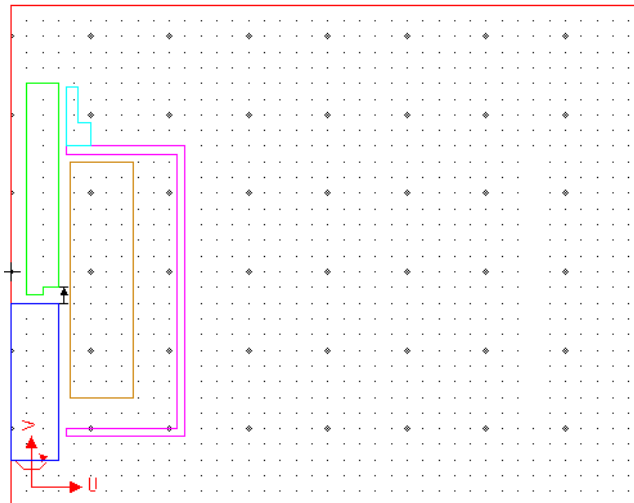
U	V
0.50	0.8
0.50	0.95
0.425	0.95
0.425	1.177
0.35	1.177
0.35	0.8
0.35	0.8

After entering the final point in the bonnet (0.35, 0.8) twice, the **New Object** window appears.

5. Change the object's **Name** to **bonnet** and its **Color** to light blue, and click **OK**.

Completed Geometry

The geometric model is now complete.



Exit the 2D Modeler

To save your model and exit:

1. Click **File>Exit**. A message appears asking if you want to save the subwindow settings as the defaults for new ones.
2. Click **No**. You do not need to use the grid settings for the solenoid model as the default for new projects. A new message appears:

Save changes to "solenoid" before closing?

3. Click **Yes**.

The geometry is saved in the **solenoid.pjt** directory, and the **Executive Commands** menu reappears. A check mark appears next to **Define Model**, indicating that this step has been completed.

Note The **Group Objects** command under **Define Model** is not required for setting up a model in Maxwell SV. For this example, none of the objects you created are connected at any point in a 3D rendering of the model; therefore, you do not need to group any objects. See the Maxwell 2D online help for further details on grouping objects.

Setting Up the Model

Now that you have drawn the geometry for the solenoid problem and returned to the **Executive Commands** menu, you are ready to set up the model.

The goals for this chapter are to:

- Assign materials with the appropriate material attributes to each object in the geometric model.
- Define any boundary conditions and sources that need to be specified, such as the source current of the coil.
- Request that the force acting on the core be calculated during the solution.

Time This chapter should take approximately 30 minutes to work through.

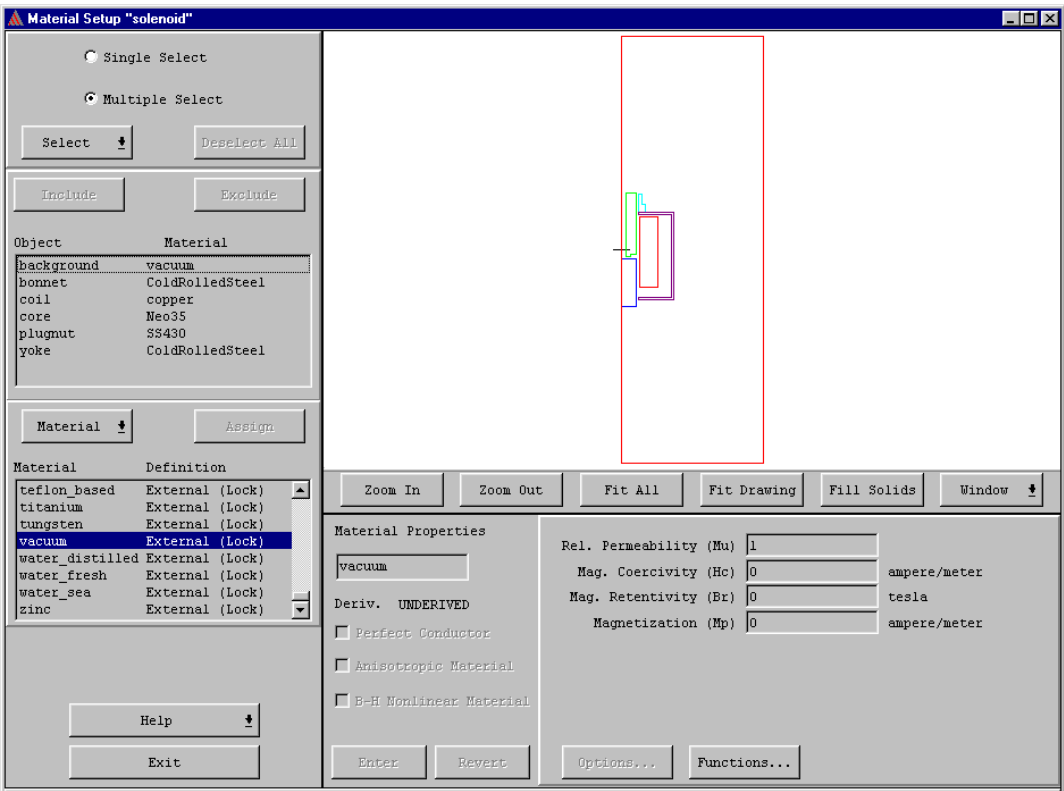
Assign Materials to Objects

The next step in setting up the solenoid model is to assign materials to the objects in the model. Materials are assigned to objects via the **Material Manager**, which is accessed through the **Setup Materials** command. You will do the following:

- Assign **copper** to the coil.
- Define the material **cold rolled steel** (a nonlinear magnetic material) and assign it to the bonnet and yoke.
- Define the material **Neo35** (a permanent magnet) and assign it to the core.
- Define the material **SS430** (a nonlinear magnetic material) and assign it to the plugnut.
- Accept the default material that is assigned to the background object, which is **vacuum**.

Access Material Manager

To access the **Material Manager**, click **Setup Materials**. The Material Manager appears.



Assign Copper to the Coil

The Material Manager should still be on the screen. In the actual solenoid, the coil is made of copper.

To assign the material **copper** to the coil:

1. Select the object that is to be assigned a material — the **coil** — in one of two ways:
 - Click the name **coil** in the **Object** list.
 - Click the object **coil** in the geometry.

Both the object and its name are highlighted.

Note To deselect an object, simply click on it. To select multiple objects from the **Object** list, click **Multiple Select**, and hold down the **Ctrl** key and click each object.

As an alternative to selecting and deselecting objects with the mouse, use the **Select** commands (which appear beneath the **Multiple Select** button in the Material Manager). They are described in Maxwell 2D's online help.

2. Select **copper** from the **Material** database listing. If it does not appear in the box, use the scroll bars to pan through the list as described in Chapter 3, "Creating the Model."
3. Click the **Assign** button, which is located above the **Material** list.

Copper now appears next to **coil** in the **Objects** list.

Assign Cold Rolled Steel to the Bonnet and Yoke

The bonnet and yoke of the solenoid are made of cold rolled steel. Since this material is not in the database, you must create a new material, **ColdRolledSteel**. This material is nonlinear — that is, its relative permeability is not constant and must be defined using a B vs. H curve. Therefore, when you enter the material attributes for cold rolled steel, you will also define a B-H curve.

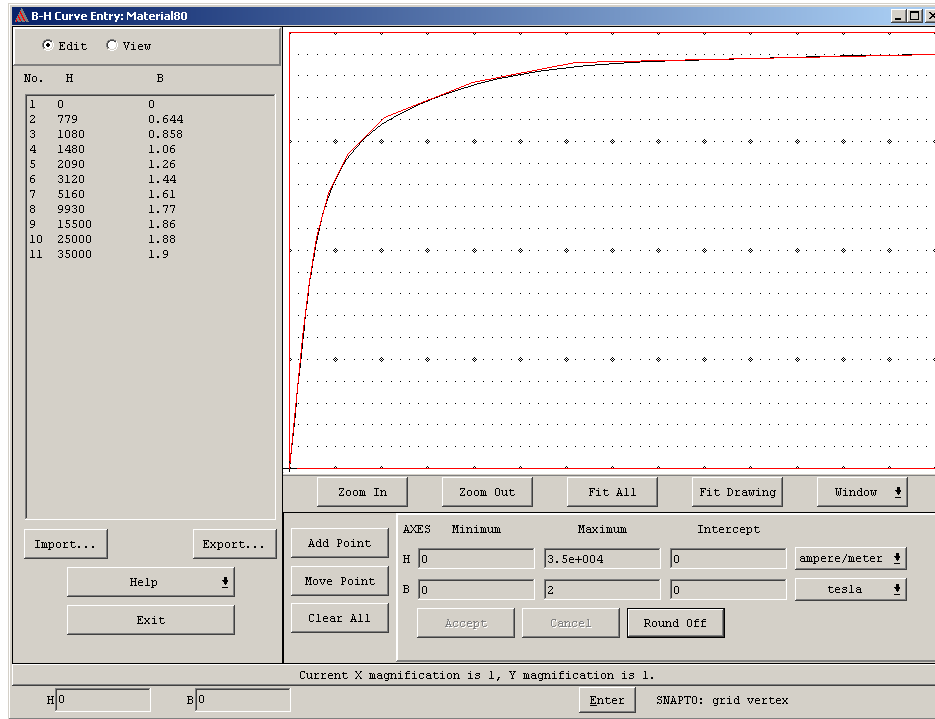
Select Objects and Create the Material

To select the objects and create the new material:

1. If it is not already selected, select the **Multiple Select** radio button from the top of the menu.
2. Select **bonnet** and **yoke**.
3. Click **Material>Add** from the **Material** pull-down menu.
4. Under **Material Properties**, change the name of the new material from **Material80** to **Cold-RolledSteel**.
5. Select **B-H Nonlinear Material**. The **Rel. Permeability (Mu)** field changes to a button labeled **B H Curve**.

Define the B-H Curve

To define the B-H curve, click **B H Curve**. The **B-H Curve Entry** window appears.



On the left is a blank BH-table where the **B** and **H** values of individual points in the B-H curve are displayed as they are entered. On the right is a graph where the points in the B-H curve are plotted as they are entered.

Note When defining B-H curves, keep the following in mind:

- A B-H curve may be used in more than one model. To do so, save (export) it to a disk file, which can then be imported into other Maxwell 2D and 3D models.

In this guide, you will not be exporting them to files.

Define the Minimum and Maximum Values for B and H

Before entering the curve's **B** and **H** values, you need to make sure the data will fit on the graph. Minimum and maximum values for the graph are entered under **AXES**. The **H** values for cold rolled steel will fall between zero and 35,000 amperes per meter. The **B** values will fall between zero and 2.0 tesla.

To change the graph size:

1. Enter **0** in the **Minimum H** field.
2. Enter **0** in the **Minimum B** field.
3. Enter **35000** in the **Maximum H** field.
4. Enter **2** in the **Maximum B** field.
5. Click **Accept**.

Note Numeric values — like the minimum and maximum B and H values — may be entered and displayed in Ansoft's shorthand for scientific notation. For instance, **35000** could also be entered as **3.5e4**. When entering numeric values, you can use either notation.

Add B-H Curve Points

To enter the points in the B-H curve:

1. Click **Add Point**.
2. Click the left mouse button at the point **(0, 0)** — the origin of the B- and H-axes. The coordinates are displayed in the **H** and **B** fields in the status bar. The values for H and B are entered in the **B-H Curve** table, and a point is entered on the graph.
3. To enter the rest of the values for the B-H curve, do the following:
 - a. To enter the next point, double-click in the **H** field in the status bar.
 - b. Enter **779**. Press the **Tab** key to go to the **B** field.
 - c. Enter **0.644**.
 - d. Click **Enter**. Press **Tab** again to return to the **H** field.

A line connects the two points on the curve.

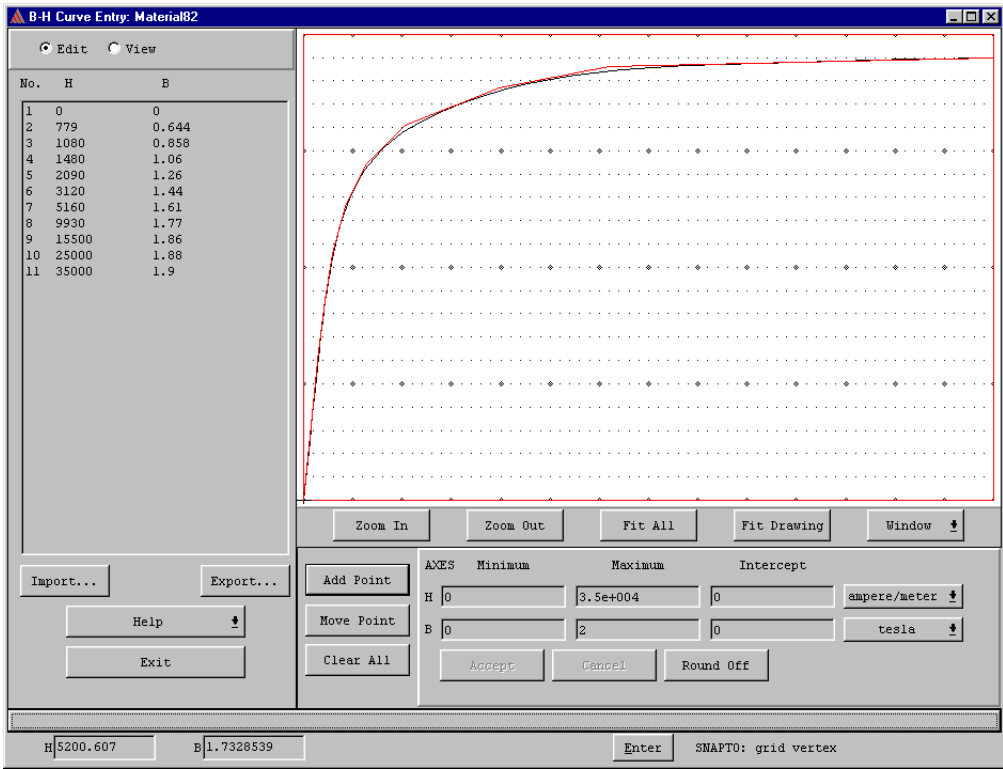
4. Repeat step three, entering the following data:

H	B
1080	0.858
1480	1.06
2090	1.26
3120	1.44
5160	1.61
9930	1.77
1.55e4	1.86

Getting Started: A 2D Magnetostatic Problem

H	B
2.50e4	1.88
3.50e4	1.90

5. After you enter the last value, click the mouse button on **Enter** twice. The software automatically fits a curve to the points you entered and displays a list of all B-H curve points, as shown below:



6. Click **Exit** to return to the **Material Setup** window.

Assign ColdRolledSteel to the Yoke and Bonnet

Now that you have returned to the Material Manager, enter **ColdRolledSteel** into the database and assign it to the selected objects:

To save and assign the new material:

1. Click **Enter** to save the material properties you have entered for **ColdRolledSteel** — including the B-H curve you have just defined — and add it to the material database. **ColdRolledSteel** appears in the **Material** list. The word **Local** appears next to it, indicating that this material is specific to the **solenoid** project.
2. Click **Assign** to assign **ColdRolledSteel** to the yoke and bonnet.

Assign Neo35 to the Core

Next, create the new material **Neo35**, and assign it to the core. **Neo35** is a permanently magnetic material.

Note In the actual solenoid, the core was assigned the same material as the plugnut — **SS430**. However, for this problem, a permanent magnet is assigned to the core to demonstrate how to set up a permanent magnetic material.

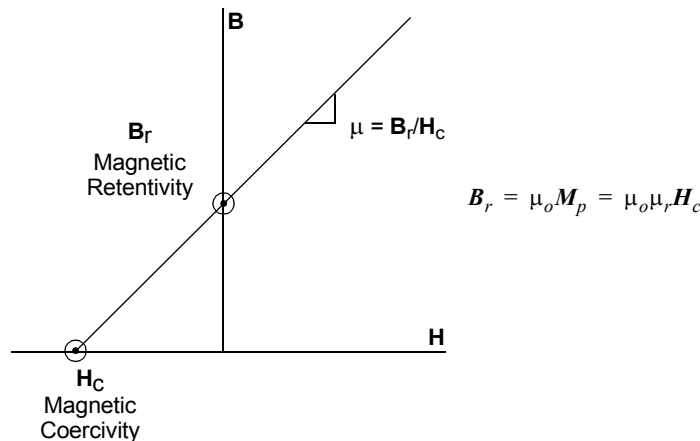
Select the Object and Create the Material

To select the core and create the material:

1. Select **core** from the **Object** list.
2. Click **Material>Add**.
3. Under **Material Properties**, change the name of the new material to **Neo35**.

Select Independent Material Properties

In magnetostatic problems, only two of the four available material properties need to be specified. The values of the other two properties are dependent upon these properties, according to the relationships shown below.

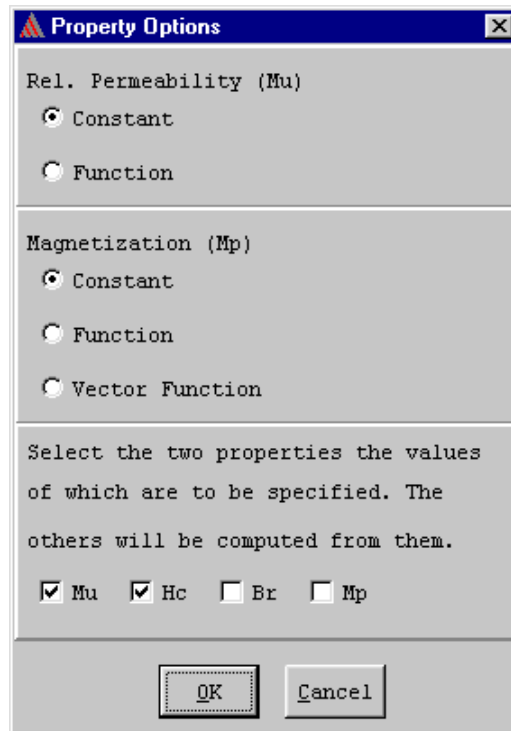


You will be specifying values for relative permeability, μ , and magnetic retentivity, B_r . The values of the other two properties, magnetic coercivity, H_c , and magnetization, M_p , will be computed using the relationships shown above.

Getting Started: A 2D Magnetostatic Problem

To specify which properties are to be entered:

1. Click **Options**. The **Property Options** window appears.



Initially, the only properties that can be specified are the relative permeability (μ_r) and the magnetic coercivity (H_c). To define the permanent magnet, you will need to specify the magnetic retentivity.

2. Click on the check box next to **Hc** to deselect it.
3. Click on the check box next to **Br** to select it.
4. Click **OK**.

Enter Material Properties

Now enter the values for the material attributes of **Neo35**. Only the two material attributes that were turned on in the **Property Options** window need to be entered — the other two attributes will be calculated automatically.

To enter the properties for **Neo35**:

1. Enter **1.05** in the **Rel. Permeability (Mu)** field.
2. Enter **1.25** in the **Mag. Retentivity (Br)** field. Values automatically appear in the other fields.
3. Click **Enter**.

Neo35 is now listed as a local material in the database.

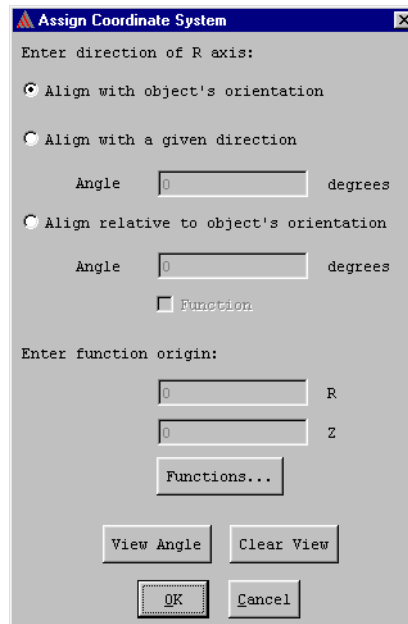
Assign Neo35 to the Core and Specify Direction of Magnetization

Now that you have created the material **Neo35**, all that remains is to assign it to the core and specify the direction of magnetization.

By default, the direction of magnetization in materials is along the R-axis (or the x-axis). However, in this problem, the direction of magnetization in the core points along the positive z-axis. To model this, you must change the direction of magnetization to act at a 90° angle from the default.

To assign **Neo35** to the core:

1. Make certain that **Neo35** is highlighted in the **Material** list.
2. Click **Assign** to assign the material **Neo35** to the **core** object. The **Assign Coordinate System** window appears.



3. Select **Align with a given direction** to align the magnetization with the entered value.
4. Enter **90** degrees in the **Angle** field.

Note Make certain you enter a positive value for the angle. For instance, if you had entered 90 degrees, the magnetization of the core would point in the opposite direction and would tend to push the core out of the solenoid.

5. Click **OK**.

Assign SS430 to the Plugnut

Next, create a new material — **SS430** — for the plugnut. Like the material **ColdRolledSteel**, it is a nonlinear material whose relative permeability must be defined using a B-H curve. The first step includes selecting the plugnut and creating the material.

Select Objects and Create Material

To select the plugnut, and create the material:

1. Select **plugnut** from the **Object** list.
2. Click **Material>Add**.
3. Under **Material Properties**, change the name of the new material to **SS430**.
4. Select **B-H Nonlinear Material**.

Define the B-H Curve

To define the B-H curve for SS430, click **B H curve**.

The **B-H Curve Entry** window appears.

Define Maximum and Minimum Values for B and H

For SS340, the values for **H** will fall between zero and 40,000 amperes per meter. The values for **B** will fall between zero and 2.0 tesla.

To change the graph size of the B-H curve:

1. Enter **0** in the **Minimum H** field.
2. Enter **0** in the **Minimum B** field.
3. Enter **40000** in the **Maximum H** field.
4. Enter **2.0** in the **Maximum B** field.
5. Click **Accept**.

Add B-H Curve Points

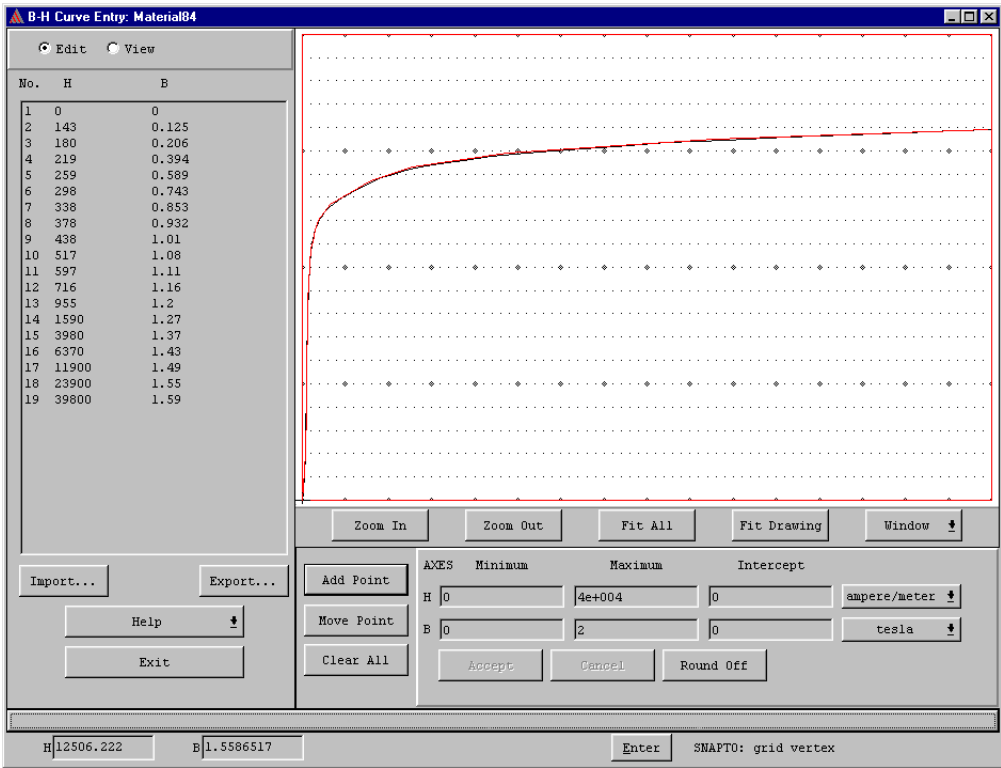
To enter the points in the B-H curve:

1. Click **Add Point**.
2. Click the left mouse button at the origin of the B- and H-axes (0, 0).
3. Enter the following points using keyboard entry, as described under “Define the B-H Curve” in “Assign Cold Rolled Steel to the Bonnet and Yoke.”

H	B
143	0.125
180	0.206
219	0.394
259	0.589
298	0.743
338	0.853
378	0.932

H	B
438	1.01
517	1.08
597	1.11
716	1.16
955	1.20
1590	1.27
3980	1.37
6370	1.43
1.19e4	1.49
2.39e4	1.55
3.98e4	1.59

4. After you enter the last value, click **Enter** twice. The software automatically fits a curve to the points you entered on the graph, as shown below:



5. Click **Exit** to return to the **Material Setup** window.

Assign SS430 to the Plugnut

Finally, add **SS430** to the material database, and assign it to the selected object:

To add and assign **SS430**:

1. Click **Enter** to save the material attributes you entered for **SS430** and add it to the material database.
2. Make certain **SS430** is selected in the **Material** list.
3. Click **Assign** to assign **SS430** to the **plugnut**.

Accept Default Material for Background

Accept the following default parameters for the object **background**:

- The material **vacuum**, which has the material attributes of a vacuum. The object **background** is the only object that is assigned a material by default.
- Include it as part of the problem region in which to generate the solution. When a material name — such as **vacuum**, the material that is assigned to **background** by default — appears next to **background** in the **Object** list, it is included as part of the solution region.

Exit the Material Manager

Now that materials have been assigned to each object, exit the **Material Setup** window:

To exit the **Material Setup** window:

1. Click **Exit** from the bottom-left of the **Material Setup** window. A window with the following prompt appears:

```
Save changes before closing?
```
2. Click **Yes**.

You are returned to the **Executive Commands** menu. A check mark now appears next to **Setup Materials**, and **Setup Boundaries/Sources** remains enabled.

If you exit the Material Manager before assigning a material to each object in the model, a check mark does not appear next to **Setup Materials** on the **Executive Commands** menu, and **Setup Boundaries/Sources** is disabled. You will not be allowed to continue setting up your model until materials are assigned to *all* objects in the model.

Set Up Boundaries and Current Sources

After you set material properties, you must define boundary conditions and sources of current for the solenoid model. Boundary conditions and sources are defined in the Boundary Manager, which is accessed through the **Setup Boundaries/Sources** command.

By default, the surfaces of all objects are Neumann or natural boundaries. That is, the magnetic field is defined to be perpendicular to the edges of the problem space and continuous across all object interfaces. To finish setting up the solenoid problem, you must explicitly define the following boundaries and sources:

- The boundary condition at all surfaces exposed to the area outside of the problem region. Because you included the background as part of the problem region as described in the “Accept Default Material for Background” section, this exposed surface is that of the object

background. Since the solenoid is assumed to be very far away from other magnetic fields or sources of current, those boundaries will be defined as “balloon boundaries.”

- The source current on the coil. Since the coil has 10,000 turns of wire, and one ampere flows through each turn, the net source current is 10,000 amperes.

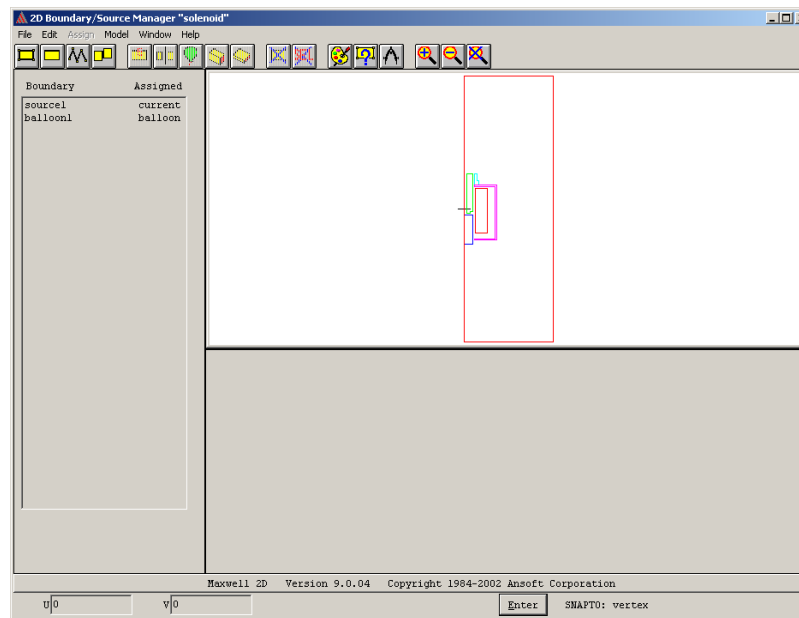
Note Maxwell SV will not solve the problem unless you specify some type of source or magnetic field — either a current source, an external field source using boundary conditions, or a permanent magnet. In this problem, both the permanent magnet assigned to the core and the current flowing in the coil act as magnetic field sources.

Access the 2D Boundary/Source Manager

Define the boundary conditions for the model.

To access the Boundary Manager:

- Click **Setup Boundaries/Sources**. The 2D Boundary/Source Manager appears.



Boundary Manager Screen Layout

Menu Bar and Toolbar

Like 2D Modeler commands, 2D Boundary/Source Manager commands may be accessed from the following:

- The menu bar, which is located along the top of the **2D Boundary/Source Manager** window.
- The toolbar, which is located on the top of the window.

Boundary

Each time you choose one of the **Assign>Boundary** or **Assign>Source** commands, an entry is added to the **Boundary** list on the left side of the menu. When you first access the 2D Boundary/Source Manager, the **Boundary** list is empty.

Display Area

The geometric model is displayed on the menu so that you can choose objects using the **Edit>Select** commands. Use the **Window>Change View** commands, described in the Maxwell 2D online help, to change your view of the model.

Boundary/Source Information

The box that appears below the geometric model allows you to assign boundaries and sources to objects and surfaces, and to display the parameters associated with the selected boundary or source.

Types of Boundary Conditions and Sources

There are two types of boundary conditions and sources that you will use in this problem:

Balloon boundary	Can only be applied to the outer boundary. Models the case in which the structure is infinitely far away from all other electromagnetic sources.
Solid source	Specifies the DC source current flowing through an object in the model.

In this example, you will assign boundary conditions and sources to the following objects in the solenoid geometry:

Background	At the outer boundary of the problem region, the outside edges of this surface are to be ballooned to simulate an insulated system.
Coil	This object is to be defined as a 10,000-amp DC current source. The current is uniformly distributed over the cross-section of the coil and flows in the positive perpendicular direction to the cross-section.

Set Source Current on the Coil

Before you identify a boundary condition or source, you must first identify the object or surface to which the condition is to be applied. In this section, you will pick the coil as a source and then assign a current to it.

Pick the Coil

To select the coil as a source:

1. Click **Edit>Select>Object>By Clicking**.
2. Click the left mouse button on the object **coil**, highlighting it.
3. Click the right mouse button anywhere in the display area to stop selecting objects.

Note If the appropriate object is not highlighted, or if more than one object is highlighted, do the following:

1. Click **Edit>Deselect All**. After you do, no objects are highlighted.
2. Follow the instructions again to pick the object.

Assign a Current Source to the Coil

To specify the current flowing through the coil:

1. Click **Assign>Source>Solid**. The name **source1** appears in the **Boundary** list, and **NEW** appears next to **source1**, indicating that the source has not yet been assigned to an object or surface. Fields appear in the boundary information area at the bottom of the window:

2. Leave the **Current** check box selected.
3. Leave the **Total** radio button selected.
4. Enter **10000** in the **Value** field, and click **Assign**. The label **current** replaces **NEW** next to **source1** in the **Boundary** list.

Assign Balloon Boundary to the Background

The structure is a magnetically isolated system. Therefore, you must create a balloon boundary by assigning balloon boundaries to the outside edges of the background object.

Pick the Background

To select the background as a boundary.

1. Click **Edit>Select>Object>By Clicking**, and click the left mouse button anywhere on the background.
2. Click the right mouse button to abort the command.

Now you are ready to balloon the background.

Assign Balloon Boundary

To assign a balloon boundary to the background:

1. Click **Assign>Boundary>Balloon**. The **balloon1** boundary name appears in the **Name** field.
2. Click **Assign**. The background is ballooned and **balloon** replaces **NEW** next to the boundary name in the **Boundary** list.

Note In cartesian (XY) models, all outer edges may be defined as boundaries. However, in axisymmetric (RZ) models, the left edge of the problem region cannot be assigned a boundary condition. Because the solenoid model is axisymmetric (representing the cross-section of a device that is revolved 360 degrees around its central axis), the simulator ignores the balloon boundary you defined on the left edge of the geometry. Instead, it automatically imposes a different boundary condition, in order to model that left edge as an axis of rotational symmetry.

Exit the Boundary Manager

To exit the Boundary Manager:

1. Click **File>Exit**. A window appears with the following message:
Save changes to "solenoid" before closing?
2. Click **Yes**.

You return to the **Executive Commands** menu. A check mark appears next to **Setup Boundaries/Sources**, indicating that this step has been completed. Note that a check mark also appears next to **Setup Solution Options**, indicating that a set of default solution parameters has been defined for the problem.

You are now ready to set up the force computation for the model.

Set Up Force Computation

One of your goals for this problem is to determine the force acting on the core of the solenoid. To find the force on this object, you must select it using the **Force** command under **Setup Executive Parameters**. The force (in newtons) acting on the core will then be computed during the solution process.

To select the core object for the force computation:

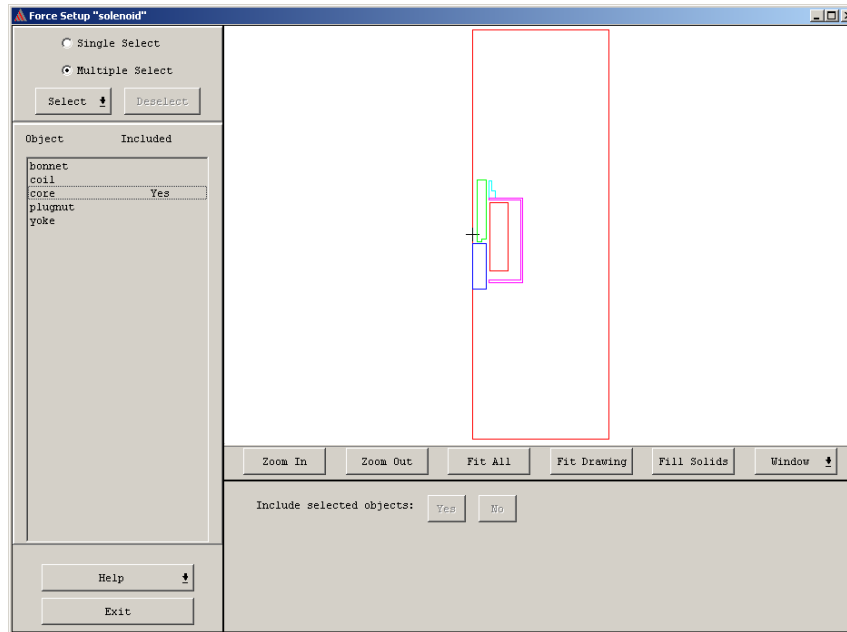
1. Click **Setup Executive Parameters**. The following menu appears:

- Force
- Core Loss
- Torque
- Flux Lines
- Post Processor Macros
- Matrix/Flux
- Select Matrix/Flux Entries

Quantities that may not be computed are grayed out and cannot be accessed. The **Select Matrix/Flux Entries** option is not enabled for Maxwell SV because the student version does not use the parametric analysis capability.

2. Click **Force**. The **Force Setup** window appears. All objects in the model are listed on the left

side of the window. The geometric model is shown on the right.



3. Select the object on which force is to be computed — the core — in one of two ways:

- Select **core** from the **Object** list.
- Click on the object **core** in the geometry.

Both the object and its name are highlighted.

4. To include the **core** object in the force computation, click **Yes** next to **Include selected objects**. The word **Yes** appears next to **core** in the **Object** list.

5. Click **Exit** to exit the **Force Setup** window. The following message appears:

Save changes before closing?

6. Click **Yes** to save the force setup and exit. You return to the **Executive Commands** menu.

The force on the core will now be computed during the solution. You are now ready to enter the solution criteria.

Generating a Solution

Now you are ready to specify solution parameters and generate a solution for the solenoid model. You will do the following:

The goals for this chapter are to:

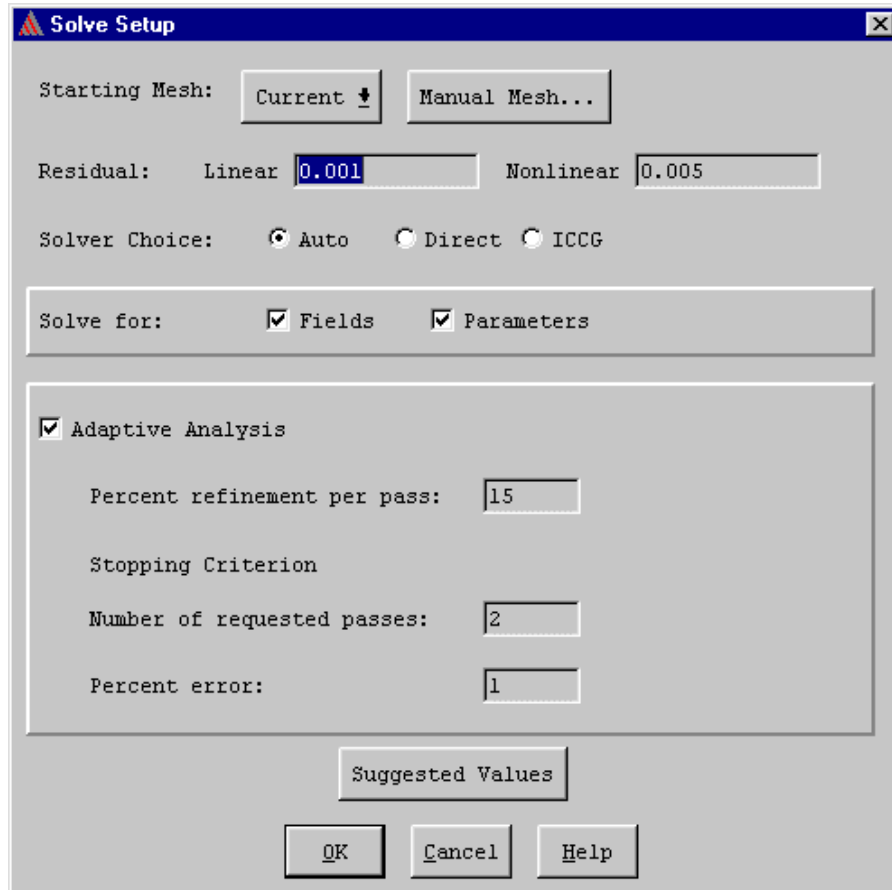
- View the criteria that affect how Maxwell SV computes the solution.
- Generate the magnetostatic solution. The axisymmetric magnetostatic solver calculates the magnetic vector potential, \mathbf{A}_ϕ , at all points in the problem region. From this, the magnetic field, \mathbf{H} , and magnetic flux density, \mathbf{B} , can be determined.
- Compute the force on the core. Since you requested force using the **Setup Executive Parameters** command, this automatically occurs during the general solution process.
- View information about how the solution converged and what computing resources were used.

Time This chapter should take approximately 40 minutes to work through.

View Solution Criteria

Use the default criteria to generate the solution for the solenoid problem.

To view the solution criteria, click **Setup Solution Options**. The **Solve Setup** window appears.



The **Solve Setup** dialog box is shown with the following settings:

- Starting Mesh:** **Current** (selected), **Manual Mesh...**
- Residual:** **Linear** **0.001**, **Nonlinear** **0.005**
- Solver Choice:** **Auto** (selected), **Direct**, **ICCG**
- Solve for:** ☒ **Fields**, ☒ **Parameters**
- ☒ **Adaptive Analysis**
 - Percent refinement per pass:** **15**
 - Stopping Criterion**
 - Number of requested passes:** **2**
 - Percent error:** **1**
- Suggested Values** button
- OK**, **Cancel**, **Help** buttons

When the system generates a solution, it explicitly calculates the field values at each node in the finite element mesh and interpolates the values at all other points in the problem region.

Starting Mesh

Leave the **Starting Mesh** command set to **Initial**. This instructs the system to use the only mesh available, the coarse initial mesh when it begins the solution process. If you had already generated a solution and wanted to take advantage of the most recently refined mesh, you would click **Current**.

Manual Mesh

To get a head start on refining the initial mesh or the current mesh, you could choose the **Manual Mesh** command. This would allow you to manually break large triangles down into smaller ones, increasing the accuracy of the solution in the desired areas.

Solver Choice

You can specify which type of matrix solver to use to solve the problem. In the default **Auto** position, the software makes the choice. The **ICCG** solver is faster for large matrices, but occasionally fails to converge (usually on magnetic problems with high permeabilities and small air-gaps). The **Direct** solver will always converge, but is much slower for large matrices. In the **Auto** position, the software evaluates the matrix before attempting to solve; if it appears to be ill-conditioned, the **Direct** solver is used, otherwise the **ICCG** solver is used. If the **ICCG** solver fails to converge while the solver choice is in the **Auto** position, the software will fall back to the **Direct** solver automatically.

Leave the **Solver Choice** option set to **Auto**.

Solver Residual

Leave the **Residual** field set to **0.001** for **Linear**, and to **0.005** for **Nonlinear**. These values specify how close each solution must come to satisfying the equations that are used to compute the magnetic field.

Solve for Fields and Parameters

The **Solve for** options tell the system what types of solutions to generate.

- | | |
|-------------------|--|
| 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 force on the core). |

Leave both the **Fields** and **Parameters** check boxes selected, to solve for both fields and parameters in this example.

Adaptive Analysis

Do the following to have the system adaptively refine the mesh and solution:

- Leave **Adaptive Analysis** selected. Doing so 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 areas of highest error, resulting in a more accurate field solution.
- Leave **Number of requested passes** and **Percent error** set to their defaults. After each iteration, the system calculates the total energy of the system and the percentage of this energy that is caused by 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.
- Leave **Percent refinement per pass** set to its default value. In most cases, the default refinement value is acceptable.

If any of these criteria has been met, the solution process is complete and no more iterations are done.

Exit Setup Solution Options

To exit the **Solve Setup** window, click **OK**. You return to the **Executive Commands** menu.

Start the Solution

Now that you have examined the solution parameters, the problem is ready to be solved.

To start the solution, click **Solve**.

The solution process begins and the following changes occur:

- The system creates the initial finite element mesh for the solenoid structure. A bar labeled **Making Initial Mesh** appears in the **Solution Monitoring** area at the bottom of the screen. It shows the system's progress as it generates the mesh.
- A bar labeled **Solving Fields** appears in the **Solution Monitoring** area.
- A button labeled **Abort** appears next to the progress bar and remains there throughout the entire solution process. If desired, you can use it to stop the solution process.
- A bar labeled **Setting up solution files** appears.

After the system makes the initial finite element mesh, the magnetostatic field solution process begins.

Values you obtain for percent energy error, total energy, or force may differ slightly from the ones given in this guide. Depending upon how closely you followed the directions for setting up the solenoid model, the results that you obtain should be approximately the same as the ones given here.

Monitoring the Solution

The following monitoring bars alternate in the **Solution Monitoring** box at the bottom of the menu. They allow you to monitor the system's progress as it computes each adaptive pass (solve — error analysis — refine cycle) of the magnetostatic solution. Periodically, the solver also displays messages beneath the progress bars. The following table explains some of the messages that may appear:

Solving Fields	Displayed as Maxwell SV computes the field solution. After it finishes computing a solution, it identifies the triangles that have the highest energy error.
Refining Mesh	Displayed as the system refines the regions of the finite element mesh with the highest error energy. Since the default specification for mesh refinement is 15 percent, the system refines triangles that have the top 15 percent of the error.
Solving matrix (nonlinear iteration)	Displayed as the system calculates a solution for the relationship between the current and the field solution. This may go on for several iterations before the general field solution continues. Because there are nonlinear materials in the model, the magnetostatic field solver uses a somewhat different solution process than if linear materials were present.
Computing force value	Displayed as the simulator solves for the force on the core due to the source current in the coil.

Viewing Convergence Data

After a few adaptive passes are completed, click the **Convergence** button at the top of the **Executive Commands** menu to monitor how the solution is progressing. Convergence information appears as shown below. In this example, the system has completed eight adaptive passes.

CONVERGENCE DATA

Number of passes:	Pass	Triangles	Total Energy(J)	Energy Error(%)	Mag Force(N)	
Completed	8	1	146	2.16419E+000	142.6010	1.16598E+002
Remaining	2	2	181	2.08579E+000	29.4621	1.19395E+002
		3	233	2.12921E+000	16.7112	1.15735E+002
		4	299	2.08319E+000	6.5286	1.21441E+002
Convergence criteria:	5	383	2.01462E+000	3.4895	1.17365E+002	
Target Error: 1%	6	493	1.99977E+000	2.4275	1.16285E+002	
Energy Error: 0.754%	7	638	1.99000E+000	1.4563	1.15634E+002	
Delta Energy: 0.144%	8	826	1.98714E+000	0.7543	1.17639E+002	

Convergence Display

Zoom In

Zoom Out

Fit All

Solution Criteria

Information about the solution criteria is displayed on the left side of the convergence display.

Number of passes	Displays how many adaptive passes have been completed and still remain.
Target Error	Displays the percent error value — 1% — that was entered using the Setup Solution Options command.
Energy Error	Displays the percent error from the last completed solution — in this case, 0.754% . Allows you to see at a glance whether the solution is close to the desired error energy.
Delta Energy	Displays the change in the total energy between the last two solutions — in this case, 0.144% . Because this value and the Energy Error value are less than the Target Error , the solution is considered to be converged.

Completed Solutions

Information about each completed solution is displayed on the right side of the screen.

Pass	Displays the number of the completed solutions. In the previous figure, eight adaptive passes were completed.
Triangles	Displays the number of triangles in the mesh for a solution. In the previous figure, the mesh used for the eighth solution had 826 triangles.
Total Energy (J)	Displays the total energy of a solution in Joules. In the previous figure, the total energy for the eighth solution was 1.98714 Joules.
Energy Error (%)	Displays the percent energy error of the completed solutions. In the previous figure, the energy error for the eighth solution was .7543% .
Mag Force (N)	Displays the magnitude of the force on the core in newtons. Force is computed after each adaptive pass, allowing you to see if it is converging to a stable value. In the previous figure, the force on the core during the tenth solution was 1.17639E+02 , or 117.459 newtons. This value only represents the total force acting on the object and does not indicate its direction.

Completing the Solution Process

When the solution is complete, a pop-up window with the following message appears:

`Solution Process is complete`

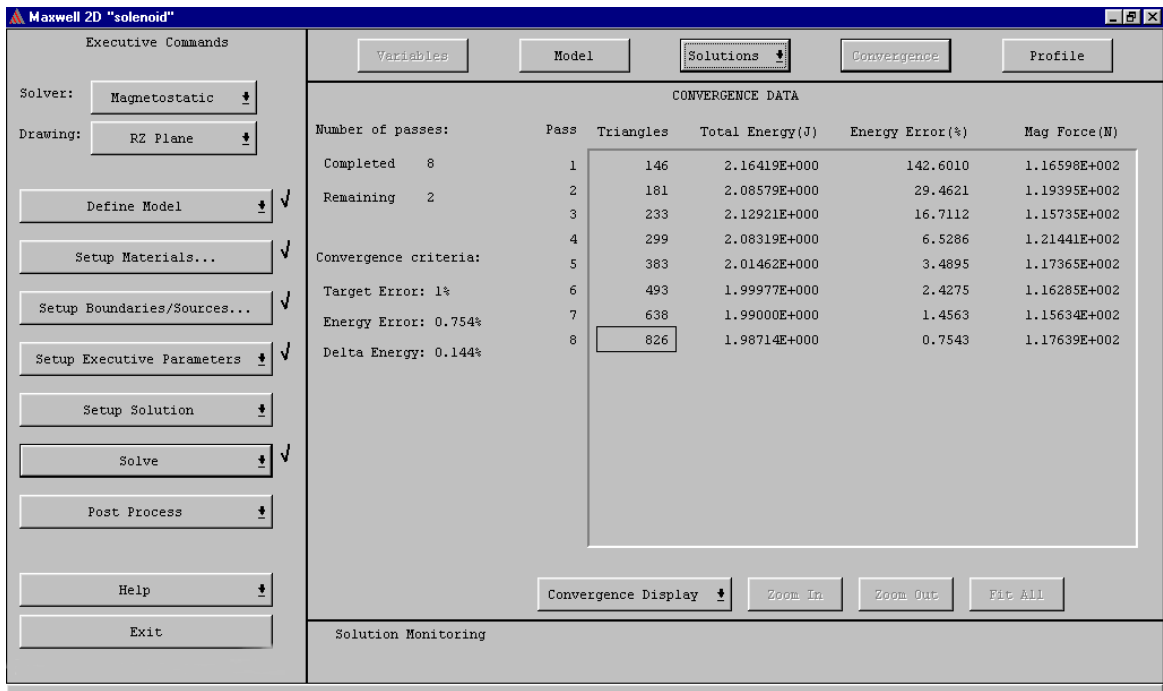
Click **OK** to continue.

You are now ready to view final convergence information for the completed solution.

Note After a solution is generated, the system does not allow you to change the geometry, material properties, or boundary conditions of the model unless you first delete the solution. Therefore, you must generate a new solution if you change the model.

Viewing Final Convergence Data

If convergence data is not displayed on the screen, click **Convergence** again.



Note If a problem does not begin to converge after several adaptive passes, the problem is probably ill-defined — for instance, boundary conditions may not have been specified correctly. If this ever happens, do the following to interrupt the solution process:

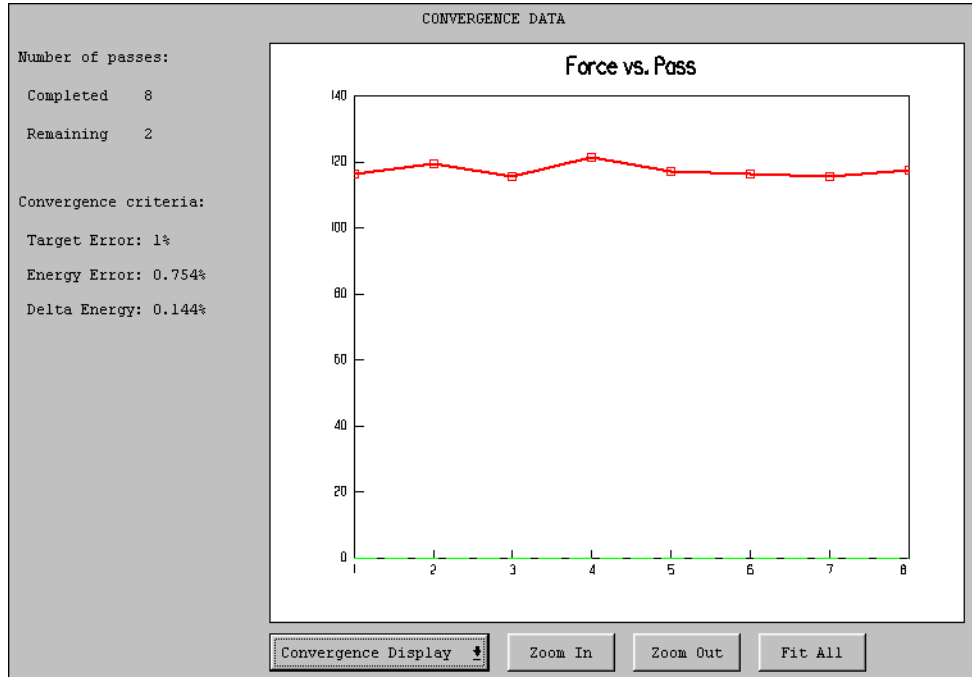
1. Click the **Abort** button in the **Solution Monitoring** box.
2. Check the problem definition, and then solve the problem again.

Plotting Convergence Data

By default, convergence data is displayed in table format as shown in the previous figure. This data can also be displayed graphically.

To plot the magnitude of force computed during each adaptive pass:

- Click **Convergence Display>Plot Force** from the bottom of the display area. The following plot appears:



Displaying this data graphically often makes it easier to see how the solution is converging. For instance, above, the curve begins to level off after the second pass as the total force starts to stabilize.

Optionally, use the other commands under **Convergence Display** to plot the number of triangles, total energy, or percent error for all adaptive passes.

Viewing Statistics

Click **Profile** from the top of the **Executive Commands** menu to see what computing resources were used during the solution process. The following screen appears.

PROFILE INFORMATION				
Command/Info	Real Time	CPU Time	Mem Size	Num Elements
Disk I/O				160 K
Solver RSS	00:00:00	00:00:00	19336K	1444 matrix
Disk I/O				160 K
Solver RSS	00:00:00	00:00:00	19336K	1444 matrix
Disk I/O				160 K
Solver RSS	00:00:01	00:00:00	19336K	1444 matrix
Disk I/O				160 K
Solver RSS	00:00:00	00:00:00	19336K	1444 matrix
Disk I/O				160 K
nlrz2d_solve	00:00:01	00:00:02	19304K	638 triangles
nlrz2d_force	00:00:01	00:00:00	19304K	638 triangles
Pass 8				
mesh_adapt	00:00:00	00:00:00	21592K	826 triangles
Solver RSS	00:00:00	00:00:00	19336K	1822 matrix
Disk I/O				203 K
Solver RSS	00:00:00	00:00:00	19336K	1822 matrix
Disk I/O				203 K
Solver RSS	00:00:00	00:00:00	19336K	1822 matrix
Disk I/O				203 K
Solver RSS	00:00:01	00:00:00	19336K	1822 matrix
Disk I/O				203 K
Solver RSS	00:00:00	00:00:00	19336K	1822 matrix
Disk I/O				203 K
nlrz2d_solve	00:00:03	00:00:03	19304K	826 triangles
nlrz2d_force	00:00:00	00:00:00	19304K	826 triangles
Total	00:00:25	00:00:20	21592K	826 triangles
Finished solenoid on ALT02 in 00:01:24 at 11/19/2002 11:53:07				

The time that the solution process began is displayed at the top of the box. Beneath it, the following information is displayed for each adaptive field solution and mesh refinement step that was completed:

- Command/Info** Displays the name of the system command that was used.
- Real time** Displays the time taken to complete the step.
- Cpu time** Displays the amount of time taken by the CPU (central processing unit) to complete the step.
- Mem size** Displays the amount of memory used.
- Num Elements** Displays the of number of triangles in the finite element mesh.

If more data is available than can fit on a single screen, scroll bars appear. To display more data, manipulate the display as described in Chapter 3, “Creating the Model”

Getting Started: A 2D Magnetostatic Problem

Analyzing the Solution

Now that you have generated a magnetostatic solution for the solenoid problem, you can analyze it using Maxwell SV's post processing features.

The goals for this chapter are to:

- Examine computed force values.
- Plot the magnetic flux and magnetic fields in and around the solenoid.
- Observe the nonlinear properties of the model using B-H curves.

Time This chapter should take approximately 25 minutes to work through.

View Force Solution

Now that the solution is complete, examine the results of the force computation.

To view the force results, click **Solutions>Force**. The final force value computed during the adaptive solution appears as shown below.

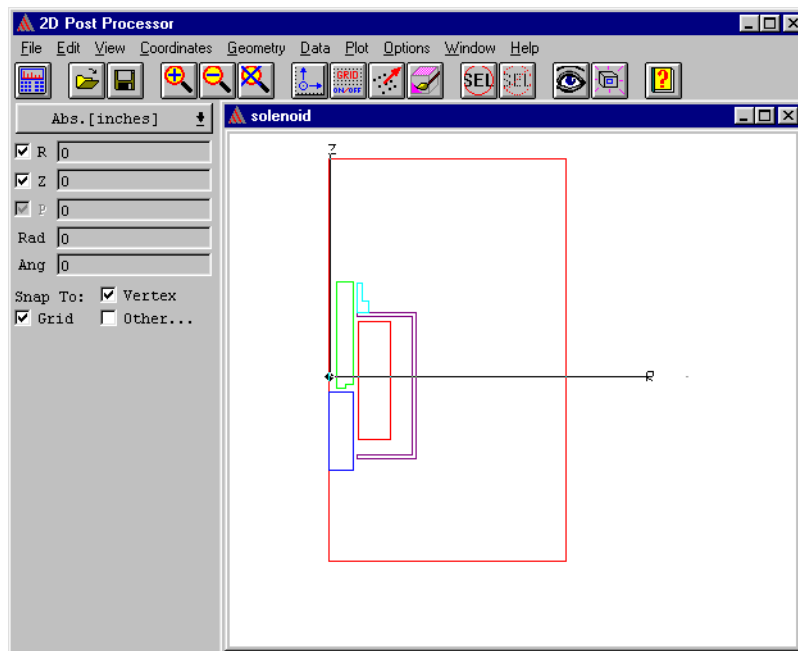
FORCE SOLUTION	
Force Setup Objects	Force Value (newtons)
core	F(Z): -117.639

- The object(s) that were selected for the force computation — in this case, the core — are listed under **Force Setup Objects**.
- The results of the force solution (each force component) are given under **Force Value (newtons)**. The net force on the core is approximately 117.459 newtons, acting in the negative z direction (pulling the core down into the solenoid). There is no component of force in the r direction because of the axial symmetry.

Access the Post Processor

Use the 2D Post Processor to plot the lines of magnetic flux and saturation levels in materials.

To access the Post Processor, click **Post Process**. The 2D Post Processor appears.



Post Processor Screen Layout

This section provides a brief overview of the 2D Post Processor. For further details, refer to the Post Process section in the Maxwell 2D online help.

General Areas

The Post Processor is divided into the following general areas:

- Menu Bar** The menu bar appears at the top of the Post Processor window.
- Viewing Window** The model appears in what is called the viewing window. This window displays the project name, **solenoid**, in its title bar.
- Side Window** This area displays the vertex and snap behavior, and lists the coordinates of the selected points.
- Status Bar** This area displays the following information:
 - The version number of Maxwell SV.
 - The function of each mouse button.

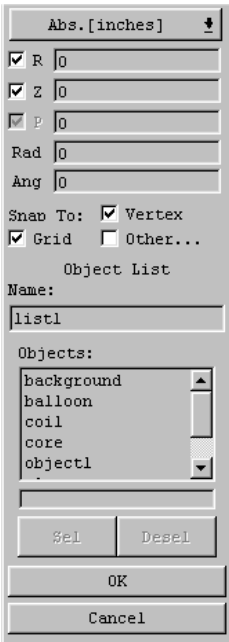
Plot the Magnetic Field

All objects created in the modeler, plus the background, represent the modeling region.

In this project, the balloon boundary is used to model the case where the modeling structure is infinitely far away from other magnetic fields or current sources. Therefore, an object called “balloon” is shown in the **On Geometry** list in the **Create New Plot** window (accessed using the **Plot>Field** command).

To define the problem region for post-processing purposes:

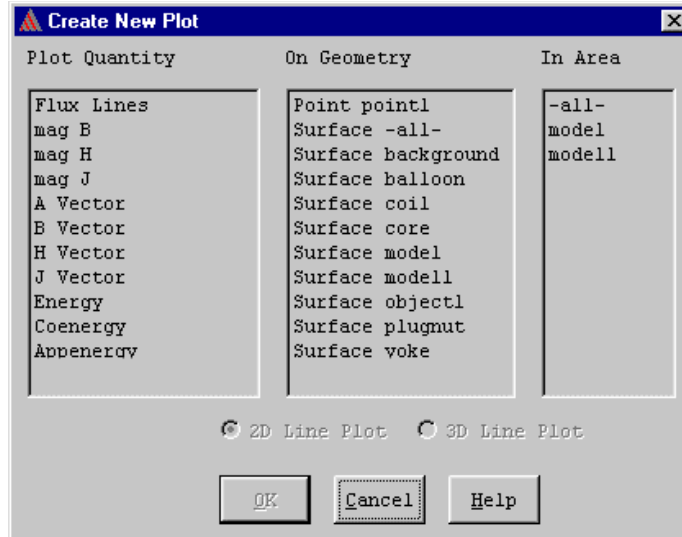
1. Click **Geometry>Create>Object List**. The object list information appears in the bottom half of the side window.
2. Change the **Name** to **model**.
3. In the **Objects** list, select all objects *except* **balloon**, and then click **OK**.



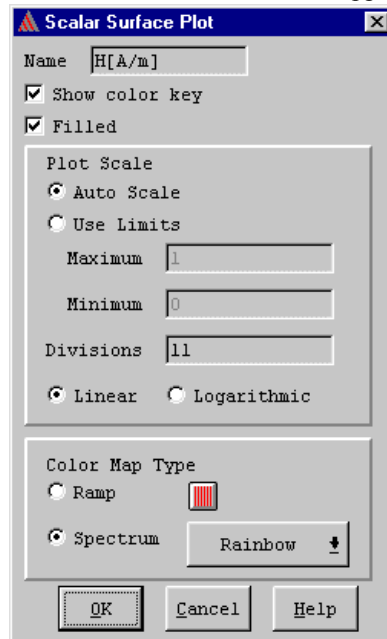
Getting Started: A 2D Magnetostatic Problem

To create a magnetic field plot (showing the magnetic field throughout the problem region):

1. Click **Plot>Field**. The **Create New Plot** window appears.

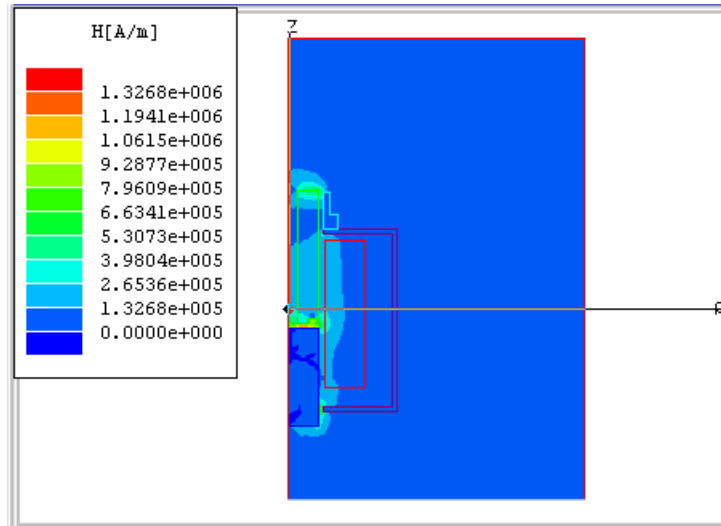


2. Under **Plot Quantity**, select **mag H**. Under **On Geometry**, select **Surface model**. Under **In Area**, select **model**.
3. Click **OK**. The **Scalar Surface Plot** window appears.



4. Make certain that **Show color key** and **Filled** are selected.

5. Click **OK**. The field plot appears. The plot should resemble the following:



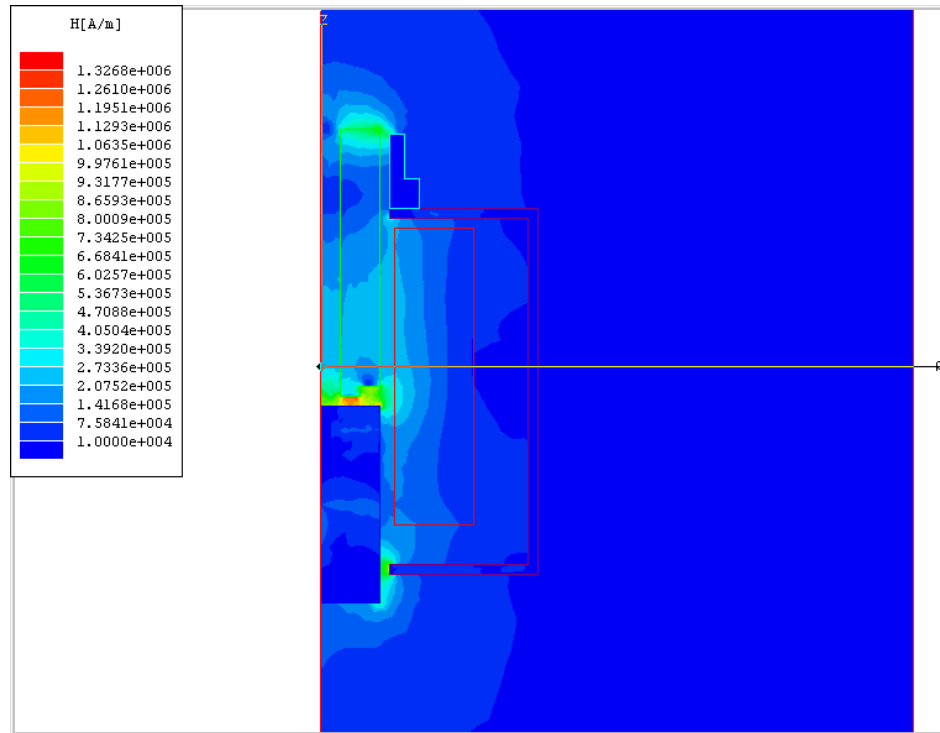
Redisplay the Magnetic Field Plot

Now, redisplay the plot to show a more definitive range of current density distribution. To do this, first zoom in on the model, then redefine the range of values that are displayed.

To redisplay the plot:

1. Click **View>Zoom In**. The cursor changes to crosshairs.
2. Click the left mouse button on a point to the upper left of the solenoid model.
3. Click the left mouse button on a point just to the lower right of the solenoid model. A more detailed image of the plot appears, allowing you to more easily observe the differences in current density.
4. Double-click on the plot key. The **Scalar Surface Plot** window appears.
5. Under **Plot Scale**, select **Use Limits**.
6. Leave the default value in the **Maximum** field.
7. Enter **10000** in the **Minimum** field.
8. Enter **21** in the **Divisions** field to display more layers.

9. Click **OK**. The plot should resemble the following one:



Identify Saturated Materials

As the final step in your analysis of the solenoid model, you will determine saturation levels in the solenoid's plugnut. In magnetically saturated materials, large changes in the magnetic field, \mathbf{H} , produce relatively small changes in the magnetic flux density, \mathbf{B} . In this problem, only the yoke, plugnut, and bonnet were assigned nonlinear materials. For this example, you will plot the saturation levels in the plugnut.

To analyze the saturation level in the plugnut, you will plot the magnetic flux density in the solenoid to see which areas have a high \mathbf{B} -field, and may be saturated. You will then select the object with the highest saturation and plot its \mathbf{B} - \mathbf{H} curve using the **Plot>BH Curves** command.

Plot Magnetic Flux Density

The problem region should still be enlarged as a result of the zoom function in the last step. Now you are ready to plot the magnetic flux density.

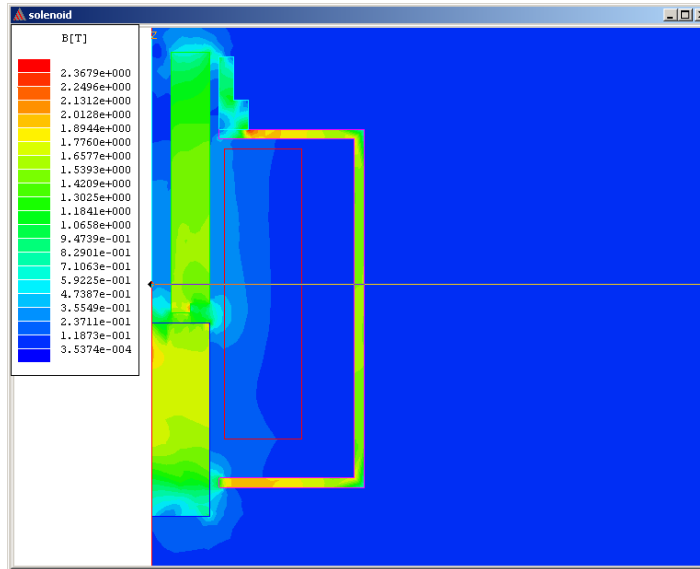
To plot the magnetic flux density:

1. Click **Plot>Visibility**. The **Plot Visibility** window appears.
2. Select **H[A/m]** from the list, and click **OK**. The magnetic field plot is hidden.
3. Click **Plot>Field**. The **Create New Plot** window appears.
4. Under **Plot Quantity**, select **mag B**. Under **On Geometry**, select **Surface model**. Under **In**

Area, select **model**.

5. Click **OK**. The **Scalar Surface Plot** window appears.
6. Under **Plot Scale**, select **Auto Scale**, and click **OK**. The flux plot appears, showing the magnetic flux density of each object in the geometry.

Any areas colored completely green, like the one on the upper-right edge of the plugnut, may be saturated.

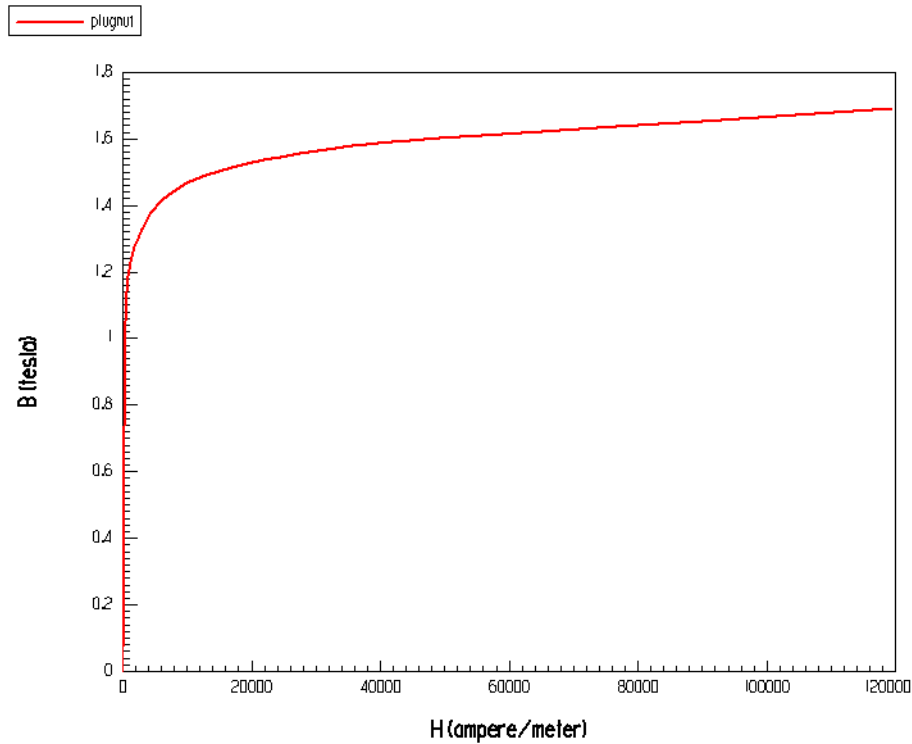


Plot the B-H Curve

Now you can plot \mathbf{H} versus \mathbf{B} , using the **Plot>BH Curves** command, and graphically view the non-linear properties of the plugnut.

To analyze the B-H curve of the plugnut:

1. Click **Plot>BH Curves**. The **Select Object B-H Curves** window appears.
2. Select **plugnut** from the list, and click **OK**. The B-H curve associated with the plugnut appears, as shown below:



When the B-field in a nonlinear material rises above the “knee” of the material’s B-H curve, further increases in \mathbf{H} result in relatively small increases in \mathbf{B} . At this point, the material is said to be magnetically saturated.

Exit the Post Processor

To exit the 2D Post Processor:

1. Click **File>Exit** from the post processor. A window with the following prompt appears:
`Exit Post Processor?`
2. Click **Yes**. The **Executive Commands** window appears.

Exit Maxwell SV

Once you have finished the post processing functions, you are ready to exit the software.

To exit Maxwell SV:

1. Click **Exit** from the bottom of the **Executive Commands** menu. The following prompt appears:
`Exit Maxwell SV?`
2. Click **Yes**. The **Executive Commands** window closes, and the Project Manager reappears.

Exit the Maxwell Software

To exit the Project Manager, click **Exit**. The Project Manager closes, and the Control Panel reappears.

To exit the Maxwell Control Panel:

1. Click **EXIT**. A window with the following prompt appears:
`Exit Maxwell?`
2. Click **Yes**. You return to Microsoft Windows.

You have completed *Getting Started: A 2D Magnetostatic Problem*, the second Getting Started guide for Maxwell SV. (The first guide in the SV Getting Started series is called *Getting Started: A 2D Electrostatic Problem*.)

Getting Started: A 2D Magnetostatic Problem

A

- adaptive analysis 5-3
- Ansoft software, installing 1-iv
- Assign/Boundary commands, Balloon 4-15
- Assign/Source commands, Solid 4-15
- axisymmetric models 1-2, 3-2
 - boundary conditions 4-16
 - drawing size 3-7

B

- background
 - and boundary conditions 4-12
 - ballooning 4-15
 - default material for 4-12
 - definition of 3-9
 - extending 4-15
 - including in solution region 4-12
- balloon boundary
 - assigning 4-15
 - definition of 4-14
- B-H curve
 - adding points to 4-5, 4-10
 - defining 4-4, 4-10
 - fitting curve to points 4-6
 - for cold rolled steel 4-4

- for SS430 4-10
 - keyboard entry 4-5
 - maximum B and H values 4-5, 4-10
 - minimum B and H values 4-5, 4-10
 - plotting saturation levels on 6-8
- B-H Plot (Post Processor Plot menu commands) 6-8
- bonnet
 - assigning a material to 4-3
 - creating 3-14
- boundary conditions 4-12
 - and axisymmetric models 4-16
 - and cartesian models 4-16
 - balloon 4-14, 4-15
 - displaying 4-14
 - for sample problem 4-14
 - screen layout 4-14
 - use of 4-12
- buttons
 - disabled 2-5

C

- cartesian models 3-2
- coil
 - assigning a material to 4-3
 - assigning source current to 4-14, 4-15
 - creating 3-12
 - picking as a current source 4-14

Getting Started: A 2D Magnetostatic Problem

- cold rolled steel
 - defining as a material 4-3
 - entering B-H curve for 4-4
- colors, changing for objects 3-10
- commands
 - accessing from menus 3-3, 4-13
 - accessing with the keyboard 3-3
 - accessing with the mouse 3-3
 - accessing with the toolbar 4-13
- computing resources
 - clock time 5-9
 - commands executed 5-9
 - CPU time 5-9
 - memory size 5-9
 - number of elements 5-9
 - viewing 5-9
- Control Panel
 - icon 2-2
- Convergence 5-5, 5-7
- convergence data
 - delta energy 5-5
 - energy error 5-5, 5-6
 - force 5-6
 - mesh size 5-6
 - number of passes 5-6
 - pass number 5-6
 - plotting 5-8
 - target error 5-5
 - total energy 5-6
 - viewing 5-5, 5-7, 5-8
- Convergence Display, Plot Force 5-8
- coordinate system, uv (local) 3-4
- coordinates
 - entering via keyboard 3-4
 - of cursor 3-4
- core
 - assigning a material to 4-10
 - convergence of force solution 5-6, 5-8
 - creating 3-11
 - selecting for force computation 4-17
- current 4-14
 - and balloon boundaries 4-15
 - assigning to coil 4-14, 4-15

D

- Define Model 1-3, 3-3
 - Draw Model 1-3
 - Group Objects 1-3
- dimensions of sample problem 3-5
- direct solver 5-3
- directories, project
 - creating 2-3
 - selecting 2-4
- Draw Model 1-3
- Drawing 1-3, 3-2
- drawing plane 3-2
- drawing region
 - adjusting view of 2-7, 3-14
 - and axisymmetric models 3-7
 - maximum x- and y-coordinates 3-7
 - minimum x- and y-coordinates 3-7
 - setting up 3-6
 - size of 3-7
- Drawing Size 3-7
- Drawing, RZ Plane 3-2

E

- energy
 - error 5-3, 5-5, 5-6
 - fluctuations in energy error 5-7
 - total 5-6
- error
 - energy 5-5, 5-6
 - fluctuations in energy error 5-7
 - of solution 5-3
- Executive Commands menu 2-7
 - Convergence 5-5, 5-7
 - Define Model 3-3
 - Drawing 3-2
 - Post Process 6-2
 - Profile 5-9
 - Setup Boundaries/Sources 4-12
 - Setup Executive Parameters 4-16
 - Setup Materials 4-2
 - Setup Solution Parameters 5-1, 5-2

- Solutions 6-2
- Solve 5-4
- Solver 3-2
- Exit 3-15, 4-16

F

- field of view
 - fitting all objects in 2-7, 3-14
 - fitting drawing space in 2-7
 - zooming in on 2-7
 - zooming out of 2-7
- fields
 - entering text and numbers in 3-7
 - selecting with the mouse 3-7
- File commands
 - Exit 3-15, 4-16
 - Open 3-4
 - Save 3-13
- files, saving geometry to 3-13
- Fill Solids 2-7
- Fit All 2-7
- Fit Drawing 2-7
- force
 - and solution process 4-16
 - components of 6-2
 - computing for the solenoid core 4-16
 - convergence of solution 5-8
 - direction of 6-2
 - magnetic 5-6
 - magnitude of 5-6, 6-2
 - objects in computation 4-17
 - plotting magnitude of 5-8
 - viewing solution results 6-2

G

- geometric models
 - axisymmetric 1-2, 3-2
 - cartesian 3-2
 - general strategy for creating 3-3
 - saving 3-13
- grid
 - changing point spacing 3-10

- in subwindows 3-4
- modify spacing of 3-8
- snapping to 3-4
- grid points, snapping to 3-4
- Group Objects 1-3

I

- ICCG solver 5-3
- installing Ansoft software 1-iv
- insulated system 4-15

K

- keyboard
 - accessing commands with 3-3
 - selecting points with 3-4, 3-10, 3-14, 4-5, 4-10

M

- magnetic coercivity 4-7
- magnetic field
 - and balloon boundaries 4-15
 - and saturation 6-6
 - solving for 5-4
- magnetic flux density
 - and saturation 6-6
 - plotting 6-6
- magnetic retentivity 4-7
- magnetization
 - relationship to other properties 4-7
 - specifying direction of 4-9
- Material, Add 4-3
- materials
 - adding materials to material database 4-2
 - adding to local database 4-2
 - assigning to objects 4-2
 - assigning to the background 4-12
 - assigning to the bonnet 4-3
 - assigning to the coil 4-3
 - assigning to the core 4-10
 - assigning to the plugnut 4-10
 - assigning to the yoke 4-3
 - material database 4-2

Getting Started: A 2D Magnetostatic Problem

- nonlinear 4-3, 4-10
- permanent magnets 4-7
- Maxwell 2D Field Simulator
 - executive commands 1-3
 - exiting 6-9
 - general procedure 1-3
 - starting 2-6
- Maxwell Control Panel
 - accessing 2-2
 - exiting 6-9
 - Projects command 2-3
- memory used during solution 5-9
- menu bar
 - for 2D Modeler 3-3
 - for Boundary Manager 4-13
 - in Post Processor 6-3
- mesh
 - and solution accuracy 5-2
 - default 5-2
 - example of 1-1
 - initial 5-4
 - number of triangles in 5-6
 - starting 5-2
 - use of 1-1
- mesh refinement
 - adaptive 5-3
 - and error energy 5-3
 - manual 5-2
 - percentage refined 5-3
- message bar
 - during solution process 5-4
 - in 2D modeler 3-4
- Model commands
 - Drawing Size 3-7
 - Drawing Units 3-6
- mouse
 - accessing commands with 3-3
 - selecting objects with 4-3
 - snap-to behavior of 3-4, 3-8
- mouse buttons functions 3-4
- mouse coordinates, displaying 3-4

N

- names, changing for objects 3-10
- Neo35
 - creating 4-7
 - specifying direction of permanent magnetization 4-9
 - specifying properties of 4-8
- nominal problem
 - generating a solution for 5-4
 - post processing 6-2
- nonlinear materials
 - adding 4-3, 4-10
 - defining B-H curve for 4-4, 4-10
 - identifying saturated materials 6-6
 - plotting saturation levels 6-8

O

- Object commands
 - Polyline 3-11, 3-13, 3-14
 - Rectangle 3-9, 3-12
- objects
 - assigning materials to 4-2
 - background 3-9
 - changing the colors of 3-10
 - changing the names of 3-10
 - creating 3-9
 - deselecting 4-3
 - dimensions of 3-5
 - displaying as filled-in solids 2-7
 - displaying as outlines 2-7
 - polylines 3-11, 3-13, 3-14
 - rectangles 3-9
 - selecting 4-3, 4-10
 - selecting for force computation 4-17

P

- permanent magnets
 - creating 4-7
 - direction of magnetization in 4-9
- Pick commands, Object 4-14

- Plot 6-6
 - Plot Force 5-8
 - plots
 - convergence data 5-8
 - force 5-8
 - saturation levels 6-8
 - plots, field
 - common quantities 6-6
 - magnetic flux density 6-6
 - shaded plots 6-6
 - plugnut
 - assigning a material to 4-10
 - creating 3-9
 - viewing saturation levels in 6-6
 - points
 - entering from the keyboard 3-4, 3-10, 3-14, 4-5, 4-10
 - Polyline 3-11, 3-13
 - Post Process 1-4, 6-2
 - post processing 6-2
 - exiting 6-9
 - Post Processor Plot menu commands
 - B-H Plot 6-8
 - Visibility 6-6
 - Profile 5-9
 - Project Manager commands
 - Add 2-3
 - Open 2-6
 - Save Notes 2-5
 - Project Manager, New 2-4
 - projects
 - creating 2-4
 - entering descriptions of 2-5
 - opening 2-6
-
- R**
- Rectangle 3-9, 3-12
 - relative permeability 4-7
 - residual 5-3
-
- S**
- saturation
 - in nonlinear materials 6-6
 - plotting levels of 6-8
 - saving geometry files 3-13, 3-15
 - scientific notation, Ansoft's 4-5
 - scroll bars 5-9
 - Setup Boundaries/Sources 1-3, 4-12
 - Setup Executive Parameters 1-3, 5-3
 - Setup Executive Parameters, Force 4-16
 - Setup Materials 1-3, 4-2
 - Setup Solution Parameters 1-3, 5-1, 5-2
 - size
 - of finite element mesh 5-6, 5-9
 - of memory during solution 5-9
 - of the drawing region 3-4, 3-7
 - snap-to behavior of mouse 3-4, 3-8
 - solutions
 - adaptive analysis of 5-3
 - and adaptive mesh refinement 5-3
 - calculated values 5-2
 - changing boundaries and sources after generating solution 5-7
 - changing geometry after generating solution 5-7
 - changing materials after generating solution 5-7
 - completing 5-6
 - computing 5-4
 - convergence 5-5
 - criteria for computing 5-2
 - effect of mesh 5-2
 - force 6-2
 - interpolated values 5-2
 - interrupting 5-4, 5-8
 - monitoring 5-4
 - plotting 6-6
 - refinement of 5-3
 - satisfying equations 5-3
 - total energy 5-6
 - types 5-3
 - viewing convergence data for 5-7
 - viewing fields 6-1
 - viewing profile data for 5-9
 - Solutions, Force 6-2
 - Solve 1-4, 5-4
 - Solver 3-2
 - AC Conduction 3-2

Getting Started: A 2D Magnetostatic Problem

- DC Conduction 3-2
- Eddy Axial 3-2
- Eddy Current 3-2
- Electrostatic 3-2
- Magnetostatic 3-2
- Solver 1-3
- solver choice 5-3
- solver residual 5-3
- solvers
 - auto 5-3
 - direct 5-3
 - ICCG 5-3
- sources 4-12
 - assigning to objects 4-15
 - current 4-14
 - displaying 4-14
 - for sample problem 4-14
 - screen layout 4-14
- SS430
 - defining material 4-10
 - entering B-H curve for 4-10
- starting mesh 5-2
- status information
 - cursor position 3-4
 - snap-to behavior of mouse 3-4
 - unit of length 3-4
- subwindows
 - and complex geometries 3-4
 - and project windows 3-4
 - coordinate system 3-4
 - drawing grid 3-4
 - size 3-4

T

- toolbar
 - in Boundary Manager 4-13
- triangles
 - manually refining 5-2
 - number in mesh 5-6, 5-9
- 2D Modeler
 - accessing 3-3
 - exiting 3-15
 - screen layout

Index-6

- accessing commands 3-3
- menu bar 3-3
- message bar 3-4
- project window 3-4
- status bar 3-4
- subwindows 3-4

U

- units
 - changing 3-6
 - default 3-4
 - displaying 3-6
 - rescaling geometric models 3-6

V

- vertex points, snapping to 3-4

W

- Window commands (2D Modeler), Grid 3-8
- windows, project 3-4
- Wire Frame 2-7

Y

- yoke
 - assigning a material to 4-3
 - creating 3-13

Z

- Zoom In 2-7
- Zoom Out 2-7