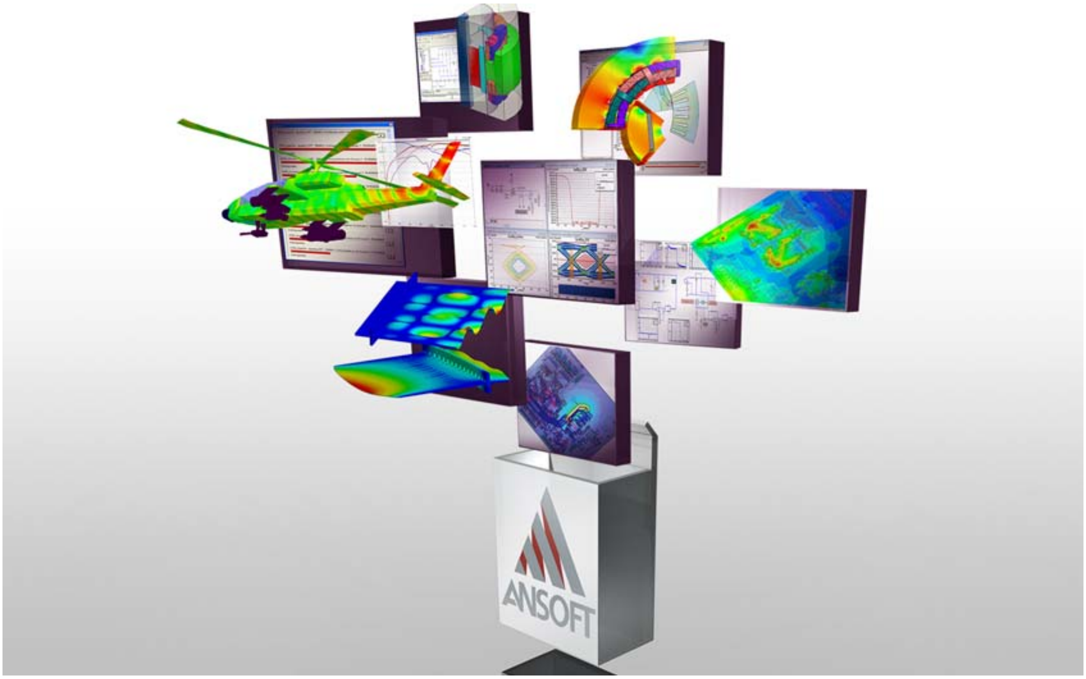


Getting Started with Maxwell:
A 2D Magnetostatic Solenoid Problem

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

November 2010
Inventory: 002880

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

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

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

Conventions Used in this Guide

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

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

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

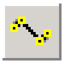
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, click **Maxwell>Excitations>Assign>Voltage**.

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

  - Emphasis.

  - The titles of publications.

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

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

Getting Help

Ansoft Technical Support

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

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

Context-Sensitive Help

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

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

Table of Contents

9. Plotting Results from a Design Variation

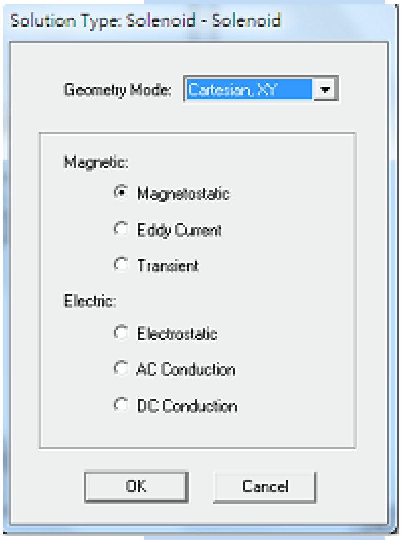
1 Introduction

Maxwell is an interactive software package that uses finite element analysis (FEA) to solve two-dimensional (2D) electromagnetic problems.

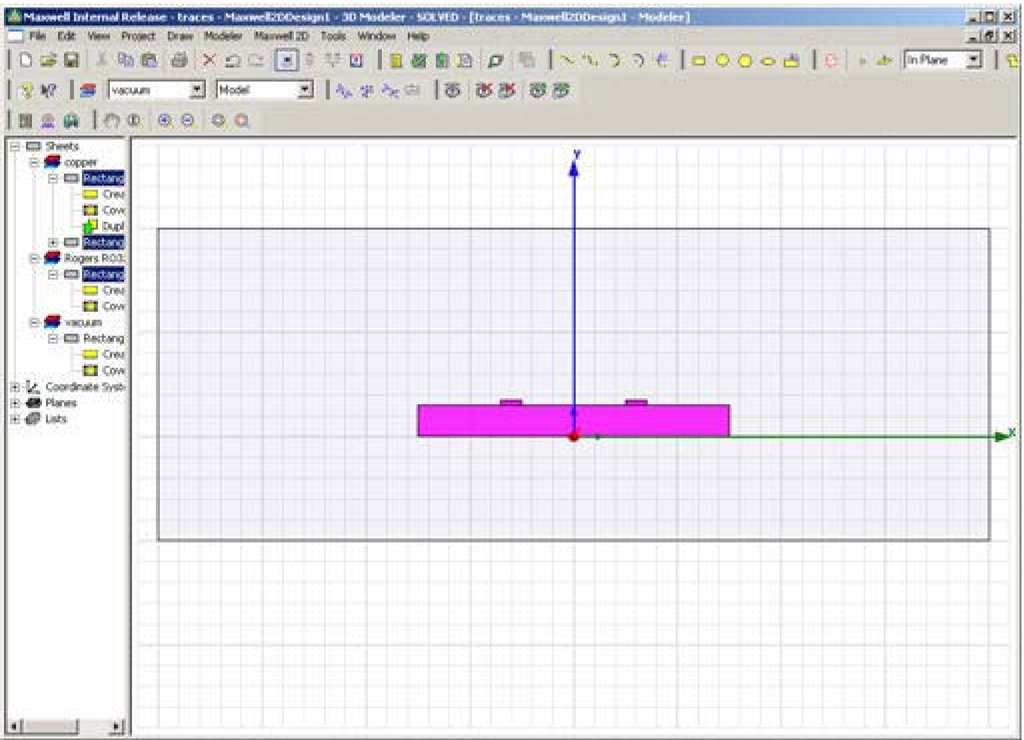
This guide has been structured to be viewed on your screen. The Next and Previous buttons will allow easy movement within the document while you are working on creating this project. Should you prefer a hardcopy, the PDF version is available here:

To analyze a problem, you 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. The specific types of field solutions and quantities that can be computed depend on which Maxwell 2D solution type you specified (electric fields, DC magnetics, AC magnetics, transient fields and data).
- Allows you to analyze, manipulate, and display field solutions.



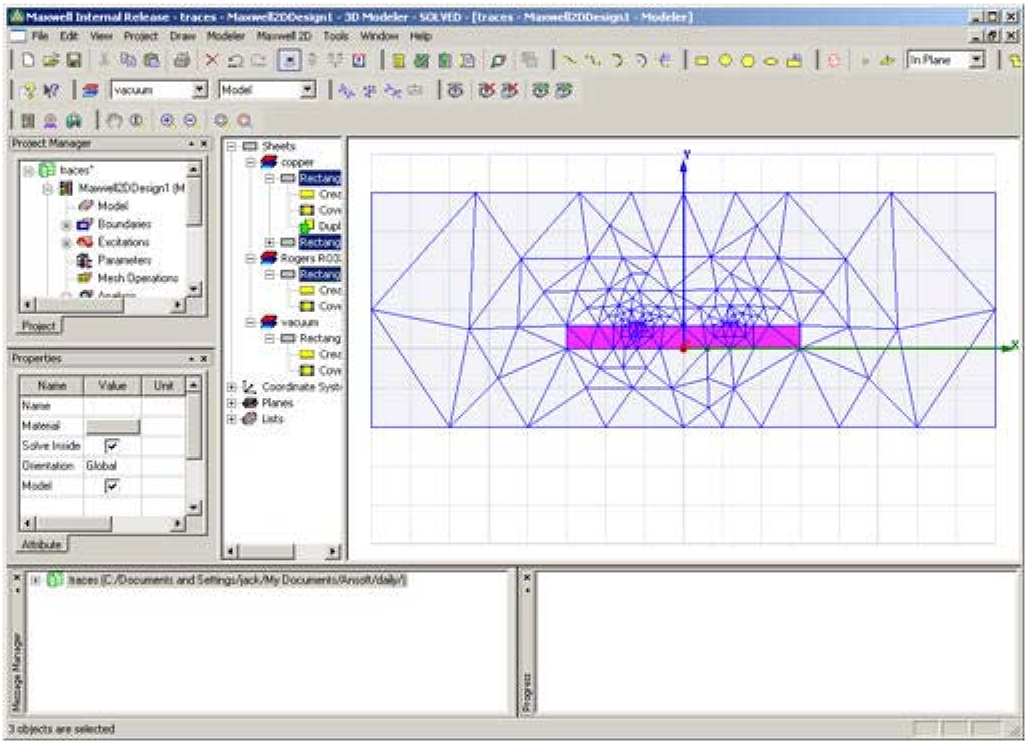
A sample geometry created with Maxwell appears below:



This model is actually a three-dimensional (3D) object. Maxwell 2D analyzes the 2D geometry as a cross-section of the model, then generates a solution for that cross-section.

In addition to XY models, Maxwell 2D may also be used to compute fields in axi-symmetric models to take advantage of 3D geometry that exhibit rotational symmetry about an axis. The geometry described in this guide exhibits such symmetry.

The following figure shows the finite element mesh that was automatically generated for the 2D geometry:

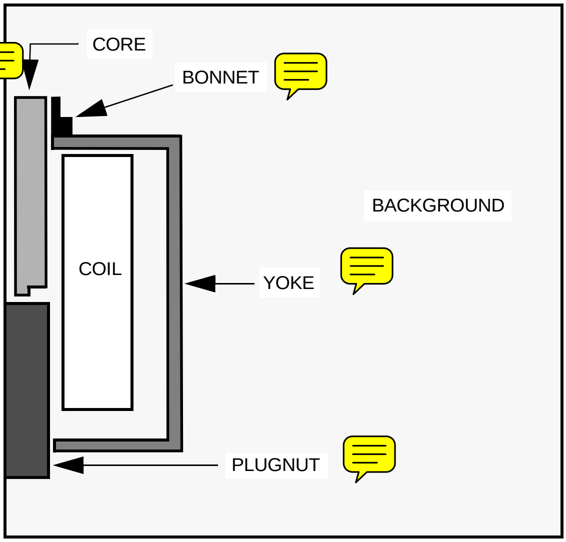


Dividing a structure into many smaller regions (finite elements) allows the system to compute the field solution separately in each element. The smaller the elements, the more accurate the final solution.

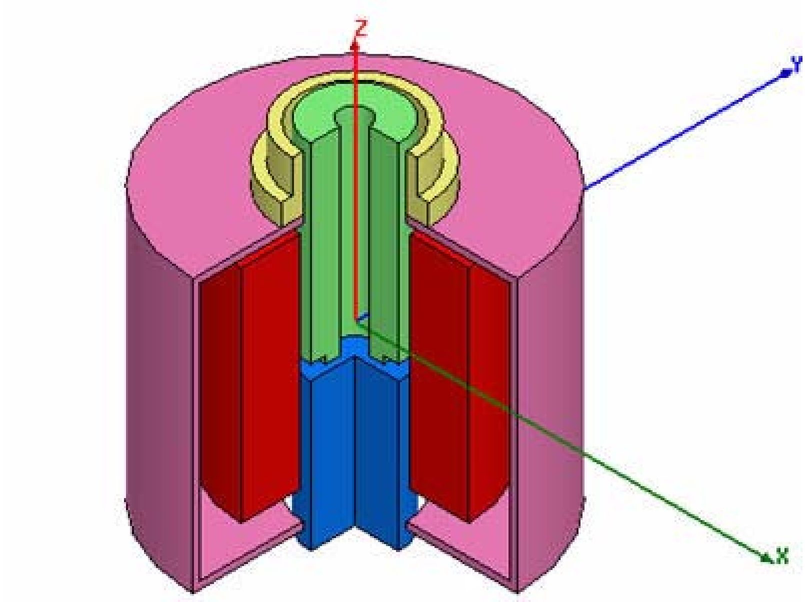
The Sample Problem

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

- Core
- Bonnet
- Coil
- Yoke
- Plugnut



The 2D diagram 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 axi-symmetric model in Maxwell 2D. Of course, material properties, excitations and boundary conditions must also be appropriately modeled by an axi-symmetric design.

Goals

Your goals in *Getting Started with Maxwell: A 2D Magnetostatic Solenoid 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.

You will accomplish these goals by doing the following:

1. Draw the plugnut, core, coil, yoke, and bonnet using the **Modeling** commands.
2. Defining and assigning materials to each object.
3. Defining boundary conditions and current sources required for the solution.
4. Requesting that the force on the core be computed, using the **Parameters** section of the project tree.
5. Specifying solution criteria and generating a solution using the **Add Solution Setup** and **Analyze** commands. You will compute both a magnetostatic field solution and the force on the core.
6. Viewing the results of the force computation.
7. Plotting saturation levels and contours of equal magnetic potential via the **Post Processor**.

This simple problem illustrates the most commonly used features of Maxwell 2D.

2

Setting Up the Design

In this chapter you will complete the following tasks:

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

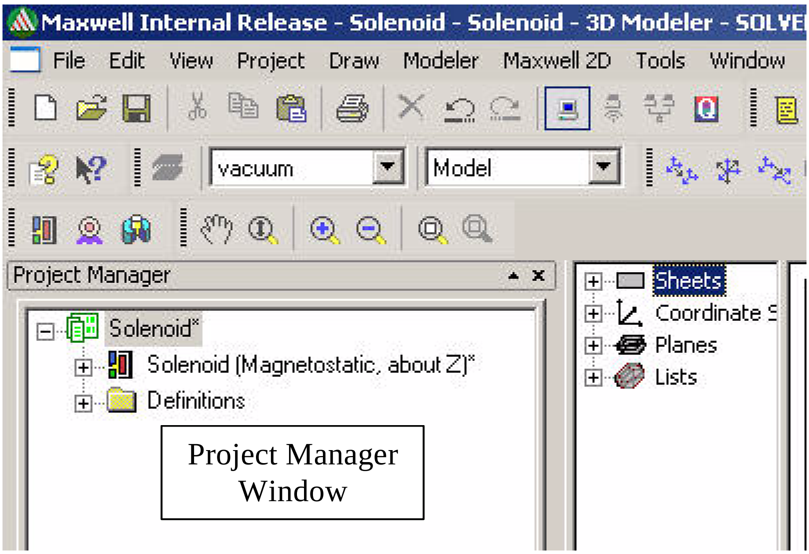
Open Maxwell and Save a New Project

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

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

To open Maxwell and save a project:

1. Double-click the **Maxwell12** icon on your desktop to launch Maxwell.
    - You can also start Maxwell by clicking **Start>All Programs>Ansoft>Maxwell12** from Windows.
2. Click **Project**>**Insert Maxwell 2D Design**.
   The new design is listed in the project tree. By default, it is named **Maxwell2DDesign1** by default. The **Modeler** window appears to the right of the Project Manager (another name for the project tree).



3. Click **File>Save As**.
   The **Save As** dialog box appears.
4. Locate and select the folder in which you want to save the project.
    a. Type **Solenoid** in the **File name** box, and click **Save**.

5 The project is saved in the specified folder under the name **Solenoid.mxwl**. Rename the design:

a. Right-click **Maxwell2DDesign1**.
A shortcut menu appears.

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

c. Type a **Solenoid** as the name for the design, and press **Enter**.
The project and design are now both named **Solenoid**.

Specify a Solution Type

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

1 Click **Maxwell2D>Solution Type** from the menus.

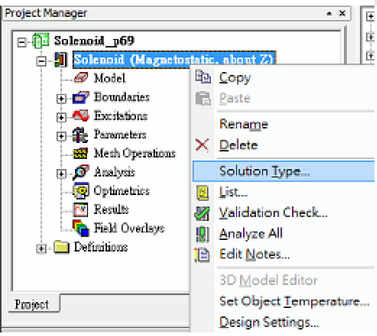
The **Solution Type** dialog box appears.

2 Select the **Magnetostatic** radio button.

3 In the **Geometry Mode** pull-down, select **Cylindrical about Z**.

4 Click **OK**.

Note Many commands in the **Maxwell2D** menu are also available by Right-Clicking on various section of the project tree. For example, Right-Click on **Solenoid (Magnetostatic about Z)** in the project tree and you can select and change the Solution Type from the pop-up menu.



Set the Drawing Units

1 Click **Modeler>Units**.

The **Set Model Units** dialog box appears.

2 Select **in** (inches) from the **Select units** pull-down menu.

3 Click **OK**.

3

Creating the Geometric Model

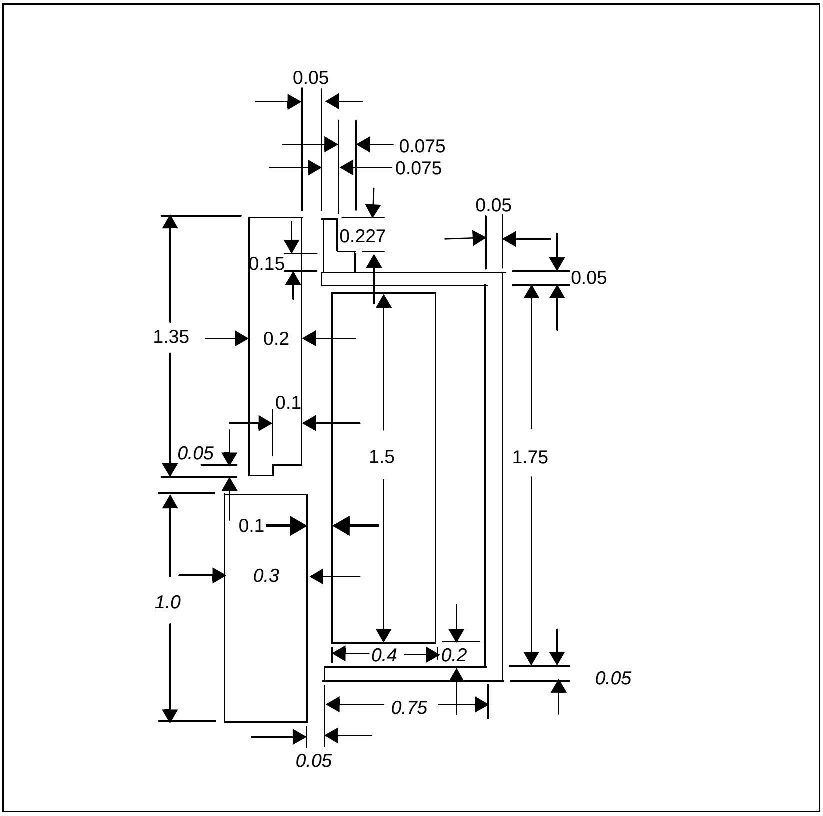
In this chapter you will complete the following tasks:

- Use the rectangle drawing mode to create a solenoid plugnut.
- Create the solenoid's core, yoke and bonnet objects using the polyline command.
- Explore the use of keyboard entry mode in creating the coil, yoke and bonnet objects.
- Create the solenoid coil.
- Create the background object.

Create the Geometry

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

The dimensions of the solenoid that you will be modeling are shown below. For axi-symmetric structures, the axis of symmetry is the Z-axis, and drawing is preformed in the XZ plane will all objects in the X>0 portion of the plane. You will use these dimensions, which are given in inches, to create the geometric model.



The following pages step you through drawing the above model.

Draw the Plugnut

First, draw the plugnut using the rectangle command.

To create the plugnut:

1 Click **Draw>Rectangle**.

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

2 Select one corner of the rectangle by clicking at the **(0,0,-0.2)** location, and press the **TAB** key to jump to the manual entry area in the Status Bar at the bottom of the screen.

3 Notice the Status Bar is prompting for the Opposite corner of the Rectangle. Type **0.3** in the **dX** box, ensure that **dY** is set to **zero**, and type **-1.0** in the **dZ** box. Press **Enter** to complete the creation of the rectangle. The default properties appear in the **Properties Window**.

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

4 In the **Attribute** tab, change the **Name** (currently **Rectangle1**) to **Plugnut** by clicking on **Rectangle1**. The field becomes editable and you can enter **Plugnut** as the new object name.

5 Optionally change the color of the rectangle to **blue**:

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

b. Select any of the blue shades from the **Basic colors** group, and click **OK** to return to the **Properties** window. The object color change will not be apparent while it is currently selected.

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

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

7 Optionally, when using the pop-up **Properties** dialog, click **OK** to close the **Properties** window. A rectangle named **Plugnut** is now part of the model.

Draw the Core

Next, use the polyline command to create the core.

To create the core:

1. First you need to adjust the grid settings by clicking **View>Grid Settings**. In the Grid Spacing dialog de-select the **Auto adjust density** checkbox and enter **0.05** in each of the **dX**, **dY**, and **dZ** value fields.
2. Click **Draw>Line**.
   The cursor changes to a small black box, indicating that you are in **Drawing** mode.
3. Select the first vertex by clicking at the **(0.1, 0, 1.2)** location. The first vertex is locked in and you may now move the cursor to the location of the second vertex **(0.1, 0, -0.15)**.

Note The first vertex may be outside the viewable drawing area. If so, hold down the **Shift** key and move the mouse slightly. The mode will switch to **Pan** mode. While continuing to hold the **Shift** key down, press and hold the left mouse button. Drag the mouse to pan the modeler window and make more of the positive **Z-axis** available. Releasing the mouse button and shift key exits **Pan** mode automatically.

4. Continue creating the core object by clicking at each of the points in the table in succession.

Table 1:

X	Y	Z
0.2	0	-0.15
0.2	0	-0.1
0.3	0	-0.1
0.3	0	1.2
0.1	0	1.2

Table 1:

X	Y	Z
0.1	0	1.2

Note Selecting the final vertex twice causes the modeler to end the polyline creation process. Since the polyline creates a closed polygon, a 2D sheet object is automatically created from the series of vertices.

5. After Entering the final vertex twice, an object named polyline1 is created. The **Properties** window contains the default information for the newly created object.
6. Click the **Attribute** tab.
7. Change the **Name** to **Core**.
8. Optionally change the color of the **Core** object to **green**.

Keyboard Entry

When creating objects in the modeler, you may use the cursor or manually enter coordinates. Manual entry is particularly useful when the dimensions fall between the grid spacing.

In order to manually enter points, click the **TAB** key and the system focus is moved from the drawing window to the keyboard entry area of the status bar allowing entry of direct vertices or dimensions depending upon the object being created. The status bar also contains prompts to assist with the manual entry process.

Once in manual entry mode, you may continue to press the **TAB** key to switch between the entry fields.

Draw the Coil

Now draw the coil object using the keyboard entry technique discussed in the previous section.

To create the coil:

1. Click **Draw>Rectangle**.

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

2 Press the **TAB** key to switch the focus to the keyboard entry area at the bottom of the screen.

Note Do not move the mouse once the focus has switched to the keyboard entry area or the system will revert back to mouse entry mode and any data that has been manually entered will be lost.

3 Enter the coordinate **(0.375, 0, 0.7)** for the rectangle position. Pressing the **TAB** key switches between the fields for easy data entry. Press **Enter** once the coordinate data is entered.

4 The Status Bar now prompts for the opposite corner of the rectangle. Using the **TAB** key to switch between fields, enter the values for the dimensions of the coil. Type **0.4** for **dX**, **0** for **dY**, and **-1.5** for **dZ**. Press **Enter** to complete the creation of the rectangle. The default properties appear in the **Properties Window**.

Note You may enter the exact location of the opposite corner of the rectangle by switching the entry mode from relative to absolute using the pull-down listbox.

5 In the **Attribute** tab, change the **Name** (currently **Rectangle1**) to **Coil**.

6 Optionally change the color of the rectangle to **Red**:

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

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

Draw the Yoke

Next, use the polyline command to create the yoke of the solenoid.

To create the yoke:

1 Click **Draw>Line**.

Press the **TAB** key to enter keyboard entry mode.

2 Create the yoke object by entering each data point from the table below followed by the **Enter** key.

Table 2:

X	Y	Y
0.35	0	-1.05
1.1	0	-1.05
1.1	0	0.8
0.35	0	0.8
0.35	0	0.75
1.05	0	0.75
1.05	0	-1.0
0.35	0	-1.0
0.35	0	-1.05

3 After Entering the final vertex, press the **Enter** key twice. An object named polyline1 is created. The **Properties** window contains the default information for the newly created object.

Note Selecting the final vertex twice causes the modeler to end the polyline creation process. Since the polyline creates a closed polygon, a 2D sheet object is automatically created from the series of vertices.

4 Click the **Attribute** tab.

5 Change the **Name** to **Yoke**.

6 Optionally change the color of the **Yoke** object to purple.

Draw the Bonnet

As in the previous section, use the polyline command in keyboard entry mode to create the bonnet of the solenoid.

To create the bonnet:

1. Click **Draw>Line**.

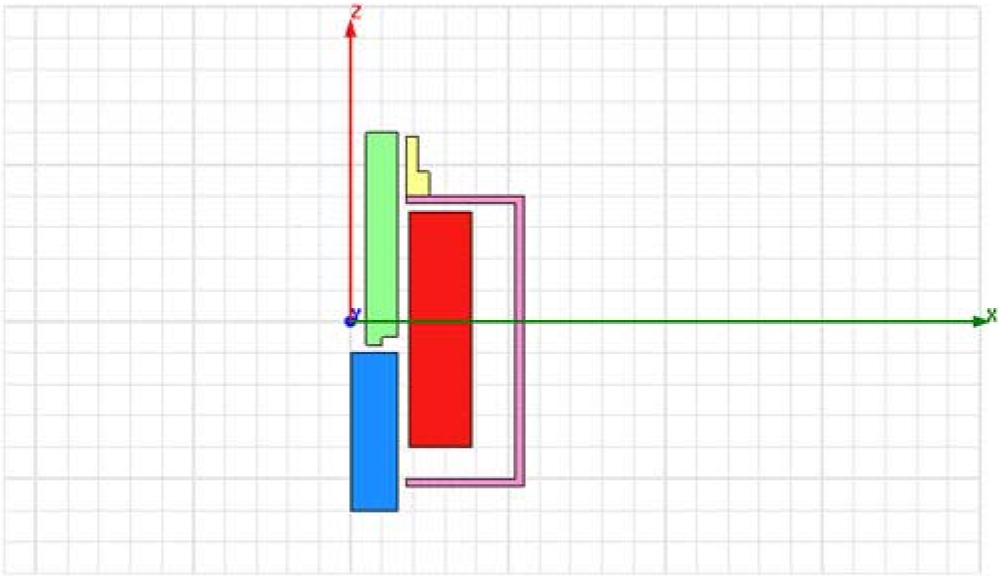
   Press the **TAB** key to enter keyboard entry mode.

2. Create the bonnet object by entering each data point from the table below followed by the **Enter** key.

Table 3:

X	Y	Y
0.35	0	0.8
0.5	0	0.8
0.5	0	0.95
0.425	0	0.95
0.425	0	1.177
0.35	0	1.177
0.35	0	0.8

3. After Entering the final vertex, press the **Enter** key twice. An object named polyline1 is created. The **Properties** window contains the default information for the newly created object.
4. Click the **Attribute** tab.
5. Change the **Name** to **Bonnet**.
6. Optionally change the color of the **Bonnet** object to light blue.
7. The geometric model should appear as shown in the following graphic:



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

Create the Background (Region)

Define a background region box with the origin at (0, 0, -2) and the dimensions of (2, 0, 4).

To create the background region box:

1 Click **Draw>Rectangle**.

2 Type the box position **(0, 0, -2)** in the **X**, **Y**, and **Z** fields at the bottom of the screen, and then press **Enter**.

3 Type the box size **(2, 0, 4)** in the **dX**, **dY**, **dZ** fields, and then press **Enter**.

The **Properties** window contains the default information for the newly created object.

4 Click the **Attribute** tab.

5 Change the **Name** (currently **Rectangle1**) to **bgnd**.

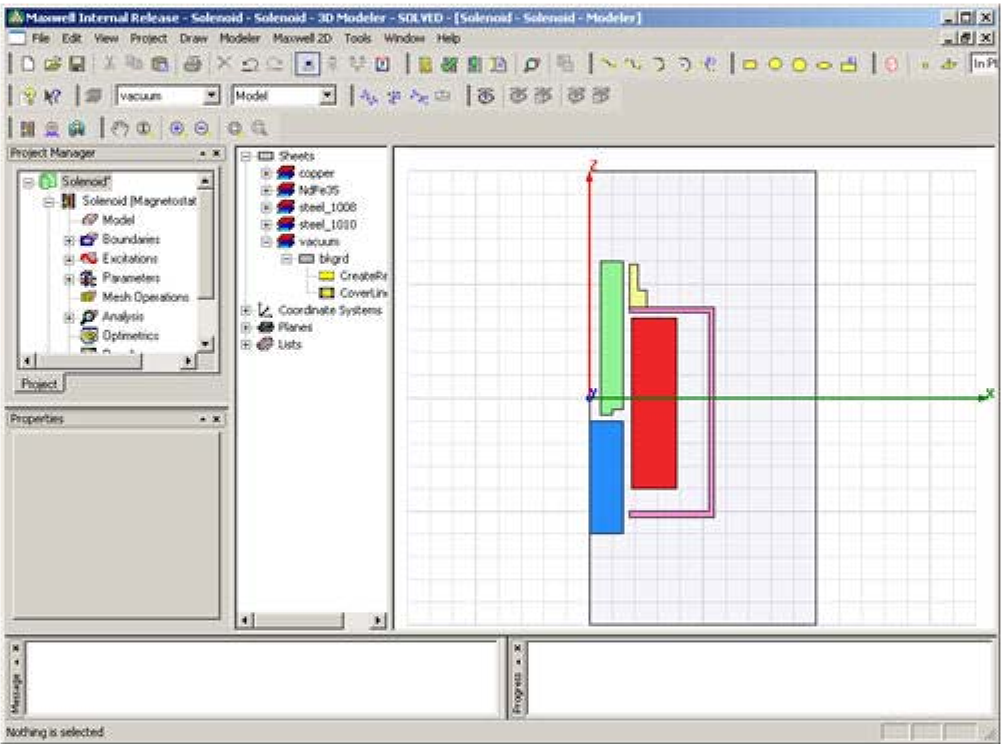
6 Set the transparency to **0.9**:

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

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

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

7 The final geometry should look similar to the following:



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

4

Setting Up the Solenoid Model

In this chapter you will complete the following tasks:

- 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.

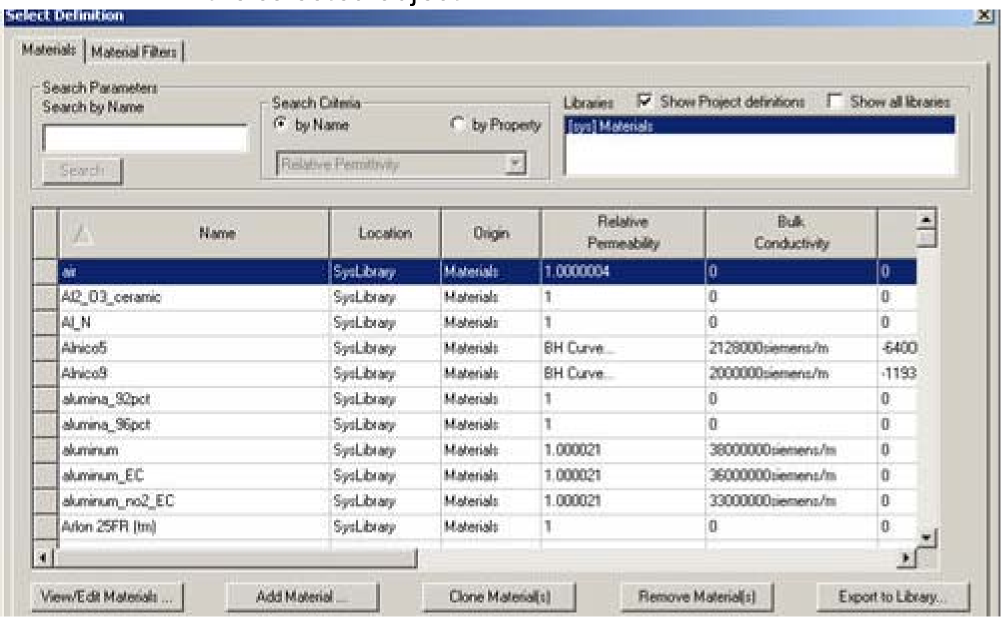
Assign Materials to Objects

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 **Properties Window**. 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 Database

To access the **Material Database**, select the coil object by clicking in the **Model Window** or by selecting it in the history tree. Once you have selected the coil, click in the Material Value field of the **Properties Window**. The Select Definition dialog appears to allow material definitions to be assigned to the selected object.



Assign Copper to the Coil

In the actual solenoid, the coil is made of copper. Scroll the database table down to the definition of copper in the Material Manager and select it by clicking on the name field. Then click **OK**. The Properties Window will now show copper in the Material Value field for the coil object and **coil** will be listed under copper in the **Object** list.

The general procedure to assign a material to an object:

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

   Both the object and its name are highlighted.
2. Select the Material Value field in the **Properties Window**.
3. Scroll to and select the material of interest from the **Material database** listing. You may also use the Search Parameters section of the dialog to narrow the search or jump quickly through the database listing.
4. Click the **OK** button to complete the assignment.

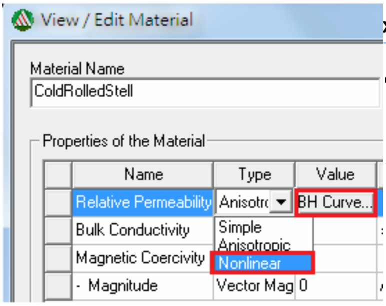
Assign Cold Rolled Steel to the Bonnet and Yoke

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. Select both the **bonnet** and **yoke** (click on the **bonnet** and then, while holding the **Ctrl** key down, click on the **yoke**).
2. Click the **Material Value** field from the **Properties Window**.
3. Select **Add Material**. The **View/Edit Material** dialog appears.



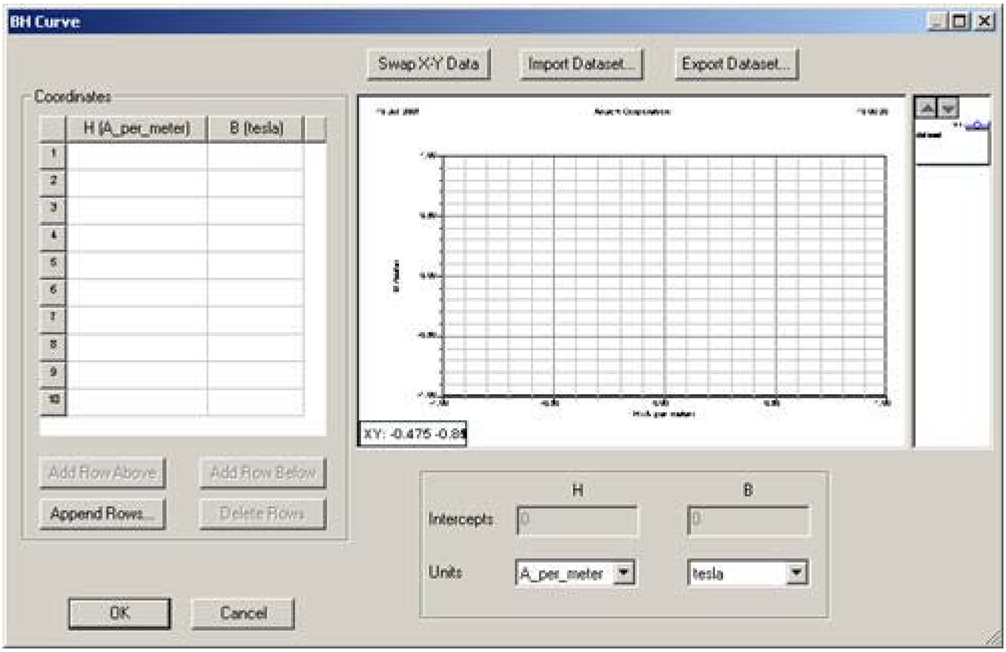
4 Under **Material Name**, change the name of the new material from **Material1** to **ColdRolledSteel**.

5 Change the **Relative Permeability** type field from Simple to Nonlinear by clicking in the Type field to view the available options and selecting Nonlinear. The **Rel. Permeability** value field changes to a button labeled **BH Curve**.

Define the B-H Curve For Cold Rolled Steel

To define the properties of cold rolled steel, use the **BH Curve** button to define a B-H curve giving the relationship between **B** and **H** in the material.

To define the B-H curve, click **BH Curve** in the **Relative Permeability** value field. 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 2D models.
- Maxwell 3D can read B-H curve files that have been exported from Maxwell, enabling you to use the same curves for both 2D and 3D models.

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

Add B-H Curve Points for Cold Rolled Steel

To enter the points in the B-H curve:

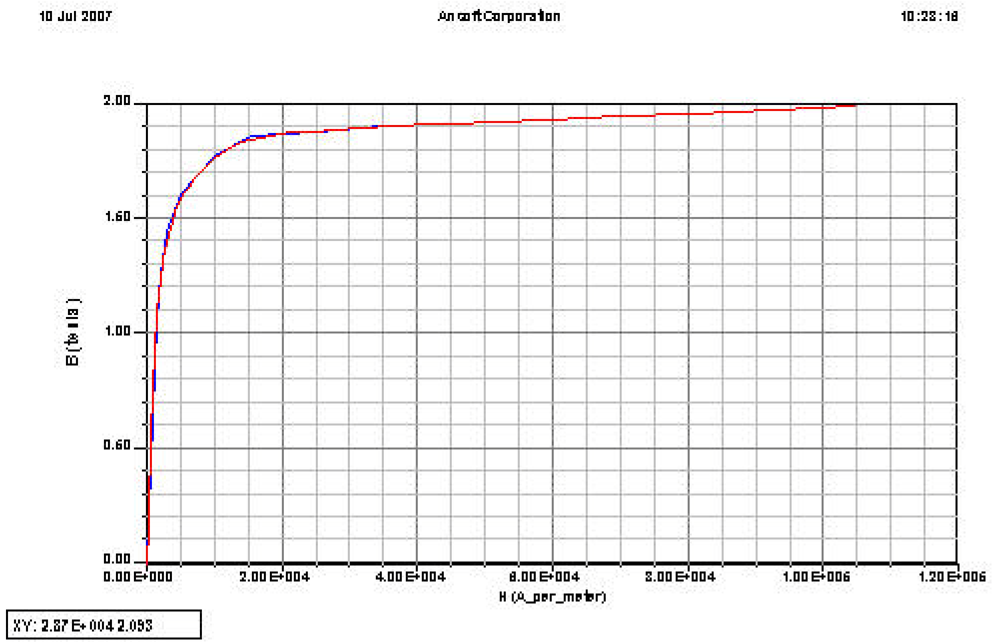
1. Click in the table entry area under the H column, row 1.
2. Enter **0.0** for the minimum H value and press the **TAB** key.
3. Enter **0.0** for the minimum B value and press the **TAB** key.
4. Enter the rest of the values for the B-H curve from the table below, using the **TAB** key to accept the entry and move to the next available cell.

Table 4:

H	B
0.0	0.0
1080	0.858
1480	1.06
2090	1.26
3120	1.44
5160	1.61
9930	1.77
1.55e4	1.86
2.50e4	1.88
3.50e4	1.90

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.

5. The graph automatically updates as each data point is entered. 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. After you enter the last value press **Enter** to exit data entry mode and click **OK** to return to the **View/Edit Material** window.

Assign ColdRolledSteel to the Yoke and Bonnet

Now that you have completed the B-H curve entry, enter **ColdRolledSteel** into the database and assign it to the selected objects:

To save and assign the new material:

1. Click **OK** in the **View/Edit Material** dialog 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 then appears highlighted in the database listing. The word **Project** appears next to it in the location field, indicating that this material is specific to the **solenoid** project.
2. Click **OK** to assign **ColdRolledSteel** to the yoke and bonnet.

Assign Neo35 to the Core

Next, create a 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. However, for this problem, a permanent magnet is assigned to the core to demonstrate how to set up a permanent magnetic material.

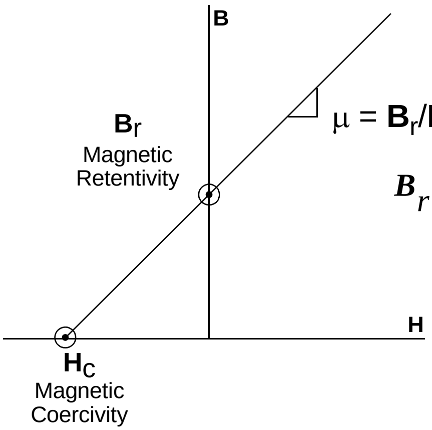
Select the Object and Create the Material

To select the core and create the material:

1. Select **core** from the **Object** list.
2. Click the Material Value field in the **Properties Window**.
3. In the Select Definition dialog, select **Add Material**. The **View /Edit Material** dialog is displayed.
4. Under **Material Name**, 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. In Maxwell, you may enter only the Permeability(m) and the Magnetic Coercivity(H_c). The values of the other two properties are dependent upon these properties, according to the relationships shown below.



$$\mu = B_r/H_c$$

$$B_r = \mu_o M_p = \mu_o \mu_r H_c$$

The values of the other two properties, magnetic retentivity, B_r, and magnetization, M_p, are computed using the relationships shown above. You may use the "Calculate Properties for" pull down to calculate one set of properties from the other.

To calculate the properties to be entered:

1. Select the **Calculate Properties for...** pull down list and choose **Permanent Magnet**. The **Properties for Permanent Magnet** window appears.
2. Enter **1.05** in the **Rel. Permeability (Mu)** field but do not press **Enter**.
3. Click on the check box next to **Hc** to de-select it.
4. Click on the check box next to **Br/Mp** to select it.
5. Enter **1.25** in the **Mag. Retentivity (Br)** field and press the TAB key or use the mouse to change the dialog focus. Values automatically appear in the remaining fields.
6. Press **OK** to accept all the values and close the dialog. When using the Calculate Properties dialog, all data is automatically transferred when the dialog closes.

Enter Material Properties

The View/Edit Material dialog should now have the values for **Relative Permeability** and **Magnetic Coercivity** entered in the dialog.

Optionally **- To enter the properties for Neo35 manually:**

1. Enter **1.05** in the **Rel. Permeability (Mu)** field.
2. Enter **-947350.85** in the **Mag. Coercivity Magnitude (Hc)** field.

   Since the **Mag. Coercivity** is a vector quantity, the dialog will update with entry fields for the **X**, **Y**, and **Z** vector components to specify the direction of the vector.

Note By default, most material properties in the database shipped with Maxwell will be oriented along the x-axis **(1, 0, 0)** when a vector orientation is required. Other orientations require the user to create a coordinate system and align the material with it in order to change the orientation vector once it has been specified in the material database.

   Also, verify that the **Material Coordinate System Type** list-box at the top of the dialog is set to Cartesian.

3. Click **OK**. **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^{o} angle from the default.

To assign **Neo35** to the core:

1. Make certain that **Neo35** is highlighted in the **Material** list.
2. Click **OK** in the **Select Definition** dialog to assign the material **Neo35** to the **core** object. In the **Properties Window**, the Orientation property is automatically set to **Global**.

Create a Relative Coordinate System for the Magnet Orientation

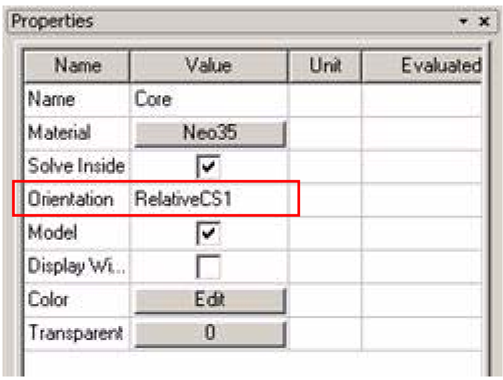
Since material properties must be aligned with a coordinate system, you will now create one that is rotated 90^{o}.

1. Click **Modeler>Coordinate System>Create>RelativeCS>Rotated**.
2. Using the mouse, align the **X-axis** of the new coordinate system with the **Z-axis** of the global coordinate system and click the mouse. A new coordinate system **RelativeCS1** is created and automatically set to be the current working coordinate system.

Note You may also use keyboard entry mode to specify the new coordinate system by pressing **TAB** on your keyboard. This will shift the focus to the data entry area at the bottom of the screen where you can enter exact values for the coordinate point, in this case (0, 0, 1). Pressing the **TAB** key also allows you to navigate between the **X**, **Y** and **Z** entry boxes quickly.

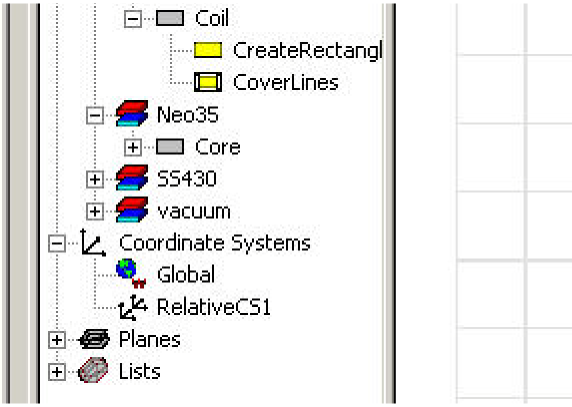
Complete the Alignment of the Magnet

1. Make sure the **core** object is selected.
2. In the **Properties Window** select the Value cell next to **Orientation**. A pop up selection is displayed listing the coordinate systems available in the project.



3. Select **RelativeCS1**. This completes the alignment of the Magnetic Coercivity for the **core** object.

4 You may wish to return to the Global Coordinate system. To do so, click on **Global** in the history tree under **Coordinate Systems** as shown in the figure below.



Create SS430 Material and Assign to 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 included 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 the Material Value field in the **Properties Window**.

3 In the **Select Definition** dialog, click on **Add Material**.

4 Under **Material Name**, change the name of the new material to **SS430**.

5 Select the **Relative Permeability** type field and change it from Simple to Nonlinear. The value field changes to a button labeled **BH Curve**.

Define the B-H Curve for SS430

To define the B-H curve for SS430, click the **B-H curve** button. The **B-H Curve Entry** window appears.

Add B-H Curve Points for SS430

Enter the points in the B-H curve according to the table below:

1. Select the **H** column, row 1 and enter **0.0**. Press the **TAB** key to accept the entry and move to the next cell.
2. Use the **Append Rows** button to add **9** additional rows to the table to accommodate the data below.
3. Enter the following points using keyboard entry.

Table 5:

H	B
0.0	0.0
143	0.125
180	0.206
219	0.394
259	0.589
298	0.743
338	0.853
378	0.932
438	1.01
517	1.08
597	1.11
716	1.16
955	1.20
1590	1.27

Table 5:

H	B
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 press **Enter** to accept the last data point and click **OK** to return to the **View/Edit Material** window.

Assign SS430 to the Plugnut

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

To save and assign **SS430**:

1 Click **OK** in the **View/Edit Material** dialog to save the material properties you have entered for **SS430** — including the B-H curve you have just defined — and add it to the material database.

2 Make certain **SS430** is highlighted. Click **OK** in the **Select Definition** dialog to save the material attributes you entered for **SS430** and assign it to the **plugnut** object

Accept Default Material for Background

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

- The object **bgnd** is the only object that will use the material assigned by default. At the time an object is created, a default material is assigned and is visible in the **Properties Window**.
- The default material, vacuum, is acceptable to use for the **bgnd** in this model.

Note During the model creation process, the material may be assigned immediately to the object before continuing to create the next object. This may have some advantages since copying an object with a material assigned will preserve the material assignment for the copy objects and may reduce the need for material assignments.

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 through the Boundaries and Excitations entries in the Project tree or through the **Maxwell2D>Boundaries** and **Maxwell2D>Excitations** menus respectively.

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, this exposed surface is that of the object **bgnd**. 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 2D 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.

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.
Current source	Specifies the DC source current flowing through an object in the model.

You will assign boundary conditions and sources to the following objects in the solenoid geometry:

bgnd	At the outer boundary of the problem region, the outside edges of this surface are to be ballooned to simulate an insulated system. The edge along the Z axis will not be assigned since this is the axis of rotation for this Cylindrical about Z problem.
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.

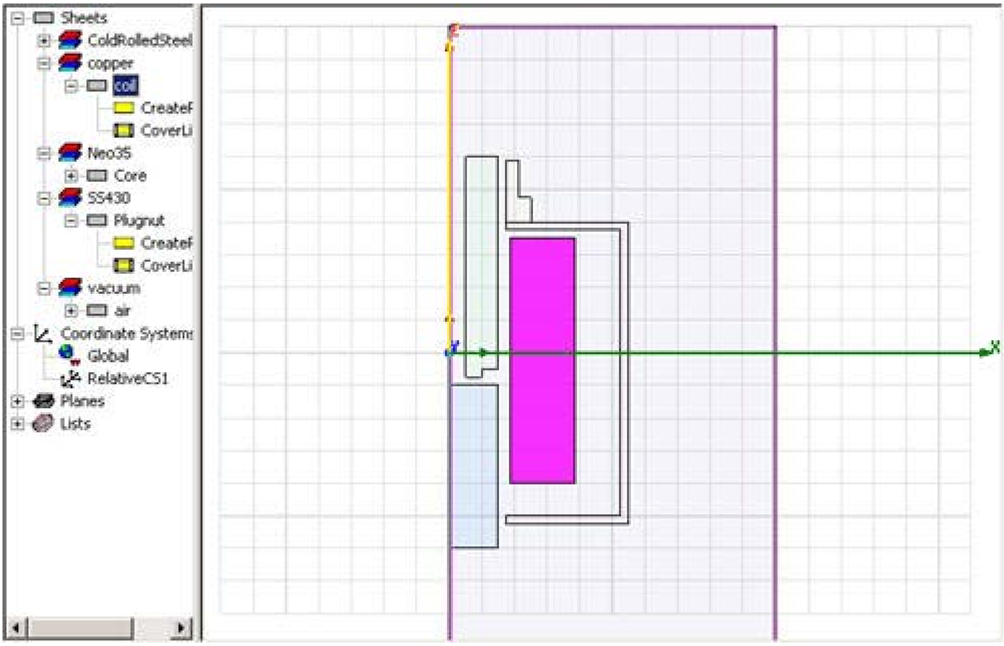
Set Source Current on the Coil

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.

Assign a Current Source to the Coil

To specify the current flowing through the coil:

1. Select the **coil** object by clicking on the object in the modeler window or selecting the object in the history tree.



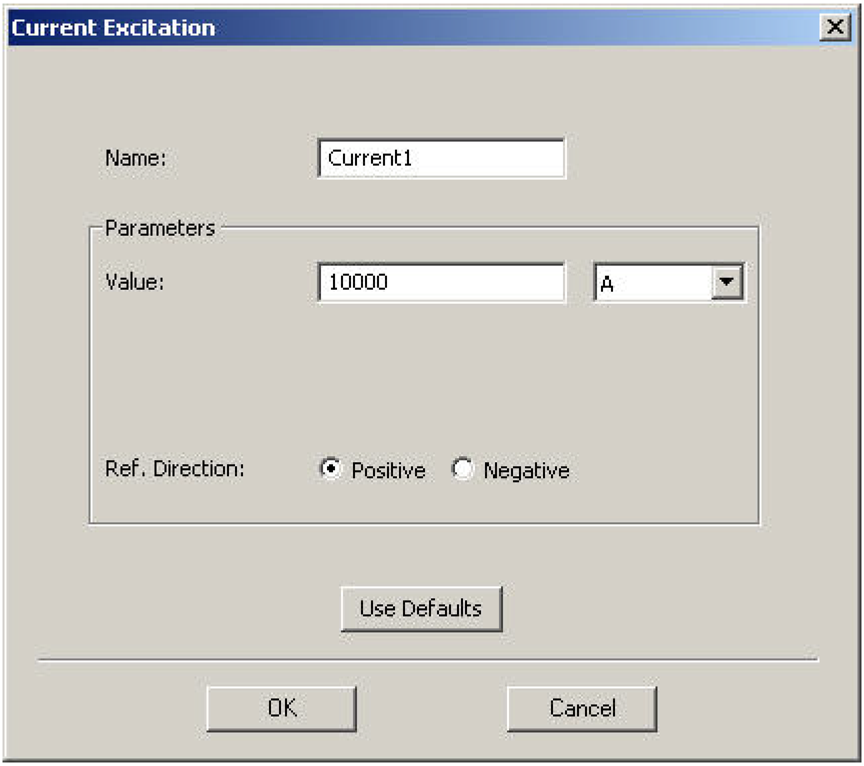
2. Click **Maxwell2D>Excitations>Assign>Current**. The Current Excitation dialog appears.

3 Enter **10000** in the Value field and ensure the units are set to Amps.

4 Ensure the **Ref. Direction** is set to **Positive** to indicate current flowing in the positive PHI direction, in this case, into the screen.

Note In Magnetostatic problems, the current is always distributed uniformly through the object. In addition, **Positive** is in the positive Z direction for XY problems, and positive PHI for RZ problems.

5 Click **OK** to complete the assignment of the source named **Current1** to the **coil** object.



6 **Current1** is now listed under the **Excitations** section of the Project tree.

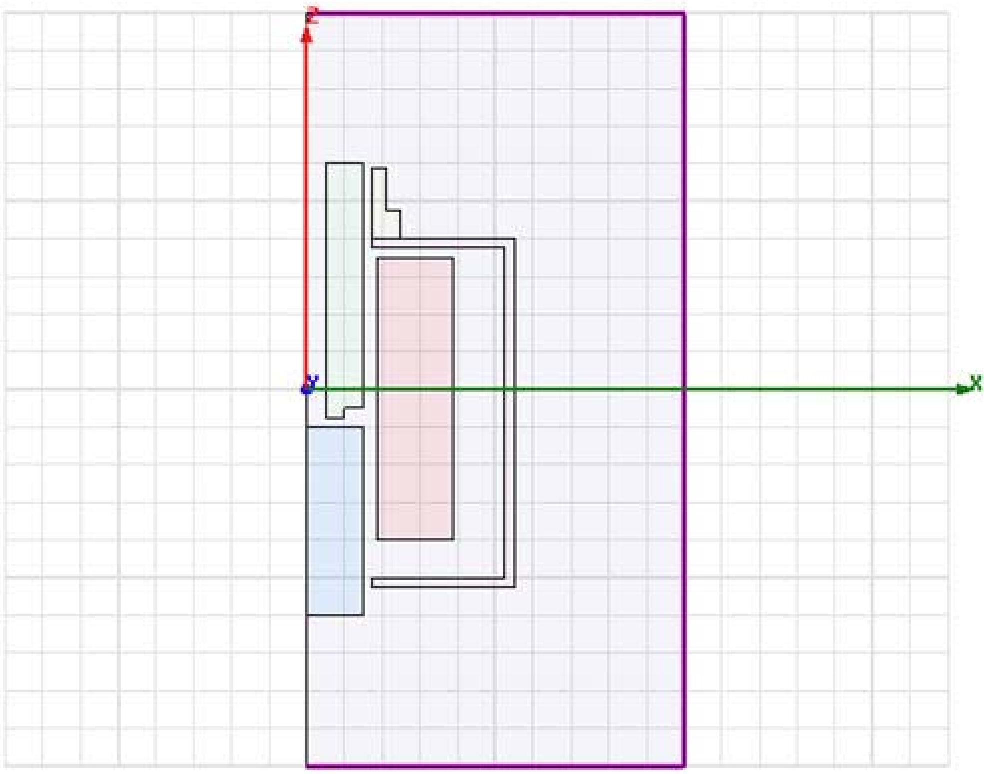
Assign Balloon Boundary to the Background

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 edges of the background to use as a boundary:

1. Click **Edit>Select>Edges**, select the three edges of the background object that correspond to the open region as shown. Click the first edge and the click the remaining



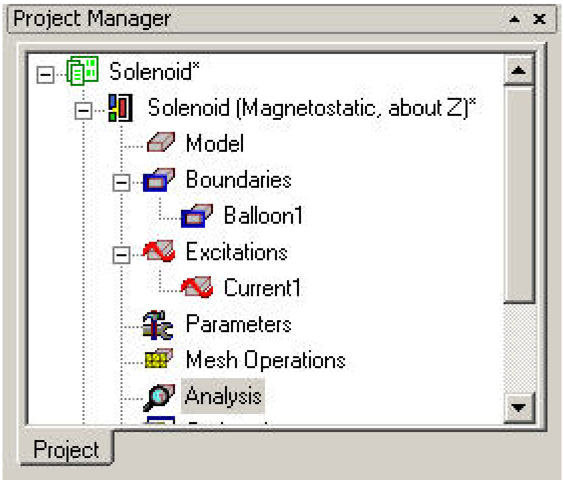
edges while holding down the **Ctrl** key.

2. Click **Maxwell2D>Boundaries>Assign>Balloon**.

3 The Balloon Boundary dialog appears with **balloon1** in the **Name** field. Click **OK** to accept the default name.

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 balloon boundary condition. Because the solenoid model is axisymmetric (representing the cross-section of a device that's revolved 360 degrees around its central axis). Instead, it automatically imposes a different boundary condition to model that edge as an axis of rotational symmetry.

4 **Balloon1** now shows up in the Project tree under the Boundaries section



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 and assign the force parameter. 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 **Edit>Select>Objects** in the menu. Click on the **core** object in the modeler window.

2 Click **Maxwell2D>Parameters>Assign>Force** in the menu. The **Force Setup** dialog appears.

3 Click **OK** to select the default name and assign the force computation to the **core** object.

4 The force computation now appears tin the project tree under Parameters.

You are now ready to enter the Inductance computation.

Set Up Inductance Computation

In addition to the force on the core, the coil inductance is of interest.

To set up the inductance computation:

1 Click **Maxwell2D>Parameters>Assign>Matrix** in the menu. The **Matrix** setup dialog appears.

2 Click the **Include** checkbox next to **Current1** to select the current excitation for use in a matrix calculation.

   Since there is only one excitation defined in this problem, the return current must be set to the default (infinite); however, in an axisymmetric model the current returns in the coil itself since it is rotated 360 degrees about the Z-axis.

3 Click **OK**.

4 Click **File>Save** to save all the changes for Boundary, Excitation, and Parameter setup.

You are now ready to enter the solution criteria.

5 Generating a Solution

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

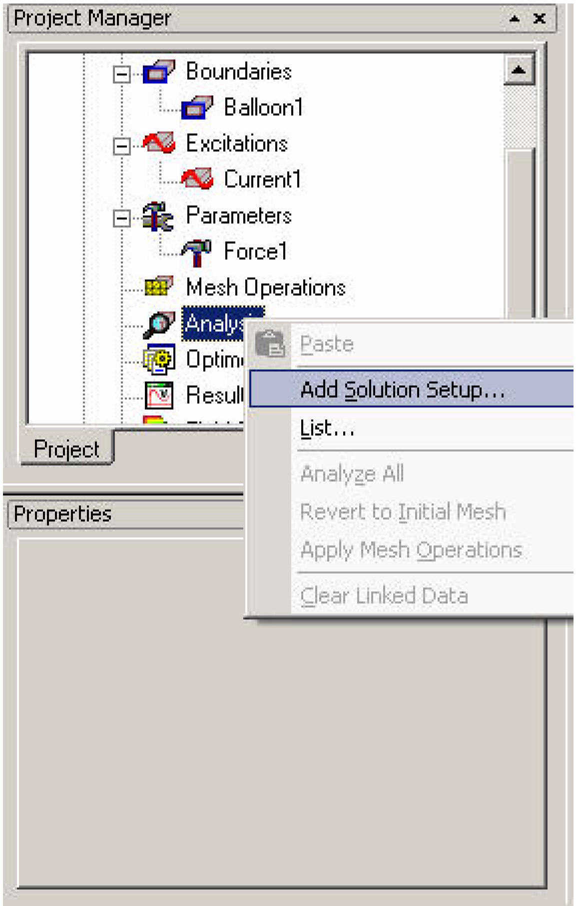
- View the criteria that affect how Maxwell 2D computes the solution.
- Generate the magnetostatic solution. The axisymmetric magnetostatic solver calculates the magnetic vector potential, $\mathbf{A}_{\phi}$, at all points in the problem region. From this, the magnetic field, **H**, and magnetic flux density, **B**, can be determined.
- Compute the force on the core. Since you requested force using the **Parameters** command, this automatically occurs during the general solution process.
- View information about how the solution converged and what computing resources were used.

Add Solution Setup

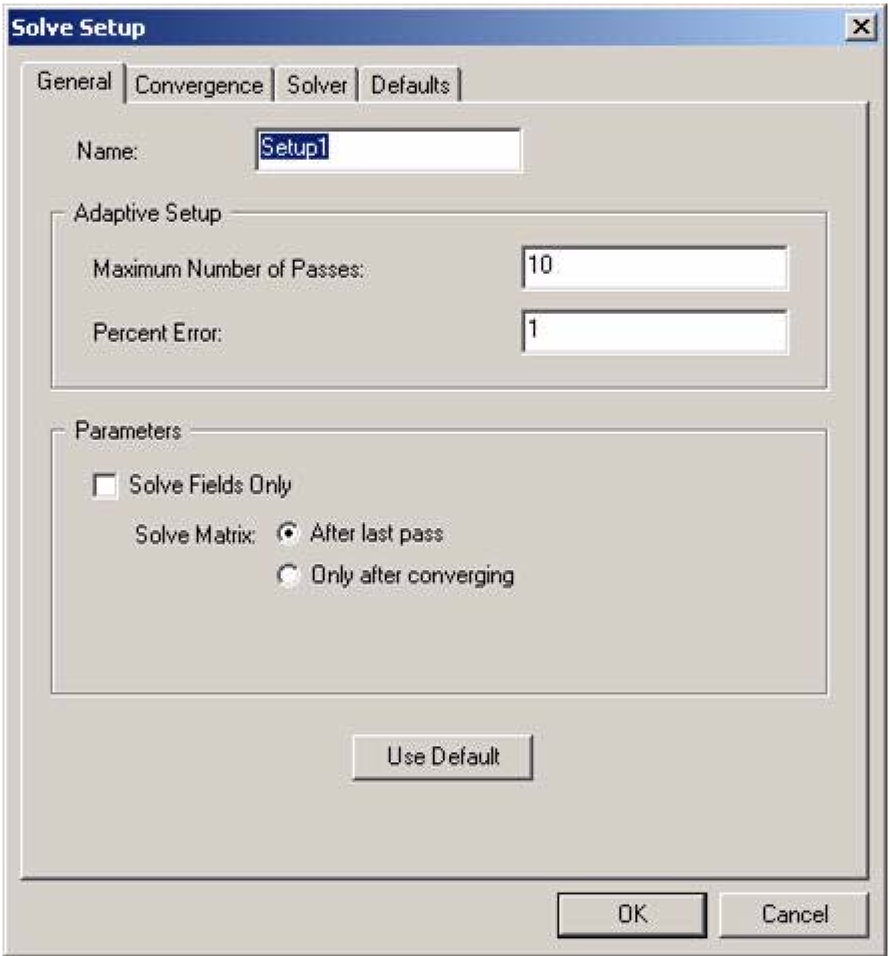
Use the default criteria to generate the solution for the solenoid problem.

1 In the Project tree right-click **Analyze** and select **Add Solution Setup**.

.



2 The **Solve Setup** window appears.



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.

Adaptive Analysis

In the **Adaptive Setup** section of the General Tab, do the following to have the system adaptively refine the mesh and solution:

1 Enter 10 in the **Maximum Number of Passes** field for the

maximum number of adaptively refined solution passes to complete.

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.

Note 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.

2 Leave **Percent refinement per pass** set to its default value of 1%.

The Percent Error field tells the software what target error to achieve within the number of passes allowed. If this percent error is reached the solution process will terminate even though additional adaptive passes may be available. In most cases, the default refinement value is acceptable to provide an accurate solution in reasonable time.

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

Parameters

The **Parameters** section of the General Tab allows the user to specify when requested Parameters should be solved. For this solution, make sure that the **Solve Fields Only** box is not checked, allowing the force solution to be calculated after each adaptive solution.

Mesh Refinement Criteria

In the Convergence Tab, the Standard section refers to the mesh refinement to be used during adaptive analysis.

1. Set the **Refinement Per Pass** to 30% to tells the software to increase the number of mesh triangles by up to 30 percent after each adaptive solution.
2. Set **Minimum Number of Passes** to 2 to force the system to solve at least two passes, regardless of the solution accuracy calculated after the initial solution.
3. Set the **Minimum Converged Passes** to 1. Setting a higher number will force multiple successive solutions to be below the Percent Error criteria before stopping the solution process.

Solver Residual

In the Solver Tab, leave the **Nonlinear Residual** field set to **.0001**. This value specifies how close each solution must come to satisfying the equations that are used to compute the magnetic field. All other fields should remain at their default values.

Select **OK** at the bottom of the dialog to complete the **Solve Setup** process

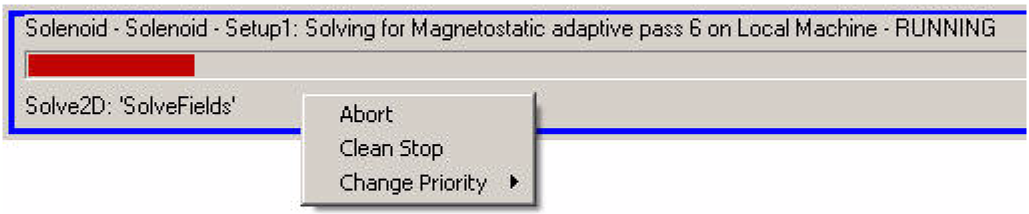
Start the Solution

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

- To start the solution, right-click **Setup1>Analyze** in the Project Manager window.

The system creates the initial finite element mesh for the solenoid structure. A progress bar appears in the **Progress** box at the bottom of the screen. It shows the system's progress as it generates the mesh and computes the adaptive solutions.

The solution may be stopped by right-clicking on the progress window as shown.



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.

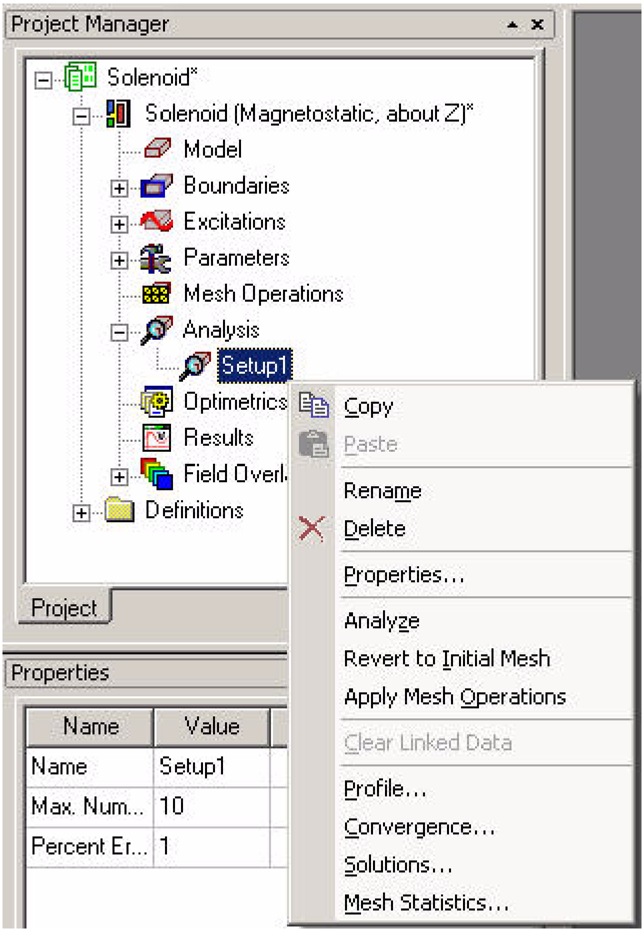
Once the solution process has completed, or in the case of an error, the Message Manager window will display information regarding the reason for the solution process termination. In this case you should see the following message:

Normal completion of simulation on server: Local Machine

Note After a solution is generated, the system will invalidate your solution if you change the geometry, material properties, or boundary conditions of the model. Therefore, you must generate a new solution if you change the model.

Monitoring The Solution

You may monitor the solution progress while the simulation is running by right-clicking on the solution setup entry in the Project Manager. The following information is available while the simulation is running.



- **Profile** displays the Profile Tab of the Solution dialog which lists the computer resource (memory and computation time) usage for each process in the simulation and the running total.
- **Convergence** shows the mesh size, error calculation, and delta energy for each adaptive pass in the solution.

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:

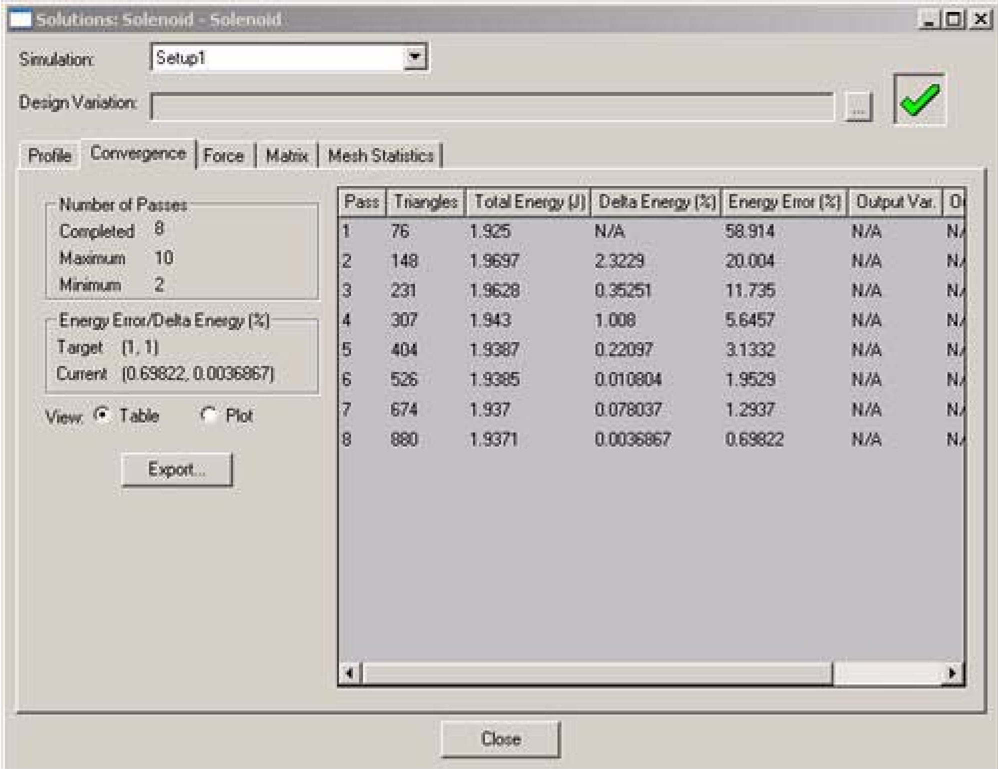
1. Right-Click the **Progress Bar** and select **Abort**.
2. Check the problem definition, and then solve the problem again.

- **Solutions** displays the results of the Parameter calculations, in this case, the force calculated on the **core**.
- The **Mesh Statistics** option displays the Mesh Statistics Tab of the Solution dialog. Here you can view the number of triangles and various triangle properties for each object in the solution.

Viewing Convergence Data

Now that the solution has completed, you can review the solution information to judge the accuracy and suitability of the solution. Right-Click on **Setup1** in the Project Manager and select **Convergence** to monitor how the solution is progressing.

Convergence information appears as shown below. In this example, the system has completed 8 adaptive passes.



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 during Add Solution Setup.
Energy Error	Displays the percent error from the last completed solution — in this case, **0.69%**. Allows you to see at a glance whether the solution is close to the desired error energy. Because this value is less than the **Target Error**, the solution was considered to be converged.
Delta Energy	Displays the change in the percent error between the last two solutions — in this case, **0.003%**.

Completed Solutions

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

Note Your individual solution may differ slightly due to machine differences and meshing differences with each release of the software.

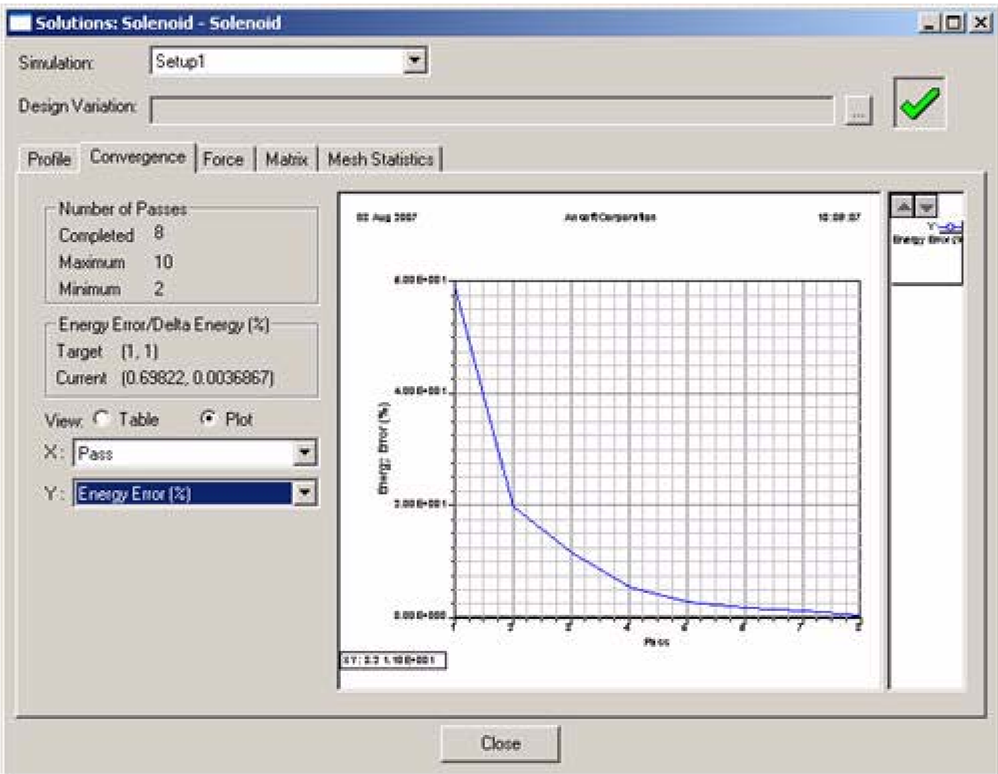
Pass	Displays the number of the completed solutions. In the previous figure, 8 adaptive passes were completed.
Triangles	Displays the number of tetrahedrons in the mesh for a solution. In the previous figure, the mesh used for the eighth solution had 880 triangles.
Total Energy (J)	Displays the total energy of a solution in Joules. In the previous figure, the total energy for the sixth solution was **1.9385** Joules.
Delta Energy (%)	Displays the change in Total Energy between the current and previous expressed as a percentage of the previous pass energy. In the previous figure, the Delta Energy for the sixth pass was **0.010804%**.
Energy Error (%)	Displays the percent energy error of the completed solutions. In the previous figure, the energy error for the eighth solution was **0.698%**.

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 Energy Error computed during each adaptive pass:

- On the **Convergence Tab,** click the **Plot** radio button. Use the drop-down menu to select **Energy Error** for the Y-axis. The following plot appears:

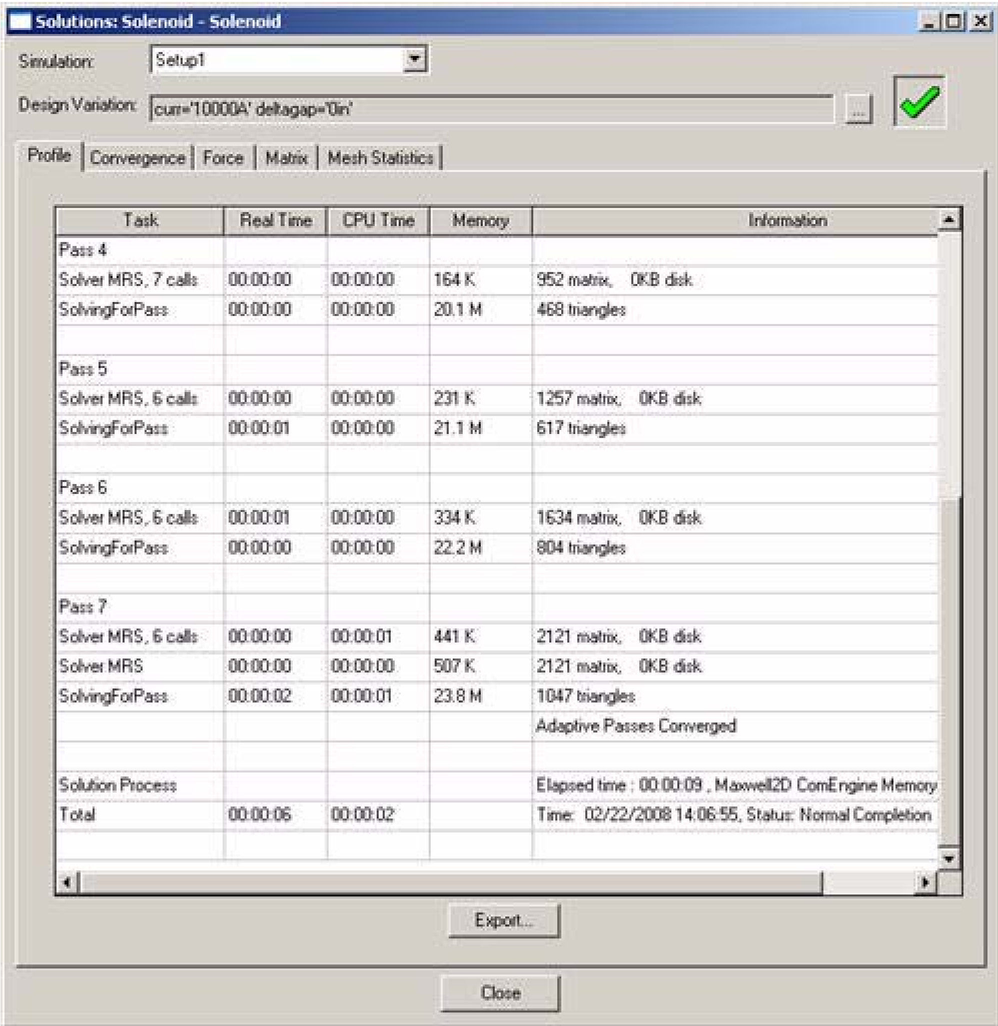


Displaying this data graphically often makes it easier to see how the solution is converging.

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

Viewing Statistics

Click the **Profile** tab to see what computing resources were used during the solution process. The following screen appears.



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:

Task	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.
Memory	Displays the amount of memory used.
Information	Displays the of number of triangles, number of CPUs, and various other information for the process.

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

Click the **Close** button at the bottom of the Solutions dialog. You are now ready to move on to post processing the solution and evaluate the fields and parameters calculated for this model.

6 Analyzing the Solution

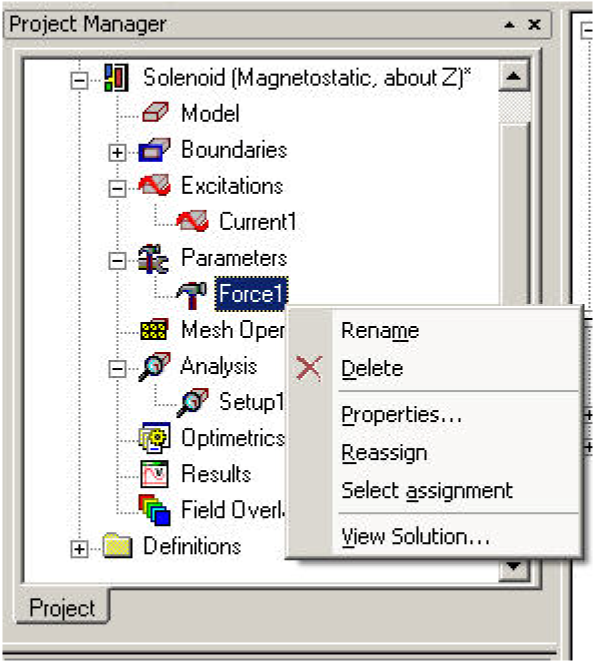
Now that you have generated a magnetostatic solution for the solenoid problem, you can analyze it using Maxwell 2D's post processing features.

- Examine the computed force values.
- Plot the magnetic flux and magnetic fields in and around the solenoid.

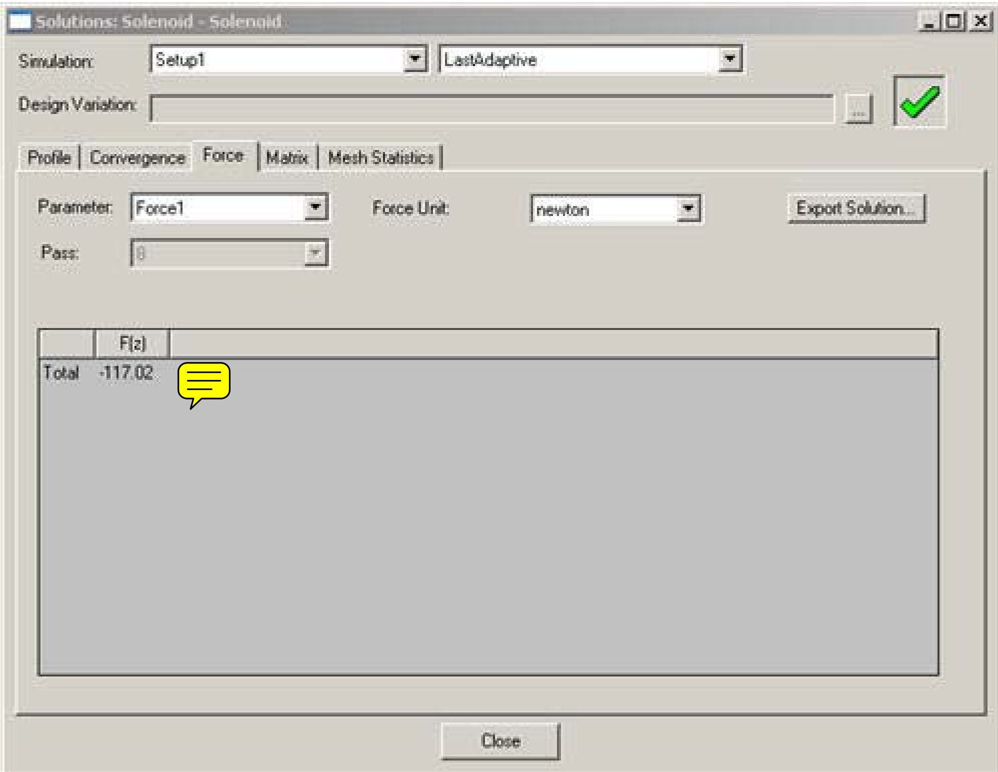
View Force Solution

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

To view the force results, right-click on the **Force1** entry in the Parameter section of the Project Manager as shown and select **View Solution**.



The final force value computed during the adaptive solution appears as shown below. Note that your values may differ slightly from those shown:



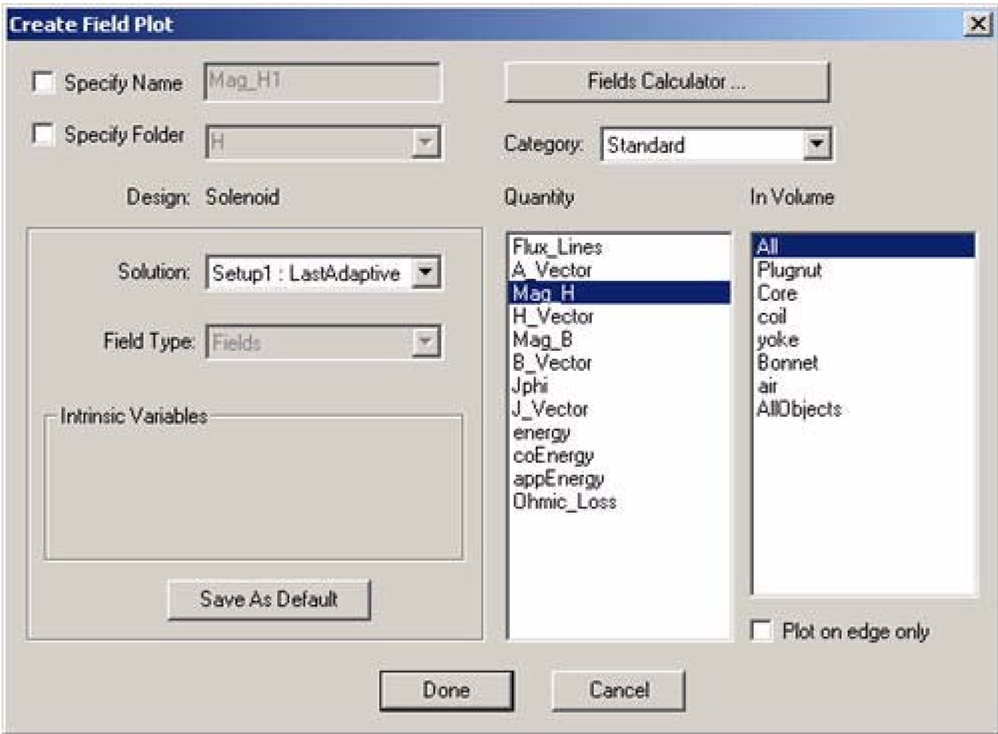
The net force on the core is approximately 117 newtons, acting in the negative Z direction (pulling the core down into the solenoid). There is no component of force in the R direction because of the axial symmetry. Click **Close** to dismiss the window.

Plot the Magnetic Field

You will now create a field plot of the Magnetic Field in the entire problem region.

1 You must first select one or several objects in the problem region on which to create a field plot. Click the mouse anywhere in the Modeler window and press **Ctrl+A**. This will select all objects in the model.

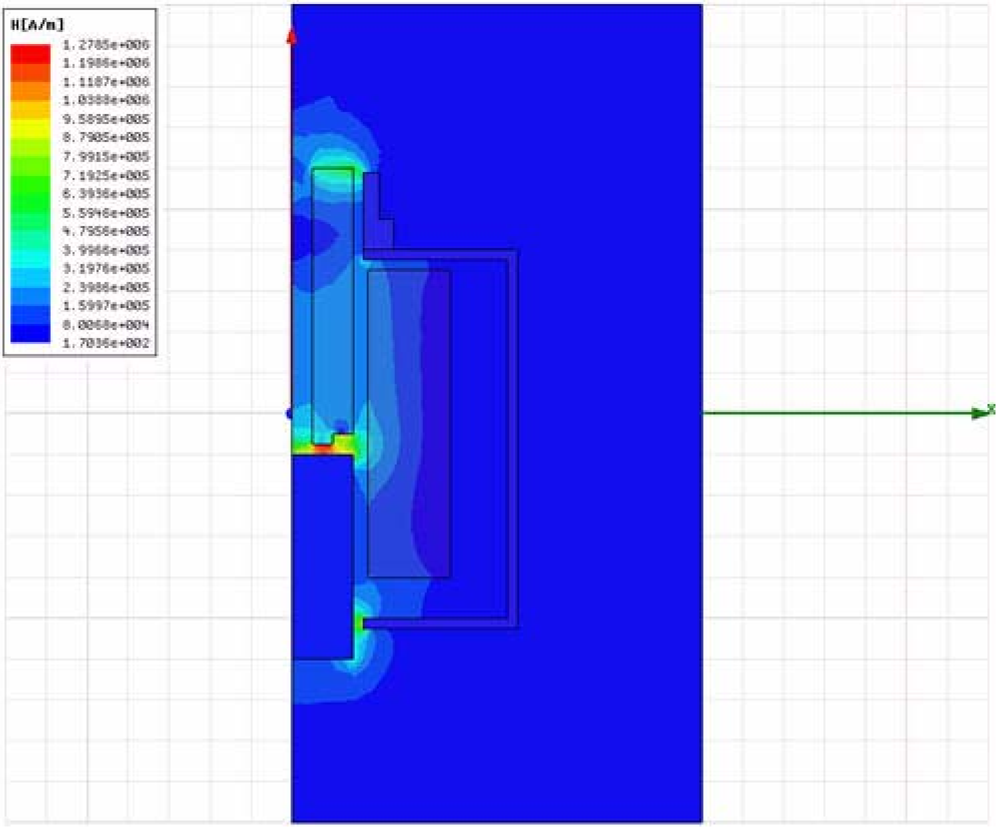
2 From the Field Overlays section of the Project Manager (or the **Maxwell2D>Fields** menu), select **Fields>H>MagH** command to plot the magnetic field magnitude throughout the selected problem region.
The **Create Field Plot** window appears.



3 **Mag H**, and In Volume **All** are selected by default due to the procedure used to launch the dialog. Also note that the dialog defaults to the LastAdaptive pass data to plot.

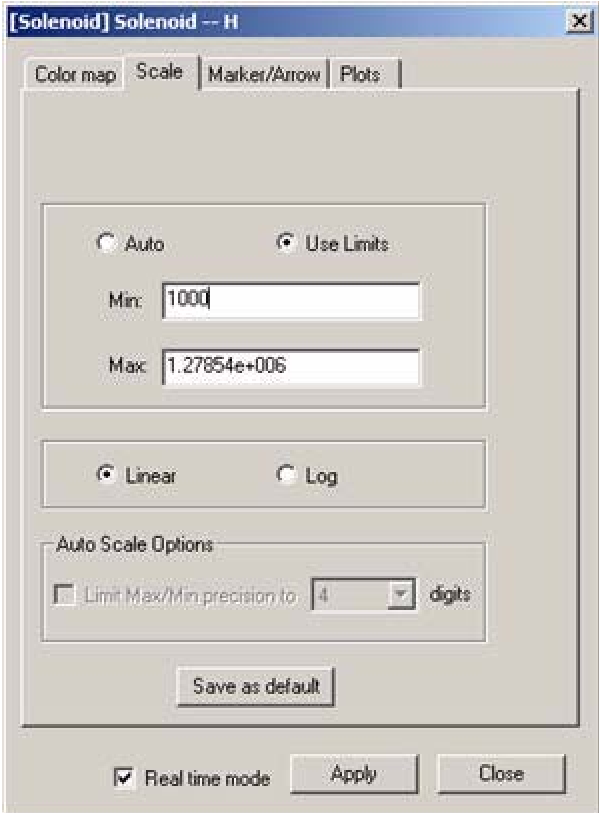
4 *Optionally*, a plot on only the edge of the selected Volume may be obtained by selecting the **Plot on edge only** checkbox.

5 Click **Done**. The **Mag H** plot shown below appears.



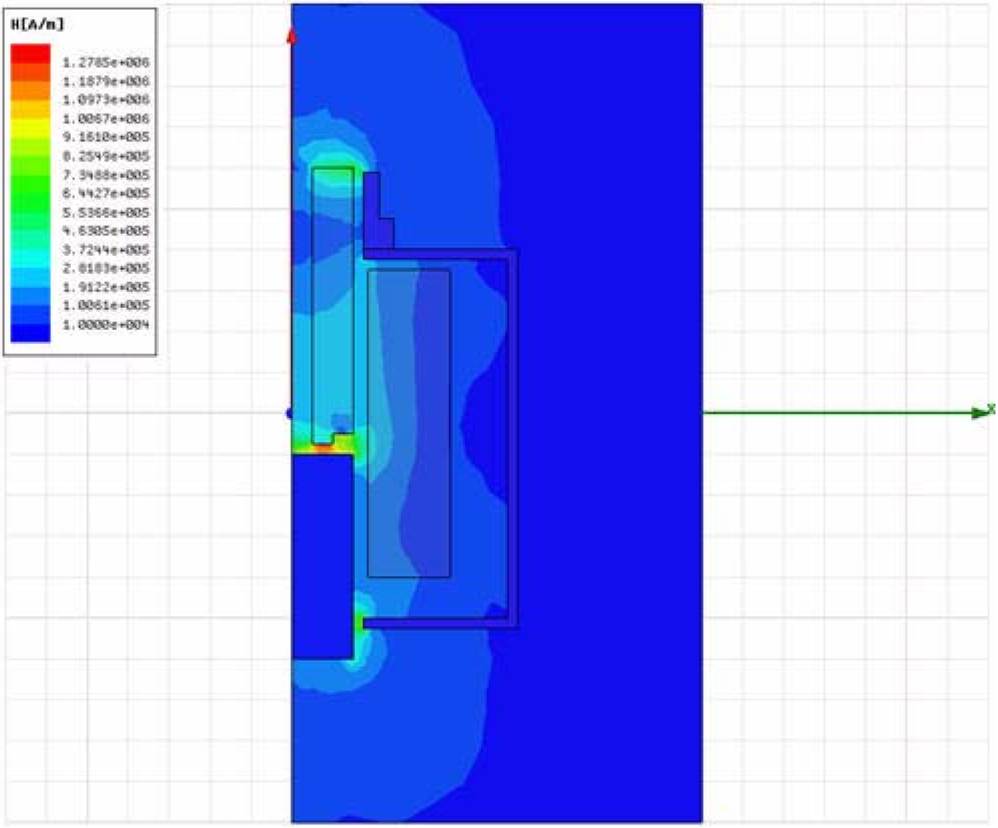
6 Double-click on the **color key** and the Modify Plot dialog is displayed. Click the **Scale** tab.

7 Click the **Use Limits** radio button and enter **1000** in the **Min** field as shown in the figure below.



8 Click **Apply** and the plot will update with a new minimum field display.

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



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

7

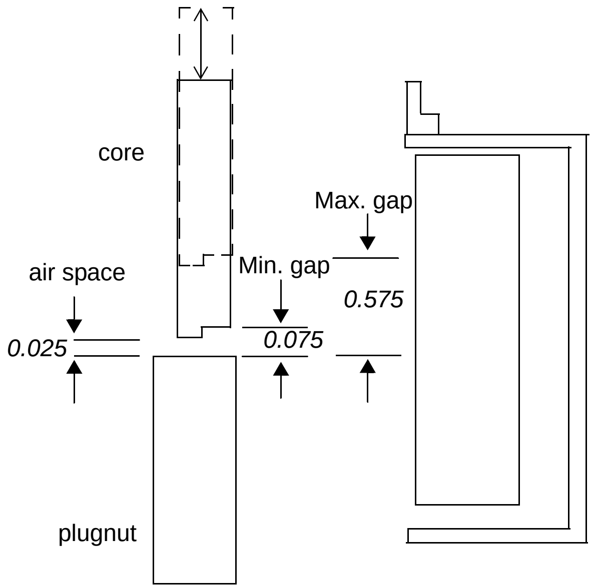
Adding Variables to the Solenoid Model

Now you are ready to use Maxwell to solve the solenoid problem parametrically. This chapter shows you how to add design variations to the solenoid model.

- Add a geometric variation to the solenoid model. This will define the distance between the core and the plugnut as a variable that can be swept during the solution.
- Define a source current function and assign it to the coil
- Request that the force acting on the core and the coil inductance be calculated as a function of core position and coil current during the solution.

The Solenoid Model

The geometric design variable that you are going to define in this chapter represents the gap increase from the nominal design between the solenoid's core and plugnut. By varying the distance between these objects, you can model the solenoid's behavior over a range of core positions. During the solution, **deltagap** will vary from 0.0 to 0.5 inches, representing a spacing between the core and plugnut of 0.075 to 0.575 inches as shown. The core never actually touches the plugnut — there is a minimum air space of 0.025 inches.



11 Choose **File>Save As** from the menu and save a copy of the project to **Actuator_param**. This will become our parameterized project.

Adding Geometric Variables

There are many way to parameterize a geometric model in Maxwell's modeler. For this example, you will use the **Edit>Arrange>Move** command and assign a variable to the move distance. The variable can then be modified by the Parametric Analysis system to move the core location for the sweep analysis.

Other options for varying an object would be to use variables in place of exact coordinates in the rectangle command used to create the core as an example. Then the corners of the rectangle can be varied allowing the core to move and change shape as well. However, all variations of geometric parameterization use the basic procedure you will follow here:

- Assign a variable to one or several points in the creation of a geometric object.
- Provide a default value for the variable.
- Specify the variable is to be used for Optimization, Sensitivity, or Tuning Analysis. Local design variables are automatically available for Parametric Analysis.
- Indicate the range of values the variable may take during the Parametric Analysis.

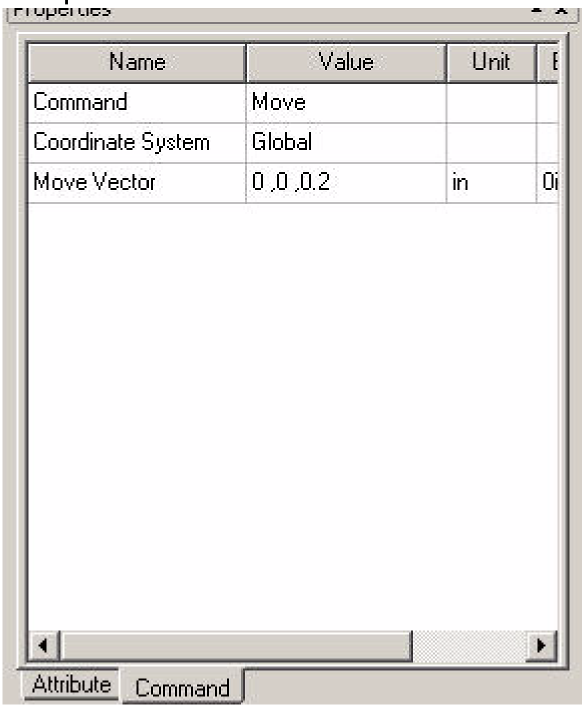
Add a Variable to the Core Object

You will now add a simple linear movement to the **core** object using the move command.

1. Select the **core** object by clicking on it in the Modeler Window or by selecting it in the history tree.
2. Choose **Edit>Arrange>Move** from the menu. The modeler switches to move mode and prompts you for a reference point in the status bar.
3. Enter a reference point by clicking at the origin. Optionally, you may enter **(0, 0, 0)** in the keyboard entry area of the status bar and press the **Enter** key.
4. Enter a target point along the Z-axis by clicking the mouse along the axis. Optionally enter **(0, 0, 0.2)** in the keyboard entry area and press the **Enter** key.

Note It is not important what points are selected for the move command. You are just creating the move command in the model history. The exact movement will be controlled by the variable you set up next.

5 After entering the target point, the properties window will update with the Command tab as shown.

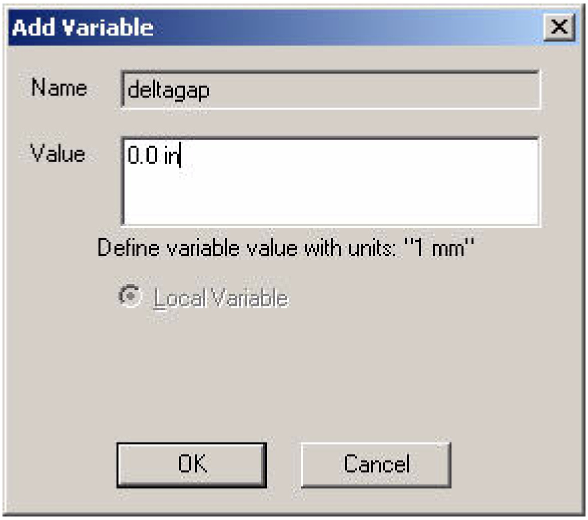


Alternatively, you may have the Properties dialog appear if you have the option **Edit Properties of New Primitives** set under the **Modeler Options** command

Select the Value field of the Move Vector row and enter **deltagap** in the **Z**-axis direction. In addition, make sure that the **X**-axis and **y**-axis movement is zero. Press **Enter**.

6 The Add Variable dialog will be displayed indicating that you have entered a variable in a numeric data field.

Enter **0.0 in** into the value field of the dialog as shown.



7 Click **OK** to complete the geometric variable assignment.

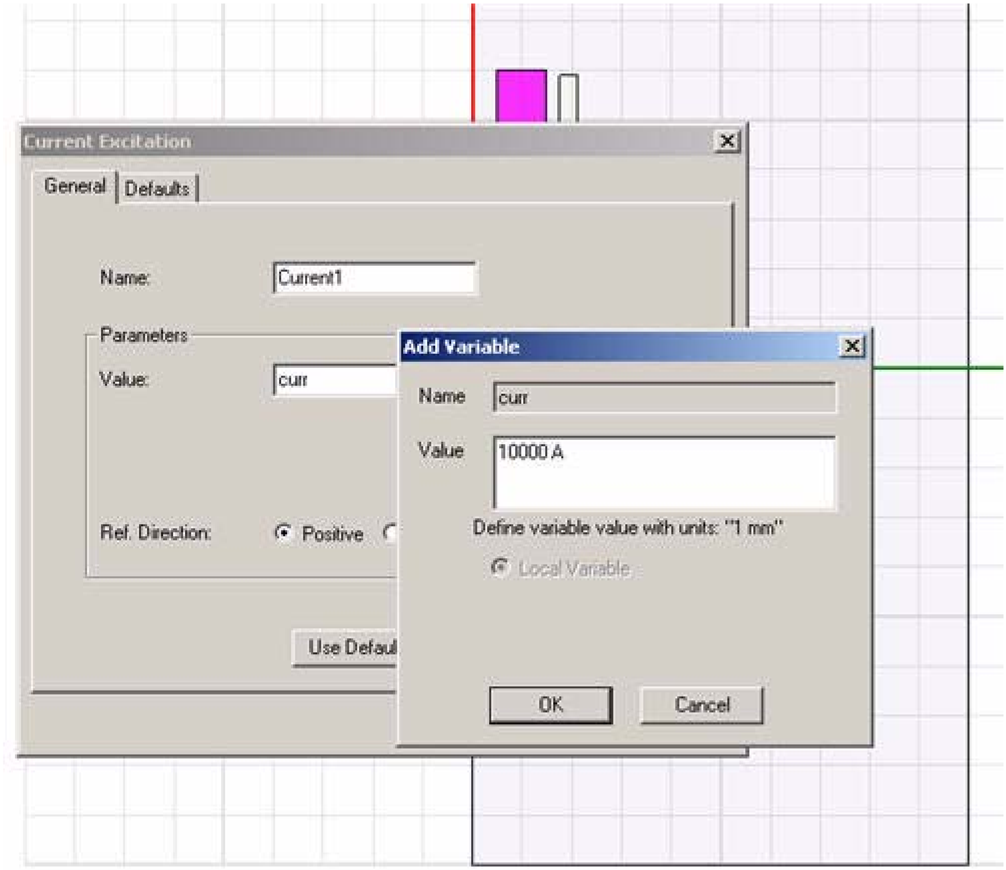
Set the Coil Current to a Variable

Variables may be used in material, boundary and source excitations as well as for geometric movement or shape alteration. In this section we will modify the coil current excitation to make it a variable available for parametric analysis.

1 In the Excitations area of the Project Tree, select the **Current1** excitation by **Double-Clicking**.

2 The Current Excitation dialog will be displayed. Replace **10000** in the value field with the variable **curr** and click **OK.**

3 In the Add Variable dialog, enter **10000 A** into the value field as shown below. Click **OK** to complete the variable assignment.



Set Variable Ranges for Parametric Analysis

Once variables have been added to the project, you must specify the range over which you want the variables to be varied in the analysis.

1 Begin by selecting **Maxwell2D>Optimetrics Analysis>Add Parametric** from the menu. The Setup Sweep Analysis dialog is displayed.

2 In the Sweep Definitions tab click **Add**. The Add/Edit Sweep dialog is displayed with a default variable selected.

3 Verify that the variable is set to **deltagap** and **Linear Step** is selected as the sweep type. Enter the Start, Stop, and

Step size information from the table below.

Value	Data
Start	0.000
Stop	0.500
Step	0.050

4 After entering the data listed, click the **Add** button to transfer the sweep spec to the table on the right.

5 Switch the selected Variable to **curr** and the sweep type to **Linear Count**. Complete the current sweep entry from the following table.

Value	Data
Start	0
Stop	1500
Count	11

6 After entering the data listed, click the **Add** button to transfer the sweep spec to the table on the right.

7 Verify that two sweep entries exist in the table on the right side of the dialog and click **OK**.

8 In the Setup Sweep Analysis dialog, select the **Options Tab** and click the checkbox next to **Copy geometrically equivalent meshes**. This will allow the simulator to reuse meshes for the coil current variations and save meshing time and overall analysis time.

9 Also in the **Options Tab**, make sure the **Save Fields and Mesh** checkbox is selected.

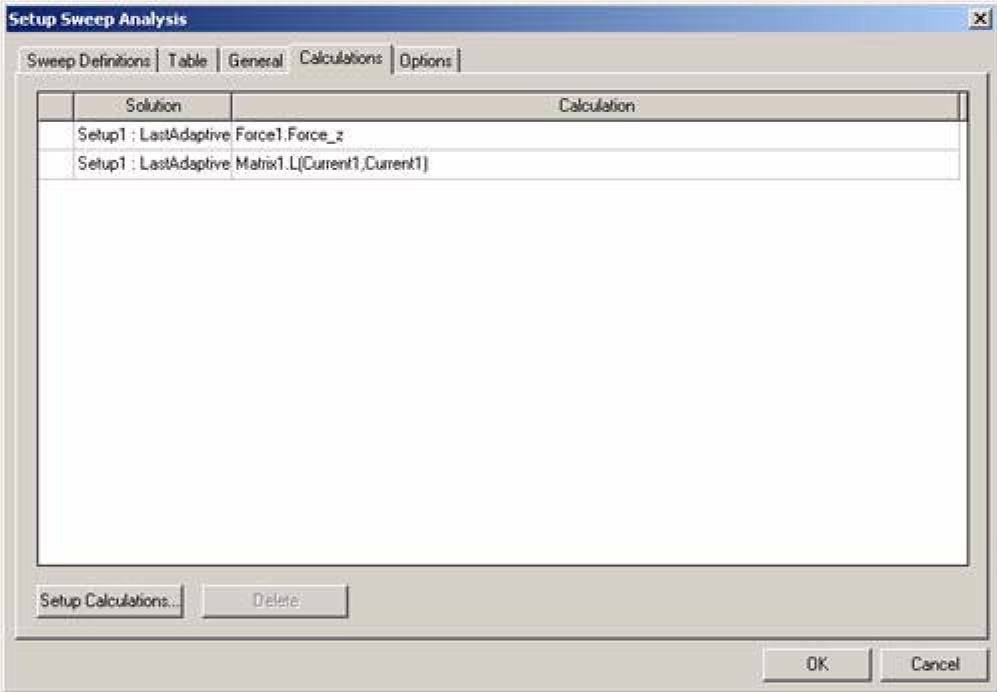
10 Finally, select the **Calculations Tab**. Click on the **Setup Calculations** button to display the Add/Edit Calculation dialog.

11 Select **Force** under Category in the **Trace Tab** and click the **Add Calculation** button.

12 Select **L** under Category in the **Trace Tab** and click the **Add Calculation** button.

13 Select **Done** to return to the Setup Sweep Analysis dia-

log. The dialog should now contain two calculations to be performed during each parametric analysis step as shown below.



The basic Parametric analysis setup is complete; however, it is instructive to note a few additional capabilities of the Setup Sweep Analysis before dismissing the dialog.

- The **Table Tab** can be used to edit individual entries or to add or delete entire rows of the table.
- The **General Tab** allows you to set the values for any variables the design may contain that you have chosen not to include in a sweep. In addition, you can select the solution process parameters by selecting a setup in the Sim Parameters section of the dialog.
- In the **Options Tab** you specify whether you want to save the Fields and Mesh for post processing purposes.

14 Click **OK** to dismiss the dialog.

Redefining Zero Current Sources

The variable spreadsheet is now filled. It has 121 entries (called setups) in it, one for each combination of the values of **deltagap** and **curr**. But you still have a little more work to do. You swept the **curr** variable starting at 0 amperes; however, it would be a waste of time to solve for a zero solution. In addition, since we want to re-use the mesh, we need to make sure that we start off with a solution that will provide good meshing overall. Therefore, you need to edit the spreadsheet and replace all the zeroes in the **curr** column with -1 ampere.

To accomplish this, do the following:

1. In the Project Tree, double-click on **ParametricSetup1** in the **Optimetrics** folder.

   The **Setup Sweep Analysis** dialog appears.
2. Select the **Table** tab.
3. Scroll through the table and change each row containing **0A** for the variable **curr** to **-1A**.
4. Click **OK** to complete the changes.

Save Variables and Parameter Setup

Having added variables for both the geometric variation and the coil current; as well as, defining the sweep ranges for each, it is a good time to save the setup.

- Choose **File>Save**.

The **Solenoid_param** geometry is saved and you are now ready to move on to the solution of the parametric model.

8 Generating a Parametric Solution

Now that you have added physical constraints to the geometry and set up the parametric variable table, you are ready to generate a parametric solution for the solenoid model. You will do the following:

- Generate the magnetostatic solution for each variant on the original model.
- Compute the force on the core as a function of the core position and the current in the coil. (Since you requested force using the **Parameters** command, this automatically occurs during the parametric solution process.).
- Compute the inductance in the coil as a function of core position and the current in the coil. (Since you requested coil inductance using the **Parameters** command, this automatically occurs during the parametric solution process.).
- Review the **Convergence**, **Profile**, and **Force** results of the Parametric analysis.

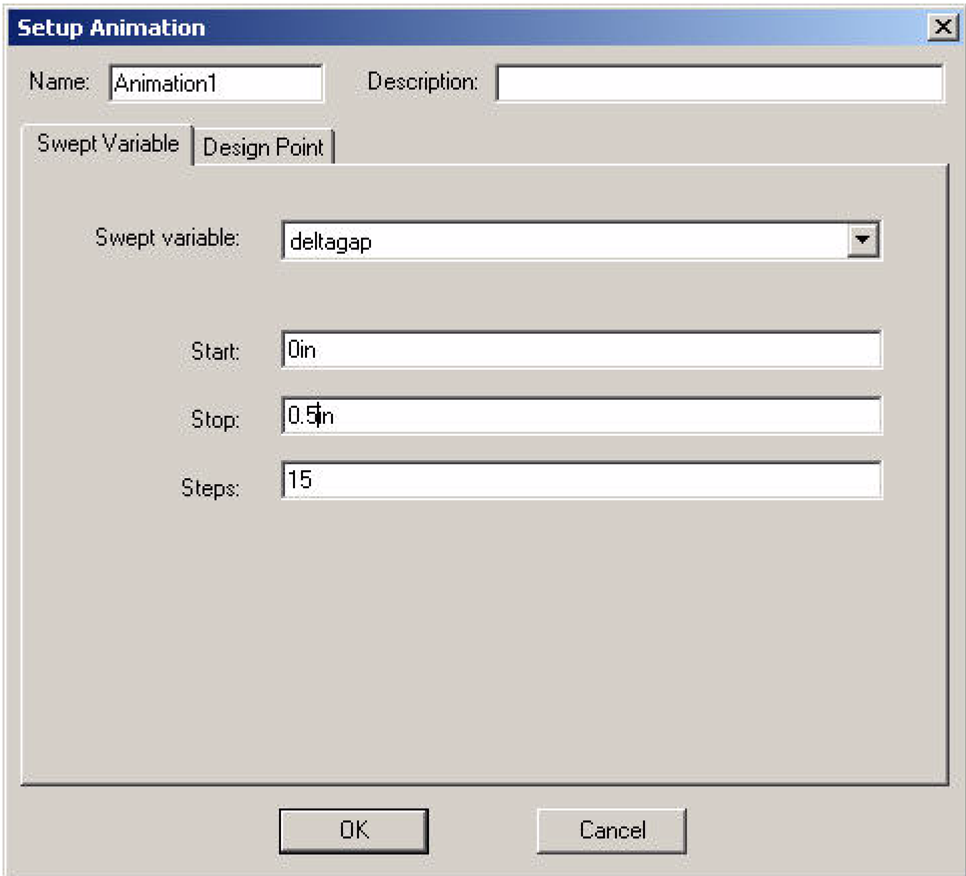
Model Verification

You can quickly verify that the spreadsheet contains only valid geometric sweep parameters for the geometry.

To verify the model:

1 Choose **View/Animate**.

The setup animation window appears.



2 Enter **0in** for **Start**, and **0.5in** for **Stop**. These values are consistent with the values used in the sweep setup.

3 Click **OK**.

4 The **Animation** dialog is displayed and the modeler win-dowshows the geometry animated with the motions of the **core** object over the range of values for **deltagap**.

Note Note that over the range of values used for deltagap, there is no geometry overlap.

5 Choose **Close** in the Animation dialog to end the model animation.

Start the Parametric Solution

Now that you have examined the geometric parameters, the problem is ready to be solved. As a general practice, you should first solve the nominal problem to make sure that the problem is set up correctly. If the nominal problem solves properly, then the parametric solution should be sufficient.

Solving the Nominal Problem

To solve the nominal problem:

1. In the **Project Manager** window, right-click **Setup1** under **Analysis** and select **Analyze** on the shortcut menu.

   A solution is generated for the nominal values of the solenoid parameters.
2. Once the solution has completed, right-click **Setup1** under **Analysis** and select **Convergence** to view the results. If the solution progressed normally, you will see the number of **Triangles** increasing with each pass, and the **Energy Error** decreasing to less than 1%.

Solving the Parametric Problem

You have set up your parametric table with each row to be solved. Depending on your computer, each row will require a few minutes to solve. During the setup you selected **Copy Geometrically Equivalent Meshes** which will improve the solution time; however, you may want to start the solution when there is at least 10 to 15 minutes available for processing.

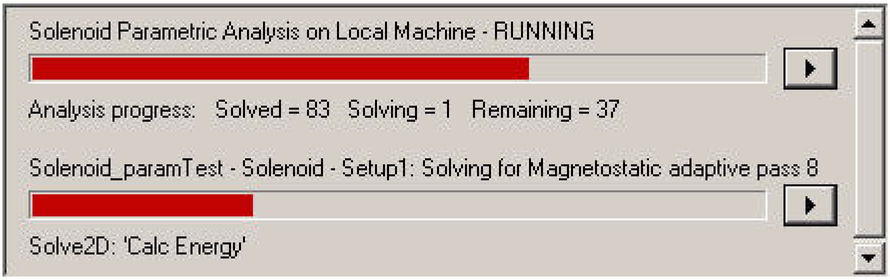
To start the parametric solution:

- Right-click **ParametricSetup1** under **Optimetrics** in the Project Manager window and select **Analyze** on the shortcut menu. The solution process begins.

Note Do not be alarmed if the values you obtain for percent energy error, total energy, inductance, or force differ slightly from the ones given in this guide. The results that you obtain should be approximately the same as the ones given here.

Monitoring the Solution

When performing Parametric Analysis the dual monitoring bar shown below is displayed in the **Progress** window. The top bar displays the progress regarding the solutions in the analysis table. In this case, 120 total analyses are to be performed as a result of the variation of the **deltagap** and **curr** variables..



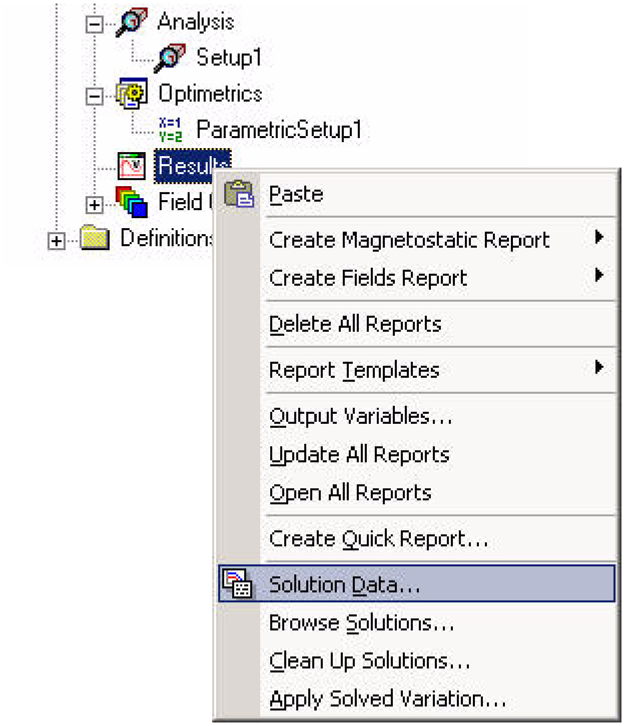
The bottom progress bar shows the standard progress in the solution of each individual row of the analysis table.

Viewing Parametric Solution Data

During the solution process, you may view solutions, the solution convergence, and the solution status or profile of any row in the table.

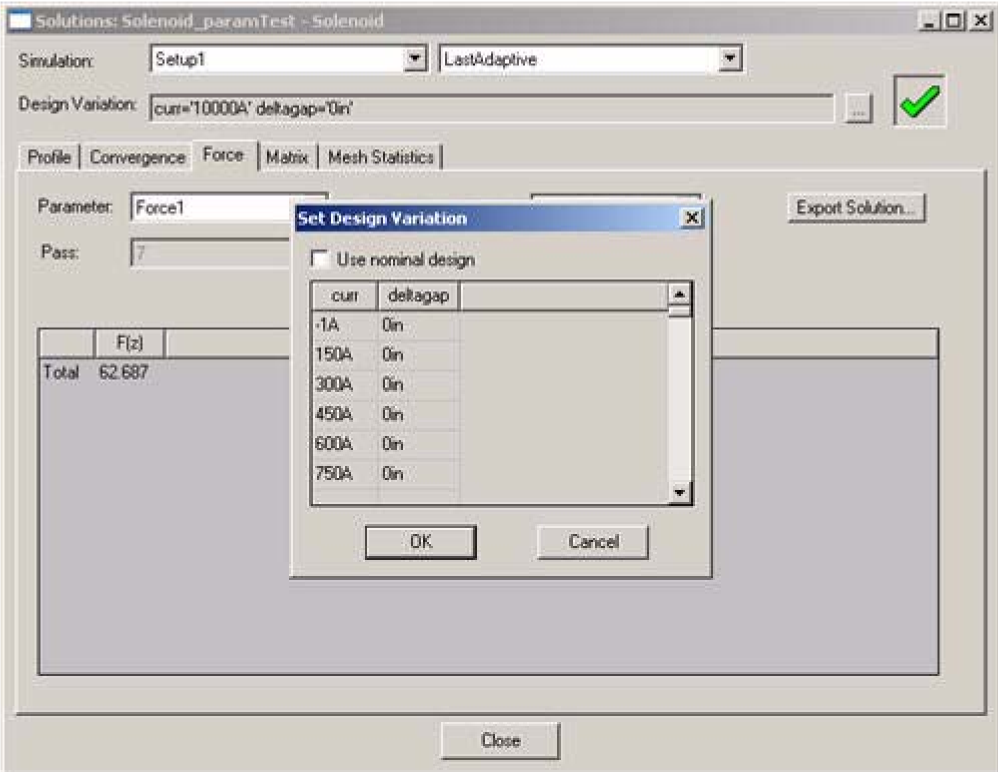
To view the parametric solution data during the solution process:

1. In the Project Manager window, right-click on **Results** and select **Solution Data** from the shortcut menu as shown.



The **Solutions** dialog is displayed. By default, the design variation is set to the **Nominal** problem.

2. Click on the ellipsis button next to the design variation to display the **Set Design Variation** dialog as shown.



3 Uncheck the **Use nominal design** checkbox.

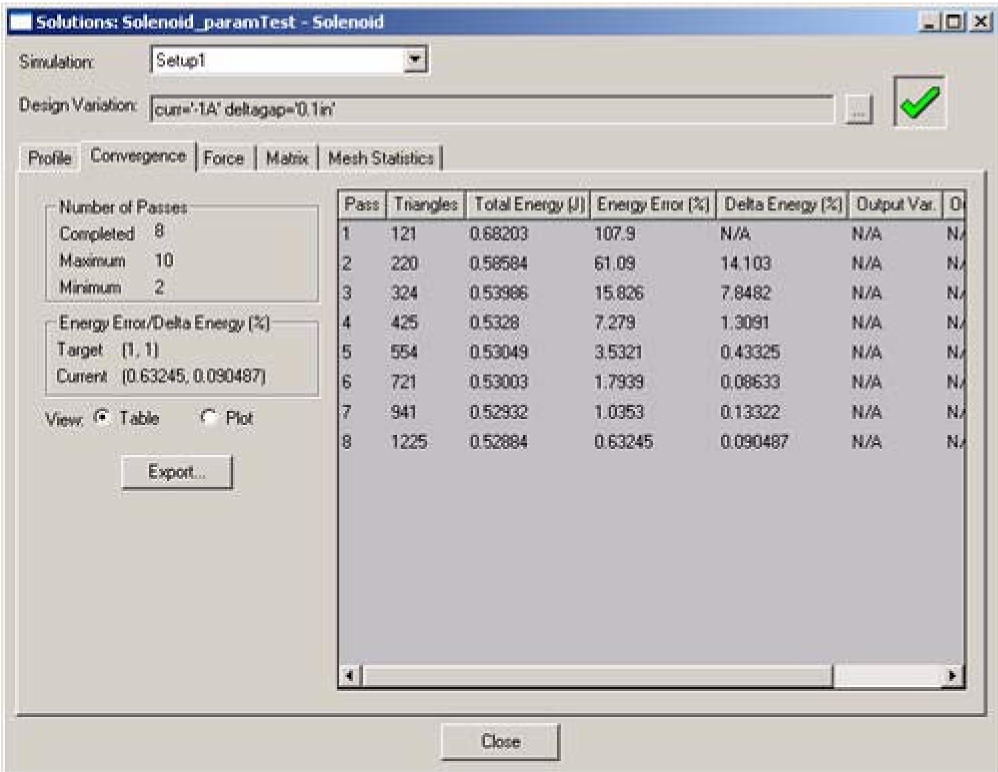
4 Select any row in the table by clicking to highlight it and click **OK**

5 The design variation in the Solutions dialog now shows the profile, convergence information, force, etc. associated with the selected variable values.

Viewing Parametric Convergence Data

After selecting a design variation, choose **Convergence** to monitor how the solution is progressing.

- For example, if you choose the variation corresponding to **Curr=-1A** and **deltagap=0.1in**, then choose **Convergence**, you will see something like what is shown below:



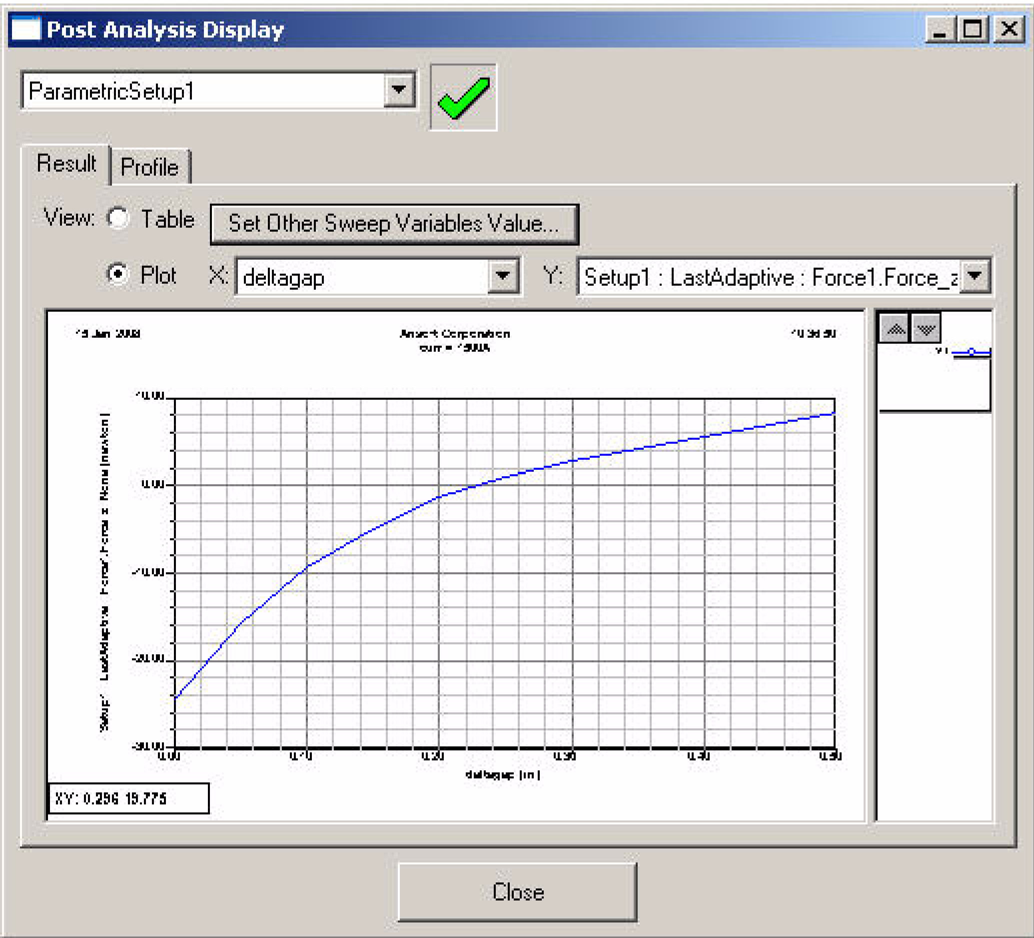
- The mesh has been iteratively increased as shown in the **Triangles** column and the corresponding **Energy Error** has been decreased to less than 1%.
- You may change the View from **Table** to **Plot** by clicking the **Plot** radio button and selecting which quantities should be plotted on the **X** and **Y** axes.

Plotting Parametric Convergence Data

In addition to viewing the results for each design variation, you may view the results as a function of the design variable values.

1. In the Project Manager window, right-click **ParametricSetup1** under the **Optimetrics** folder and select **View Analysis Results**.

   The **Post Analysis Display** dialog appears as shown below:



2. Using the pull down menu, select **deltagap** as the variable to plot on the X axis.

For parametric analyses involving multiple variables, the **Setup Plot** dialog appears.

3 Select **1050A** for the variable **curr** and click **OK.**

4 In order to review the Inductance of the coil, use the **Y axis** pull down menu and select **L(Current1, Current1)**.

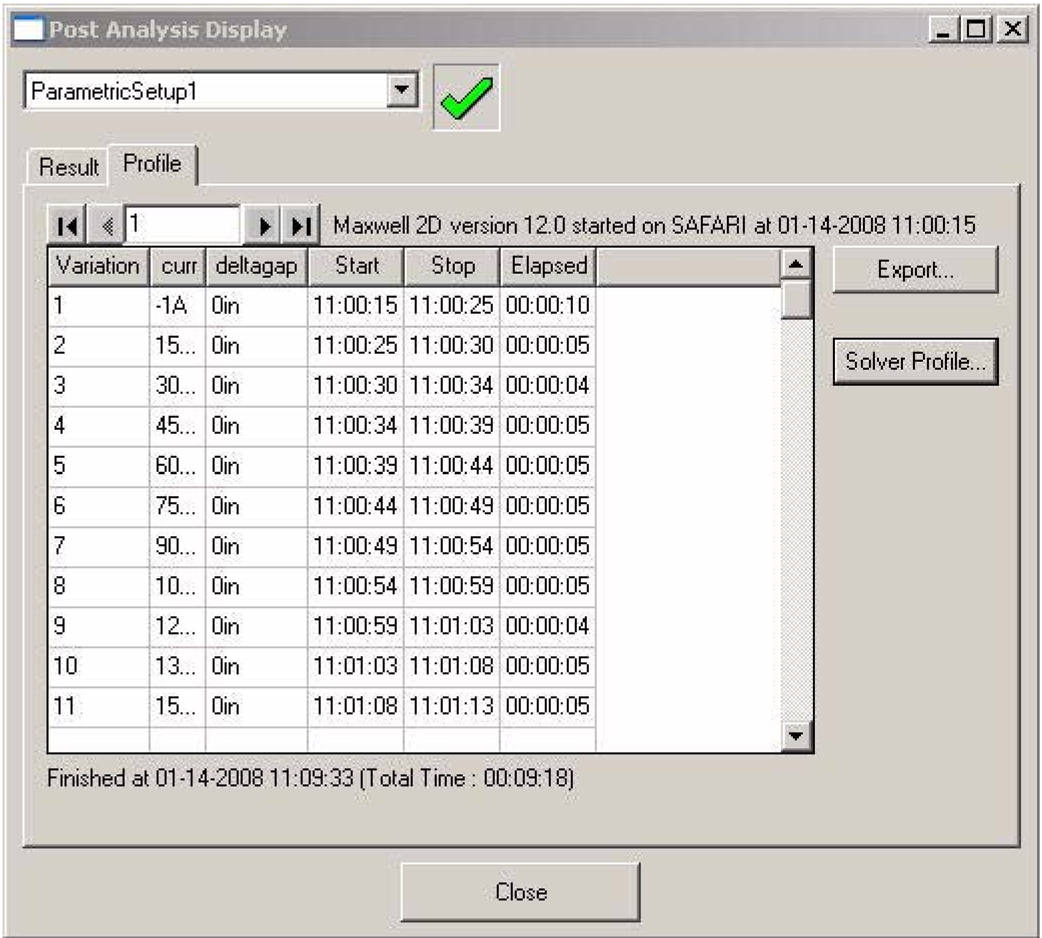
5 You may inspect the performance at other values of **curr** by clicking the **Set Other Sweep Variables Value** button and selecting the value of interest.

Viewing Parametric Solver Profile

Individual design variations may be inspected to see the solver profile as well.

1 In the **Post Analysis Display** dialog, select the **Profile** tab.

The following screen appears:



2 The **Start**, **Stop** and **Elapsed** time for each variation is shown in the table. Select **Variation 8** by clicking on it in the table.

3 Click the **Solver Profile** button.

4 The **Solutions** dialog is displayed with the selected design variation loaded and the profile tab selected.

The following information is displayed for each completed adaptive field solution and mesh refinement step. If more data is available than can fit on a single screen, scroll bars appear.

Task	Displays the system command that was used.
Real time	Displays the time taken to complete the step.
CPU time	Displays the time taken by the CPU to complete the step.
Memory	Displays the amount of memory used.
Information	Displays the number of triangles in the finite element mesh, size of the matrix, disk space used and other information relevant to the solution process.

5 Click **Close** to dismiss the **Solutions** dialog and return to the **Post Analysis Display** dialog.

6 Click **Close** to dismiss the **Post Analysis Display** dialog.

9

Plotting Results from a Design Variation

With the result of design variations available, you can use the post processor to create reports and plot fields with multiple variations. You will do the following:

- Create a report of force vs. gap with multiple traces for each current level.
- Set the Design Variation for plotting fields from a design variation.
- Animate a field plot using the Design Variation values for the gap.

Access Parametric Post Processor

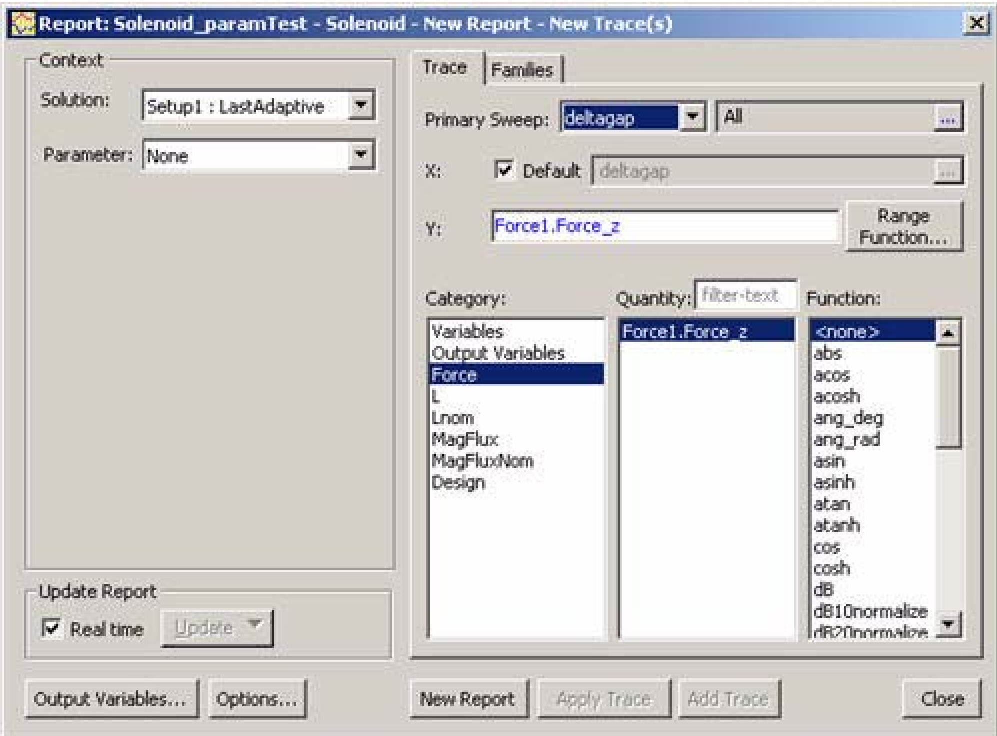
You will use the Post Processor to plot the force on the core as a function of position for different values of coil current.

To access the Post Processor:

1 Select **Maxwell2D>Results>Create Magnetostatic Report>Rectangular Plot.**

Alternatively, right-click on **Results** in the Project Manager window and select Create Magnetostatic Report>Rectangular Plot from the shortcut menu.

The **Report** dialog appears.

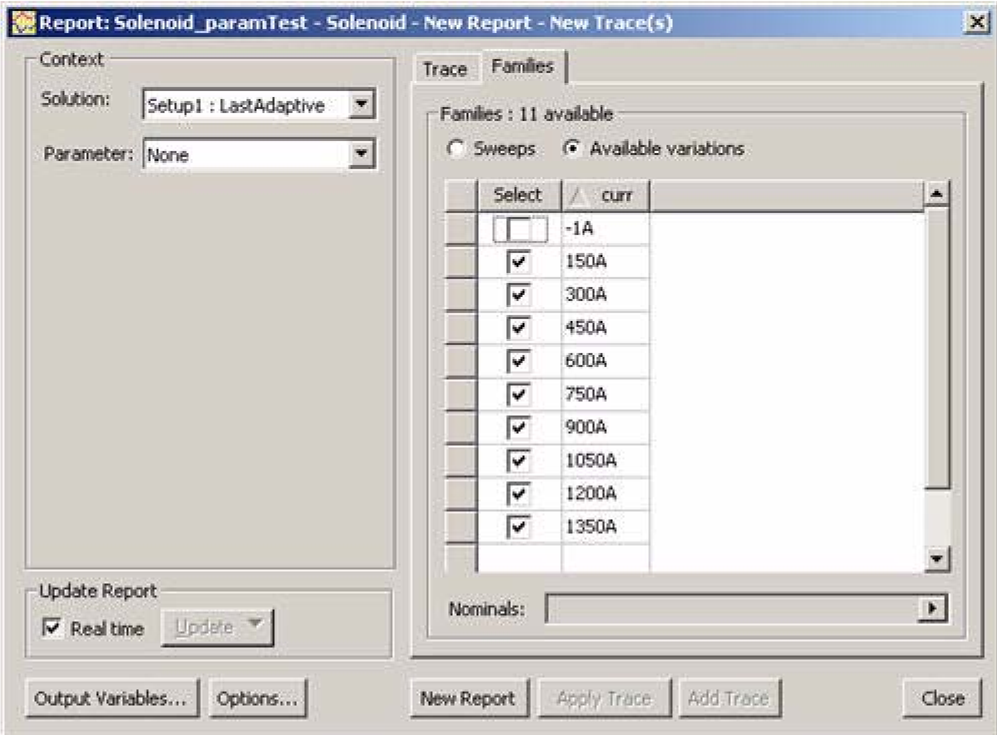


2 Since we want to plot Force as a function of position, make sure that the variable **deltagap** is selected as the **Primary Sweep** variable.

3 Also, make sure that **Force** is selected under the Category list.

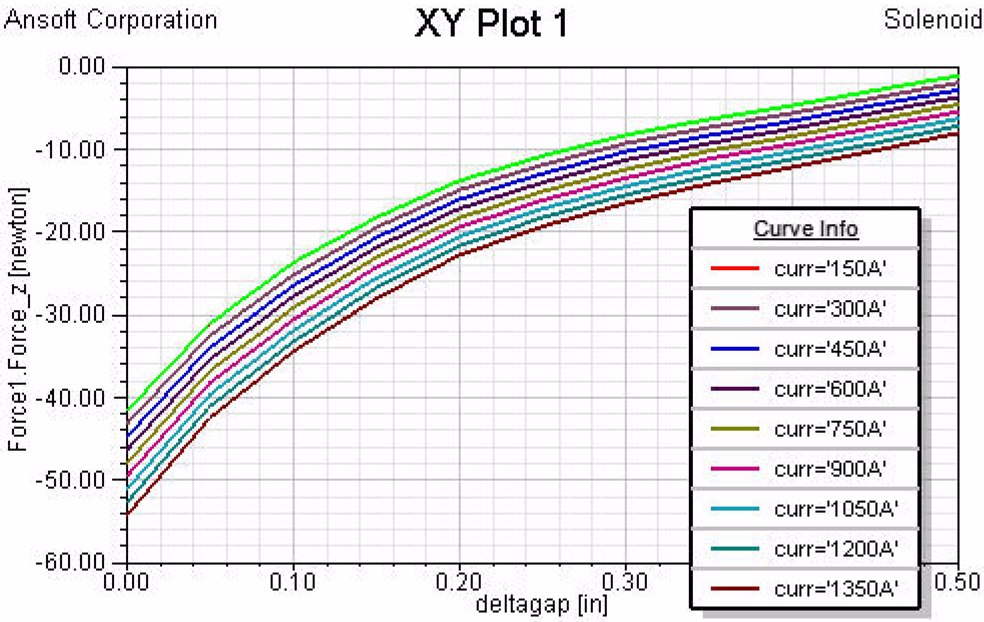
4 Click on the **Families** tab to set the values of **curr** to plot.

5 In the **Families** tab, you may select **Sweeps** or choose individual variations for the variable **curr** as shown.



6 Click **New Report** to plot the force versus gap spacing for various current values.

7 Click **Close** to dismiss the **Report** dialog and view the family of curves as shown below.



Plotting Fields of a Design Variation

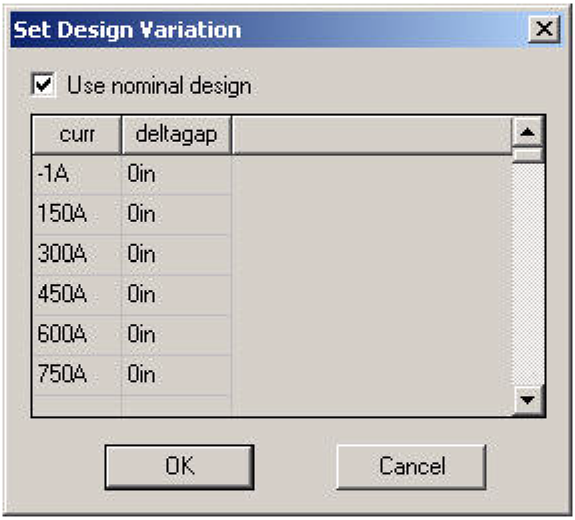
In order to plot the fields from a particular design variation, you must first have told the software to save the fields from the parametric analysis during the setup process. You did this in the **Set Variable Ranges for Parametric Analysis** section of this guide.

To plot the fields from a particular design variation:

Apply Solved Variation

1. Tell the software which variation to use by clicking **Results>Apply Solved Variation** from the Maxwell2D menu or the shortcut menu.

The **Set Design Variation** dialog is displayed as shown.



2 Uncheck the **Use nominal design** checkbox.

3 Scroll to the **curr=1500A, deltagap=0.5in** line and highlight it by clicking on it.

4 Click **OK** to make it the new "Nominal Design" and be able to plot fields.

Note Note the change in the model view as the design variation is applied.

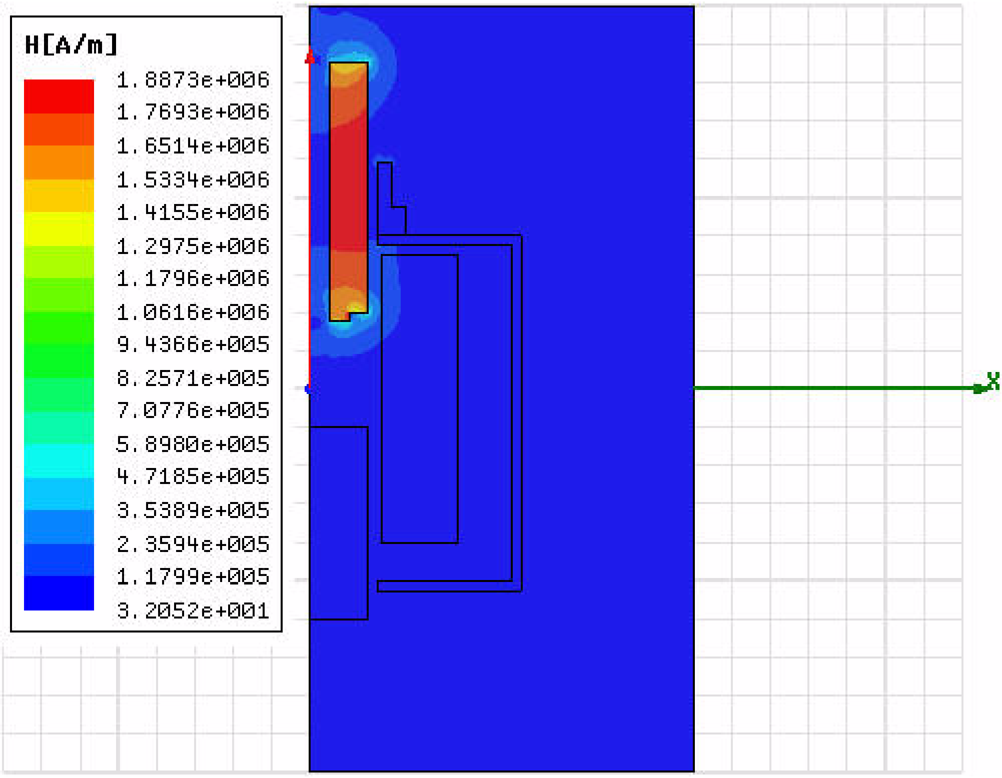
Plot Fields for the Variation

To plot the magnetic field for the variation:

1 Click **Edit>Select All** in the menu, or click in the modeler window and press **Ctrl-A**. All objects in the model will be highlighted.

2 Click **Maxwell2D>Fields>Fields>H>Mag_H**.
The **Create Field Plot** dialog appears.

3 Click **Done** to accept the plot definition and create the field plot similar to the one shown below.



4 Right-Click on the plot **Key** and select **Modify** from the shortcut menu.

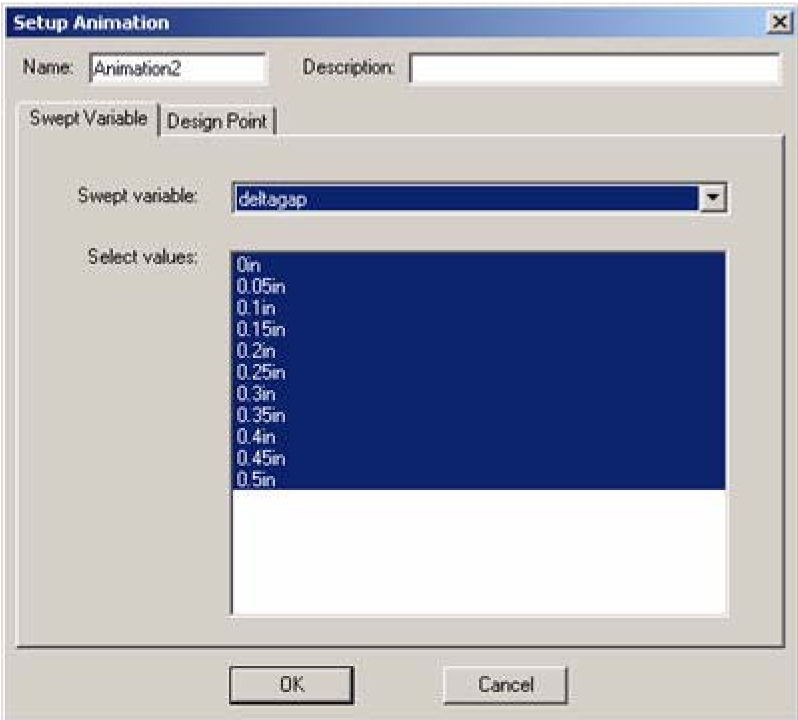
5 Click on the **Scale** tab.

6 Since the field plot is dominated by the field in the core, click on the **Log** radio button to see more field pattern in the low field regions.
7 Click **Close** to dismiss the dialog.

Animate the Field Plot Across Variations

In order to see how the field changes with the size of the gap you can animate the field as follows:

1 In the **Project Manager** window, right-click on **Mag_H** in the **Field Overlays** folder.
2 In the shortcut menu, click on **Animate**.
3 The **Setup Animation** dialog appears.



4 All values for deltagap are selected by default. Click **OK** to accept and animate the field plot.
5 Click **Export** to save the animation as a .gif or .avi file.
6 Click **Close** in the **Animation** dialog.

Exit the Maxwell Software

You have successfully completed the 2D Magnetostatic Solenoid Getting Started Problem.

1. Save the project plots and reports by clicking **File>Save**.

   Maxwell V12 will save all data including the plots you have created for later use.

2. To exit the software, click **File>Exit**.

Index

M

N

O

P

R

S

T

V

Y

Z