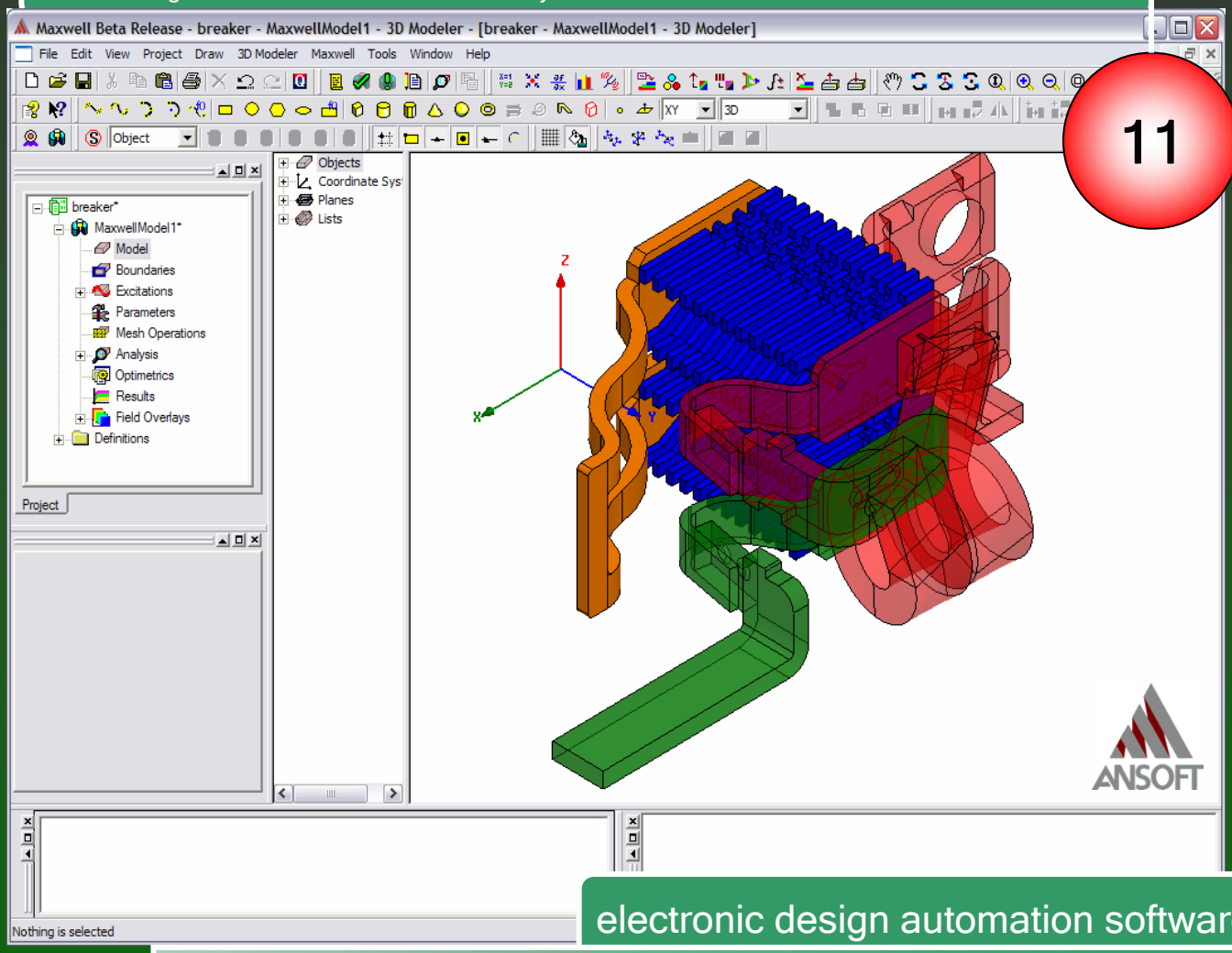
WWW.MOHANDESYAR.COM

Ansoft

Maxwell 3D

Electromagnetic and Electromechanical Analysis



electronic design automation software

user's guide – Maxwell 3D

ANSOFT Korea http://ansoft.co.kr mail to : ehan@ansoft.co.kr

WWW.MOHANDESYAR.COM

Contents

This document discusses some basic concepts and terminology used throughout the Ansoft Maxwell application. It provides an overview of the following topics:

0. Fundamentals
   - Ansoft Maxwell Desktop
   - Opening a Design
   - Setting Model Type
1. Parametric Model Creation
   - 1.1 Boundary Conditions
   - 1.2 Excitations
5. Examples

WWW.MOHANDESYAR.COM

What is Maxwell?

- Maxwell is a high-performance interactive software package that uses finite element analysis (FEA) to solve three-dimensional (3D) electric, magnetostatic, eddy current, and transient problems.
- Use it to compute:
  - Static electric fields, forces, torques, and capacitances caused by voltage distributions and charges.
  - Static magnetic fields, forces, torques, and inductances caused by DC currents, static external magnetic fields, and permanent magnets.
  - Time-varying magnetic fields, forces, torques, and impedances caused by AC currents and oscillating external magnetic fields.
  - Transient magnetic fields caused by electrical sources and permanent magnets

WWW.MOHANDESYAR.COM

Installing the Ansoft Maxwell Software

System Requirements

- Microsoft Windows XP(32/64), Windows 2000, or Windows 2003 Server. For up-to-date information, refer to the Maxwell Release Notes.
- Pentium -based computer
- 128MB RAM minimum
- 8MB Video Card minimum
- Mouse or other pointing device
- CD-ROM drive

NOTE: You should make backup copies of all Maxwell projects created with a previous version of the software before opening them in Maxwell v11

Installing the Ansoft Maxwell Software

- For up-to-date information, refer to the Maxwell Installation Guide

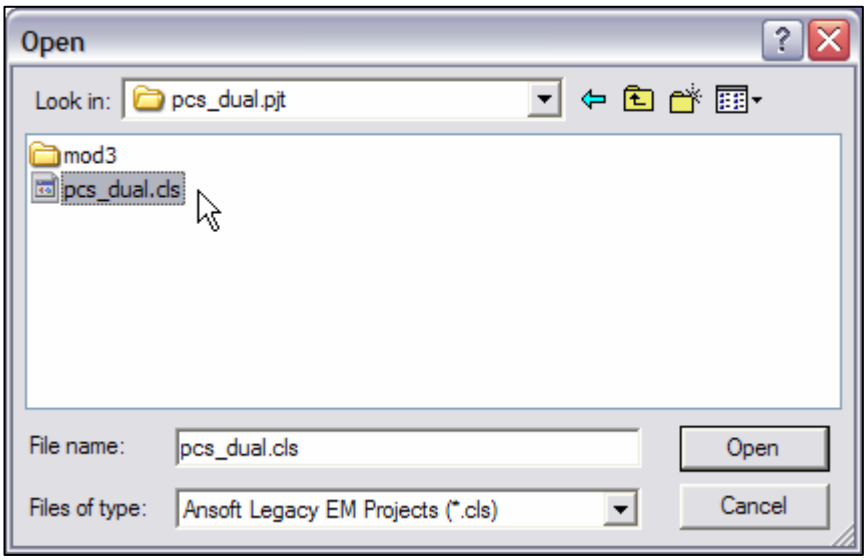
Starting Ansoft Maxwell

1. Click the Microsoft **Start** button, select **Programs**, and select the **Ansoft, Maxwell 11** program group. Click **Maxwell 11**.
2. **Or** Double click on the Maxwell 11 icon on the Windows Desktop

WWW.MOHANDESYAR.COM

Converting Older Maxwell file to Maxwell v11

- Because of changes to the Maxwell files with the development of Maxwell v11, opening a Maxwell document from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time will return to normal
- Ansoft Maxwell v11 provides a way for you to automatically convert your Maxwell projects from an earlier version to the Maxwell v11 format.
- To access Maxwell projects in an earlier version.
  - **From Maxwell v11**,
    1. Select the menu item *File > Open*
    2. Open dialog
       1. Files of Type: **Ansoft Legacy EM Projects (.cls)**
       2. Browse to the existing project and select the .cls file
       3. Click the **Open** button



WWW.MOHANDESYAR.COM

Getting Help

- If you have any questions while you are using Ansoft Maxwell you can find answers in several ways:
  - **Ansoft Maxwell Online Help** provides assistance while you are working.
    - To get help about a specific, active dialog box, click the **Help** button in the dialog box or press the **F1** key.
    - Select the menu item *Help > Contents* to access the online help system.
    - **Tooltip**s are available to provide information about tools on the toolbars or dialog boxes. When you hold the pointer over a tool for a brief time, a tooltip appears to display the name of the tool.
    - As you move the pointer over a tool or click a menu item, the **Status Bar** at the bottom of the Ansoft Maxwell window provides a brief description of the function of the tool or menu item.
    - The Ansoft Maxwell Getting Started guide provides detailed information about using Maxwell to create and solve 3D EM projects.
  - **Ansoft Technical Support**
    - To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, www.ansoft.com and select **Contact**.
      - Your Ansoft sales engineer may also be contacted in order to obtain this information.

Visiting the Ansoft Web Site

- If your computer is connected to the Internet, you can visit the Ansoft Web site to learn more about the Ansoft company and products.
  - From the Ansoft Desktop
    - Select the menu item *Help > Ansoft Corporate Website* to access the Online Technical Support (OTS) system.
  - From your Internet browser
    - Visit www.ansoft.com

WWW.MOHANDESYAR.COM

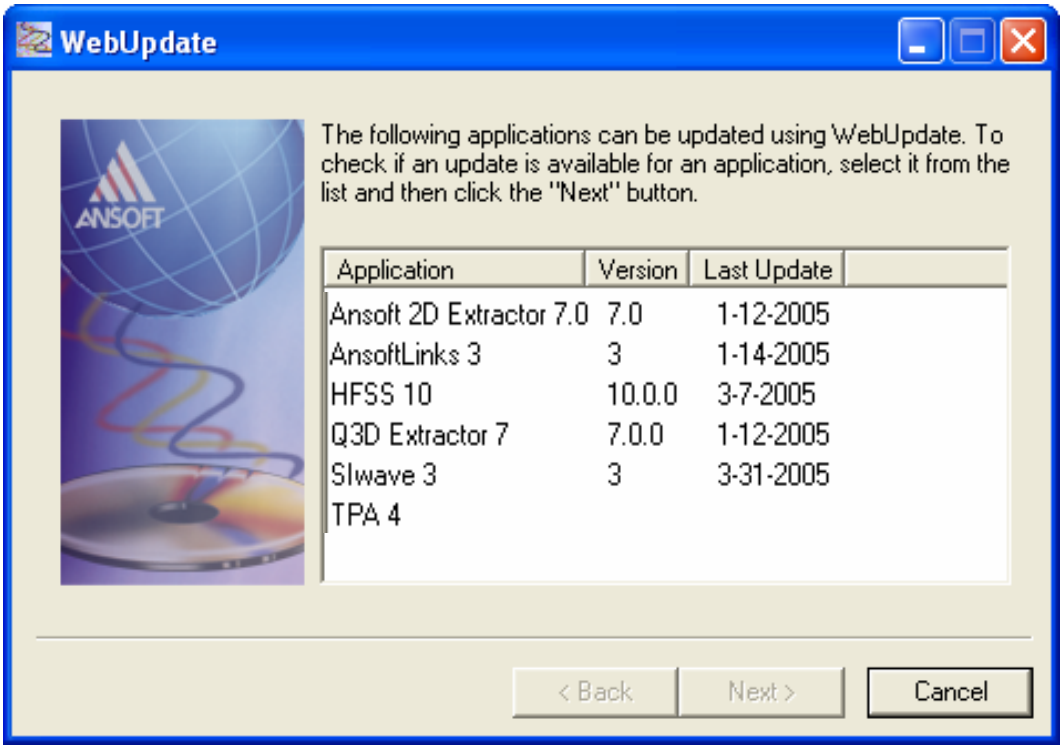
For Technical Support

- The following link will direct you to the Ansoft Support Page. The Ansoft Support Pages provide additional documentation, training, and application notes.
  - Web Site: http://www.ansoft.com/support.cfm
  - Technical Support:
    - 9-4 EST:
      - **Pittsburgh, PA**
      - (412) 261-3200 x0 – Ask for Technical Support
      - **Burlington, MA**
      - (781) 229-8900 x0 – Ask for Technical Support
    - 9-4 PST:
      - **San Jose, CA**
      - (408) 261-9095 x0 – Ask for Technical Support
      - **Portland, OR**
      - (503) 906-7944 or (503) 906-7947
      - **El Segundo, CA**
      - (310) 426-2287 – Ask for Technical Support

WWW.MOHANDESYAR.COM

WebUpdate

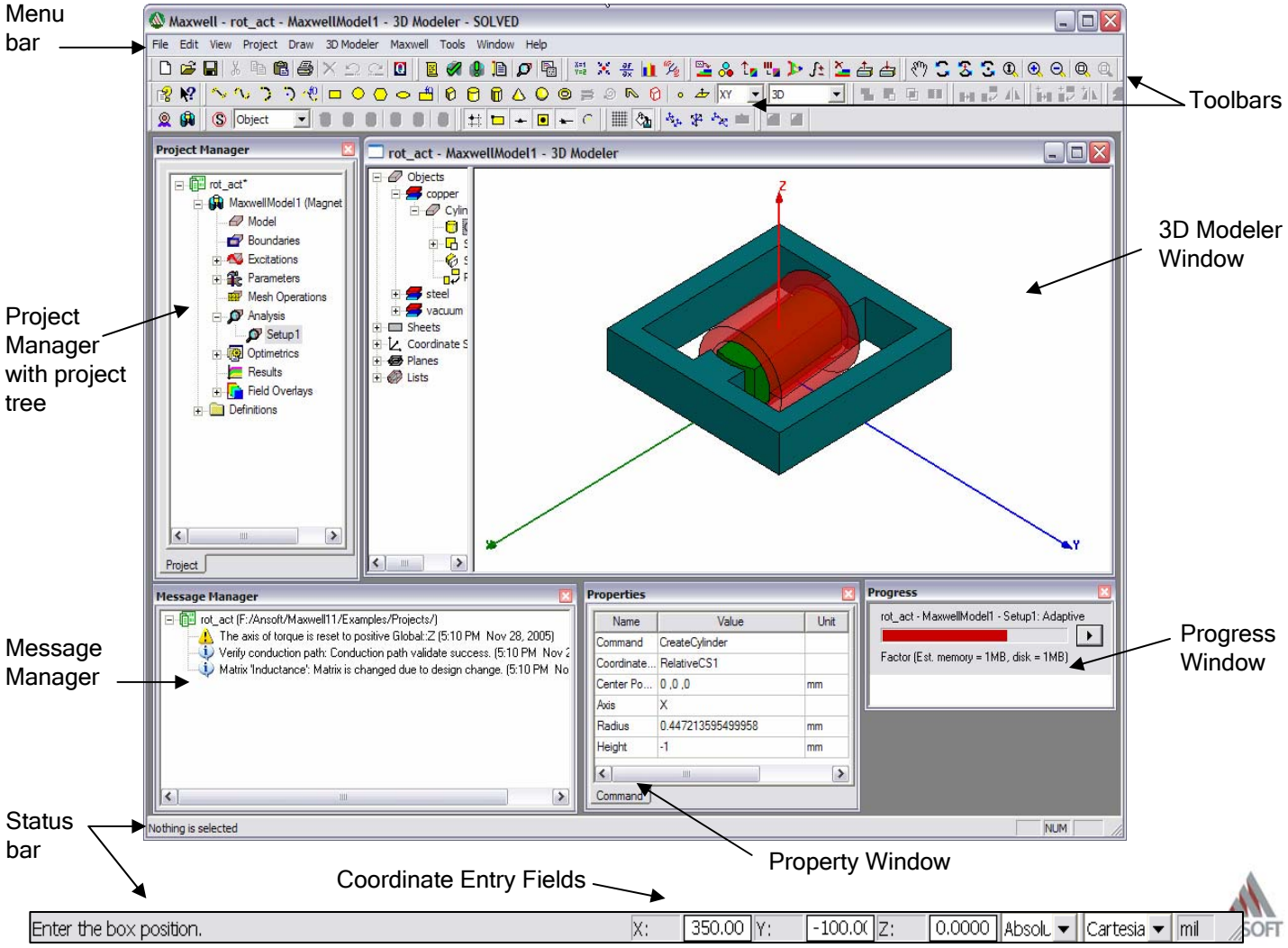
- This new feature allows you to update any existing Ansoft software from the WebUpdate window. This feature automatically scans your system to find any Ansoft software, and then allows you to download any updates if they are available.



WWW.MOHANDESYAR.COM

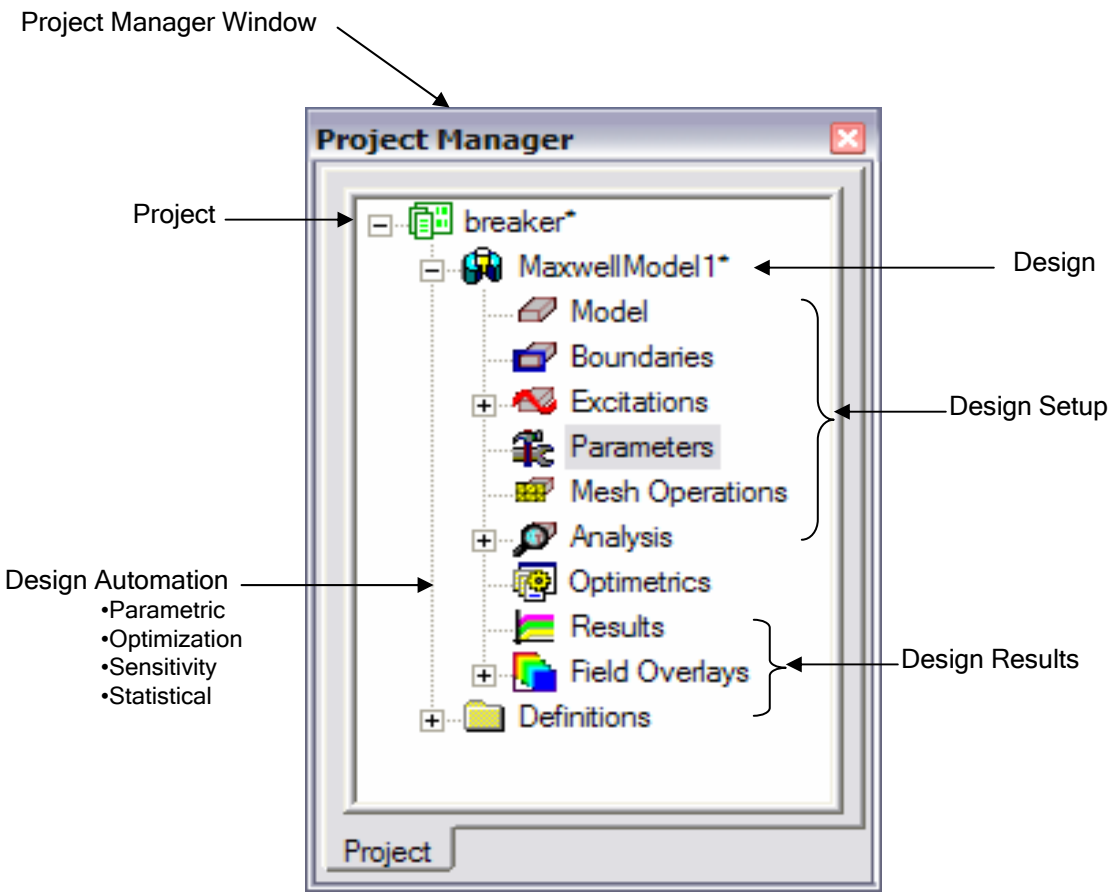
Ansoft Terms

- The Ansoft Maxwell window has several optional panels:
  - A **Project Manager** which contains a design tree which lists the structure of the project.
  - A **Message Manager** that allows you to view any errors or warnings that occur before you begin a simulation.
  - A **Property Window** that displays and allows you to change model parameters or attributes.
  - A **Progress Window** that displays solution progress.
  - A **3D Modeler Window** which contains the model and model tree for the active design. For more information about the3D Modeler Window, see chapter 1.



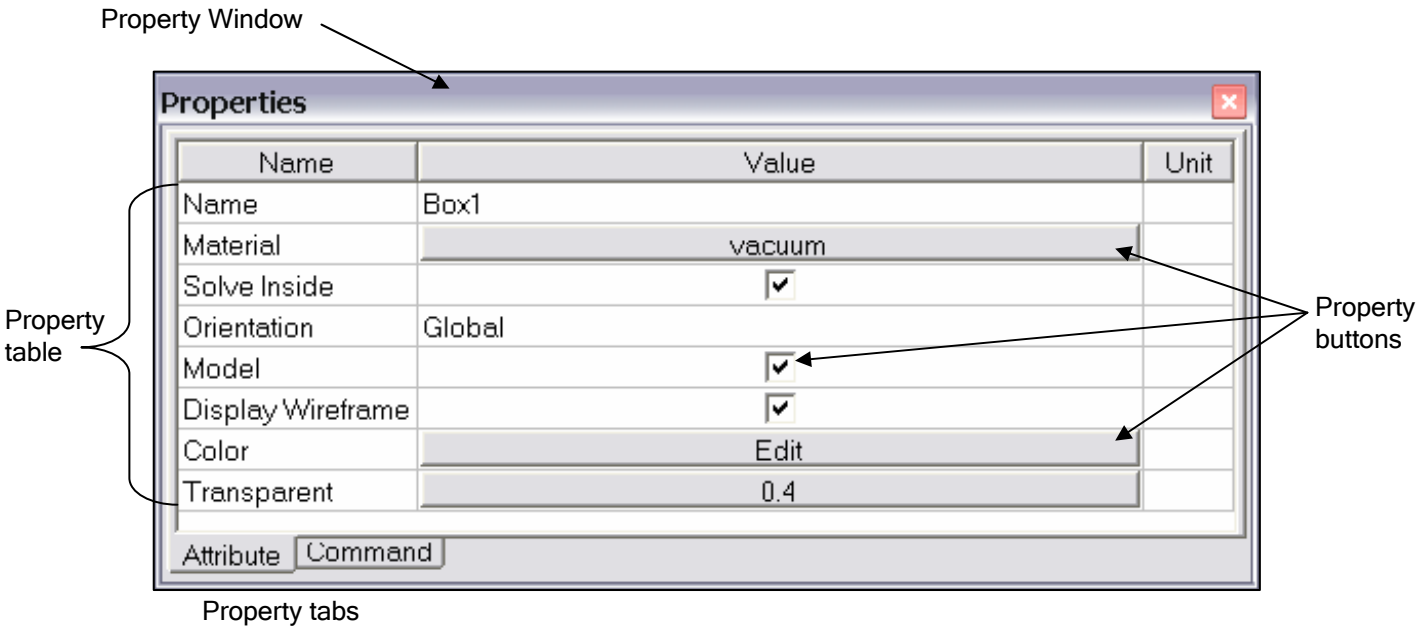
WWW.MOHANDESYAR.COM

Project Manager



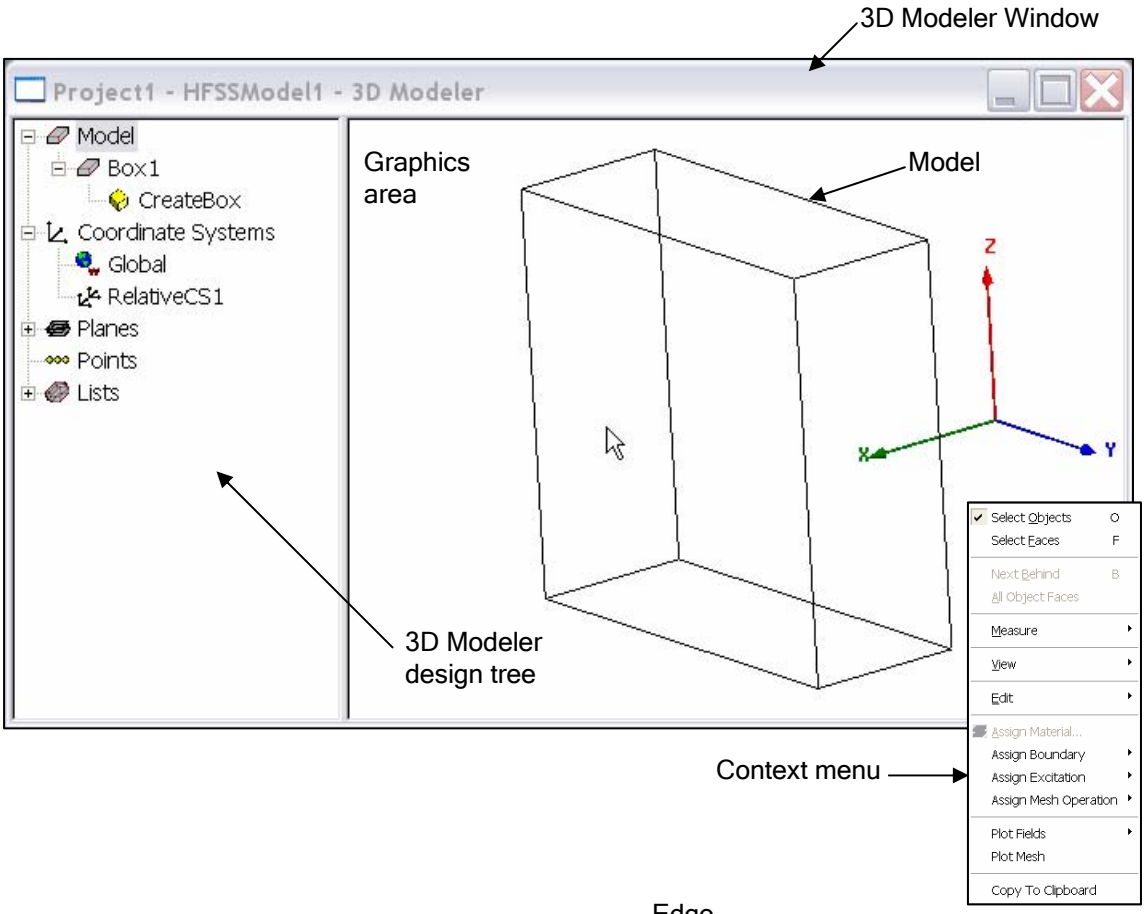
WWW.MOHANDESYAR.COM

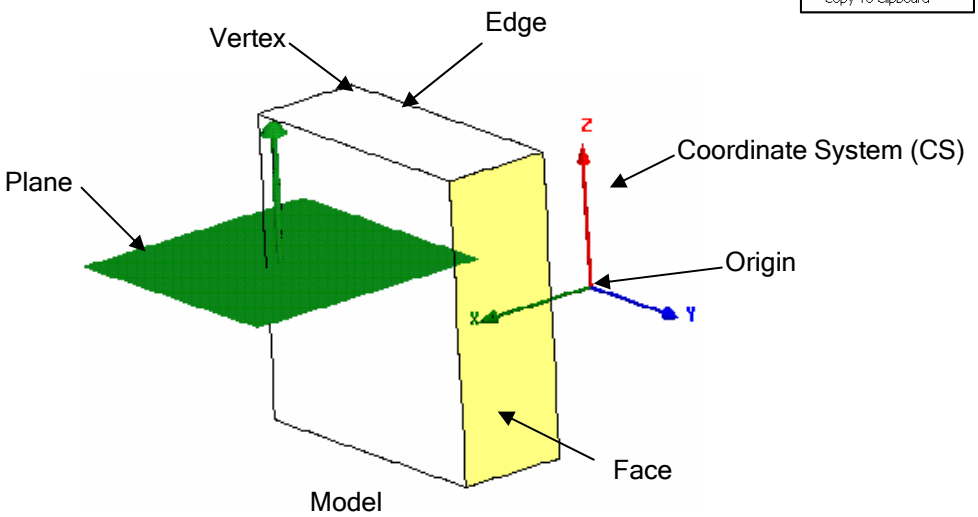
Property Window



WWW.MOHANDESYAR.COM

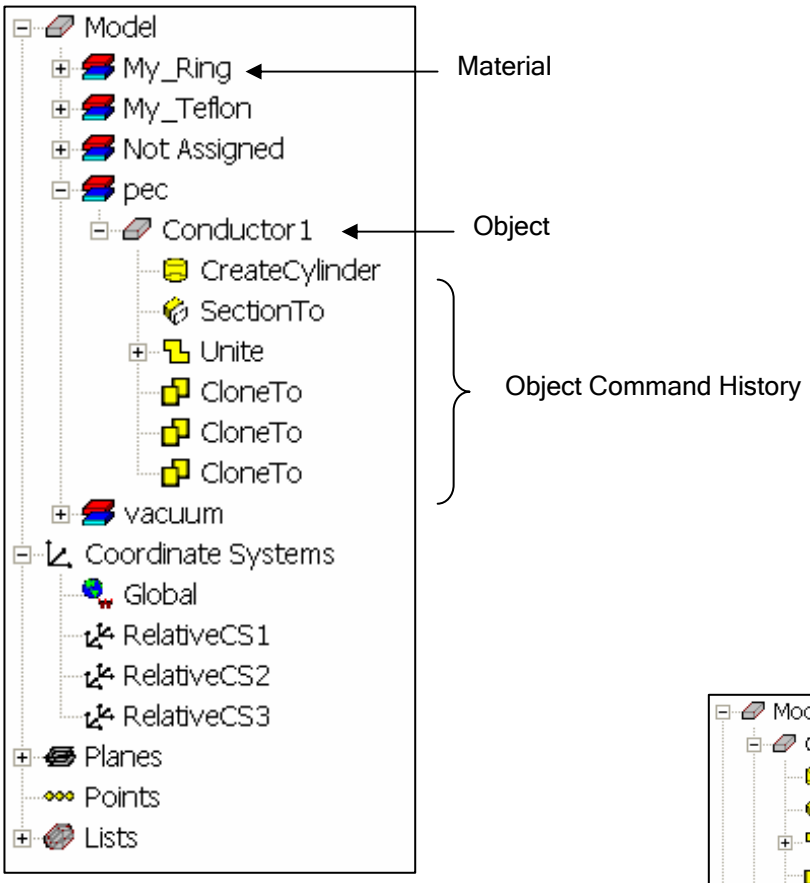
Ansoft 3D Modeler



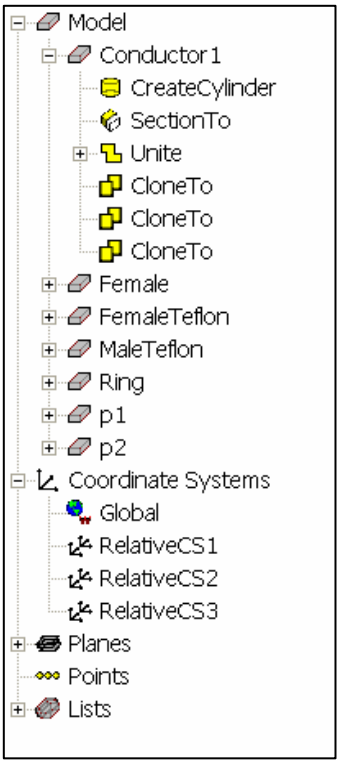


WWW.MOHANDESYAR.COM

3D Modeler Design Tree



Grouped by Material

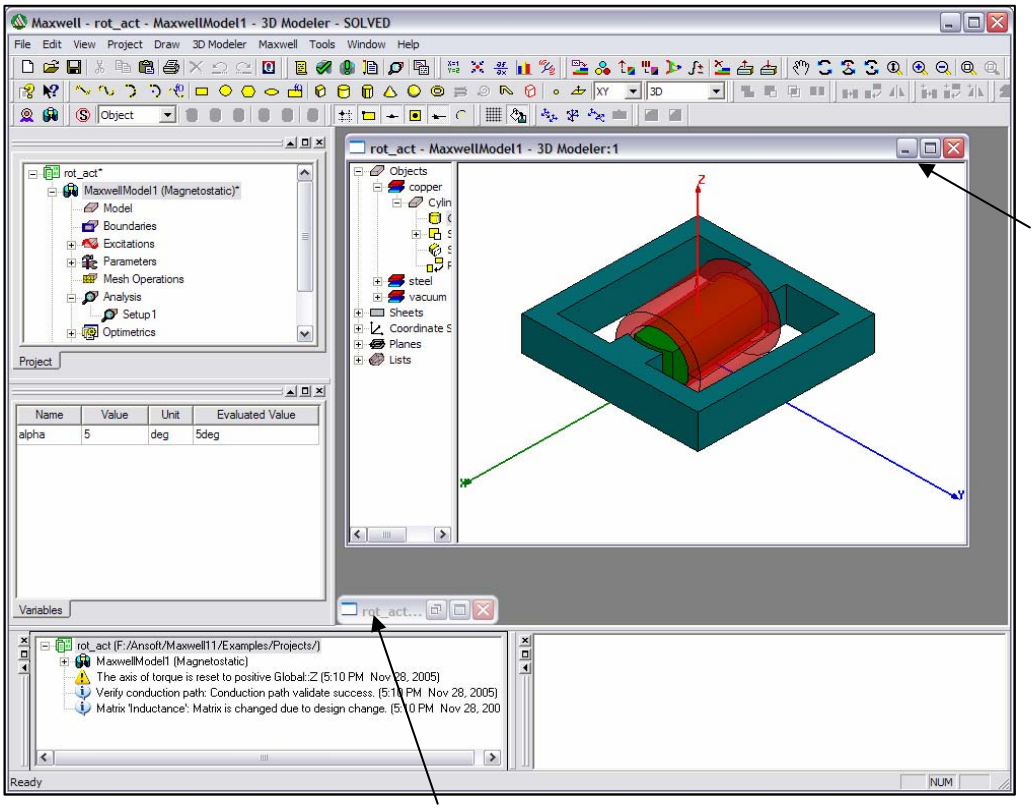


Object View

WWW.MOHANDESYAR.COM

Design Windows

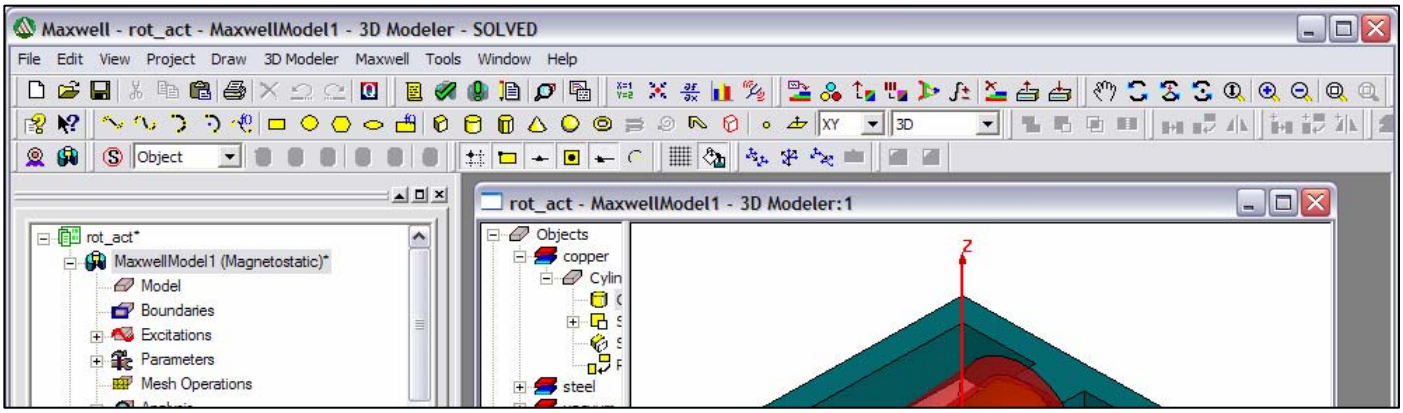
- In the Ansoft Maxwell Desktop, each project can have multiple designs and each design is displayed in a separate window.
- You can have multiple projects and design windows open at the same time. Also, you can have multiple views of the same design visible at the same time.
- To arrange the windows, you can drag them by the title bar, and resize them by dragging a corner or border. Also, you can select one of the following menu options: ***Window >Cascade***, ***Window >Tile Vertically***, or ***Window > Tile Horizontally.***
- To organize your Ansoft Maxwell window, you can iconize open designs. Click the Iconize ** symbol in the upper right corner of the document border. An icon appears in the lower part of the Ansoft Maxwell window. If the icon is not visible, it may be behind another open document. Resize any open documents as necessary. Select the menu item ***Window > Arrange Icons*** to arrange them at the bottom of the Ansoft Maxwell window.
- Select the menu item ***Window > Close All*** to close all open design. You are prompted to **Save** unsaved designs.



WWW.MOHANDESYAR.COM

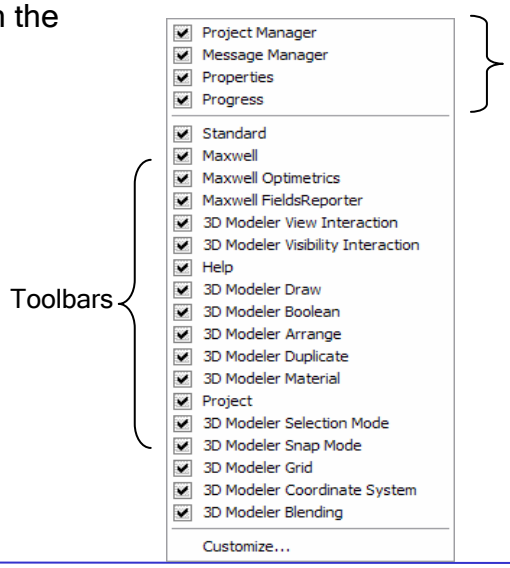
Toolbars

- The toolbar buttons are shortcuts for frequently used commands. Most of the available toolbars are displayed in this illustration of the Ansoft Maxwell initial screen, but your Ansoft Maxwell window probably will not be arranged this way. You can customize your toolbar display in a way that is convenient for you.
- Some toolbars are always displayed; other toolbars display automatically when you select a document of the related type. For example, when you select a 2D report from the project tree, the 2D report toolbar displays.



To display or hide individual toolbars:

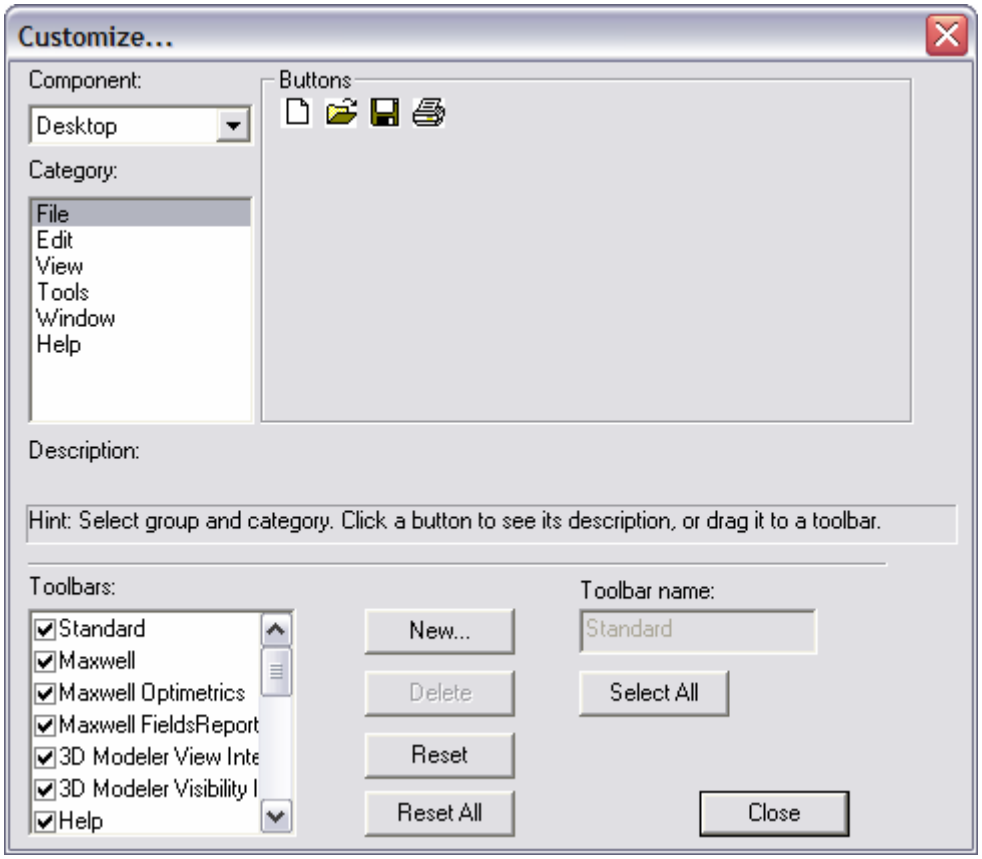
- Right-click the Ansoft Maxwell window frame.
  - A list of all the toolbars is displayed. The toolbars with a check mark beside them are visible; the toolbars without a check mark are hidden. Click the toolbar name to turn its display on or off
- To make changes to the toolbars, select the menu item *Tools > Customize*. See **Customize and Arrange Toolbars** on the next page.



WWW.MOHANDESYAR.COM

Customize and Arrange Toolbars

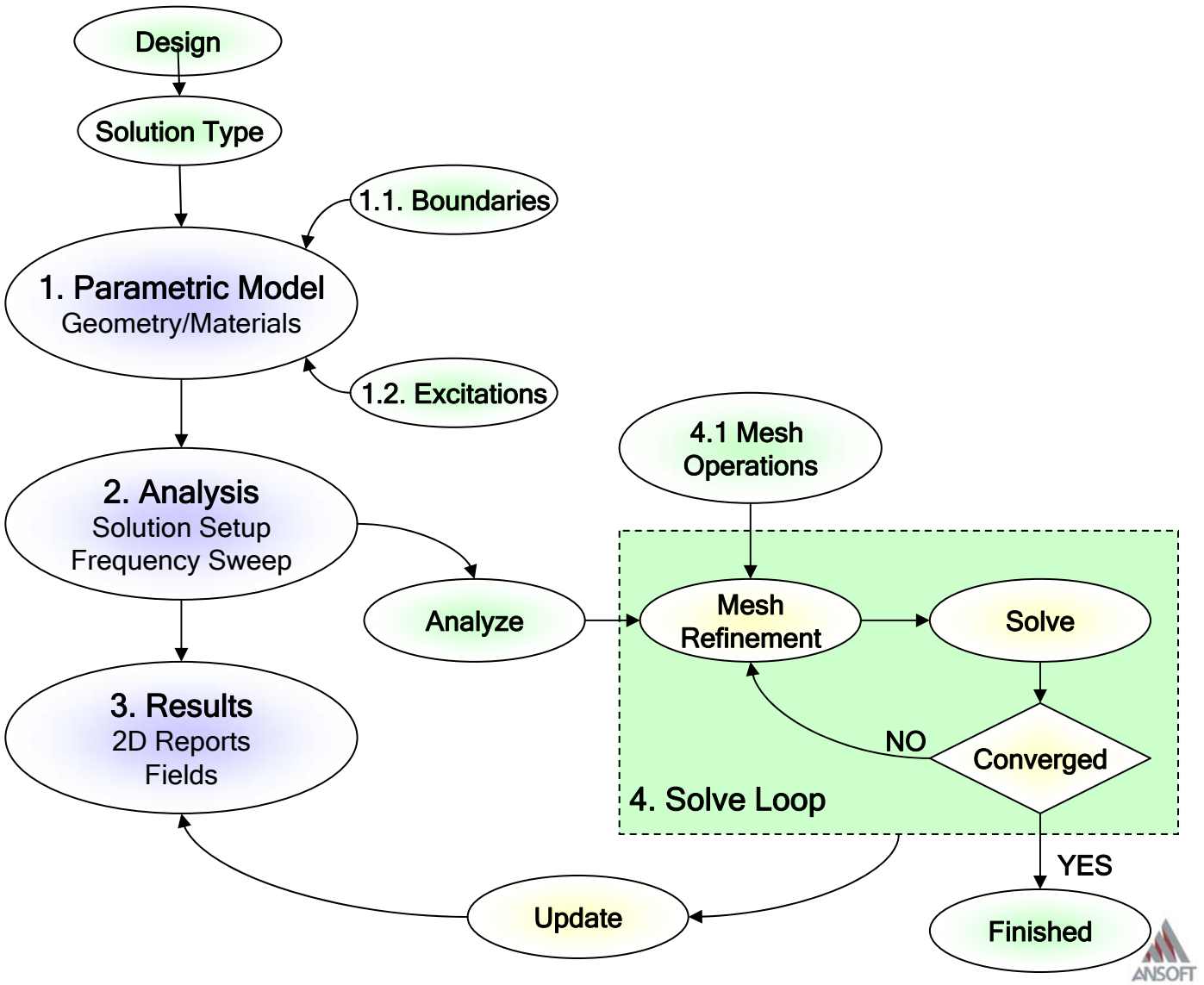
- To customize toolbars:
  - Select the menu item ***Tools > Customize,*** or right-click the Ansoft Maxwell window frame and click **Customize** at the bottom of the toolbar list.
  - In the Customize dialog, you can do the following:
    - **View a Description of the toolbar commands**
      1. Select an item from the Component pull-down list
      2. Select an item from the Category list
      3. Using the mouse click on the Buttons to display the Description
      4. Click the **Close** button when you are finished
    - **Toggle the visibility of toolbars**
      1. From the Toolbar list, toggle the check boxes to control the visibility of the toolbars
      2. Click the **Close** button when you are finished



WWW.MOHANDESYAR.COM

Ansoft Maxwell Desktop

- The Ansoft Maxwell Desktop provides an intuitive, easy-to-use interface for developing passive RF device models. Creating designs, involves the following:
  1. **Parametric Model Generation** – creating the geometry, boundaries and excitations
  2. **Analysis Setup** – defining solution setup and frequency sweeps
  3. **Results** – creating 2D reports and field plots
  4. **Solve Loop** - the solution process is fully automated
- To understand how these processes co-exist, examine the illustration shown below.



WWW.MOHANDESYAR.COM

Opening a Maxwell project

This section describes how to open a new or existing project.

Opening a New project

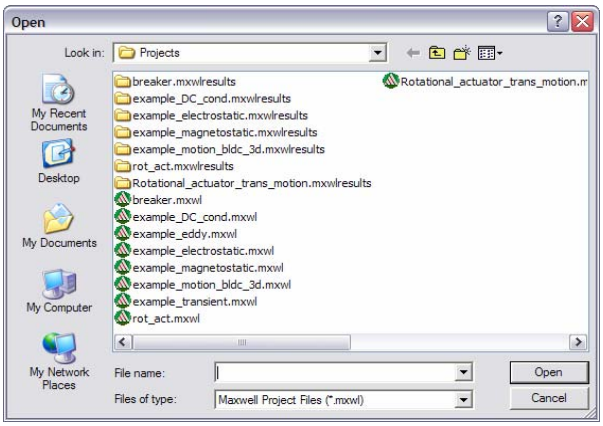
To open a new project:

1. In an Ansoft Maxwell window, select the menu item *File > New*.
2. Select the menu *Project > Insert Maxwell Design*.

Opening an Existing Maxwell project

To open an existing project:

1. In an Ansoft Maxwell window, select the menu *File > Open*. Use the Open dialog to select the project.
2. Click **Open** to open the project



Opening an Existing Project from Explorer

You can open a project directly from the Microsoft Windows Explorer.

To open a project from Windows Explorer, do one of the following:

- Double-click on the name of the project in Windows Explorer.
- Right-click the name of the project in Windows Explorer and select **Open** from the shortcut menu.

WWW.MOHANDESYAR.COM

Set Solution Type

This section describes how to set the Solution Type. The Solution Type defines the type of results, how the excitations are defined, and the convergence. The following Solution Types are available:

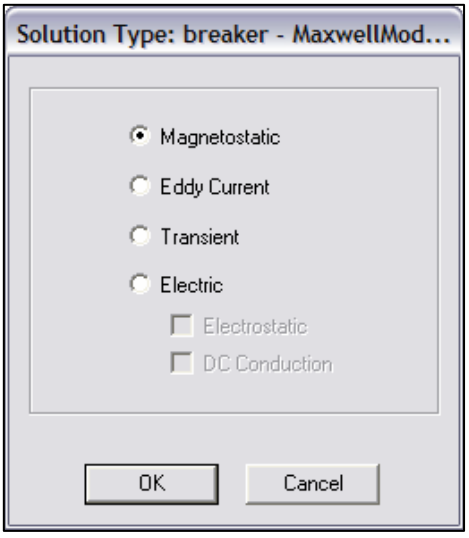
1. **Magneto static** - calculates the
2. **Eddy Current** - calculates
3. **Transient** - calculates the
4. **Electric** - calculates the

Convergence

1. **Magneto static** -
2. **Eddy Current** -
3. **Transient** -
4. **Electric** -

To set the solution type:

1. Select the menu item ***Maxwell > Solution Type***
2. Solution Type Window:
    1. Choose one of the following:
        1. **Magnetostatic**
        2. **Eddy Current**
        3. **Transient**
        4. **Electric**
    2. Click the **OK** button



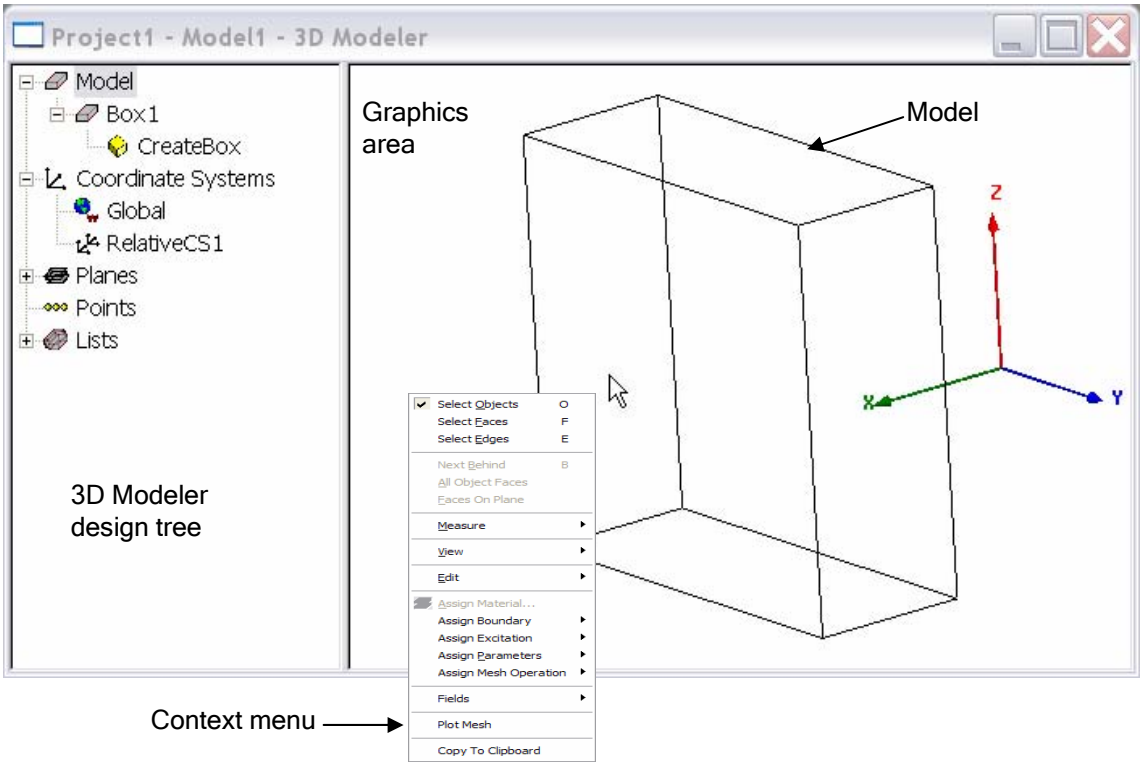
WWW.MOHANDESYAR.COM

Parametric Model Creation

- The Ansoft Maxwell 3D Modeler is designed for ease of use and flexibility. The power of the 3D Modeler is in its unique ability to create fully parametric designs without editing complex macros/model history.
- The purpose of this chapter is to provide an overview of the 3D Modeling capabilities. By understanding the basic concepts outlined here you will be able to quickly take advantage of the full feature set offered by the 3D Parametric Modeler.

Overview of the 3D Modeler User Interface

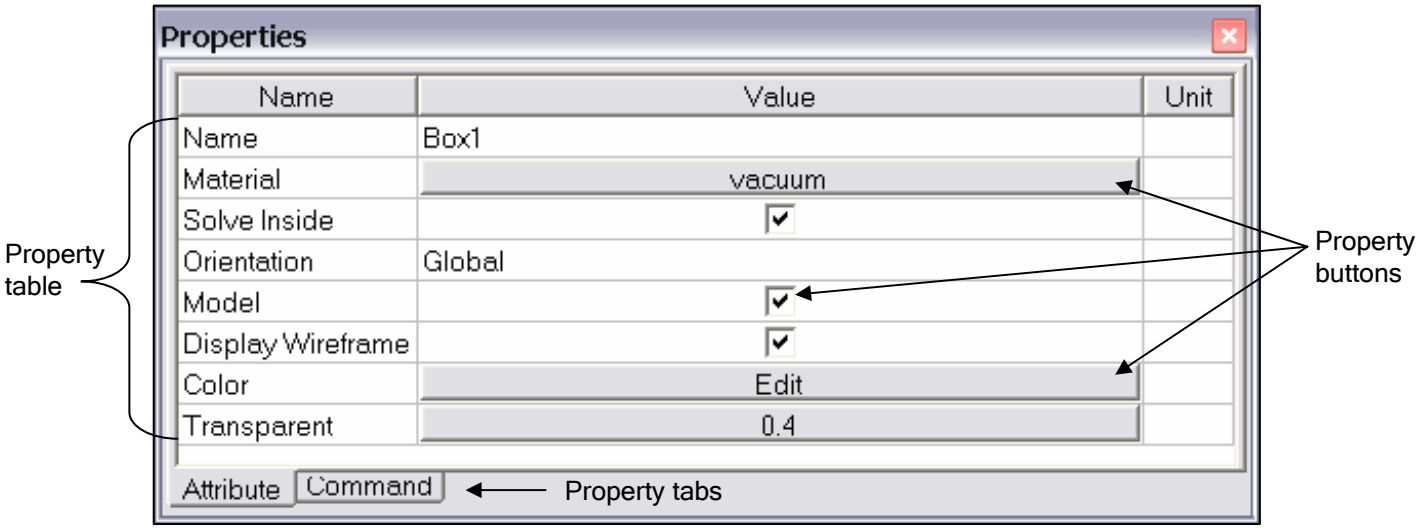
- The following picture shows the 3D Modeler window.
  - **3D Modeler Design Tree** – The 3D Modeler Design Tree is an essential part of the user interface. From here you may access the structural elements in addition to any object dependencies and attributes.
  - **Context Menus** – Context menus are a flexible way of accessing frequently used menu commands for the current context. The contents of these menus change dynamically and are available throughout the interface by clicking the right mouse button.
  - **Graphics Area** – The graphics area is used to interact with the structural elements.



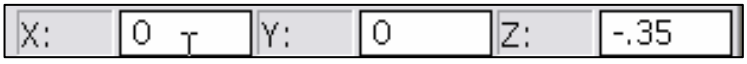
WWW.MOHANDESYAR.COM

Overview of the 3D Modeler User Interface (Continued)

- When using the 3D Modeler interface you will also interact with two additional interfaces:
  - **Property Window** – The Property Window is used to view or modify the attributes and dimensions of structural objects



  - **Status Bar/Coordinate Entry** – The Status Bar on the Ansoft Maxwell Desktop Window displays the Coordinate Entry fields that can be used to define points or offsets during the creation of structural objects



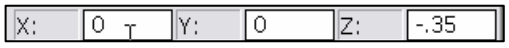
WWW.MOHANDESYAR.COM

Grid Plane

- To simplify the creation of structural primitives, a grid or drawing plane is used. The drawing plane does not in any way limit the user to two dimensional coordinates but instead is used as a guide to simplify the creation of structural primitives. The drawing plane is represented by the active grid plane (The grid does not have to be visible). To demonstrate how drawing planes are used, review the following section: Creating and Viewing Simple Structures.

Active Cursor

- The active cursor refers to the cursor that is available during object creation. The cursor allows you to graphically change the current position. The position is displayed on the status bar of the Ansoft Maxwell Desktop Window.



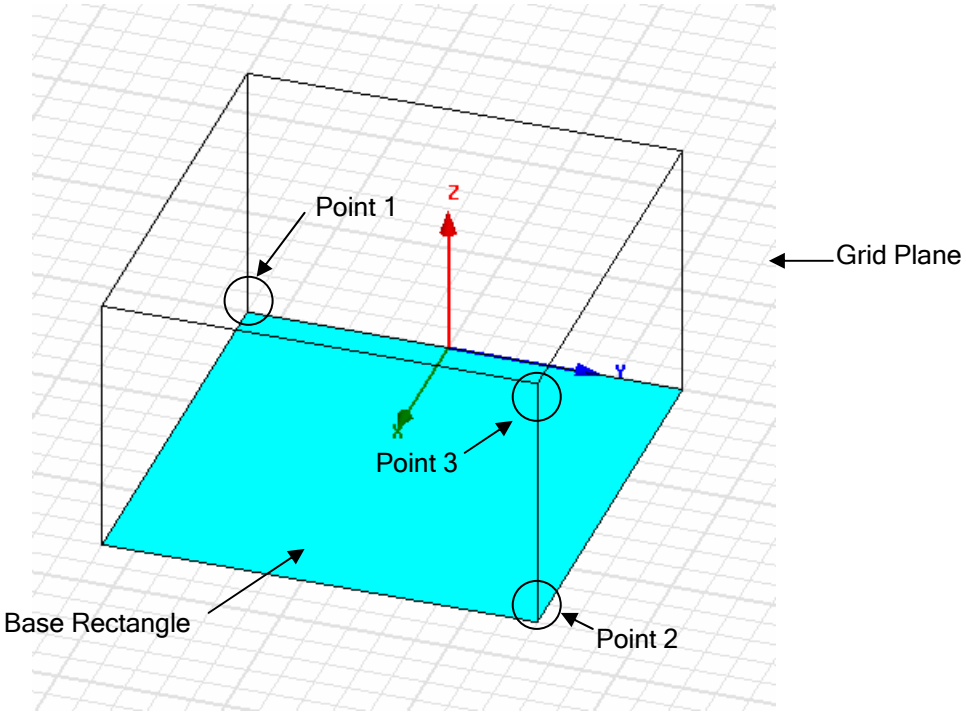
- When objects are not being constructed, the cursor remains passive and is set for dynamic selection. See the Overview of Selecting Objects for more details.

WWW.MOHANDESYAR.COM

Creating and Viewing a Simple Structure

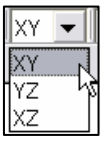
- Creating 3D structural objects is accomplished by performing the following steps:
  1. Set the grid plane
  2. Create the base shape of the object
  3. Set the Height

- **Create a Box**
  - We will investigate creating a box to demonstrate these steps. These steps assume that project and a Maxwell design have already been created. Three points are required to create the box. The first two form the base rectangle and the third sets the height.
    - Point 1: Defines the start point of the base rectangle
    - Point 2: Defines the size of the base rectangle
    - Point 3: Defines the height of the Box

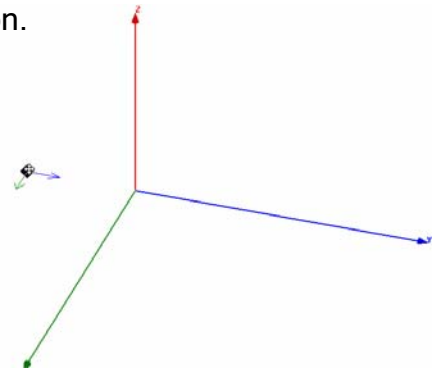


WWW.MOHANDESYAR.COM

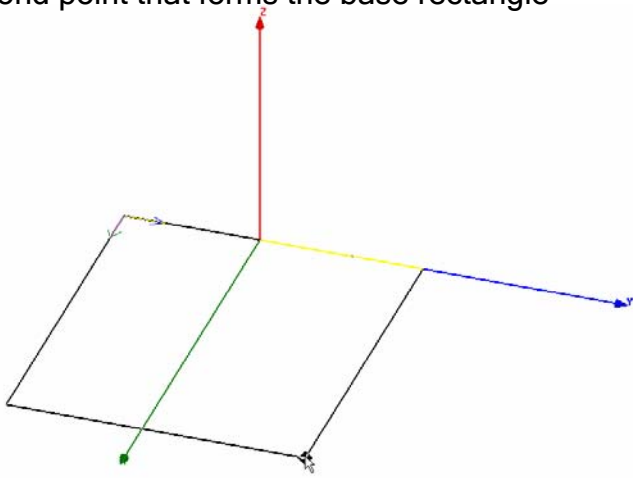
- **Create a Box (Continued)**



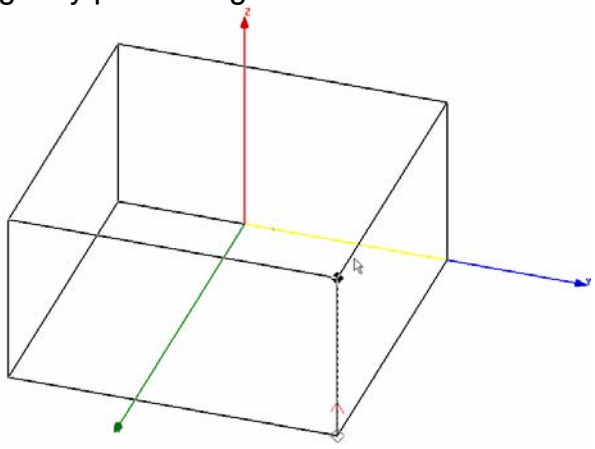
1. Select the menu item *3D Modeler > Grid Plane > XY*
2. Use the mouse to create the base shape
    1. Set the start point by positioning the active cursor and click the left mouse button.



    2. Position the active cursor and click the left mouse button to set the second point that forms the base rectangle



3. Set the Height by positioning the active cursor and clicking left mouse button.

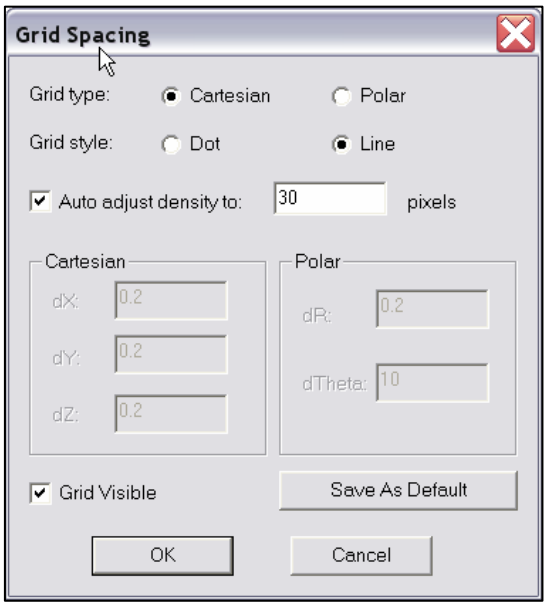


WWW.MOHANDESYAR.COM

Specifying Points

Grid

- From the example, we saw that the simplest way to set a point is by clicking its position on the grid plane. To set the precision of the grid plane, select the menu item *View > Grid Settings.* From here you may specify the Grid Type, Style, Visibility, and Precision. By pressing the **Save As Default** button, you can set the default behavior for future Maxwell Designs.

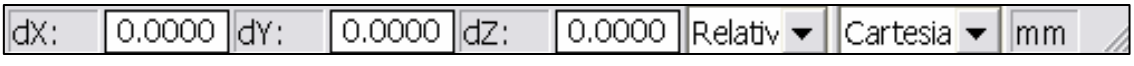


Coordinate Entry

- Another way to specify a coordinate is to use the Coordinate Entry fields which are located on the status bar of the Ansoft Maxwell Desktop. The position may be specified in **Cartesian, Cylindrical**, or **Spherical** coordinates. Once the first point is set, the Coordinate Entry will default to Relative coordinates. In Relative mode the coordinates are no longer absolute (measured from the origin of the working coordinate system), but relative to the last point entered.

Equations

- The Coordinate Entry fields allow equations to be entered for position values. Examples: 2*5, 2+6+8, 2*cos(10*(pi/180)).
- Variables are not allowed in the Coordinate Entry Field
- **Note**: Trig functions are in radians



Relative mode

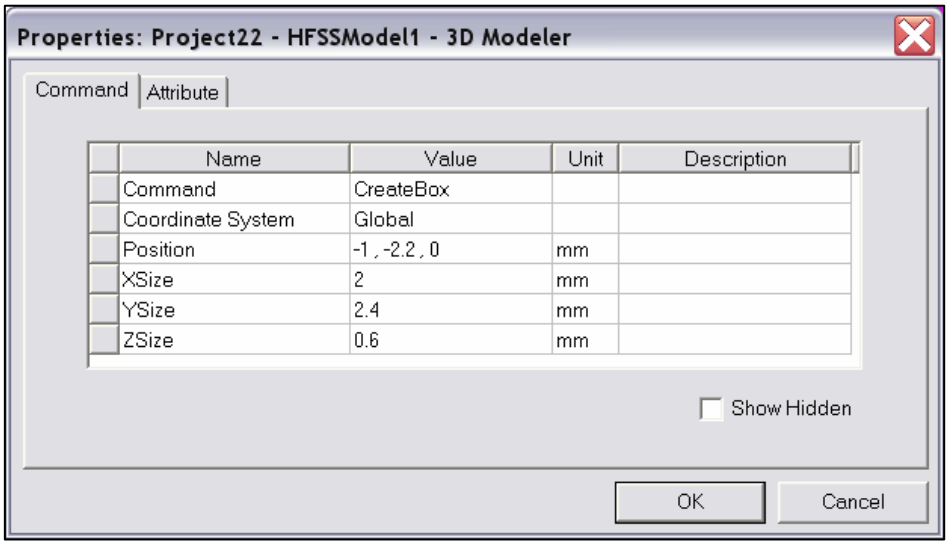
WWW.MOHANDESYAR.COM

Specifying Points (Continued)

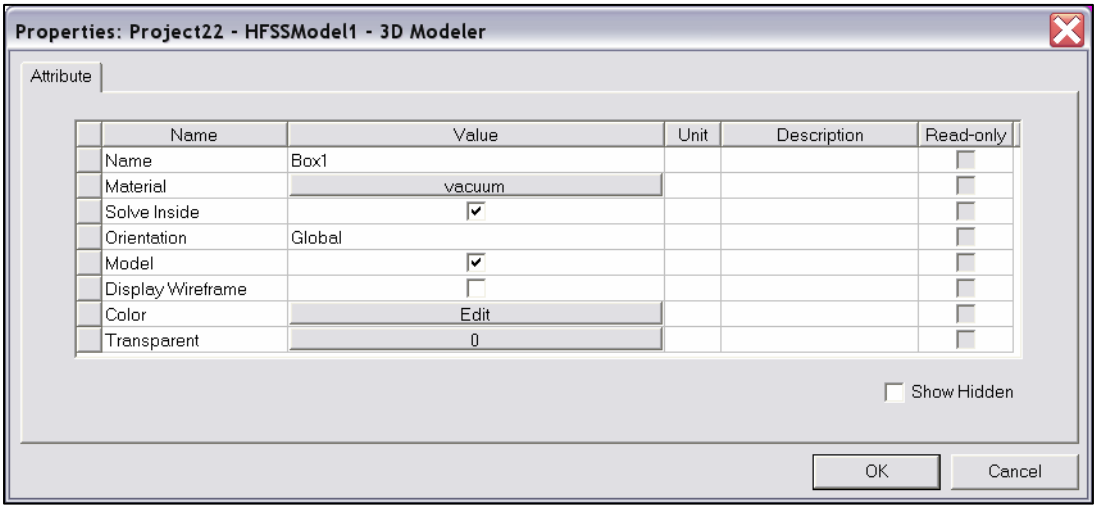
Object Properties

- By default the Properties dialog will appear after you have finished sketching an object. The position and size of objects can be modified from the dialog. This method allows you to create objects by clicking the estimated values using the mouse and then correcting the values in the final dialog.
- The Property dialog accepts equations, variables, and units. See the Overview of Entering Parameters for more detail.
- Every object has two types of properties
  1. **Command** – Defines the structural primitive
  2. **Attributes** – Defines the material, display, and solve properties

Commands



Attributes



WWW.MOHANDESYAR.COM

Overview of Draw

- **Primitives**
  - In solid modeling, the basic element or object is often called a primitive. Examples of primitives are boxes, cylinders, rectangles, circles, etc. There are two types of primitives: 3D primitives or solids, and 2D primitives or surfaces. By placing a collection of primitives in the correct location and of the correct size we can create a represent complex structural objects.
  - To create complex objects, primitives can be used as “tools” to cut holes, carve away, or join. The operations that are performed with these “tools” are often referred to as Boolean operations.
  - 2D primitives can be swept to create arbitrarily shaped solid primitives

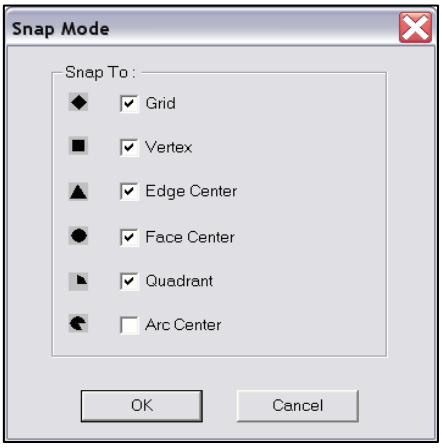
- **2D Draw Objects**
  - The following 2D Draw objects are available:
    - Rectangle, Circle, Line, Point, Spline, Ellipse, Regular Polygon (v10 circle)

- **3D Draw Objects**
  - The following 3D Draw objects are available:
    - Box, Cylinder, Sphere, Torus, Helix, Bond Wire, Cone, Regular Polyhedron (v10 cylinder)

- **True Surfaces**
  - Circles, Cylinders, Spheres, etc are represented as true surfaces. In versions prior to release 11, these primitives would be represented as faceted objects. If you wish to use the faceted primitives (Cylinders or Circles), select the Regular Polyhedron or Regular Polygon.
  - To control the mesh generation of true surfaces objects, see the section on Mesh Control.

WWW.MOHANDESYAR.COM

Overview of Draw (Continued)

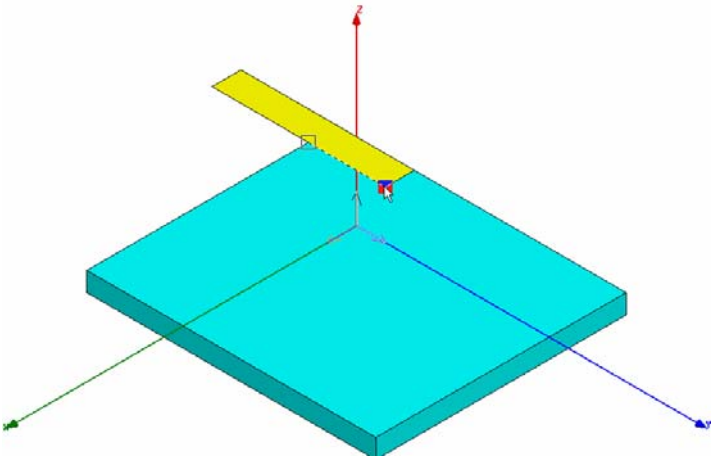


- **Snap Mode**
  - As an aid for graphical selection, the modeler provides Snap options. The default is to snaps are shown here. The shape of the active cursor will dynamically change as the cursor is moved over the snap positions.

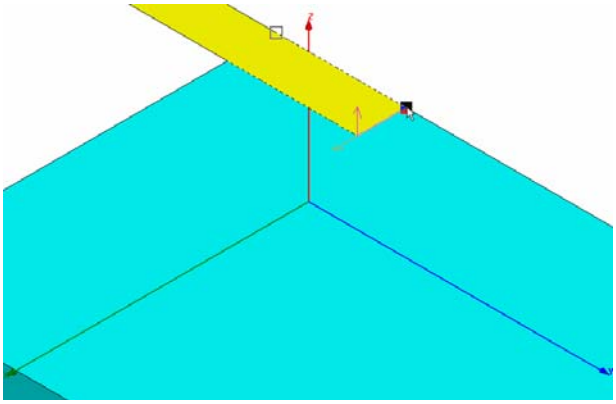
- **Moving**
  - By default all active cursor movement is in three dimensions. The modeler can also be set to allow the active cursor to only move in a plane or out of plane. These are set from the menu item **3D Modeler > Movement Mode.**
  - In addition, the movement can be limited to a specific direction (x, y, or z) by holding down the x, y, or z key. This prevents movement in the other directions.
  - Pressing the **CTRL+Enter** key sets a local reference point. This can be useful for creating geometry graphically that is based on an existing objects. This is outlined on the next page:

WWW.MOHANDESYAR.COM

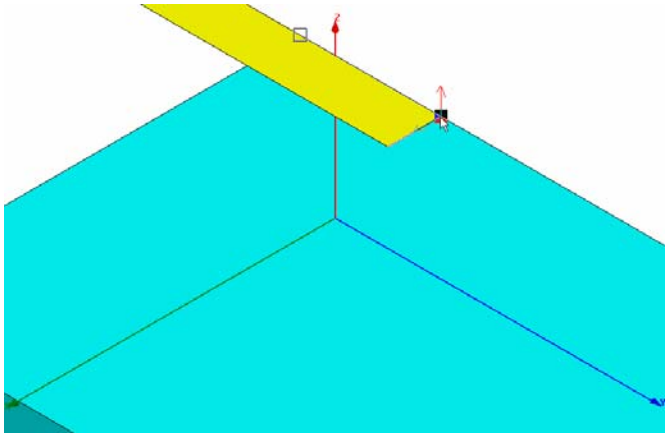
- Moving (Continued)



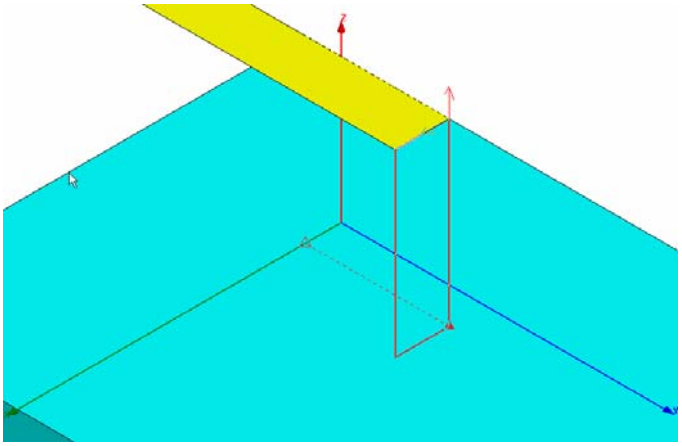
Step 1: Start Point



Step 2: Hold X key and select vertex point



Step 3: CTRL+Enter Keys set a local reference



Step 4: Hold Z key and set height

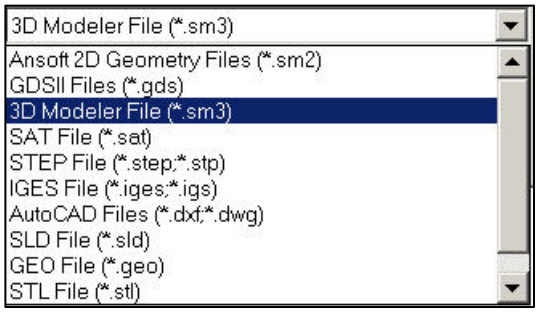
WWW.MOHANDESYAR.COM

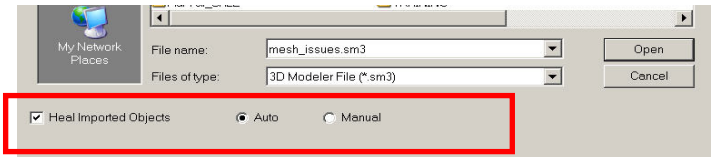
Overview of Draw

Import

- In 3D modeler you can import a drawing file from outside.

Choose option **3D Modeler -> Import** . Here is the list of import files that we support. For some of these import option you will need an add-on translator feature in your license file.



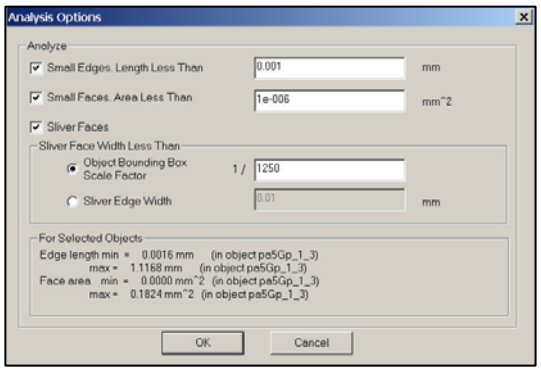


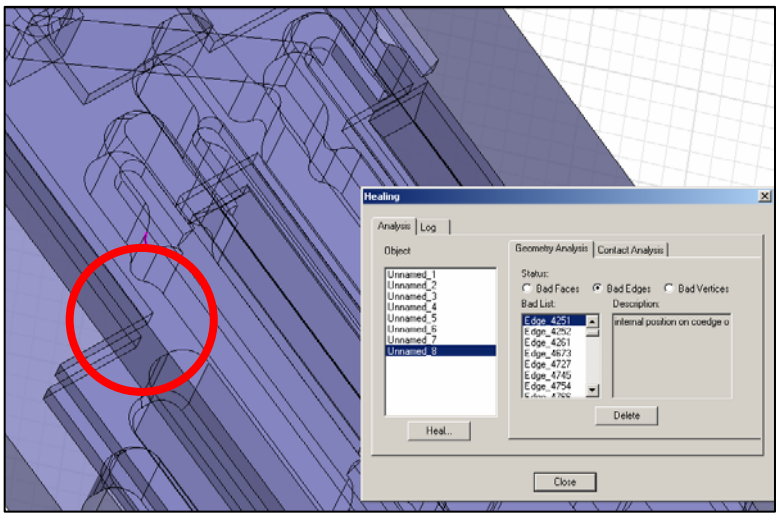
Healing

- Automated healing for imported solid models
- Post-translation user controlled healing

3D Model Analysis – 3D Modeler/Analyze

- Face, Object , Area analysis based on user inputs
- List of problems (faces, edges, vertices)
- Auto Zoom In into region where problem exists
- Remove Face
- Remove Edge
- Remove Sliver
- Remove Vertices





WWW.MOHANDESYAR.COM

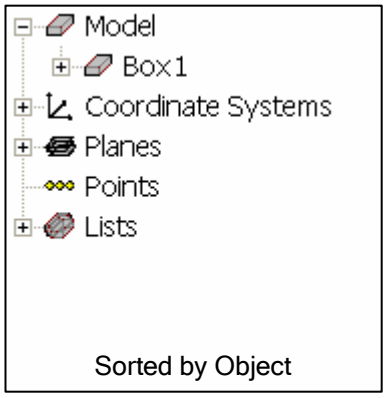
Selecting Previously Defined Shapes

- You may select an object by moving the mouse over the object in the graphics area and clicking on it. The default mode is Dynamic selection which will display the object to be selected with a unique outline color. Please note that after selecting (Clicking on the object) the object it will be displayed solid pink while all other objects are drawn transparent.
- **Types of Selection**
  - The default is to select objects. Sometimes is necessary to select faces, edges, or vertices. To change the selection mode, select the menu item *Edit > Select* and choose the appropriate selection mode. The shortcut keys **o** (Object selection) and **f** (face selection) are useful for quickly switching between the most common selection modes
- **Multiple Select or Toggle Selection**
  - Multiple objects can be selected graphically by holding down the CTRL key while selecting. In addition, with the CTRL key pressed, the selection of an object can be toggled between selected or unselected.
- **Blocked Objects**
  - If the object you wish to select is located behind another object, select the object that is blocking the desired object and press the **b** key or right-click and select **Next Behind** from the context menu. You may repeat this as many times as needed to select the correct object.
- **Select All Visible**
  - You can select all visible objects by pressing the **CTRL+a** key or by selecting the menu item *Edit > Select All Visible.*
- **Select by Name**
  - To select objects by Name you can use anyone of the following:
    - Select the menu item *Edit > Select > By Name*
    - Select the menu item *Maxwell > List*
      - Select the **Model** tab
      - Select objects from the list
    - Use the *Model Tree*. See the next page

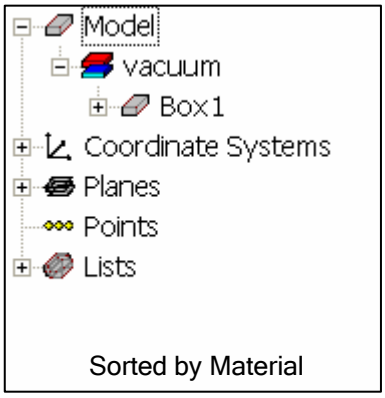
WWW.MOHANDESYAR.COM

Selecting Previously Defined Shapes (Continued)

- **Model Tree**
  - After an object has been created, it is automatically added to the Model Tree. All objects can be found in the Model Tree. If you open the Model folder you will find the objects sorted by Object or by Material. You can toggle between the views by toggling the menu item *3D Modeler > Group Object by Material*.

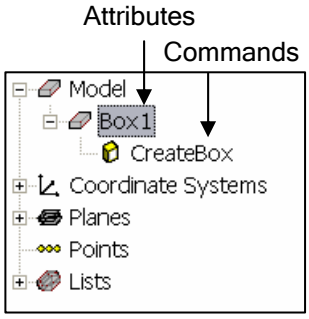


Sorted by Object



Sorted by Material

  - As stated previously, every object has two types of properties:
    - **Attributes**
      - You may select an object by clicking on the corresponding item in the Model Tree.
      - When the object is selected the attributes will be displayed in the Property Window. Double-clicking on the object will open a properties dialog. Use the Property Window or properties dialog to modify the attributes.
    - **Commands**
      - From the Model Tree, the Command Properties can be selected by expanding the object folder to display the command list. Using the mouse, select the corresponding command from the tree. The properties will be displayed in the Property Window. Double-clicking on the command will open a properties dialog. Use the Property Window or properties dialog to modify the command.
      - When the command is selected, the object will be outlined with bold lines in the 3D Model Window. Since an object can be a combination of several primitives, the command list may contain several objects. Anyone of these commands can be selected to visualize or modify the object.

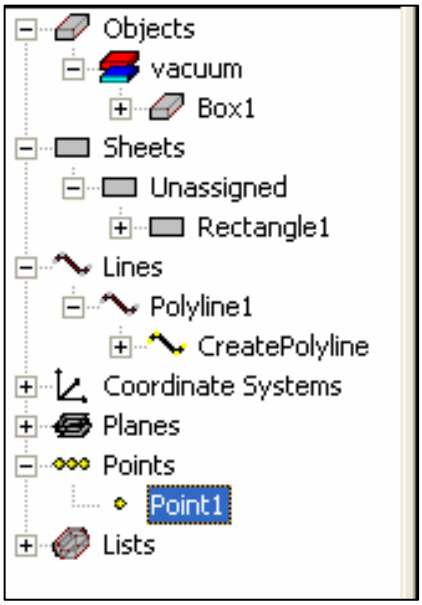


WWW.MOHANDESYAR.COM

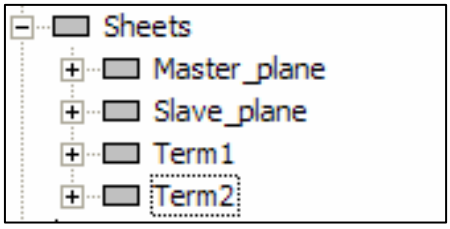
Selecting Previously Defined Shapes (Continued)

Model Tree

- Geometry in the 3D modeler is also grouped according to their model definition. Objects, Sheets, Lines, and Points are all separated so that they can be easily identified in the model tree



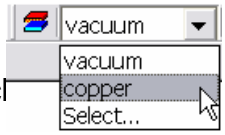
- If a boundary condition or an excitation is defined on a sheet object, then those 2D objects will be further separated according to their assignment.



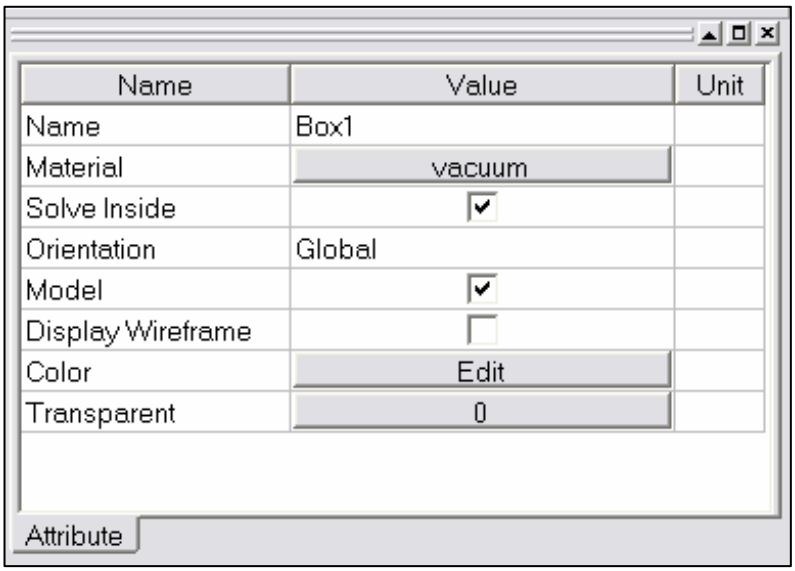
WWW.MOHANDESYAR.COM

Object Attributes

- An objects attributes set the following user defined properties:
  - **Name** – User defined name. Default names start with the primitive type followed by an increasing number: Box1, Box2, etc.
  - **Material** – User defined material property. The default property is vacuum. This can be changed by using the material toolbar



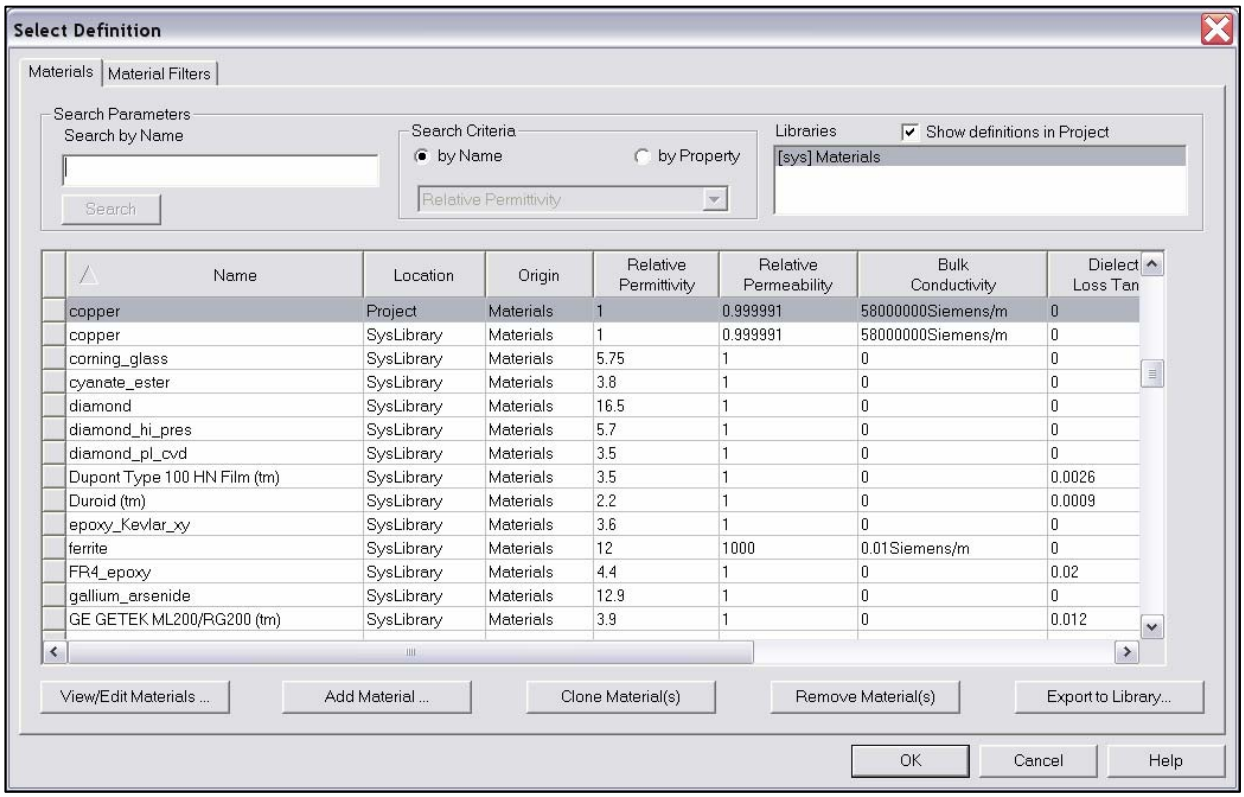
  - **Solve Inside** – By default Maxwell only solves for fields inside dielectrics. To force Maxwell to solve inside conductors, check solve inside.
  - Orientation
  - **Model Object** – Controls if the object is included in the solve
  - **Display Wireframe** – Forces the object to always be displayed as wireframe
  - **Color** – Set object color
  - **Transparency** – Set the transparency of an object. 0-Solid, 1- Wireframe
- **Note:** Visibility is not an object property.



WWW.MOHANDESYAR.COM

Materials

- By clicking on the property button for the material name, the material definition window will appear. You can select from the existing database or define a custom project material.

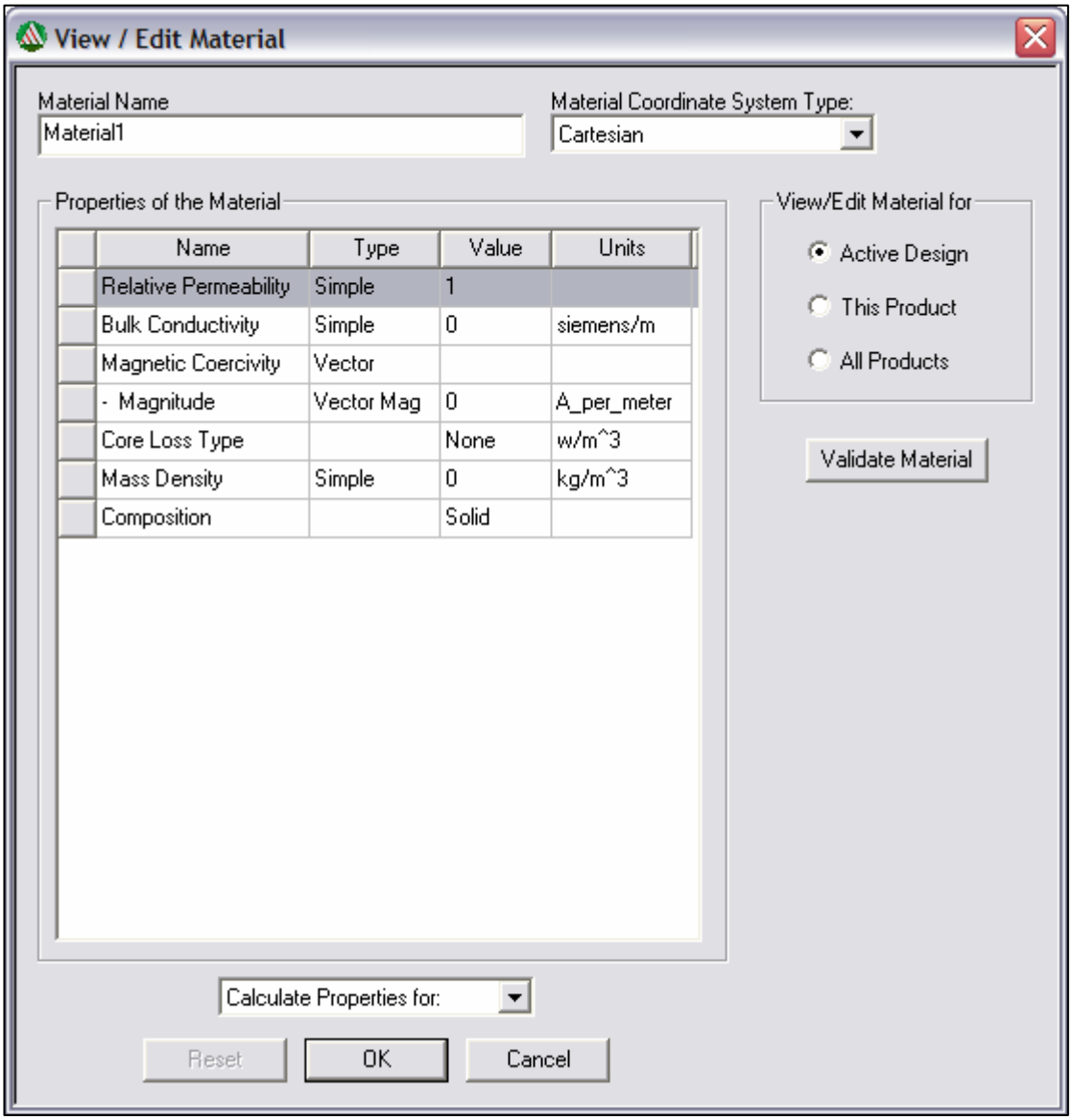


WWW.MOHANDESYAR.COM

Materials (Continued)

User Defined Project Material

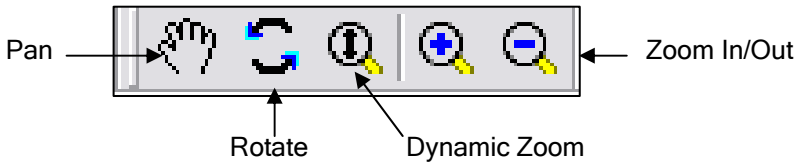
- To define a custom material click the **Add Material** button from the material definition window. The following dialog will appear. Enter the material definitions and click the **OK** button.



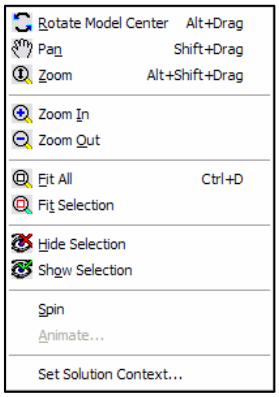
WWW.MOHANDESYAR.COM

Changing the View

- You can change the view at any time (even during shape generation) by using the following commands:
- **Toolbar**
  - **Rotate Model Center** – The structure will be rotated around the Model
  - **Pan** – The structure will be translated in the graphical area
  - **Dynamic Zoom** – Moving the mouse upwards will increase the zoom factor while moving the mouse downwards will decrease the zoom factor
  - **Zoom In/Out** – In this mode a rubber band rectangle will be defined by dragging the mouse. After releasing the mouse button the zoom factor will be applied.



- **Context Menu**
  - Right click in the graphics area and select the menu item **View** and choose from the options outlined in the Toolbar section. The context menu also offers the following:



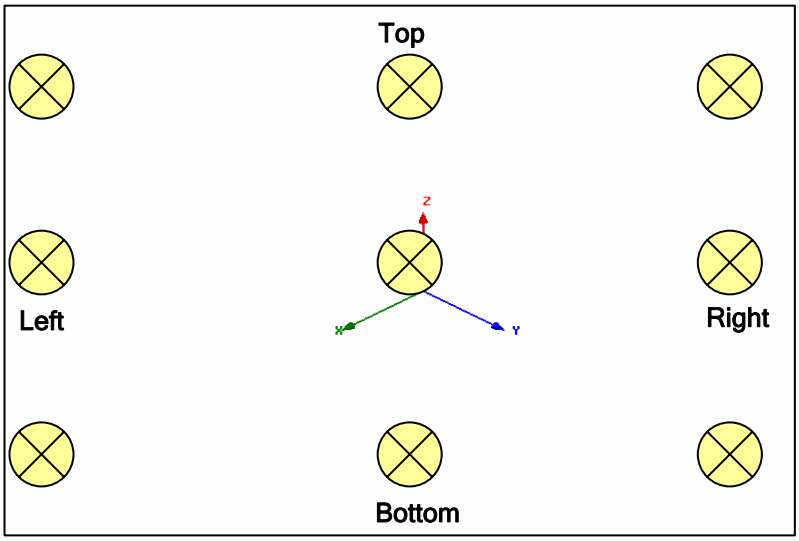
    - **Fit All** – This will zoom the defined structure to a point where it fits in the drawing area
    - **Fit Selection** – This fits only the selected objects into the drawing area.
    - **Spin** – Drag the mouse and release the mouse button to start the object spinning. The speed of the dragging prior to releasing the mouse controls the speed of the spin.
    - **Animate** – Create or display the animation of parametric geometry

- **Shortcuts**
  - Since changing the view is a frequently used operation, some useful shortcut keys exist. Press the appropriate keys and drag the mouse with the left button pressed:
    - **ALT + Drag** – Rotate
      - In addition, there are 9 pre-defined view angles that can be selected by holding the ALT key and double clicking on the locations shown on the next page.
    - **Shift + Drag** – Pan
    - **ALT + Shift + Drag** – Dynamic Zoom

WWW.MOHANDESYAR.COM

Shortcuts - Predefined Views

- These 9 pre-defined views can be seen by holding the ALT key and double clicking the left mouse button on the locations shown below.



Predefined View Angles

WWW.MOHANDESYAR.COM

Changing the View (Continued)

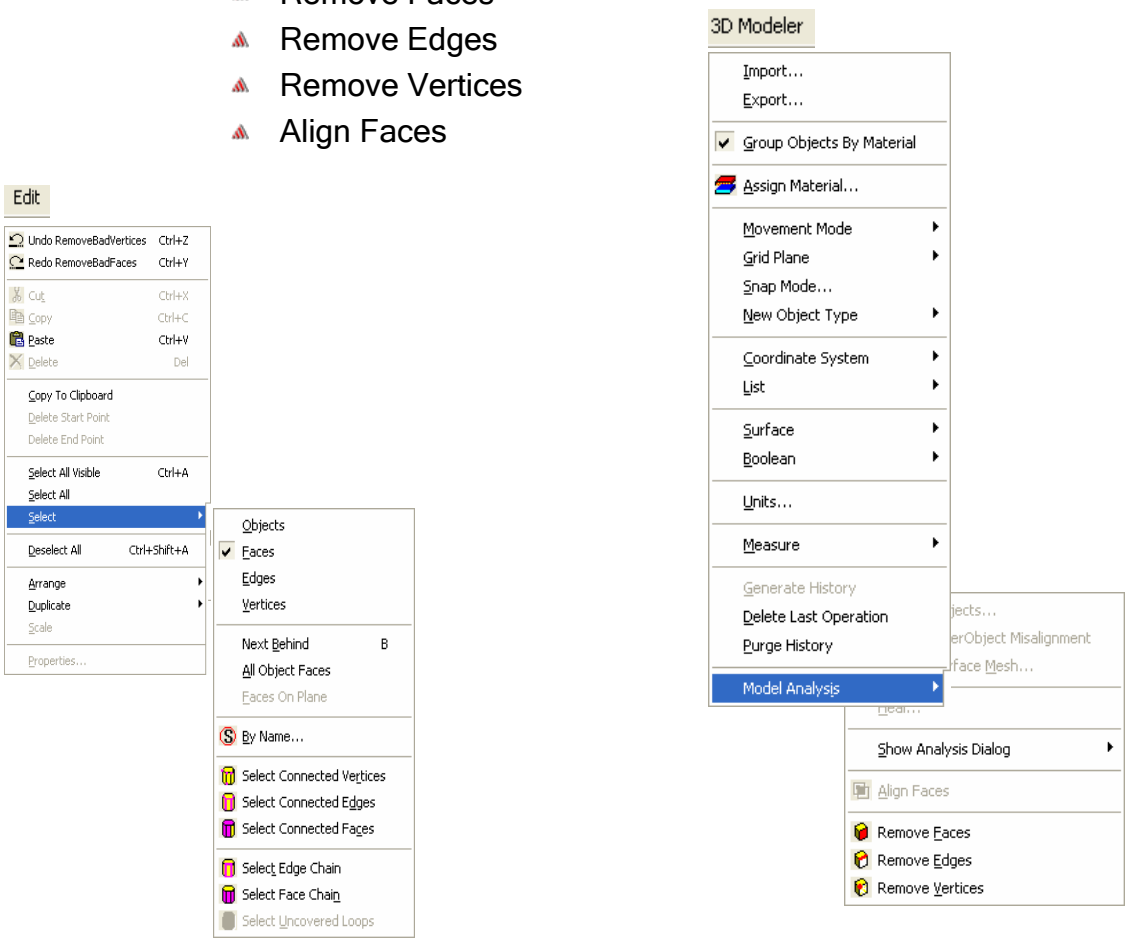
- **Visibility**
  - The visibility of objects, Boundaries, Excitations, and Field Reports can be controlled from the menu item *View > Visibility*
- **Hide Selection**
  - The visibility of selected objects can be set hidden by selecting the object(s) and choosing the menu *View > Hide Selection > All Views*.
- **Rendering**
  - To change the rendering select the menu item View > Render > Wireframe or View > Render > Smooth Shaded
- **Coordinate System**
  - To control the view of the coordinate system, select the menu item:
    - **Visibility:**
      - Toggle the menu item *View > Coordinate System > Hide (Show)*
    - **Size**:
      - Toggle the menu item *View > Coordinate System > Small (Large)*
- **Background Color**
  - To set the background color, select the menu item **View > Modify Attributes > Background Color**
- **Addition View Seetings**
  - Additional attributes of the view such as the projection, orientation, and lighting can be set from the menu item **View > Modify Attributes**

WWW.MOHANDESYAR.COM

Enhancements and New Features

- **Selection**
  - Select **Connected Vertices**
  - Select **Connected Faces**
  - Select **Connected Edges**
  - Select **Edge Chain**
  - Select **Face Chain**
  - Select **Uncovered Loops**

- **Healing**
  - **Purge History** – makes an object appear as an imported entity so that healing can be performed on it
  - Remove Faces
  - Remove Edges
  - Remove Vertices
  - Align Faces



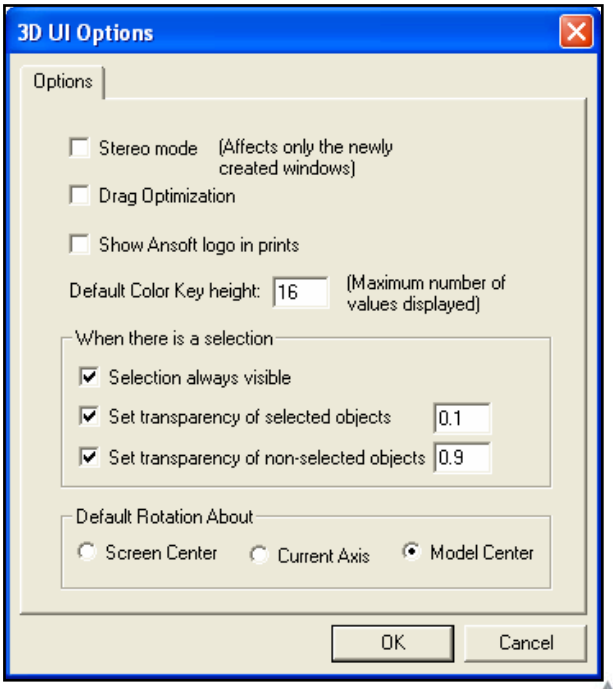
WWW.MOHANDESYAR.COM

Enhancements and New Features

- **Visibility**
  - Hide selected objects in Active View
  - Hide selected objects in All Views
  - Show selected objects in Active View
  - Show selected objects in All Views

- **3D User Interface Options**
  - When there is a selection
    - Selection is always visible
    - Set transparency of selected objects
    - Set transparency of non-selected objects
  - Default Rotation About
    - Screen Center
    - Current Axis
    - Model Center

- **3D Modeler Options**
  - Visualize history of objects



WWW.MOHANDESYAR.COM

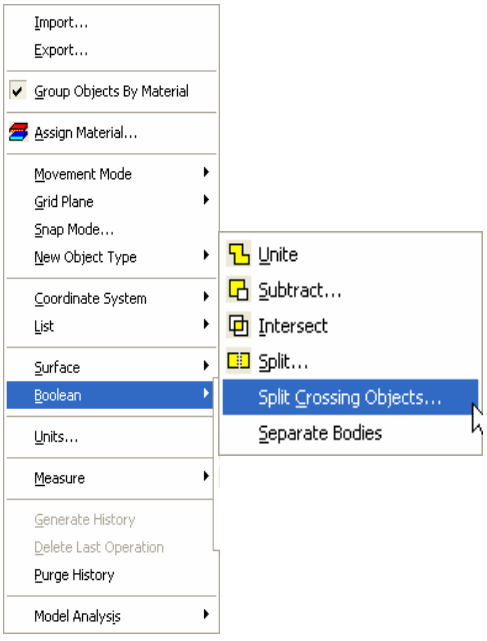
Applying Structural Transformations

- So far we have investigated hot to model simple shapes and how to change the view of the model. To create more complicated models or reduce the number of objects that need to be created manually we can apply various transformations.
- The following examples assume that you have already selected the object(s) that you wish to apply a transformation.
- You can select the transformation options from the menu item *Edit >*
  - *Arrange >*
    - **Move** – Translates the structure along a vector
    - **Rotate** – Rotates the shape around a coordinate axis by an angle
    - **Mirror** – Mirrors the shape around a specified plane
    - **Offset** – Performs a uniform scale in x, y, and z.
  - *Duplicate >*
    - **Along Lines** – Create multiple copies of an object along a vector
    - **Around Axis** – Create multiple copies of an object rotated by a fixed angle around the x, y, or z axis
    - **Mirror** - Mirrors the shape around a specified plane and creates a duplicate
  - *Scale* – Allows non-uniform scaling in the x, y, or z direction
- The faces of an object can also be moved to alter the shape of an existing object. To move the faces of an object select the menu item *3D Modeler > Surfaces > Move Faces* and select *Along Normal* or *Along Vector*.

WWW.MOHANDESYAR.COM

Combine Objects by Using Boolean Operations

- Most complex structures can be reduced to combinations of simple primitives. Even the solid primitives can be reduced to simple 2D primitives that are swept along a vector or around an axis(Box is a square that is swept along a vector to give it thickness). The solid modeler supports the following Boolean operations:
  - **Unite** – combine multiple primitives
    - Unite disjoint objects
      - Separate Bodies to separate
  - **Subtract** – remove part of a primitive from another
  - **Split** – break primitives into multiple parts
  - **Intersect**– keep only the parts of primitives that overlap
  - **Sweep** – turn a 2D primitive into a solid by sweeping: Along a Vector, Around an Axis, Along a Path
  - **Connect** – connect 2D primitives. Use Cover Surfaces to turn the connected object into a solid
  - **Section** – generate 2D cross-sections of a 3D object
- Most Boolean operations require a base primitive in which the Boolean operation is performed. Only the base object will be preserved.
- The Boolean functions provide the option to Clone objects.
- **Split Crossing Objects** – When a group of objects are selected, a Boolean split is performed on ANY objects that overlap

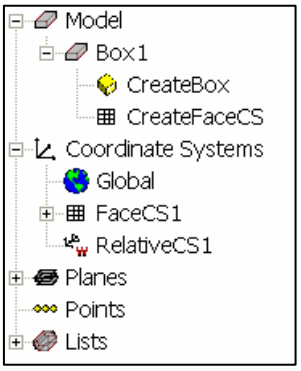


WWW.MOHANDESYAR.COM

Local Coordinate Systems

- The ability to create local coordinate systems adds a great deal of flexibility to the creations of structural objects. In previous sections we have only discussed objects that are aligned to the global coordinate system. The local coordinate system simplifies the definition of objects that do not align with the global coordinate system. In addition, the object history is defined relative to a coordinate system. If the coordinate system is moved, the geometry will automatically move with it. The definition of coordinate systems are maintained in the Model Tree.

- **Working Coordinate System**
  - The working coordinate system is the currently selected CS. This can be a local or global CS



- **Global CS**
  - The default fixed coordinate system

- **Relative CS**
  - User defined local coordinate system.
    - Offset
    - Rotated
    - Both

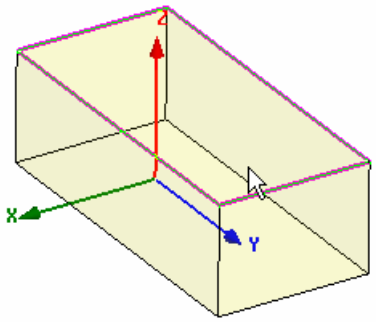
- **Face CS**
  - User defined local coordinate system. It is tied to the location of the object face it was created on. If the size of the base object changes, all objects created relative to the face CS will be updated automatically.
  - Continued on Next Page

WWW.MOHANDESYAR.COM

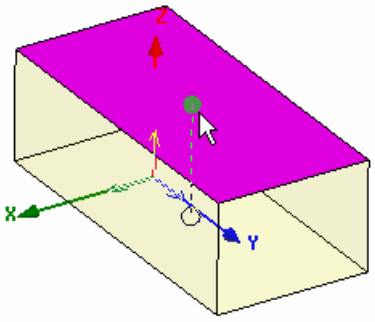
Local Coordinate Systems (Continued)

Face CS (Continued)

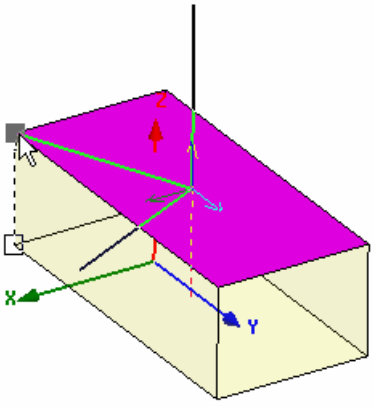
- To create a face CS, select the menu item *3D Modeler > Coordinate System > Face*
  1. Graphically select Face (Highlighted in model)
  2. Select Origin for Face CS
  3. Set X-Axis



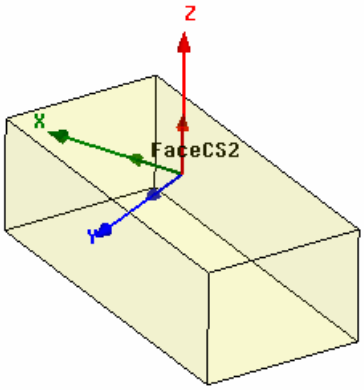
Step 1: Select Face



Step 2: Select Origin



Step 3: Set X-Axis

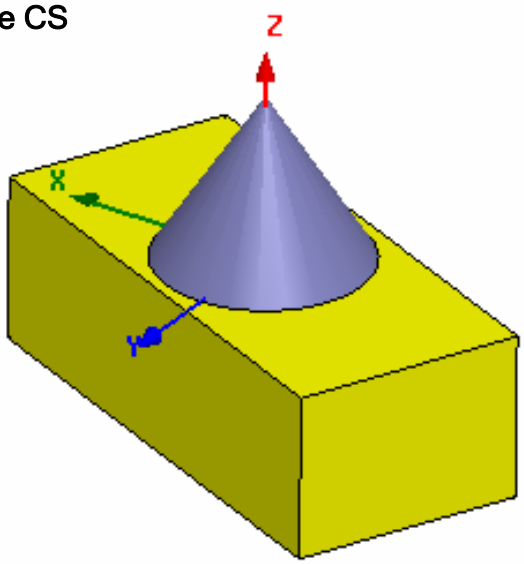


New Working CS

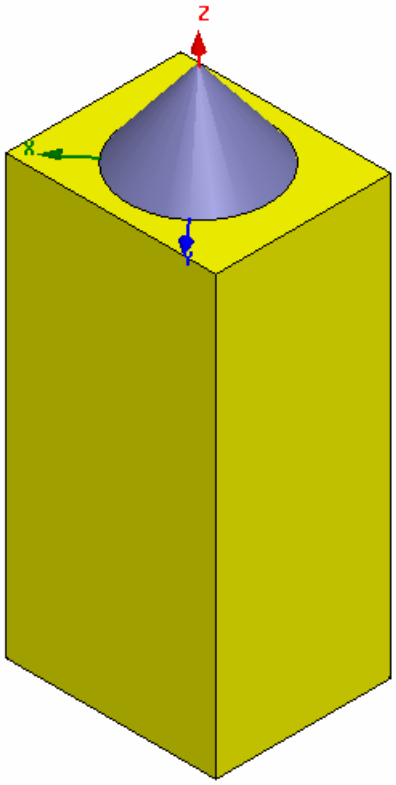
WWW.MOHANDESYAR.COM

Local Coordinate Systems (Continued)

Example of Face CS



Cone is created with Face CS



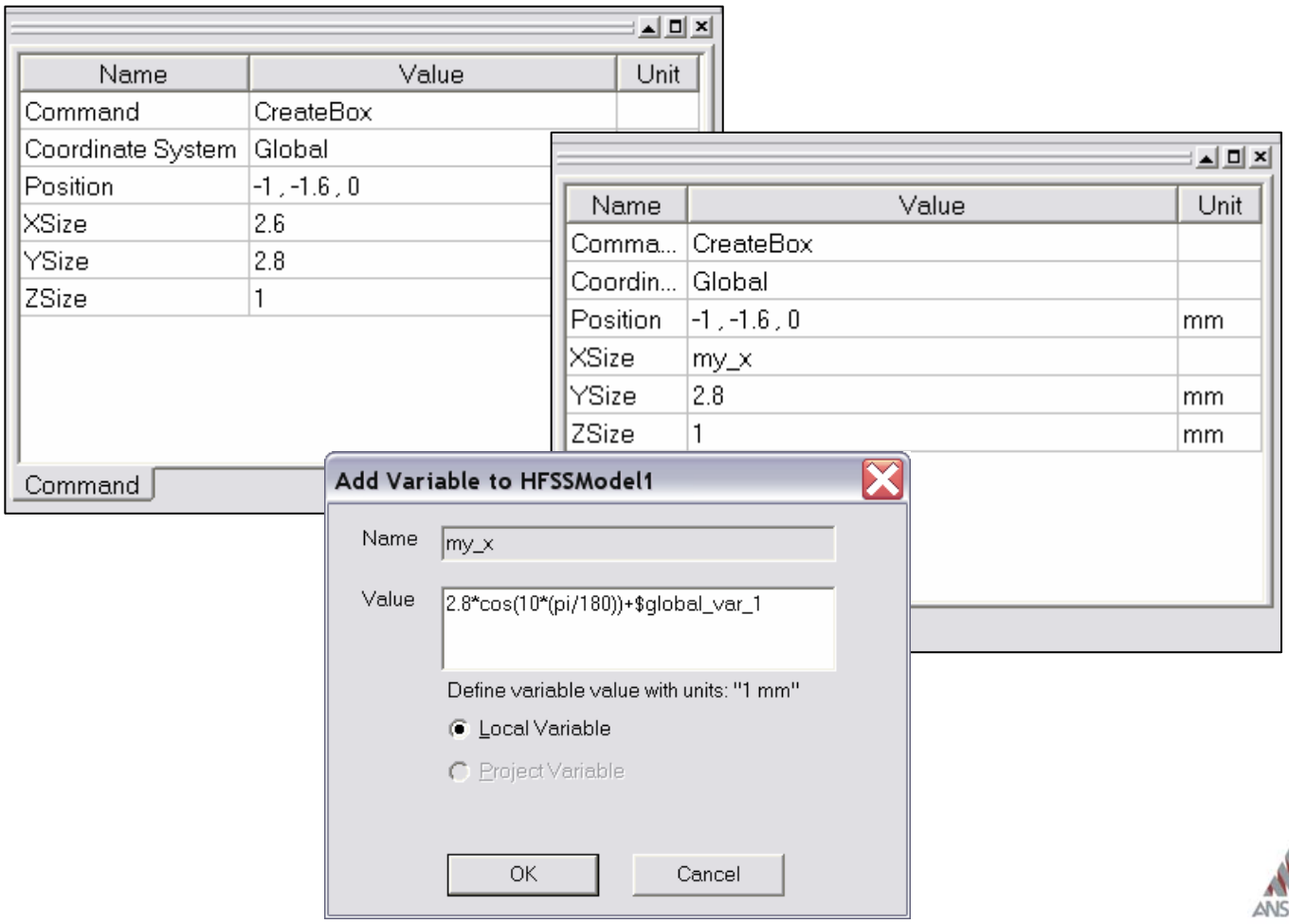
Change the size of the box and the Cone is automatically moved with the Face CS

WWW.MOHANDESYAR.COM

Parametric Geometry

- The parametric modeler capability allows us to define variables in replace of a fixed position or size. Once this has been defined the variable can be changed by the user or by Optimetrics. Optimetrics can then be used to perform automatic Optimization, Parametric Sweeps, Statistical, or Sensitivity Analysis.

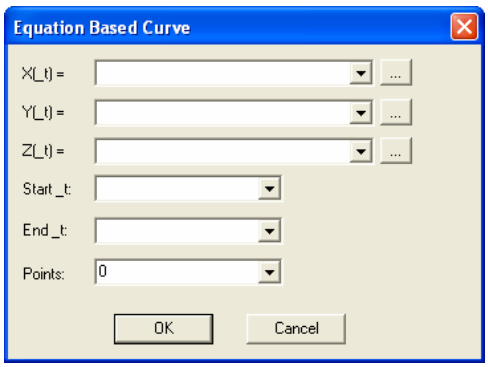
- **Defining Parameters**
  - Select the command to parameterized
  - Choose the value to change
  - Enter a variable in replace of the fixed value
  - Define the variable using any combination of math functions or design variables.
  - The model will automatically be updated

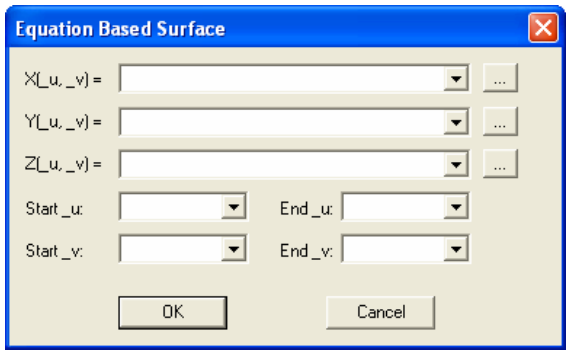


WWW.MOHANDESYAR.COM

Parametric Geometry (Continued)

- **Variables**
  - There are two types of variables that can be defined in the Maxwell Desktop
    - **Design Properties** – Local to model. To access the local variables select the menu item *Maxwell > Design Properties*
    - **Project Variables** – Global to all models in project. Start with $. To access the global or project variables, select the menu item *Project > Project Variables*
  - **Units**
    - When defining variables they must contain units. The default units for variables is meters.
  - **Equations**
    - The variables can contain complex equations. See the Online Help for a complete list of math functions
  - **Equation based Curves and Surfaces**
  - Any curve/surface that can be described by an equation in three dimensions can be drawn.





  - **Animation**
    - Right-Click in the 3D Model Window & Choose Animate to preview the parameterization
    - Note: depending on the quality of your graphics card you have the option of exporting ether AVI or GIF animation files.

WWW.MOHANDESYAR.COM

Boundary Conditions

- This chapter describes the basics for applying boundary conditions. Boundary conditions enable you to control the characteristics of planes, faces, or interfaces between objects. Boundary conditions are important to understand and are fundamental to solution of Maxwell's equations.

Why they are Important

- The field equations that are solved by Ansoft Maxwell 3D are derived from the differential form of Maxwell's Equations. For these expressions to be valid, it is assumed that the field vectors are single-valued, bounded, and have continuous distribution along with their derivatives. Along boundaries or sources, the fields are discontinuous and the derivatives have no meaning. Therefore boundary conditions define the field behavior across discontinuous boundaries.
- As a user of Ansoft Maxwell 3D you should be aware of the field assumptions made by boundary conditions. Since boundary conditions force a field behavior we want to be aware of the assumptions so we can determine if they are appropriate for the simulation. Improper use of boundary conditions may lead to inconsistent results.
- When used properly, boundary conditions can be successfully utilized to reduce the model complexity. In fact, Ansoft Maxwell 3D automatically uses boundary conditions to reduce the complexity of the model. Ansoft Maxwell 3D can be thought of as a virtual prototyping world. Unlike the real world which is bounded by infinite space, the virtual prototyping world needs to be made finite. In order to achieve this finite space, Ansoft Maxwell 3D applies a background or outer boundary condition which is applied to the region surrounding the geometric model.
- The model complexity usually is directly tied to the solution time and computer resources so it is a competitive advantage to utilize them whenever possible.

WWW.MOHANDESYAR.COM

Common Boundary Conditions

- There are three types of boundary conditions. The first two are largely the users responsibility to define them or ensure that they are defined correctly. The material boundary conditions are transparent to the user.

1. **Excitations**
   - Wave Ports (External)
   - Lumped Ports (Internal)
2. **Surface Approximations**
   - Symmetry Planes
   - Perfect Electric or Magnetic Surfaces
   - Radiation Surfaces
   - Background or Outer Surface
3. **Material Properties**
   - Boundary between two dielectrics
   - Finite Conductivity of a conductor

WWW.MOHANDESYAR.COM

How the Background Affects a Structure

- The background is the region that surrounds the geometric model and fills any space that is not occupied by an object. Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary name *outer*. You can think of your structure as being encased with a thin, perfect conductor.

- If it is necessary, you can change a surface that is exposed to the background to have properties that are different from *outer*:
  - To model losses in a surface, you can redefine the surface to be either a Finite Conductivity or Impedance boundary. A Finite Conductivity boundary can be a lossy metal, with loss as a function of frequency and defined using conductivity and relative permeability parameters. An Impedance boundary has real or complex values that by default remain constant over frequency.
  - To model a surface to allow waves to radiate infinitely far into space, redefine the surface to be radiation boundary.

- The background can affect how you make material assignments. For example, if you are modeling a simple air-filled rectangular waveguide, you can create a single object in the shape of the waveguide and define it to have the characteristics of air. The surface of the waveguide is automatically assumed to be a perfect conductor and given the boundary condition *outer*, or you can change it to a lossy conductor.

WWW.MOHANDESYAR.COM

Boundary Condition Precedence

- The order in which boundaries are assigned is important in Maxwell 3D. Latter assigned boundaries take precedence over former assigned boundaries.
- For example, if one face of an object is assigned to a Natural boundary, and a boundary in the same plane as this surface is assigned a Tangential H-Field boundary, then the Tangential H-Field will override the Natural in the area of the overlap. If this operation were performed in the reverse order, then the Tangential H-Field boundary would cover the Natural boundary.
- Once boundaries have been assigned, they can be re-prioritized by selecting *Maxwell > Boundaries > Re-prioritize*. The order of the boundaries can be changed by clicking on a boundary and dragging it further up or down in the list. NOTE: Excitations will always take the highest precedence.

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions

- **Excitation** – An excitation is a type of boundary condition that permits energy to flow into and out of a structure. See the section on Excitations.

- **Perfect E** – Perfect E is a perfect electrical conductor, also referred to as a perfect conductor. This type of boundary forces the electric field (E-Field) perpendicular to the surface. There are also two automatic Perfect E assignments:
  - Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary condition name *outer*.
  - Any object that is assigned the material pec (Perfect Electric Conductor) is automatically assigned the boundary condition Perfect E to its surface and given the boundary condition name *smetal.*

- **Perfect H** – Perfect H is a perfect magnetic conductor. Forces E-Field tangential to the surface.
  - **Natural** – for a Perfect H boundary that overlaps with a perfect E boundary, this reverts the selected area to its original material, erasing the Perfect E boundary condition. It does not affect any material assignments. It can be used, for example, to model a cut-out in a ground plane for a coax feed.

- **Finite Conductivity** –A Finite Conductivity boundary enables you to define the surface of an object as a lossy (imperfect) conductor. It is an imperfect E boundary condition, and is analogous to the lossy metal material definition. To model a lossy surface, you provide loss in Siemens/meter and permeability parameters. Loss is calculated as a function of frequency. It is only valid for good conductors. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance (Z_s) is equal to, $(1+j)/(\delta\sigma)$, where:
  - δ is the skin depth, $(2/(\omega\sigma\mu))^{0.5}$ of the conductor being modeled
  - ω is the frequency of the excitation wave.
  - σ is the conductivity of the conductor
  - μ is the permeability of the conductor

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions (Continued)

- **Impedance** – a resistive surface that calculates the field behavior and losses using analytical formulas. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance is equal to $R_s + jX_s$, where:
  - Rs is the resistance in ohms/square
  - Xs is the reactance in ohms/square

- **Layered Impedance** – Multiple thin layers in a structure can be modeled as an impedance surface. See the Online Help for additional information on how to use the Layered Impedance boundary.

- **Lumped RLC** – a parallel combination of lumped resistor, inductor, and/or capacitor surface. The simulation is similar to the **Impedance** boundary, but the software calculate the ohms/square using the user supplied R, L, C values.

- **Infinite Ground Plane** – Generally, the ground plane is treated as an infinite, Perfect E, Finite Conductivity, or Impedance boundary condition. If radiation boundaries are used in a structure, the ground plane acts as a shield for far-field energy, preventing waves from propagating past the ground plane. to simulate the effect of an infinite ground plane, check the Infinite ground plane box when defining a Perfect E, Finite Conductivity, or Impedance boundary condition. NOTE: Enabling the Infinite Ground Plane approximation ONLY affects post-processed far-field radiation patterns. It will not change the current flowing on the ground plane.

- **Radiation** – Radiation boundaries, also referred to as absorbing boundaries, enable you to model a surface as electrically open: waves can then radiate out of the structure and toward the radiation boundary. The system absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure and into space. Radiation boundaries may also be placed relatively close to a structure and can be arbitrarily shaped. This condition eliminates the need for a spherical boundary. For structures that include radiation boundaries, calculated S-parameters include the effects of radiation loss. When a radiation boundary is included in a structure, far-field calculations are performed as part of the simulation.

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions (Continued)

- **Symmetry** - represent perfect E or perfect H planes of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Symmetry boundaries, as opposed to a simple Perfect E or H plane, should be used when the plane cuts across a port. In this instance, the port has a different amount of power, voltage, and current associated with it, and thus a different impedance. To make a port with a symmetry plane look like a full-sized port, you must use the Impedance Multiplier in the boundary wizard.
  - For a single Symmetry H boundary, the Impedance Multiplier is 0.5.
  - For a single Symmetry E boundary, the Impedance Multiplier is 2.
  - Other considerations for a Symmetry boundary condition:
    - A plane of symmetry must be exposed to the background.
    - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
    - A plane of symmetry must be defined on a planar surface.
    - Only three orthogonal symmetry planes can be defined in a problem

- **Master / Slave** - Master and slave boundaries enable you to model planes of periodicity where the E-field on one surface matches the E-field on another to within a phase difference. They force the E-field at each point on the slave boundary match the E-field to within a phase difference at each corresponding point on the master boundary. They are useful for simulating devices such as infinite arrays. Some considerations for Master/Slave boundaries:
  - They can only be assigned to planar surfaces.
  - The geometry of the surface on one boundary must match the geometry on the surface of the other boundary.

WWW.MOHANDESYAR.COM

Boundary Conditions

- This chapter describes the basics for applying boundary conditions. Boundary conditions enable you to control the characteristics of planes, faces, or interfaces between objects. Boundary conditions are important to understand and are fundamental to solution of Maxwell's equations.

Why they are Important

- The field equations that are solved by Ansoft Maxwell 3D are derived from the differential form of Maxwell's Equations. For these expressions to be valid, it is assumed that the field vectors are single-valued, bounded, and have continuous distribution along with their derivatives. Along boundaries or sources, the fields are discontinuous and the derivatives have no meaning. Therefore boundary conditions define the field behavior across discontinuous boundaries.
- As a user of Ansoft Maxwell 3D you should be aware of the field assumptions made by boundary conditions. Since boundary conditions force a field behavior we want to be aware of the assumptions so we can determine if they are appropriate for the simulation. Improper use of boundary conditions may lead to inconsistent results.
- When used properly, boundary conditions can be successfully utilized to reduce the model complexity. In fact, Ansoft Maxwell 3D automatically uses boundary conditions to reduce the complexity of the model. Ansoft Maxwell 3D can be thought of as a virtual prototyping world. Unlike the real world which is bounded by infinite space, the virtual prototyping world needs to be made finite. In order to achieve this finite space, Ansoft Maxwell 3D applies a background or outer boundary condition which is applied to the region surrounding the geometric model.
- The model complexity usually is directly tied to the solution time and computer resources so it is a competitive advantage to utilize them whenever possible.

WWW.MOHANDESYAR.COM

Common Boundary Conditions

There are three types of boundary conditions. The first two are largely the users responsibility to define them or ensure that they are defined correctly. The material boundary conditions are transparent to the user.

1. **Excitations**
   - Wave Ports (External)
   - Lumped Ports (Internal)
2. **Surface Approximations**
   - Symmetry Planes
   - Perfect Electric or Magnetic Surfaces
   - Radiation Surfaces
   - Background or Outer Surface
3. **Material Properties**
   - Boundary between two dielectrics
   - Finite Conductivity of a conductor

WWW.MOHANDESYAR.COM

How the Background Affects a Structure

- The background is the region that surrounds the geometric model and fills any space that is not occupied by an object. Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary name *outer*. You can think of your structure as being encased with a thin, perfect conductor.

- If it is necessary, you can change a surface that is exposed to the background to have properties that are different from *outer*:
  - To model losses in a surface, you can redefine the surface to be either a Finite Conductivity or Impedance boundary. A Finite Conductivity boundary can be a lossy metal, with loss as a function of frequency and defined using conductivity and relative permeability parameters. An Impedance boundary has real or complex values that by default remain constant over frequency.
  - To model a surface to allow waves to radiate infinitely far into space, redefine the surface to be radiation boundary.

- The background can affect how you make material assignments. For example, if you are modeling a simple air-filled rectangular waveguide, you can create a single object in the shape of the waveguide and define it to have the characteristics of air. The surface of the waveguide is automatically assumed to be a perfect conductor and given the boundary condition *outer*, or you can change it to a lossy conductor.

WWW.MOHANDESYAR.COM

Boundary Condition Precedence

- The order in which boundaries are assigned is important in Maxwell 3D. Latter assigned boundaries take precedence over former assigned boundaries.
- For example, if one face of an object is assigned to a Natural boundary, and a boundary in the same plane as this surface is assigned a Tangential H-Field boundary, then the Tangential H-Field will override the Natural in the area of the overlap. If this operation were performed in the reverse order, then the Tangential H-Field boundary would cover the Natural boundary.
- Once boundaries have been assigned, they can be re-prioritized by selecting *Maxwell > Boundaries > Re-prioritize*. The order of the boundaries can be changed by clicking on a boundary and dragging it further up or down in the list. NOTE: Excitations will always take the highest precedence.

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions

- **Excitation** – An excitation is a type of boundary condition that permits energy to flow into and out of a structure. See the section on Excitations.

- **Perfect E** – Perfect E is a perfect electrical conductor, also referred to as a perfect conductor. This type of boundary forces the electric field (E-Field) perpendicular to the surface. There are also two automatic Perfect E assignments:
  - Any object surface that touches the background is automatically defined to be a Perfect E boundary and given the boundary condition name *outer*.
  - Any object that is assigned the material pec (Perfect Electric Conductor) is automatically assigned the boundary condition Perfect E to its surface and given the boundary condition name *smetal.*

- **Perfect H** – Perfect H is a perfect magnetic conductor. Forces E-Field tangential to the surface.
  - **Natural** – for a Perfect H boundary that overlaps with a perfect E boundary, this reverts the selected area to its original material, erasing the Perfect E boundary condition. It does not affect any material assignments. It can be used, for example, to model a cut-out in a ground plane for a coax feed.

- **Finite Conductivity** –A Finite Conductivity boundary enables you to define the surface of an object as a lossy (imperfect) conductor. It is an imperfect E boundary condition, and is analogous to the lossy metal material definition. To model a lossy surface, you provide loss in Siemens/meter and permeability parameters. Loss is calculated as a function of frequency. It is only valid for good conductors. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance (Z_s) is equal to, $(1+j)/(\delta\sigma)$, where:
  - δ is the skin depth, $(2/(\omega\sigma\mu))^{0.5}$ of the conductor being modeled
  - ω is the frequency of the excitation wave.
  - σ is the conductivity of the conductor
  - μ is the permeability of the conductor

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions (Continued)

- **Impedance** – a resistive surface that calculates the field behavior and losses using analytical formulas. Forces the tangential E-Field equal to $Z_s(n \times H_{tan})$. The surface impedance is equal to $R_s + jX_s$, where:
  - Rs is the resistance in ohms/square
  - Xs is the reactance in ohms/square

- **Layered Impedance** – Multiple thin layers in a structure can be modeled as an impedance surface. See the Online Help for additional information on how to use the Layered Impedance boundary.

- **Lumped RLC** – a parallel combination of lumped resistor, inductor, and/or capacitor surface. The simulation is similar to the **Impedance** boundary, but the software calculate the ohms/square using the user supplied R, L, C values.

- **Infinite Ground Plane** – Generally, the ground plane is treated as an infinite, Perfect E, Finite Conductivity, or Impedance boundary condition. If radiation boundaries are used in a structure, the ground plane acts as a shield for far-field energy, preventing waves from propagating past the ground plane. to simulate the effect of an infinite ground plane, check the Infinite ground plane box when defining a Perfect E, Finite Conductivity, or Impedance boundary condition. NOTE: Enabling the Infinite Ground Plane approximation ONLY affects post-processed far-field radiation patterns. It will not change the current flowing on the ground plane.

- **Radiation** – Radiation boundaries, also referred to as absorbing boundaries, enable you to model a surface as electrically open: waves can then radiate out of the structure and toward the radiation boundary. The system absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure and into space. Radiation boundaries may also be placed relatively close to a structure and can be arbitrarily shaped. This condition eliminates the need for a spherical boundary. For structures that include radiation boundaries, calculated S-parameters include the effects of radiation loss. When a radiation boundary is included in a structure, far-field calculations are performed as part of the simulation.

WWW.MOHANDESYAR.COM

Technical Definition of Boundary Conditions (Continued)

- **Symmetry** - represent perfect E or perfect H planes of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Symmetry boundaries, as opposed to a simple Perfect E or H plane, should be used when the plane cuts across a port. In this instance, the port has a different amount of power, voltage, and current associated with it, and thus a different impedance. To make a port with a symmetry plane look like a full-sized port, you must use the Impedance Multiplier in the boundary wizard.
  - For a single Symmetry H boundary, the Impedance Multiplier is 0.5.
  - For a single Symmetry E boundary, the Impedance Multiplier is 2.
  - Other considerations for a Symmetry boundary condition:
    - A plane of symmetry must be exposed to the background.
    - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
    - A plane of symmetry must be defined on a planar surface.
    - Only three orthogonal symmetry planes can be defined in a problem

- **Master / Slave** - Master and slave boundaries enable you to model planes of periodicity where the E-field on one surface matches the E-field on another to within a phase difference. They force the E-field at each point on the slave boundary match the E-field to within a phase difference at each corresponding point on the master boundary. They are useful for simulating devices such as infinite arrays. Some considerations for Master/Slave boundaries:
  - They can only be assigned to planar surfaces.
  - The geometry of the surface on one boundary must match the geometry on the surface of the other boundary.

WWW.MOHANDESYAR.COM

Examples - Contents

- Chapter 5.0 - Magnetostatic Examples
  - 5.1 - Magnetic Force
  - 5.2 - Nonlinear Inductor
  - 5.3 - Switched Reluctance Motor (Stranded Conductors)
  - 5.4 - Equivalent Circuit Extraction (ECE) Linear Movement
  - 5.5 - Anisotropic Materials
- Chapter 6.0 - Eddy Current Examples
  - 6.1 - Asymmetrical Conductor with a Hole
- Chapter 7.0 - Transient Examples
  - 7.1 - Switched Reluctance Motor (Stranded Conductors)

WWW.MOHANDESYAR.COM

Chapter 5.0 - Magnetostatic Examples

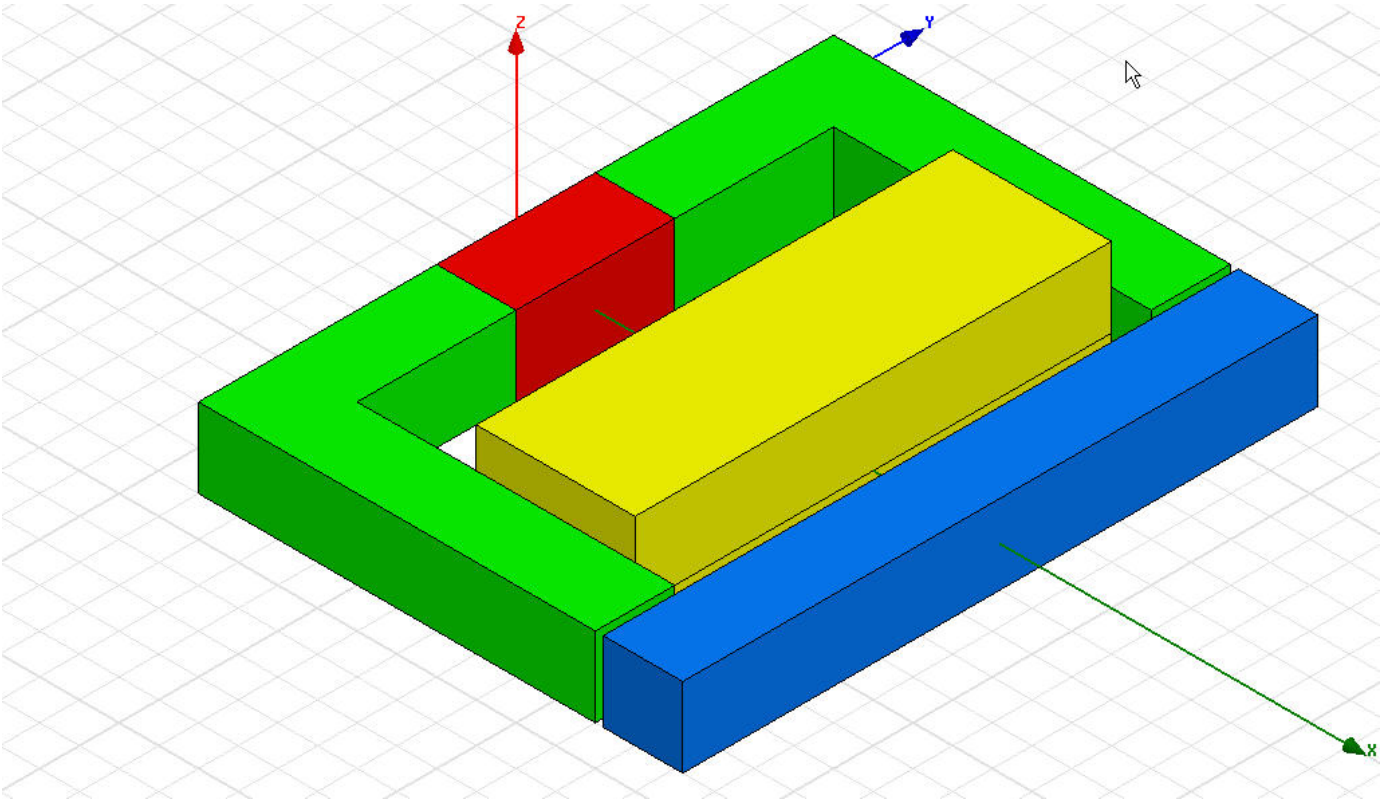
- 5.1 - Magnetic Force
- 5.2 - Nonlinear Inductor
- 5.3 - Switched Reluctance Motor (Stranded Conductors)
- 5.4 - Equivalent Circuit Extraction (ECE) Linear Movement
- 5.5 - Anisotropic Materials

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Magnetic Force

This example is intended to show you how to create and analyze a magnetostatic problem with a permanent magnet to determine the force exerted on a steel bar using the Magnetostatic solver in the Ansoft Maxwell 3D Design Environment.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Box**
    - Surface Operations: **Section**
    - Boolean Operations: **Subtract, Unite, Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Force**
  - Field Overlays:
    - **Vector B**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

Setting Tool Options

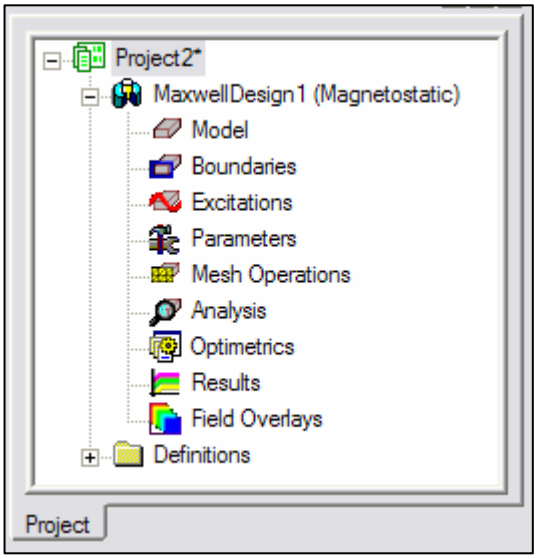
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

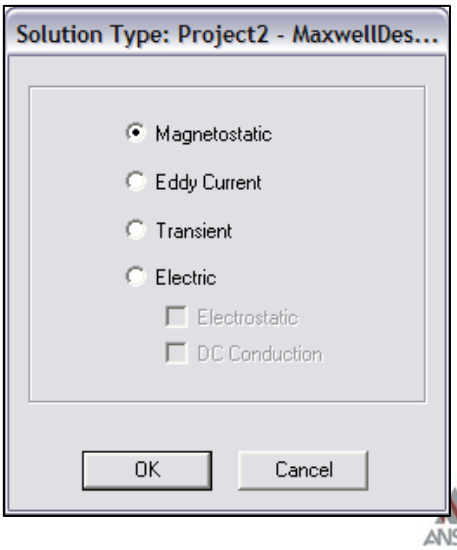
Opening a New Project

- **To open a new project:**
  - In an Ansoft Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - From the *Project* menu, select *Insert Maxwell Design*.



Set Solution Type

- **To set the solution type:**
  - Select the menu item *Maxwell > Solution Type*
  - Solution Type Window:
    - Choose **Magnetostatic**
    - Click the **OK** button



WWW.MOHANDESYAR.COM

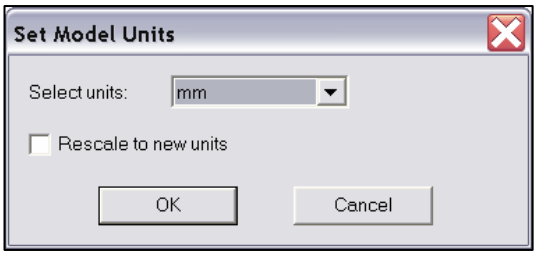
Example (Magnetostatic) - Magnetic Force

Creating the 3D Model

Set Model Units

To set the units:

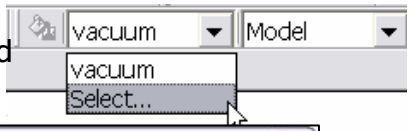
1. Select the menu item *3D Modeler > Units*
2. Set Model Units:
   1. Select Units: **mm**
   2. Click the **OK** button

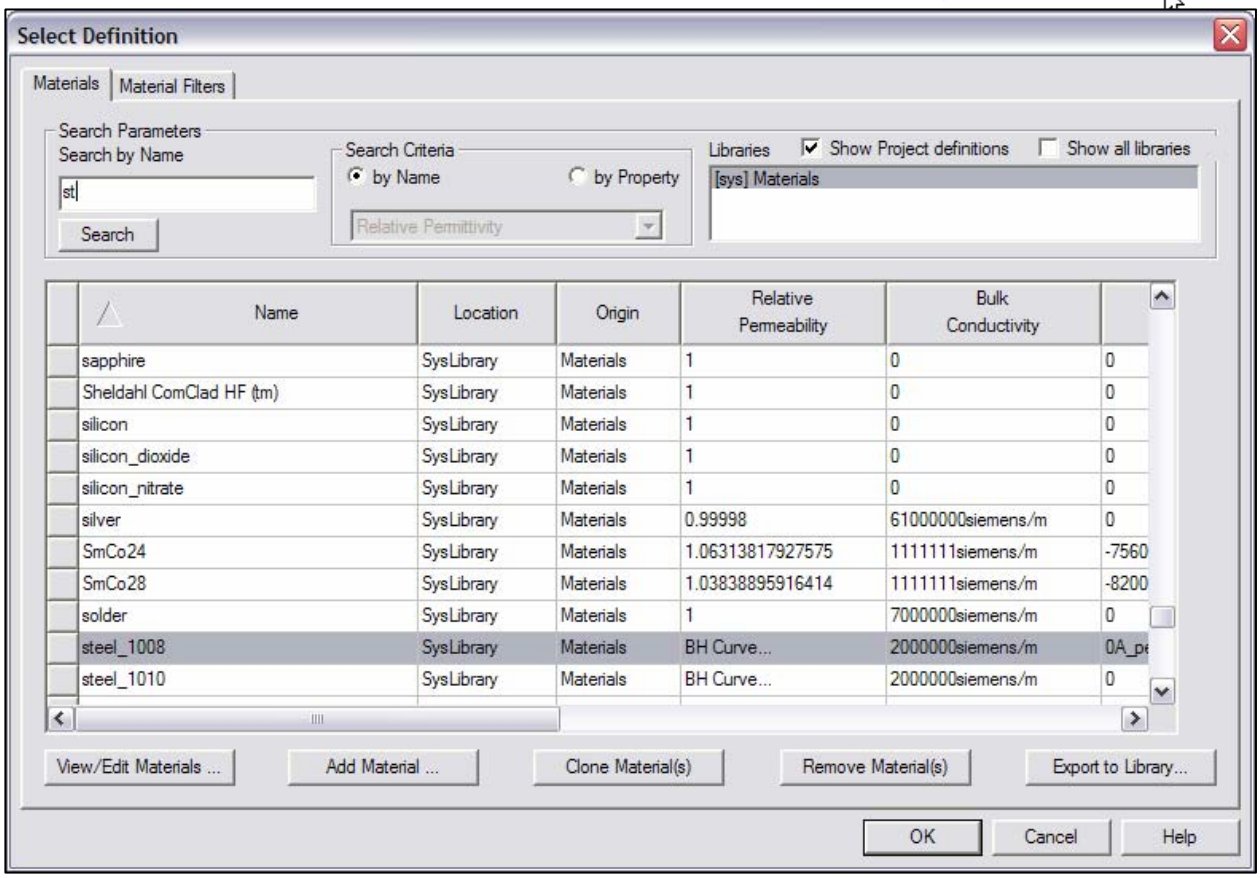


Set Default Material

To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **steel_1008** in the **Search by Name** field
   2. Click the **OK** button



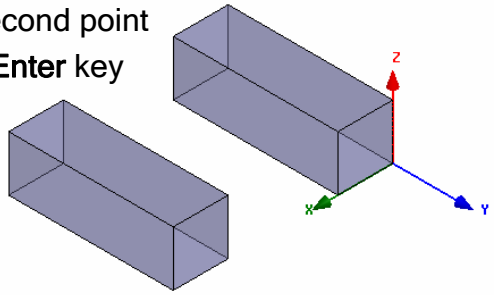


WWW.MOHANDESYAR.COM

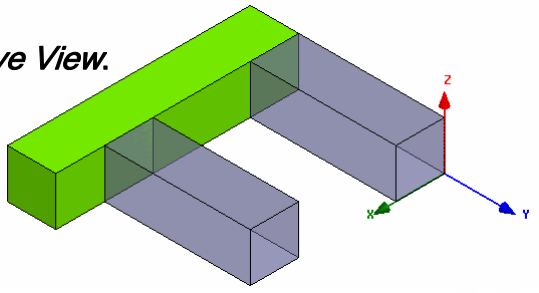
Example (Magnetostatic) - Magnetic Force

Create Core

- **To create a box:**
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **0.0**, Y: **0.0**, Z: **-5.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **10.0**, dY: **-30.0**, dZ: **10.0**, Press the **Enter** key
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.
- **Duplicate Box:**
  1. Select the menu item *Edit > Duplicate Along Line*
  2. Using the coordinate entry fields, enter the first point
     - X: **0.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the second point
     - dX: **30.0**, dY: **0.0**, dZ: **10.0**, Press the **Enter** key
  4. Duplicate Along Line Window
     1. Total Number: **2**
     2. Click the **OK** button



- **To create the core:**
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **0.0**, Y: **-30.0**, Z: **-5.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **50.0**, dY: **-10.0**, dZ: **10.0**, Press the **Enter** key
- **To set the name:**
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Core**
  3. Click the **OK** button
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Group the Core

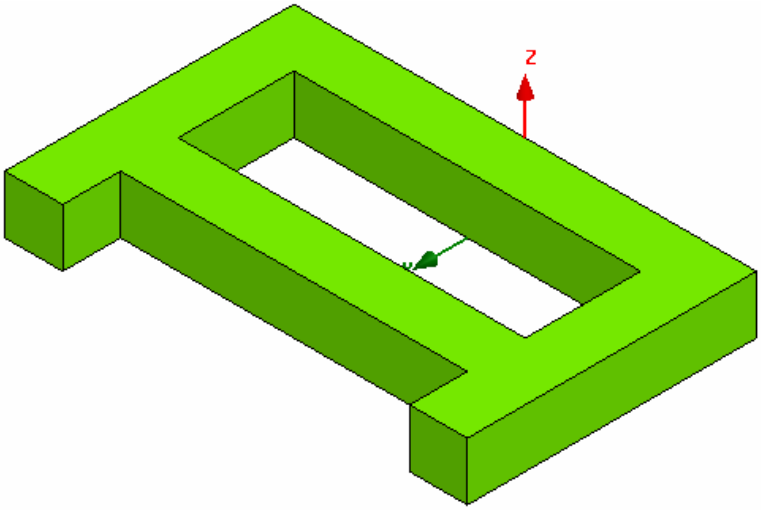
- **To select the objects**
  1. Select the menu item *Edit > Select All*
  2. Select the menu item, *3D Modeler > Boolean > Unite*
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.

Duplicate the Core

- **To select the objects**
  1. Select the menu item, *Edit > Duplicate Mirror*
  2. Using the coordinate entry fields, enter the first point
     - X: **0.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the normal point
     - dX: **0.0**, dY: **1.0**, dZ: **0.0**, Press the **Enter** key
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.

Group the Core

- **To select the objects**
  1. Select the menu item *Edit > Select All*
  2. Select the menu item, *3D Modeler > Boolean > Unite*
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Create Bar

To create the bar:

1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **51.0**, Y: **-40.0**, Z: **-5.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **10.0**, dY: **80.0**, dZ: **10.0**, Press the **Enter** key

To parameterize the object:

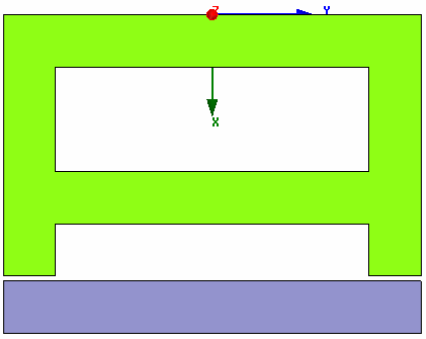
1. Select the **Command** tab from the **Properties** window
2. For **Position**, type: **50mm+mx, -40,0, -5,0**, Click the **Tab** key to accept
   - Add Variable **mx**: **1mm**, Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Bar**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View*.



Set Default Material

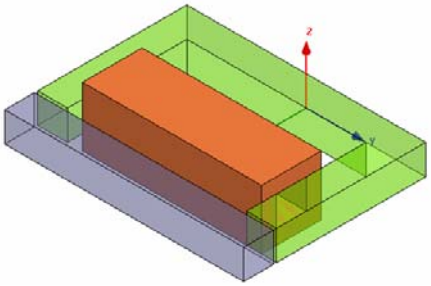
1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **copper** in the **Search by Name** field
   2. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Create Coil

- **To create the coil:**
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **45.0**, Y: **30.0**, Z: **10.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **-20.0**, dY: **-60.0**, dZ: **-20.0**, Press the **Enter** key
- **To set the name:**
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **coil**
  3. Click the **OK** button
- **To select the object for subtract**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Coil, Core**
     2. Click the **OK** button
- **To complete the coil:**
  1. Select the menu item *3D Modeler > Boolean > Subtract*
  2. Subtract Window
     1. Blank Parts: **Coil**
     2. Tool Parts: **Core**
     3. Clone tool objects before subtracting: ☑ **Checked**
     4. Click the **OK** button
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View.*



Insulate Coil

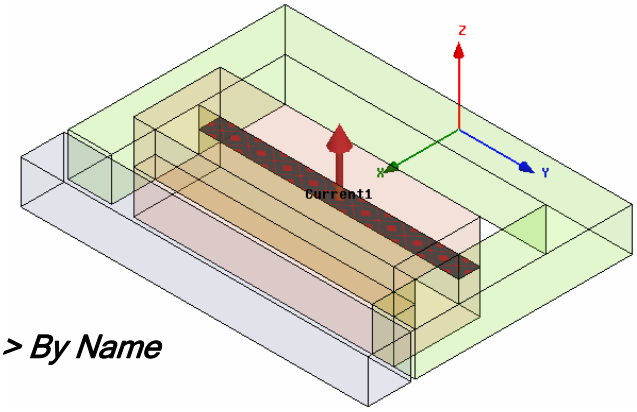
- **To assign a boundary**
  1. Select the menu item *Maxwell > Boundaries > Assign > Insulating*
  2. Insulating Boundary
     1. Name: **Insulating1**
     2. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Create Excitation

- **Section Object**
  1. Select the menu item *Edit > Surface > Section*
     1. Section Plane: **XY**
     2. Click the **OK** button
- **Separate Bodies**
  1. Select the menu item *Edit > Boolean > Separate Bodies*
- Assign Excitation
  1. Select the menu item *Maxwell > Excitations > Assign > Current*
  2. Current Excitation : General
     1. Name: **Current1**
     2. Value: **c1**
     3. Type: **Stranded**
     4. Current Direction: **positive Z direction**. (Use **Swap Direction** button)
  3. Click the **OK** button
  4. Add Variable Window
     1. Value: **100A**
     2. Click the **OK** button



Calculate Force

- **To select the object**
  - Select the menu item *Edit > Select > By Name*
  - Select Object Dialog,
    - Select the objects named: **Bar**
    - Click the **OK** button
- **Calculate Force**
  - Select the menu item *Maxwell > Parameters > Assign > Force*
  - Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Set Default Material

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **NdFe35** in the **Search by Name** field
   2. Click the **OK** button

Create Magnet

To create the magnet:

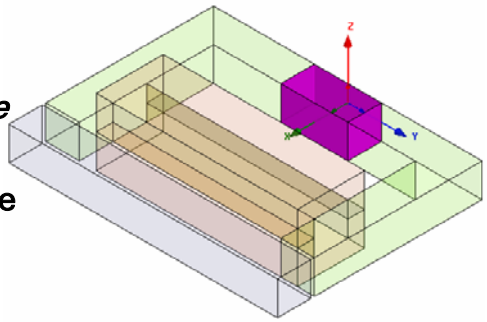
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **0.0**, Y: **-10.0**, Z: **-5.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **10.0**, dY: **20.0**, dZ: **10.0**, Press the **Enter** key

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Magnet**
3. Click the **OK** button

To select the object for subtract

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
   1. Select the objects named: **Magnet, Core**
   2. Click the **OK** button



To complete the magnet:

1. Select the menu item *3D Modeler > Boolean > Subtract*
2. Subtract Window
   1. Blank Parts: **Core**
   2. Tool Parts: **Magnet**
   3. Clone tool objects before subtracting: ☑ **Checked**
   4. Click the **OK** button

To fit the view:

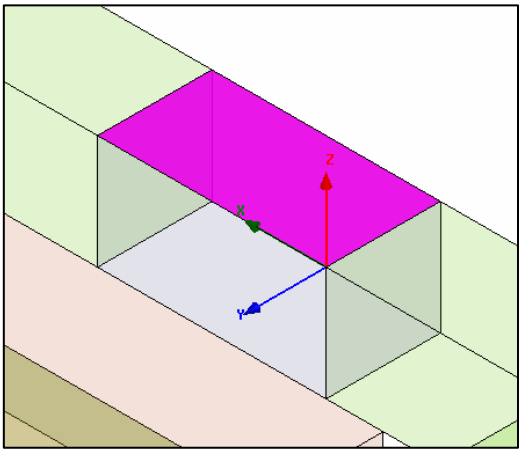
1. Select the menu item *View > Fit All > Active View*.

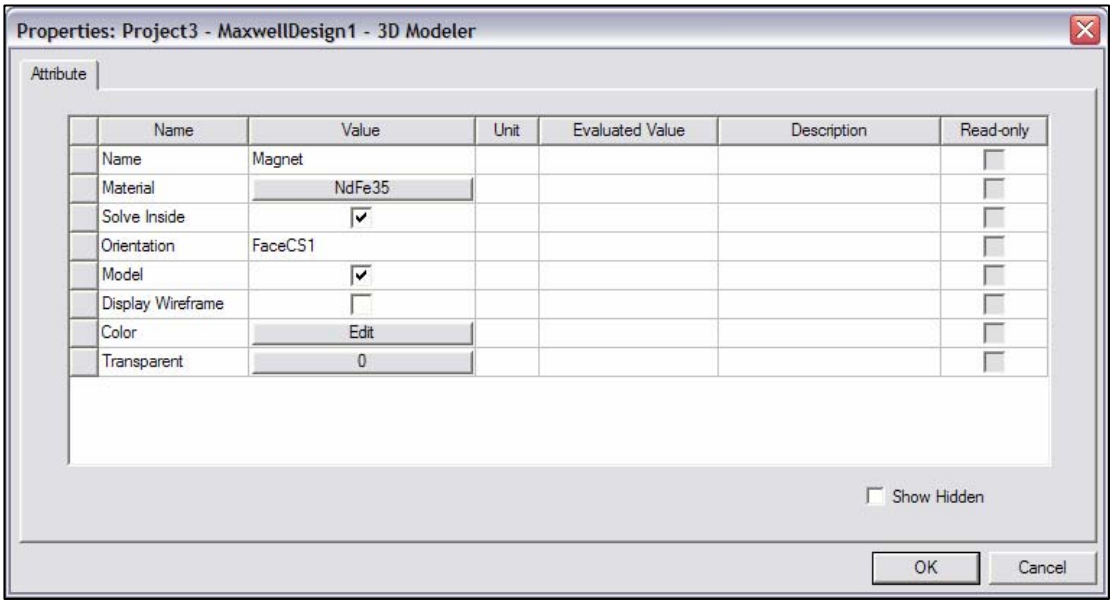
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Orient Magnet

- **Note:** By default all of the magentic material in the material libaray are oriented in the x-direction. Using a local coordinate system (CS) we can correct the orientation of the geometry to align with the material definition.
- **To create rotated CS:**
  1. Select the menu item *Edit > Select > Faces*
  2. Using the mouse, graphically select the top face of the Magnet
  3. Select the menu item 3D Modeler > Coordinate System > Create > Face CS
  4. Using the coordinate entry fields, enter the origin
     - X: **10.0**, Y: **10.0**, Z: **5.0**, Press the **Enter** key
  5. Using the coordinate entry fields, enter the axis:
     - dX: **0.0**, dY: **-20.0**, dZ: **0.0**, Press the **Enter** key
- **Change Properties**
  1. Select the menu item *Maxwell > List*
  2. Design List Window
     1. From the list, select row: **Magnet**
     2. Click the **Properties** button
     3. Properties Window
        1. Orientation: **FaceCS1**
        2. Click the **OK** button
     4. Click the **Done** button





WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Set Default Material

To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **vacuum**

Define a Region

To define a Region:

1. Select the menu item *Draw > Region*
    1. Padding Date: **One**
    2. Padding Percentage: **50**
    3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Analysis Setup

Creating an Analysis Setup

- **To create an analysis setup:**
  1. Select the menu item *Maxwell > Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item *File > Save As*.
  2. From the **Save As** window, type the Filename: **maxwell_ms_magforce**
  3. Click the **Save** button

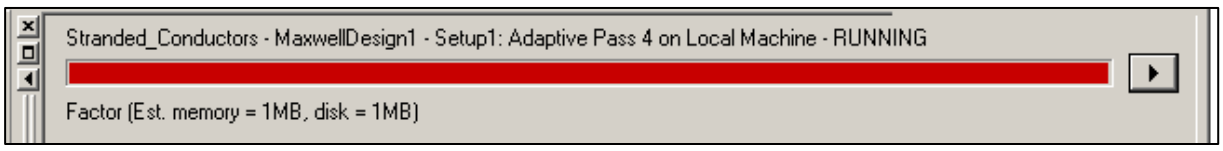
Analyze

Model Validation

- **To validate the model:**
  1. Select the menu item *Maxwell > Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item *Maxwell > Analyze All*



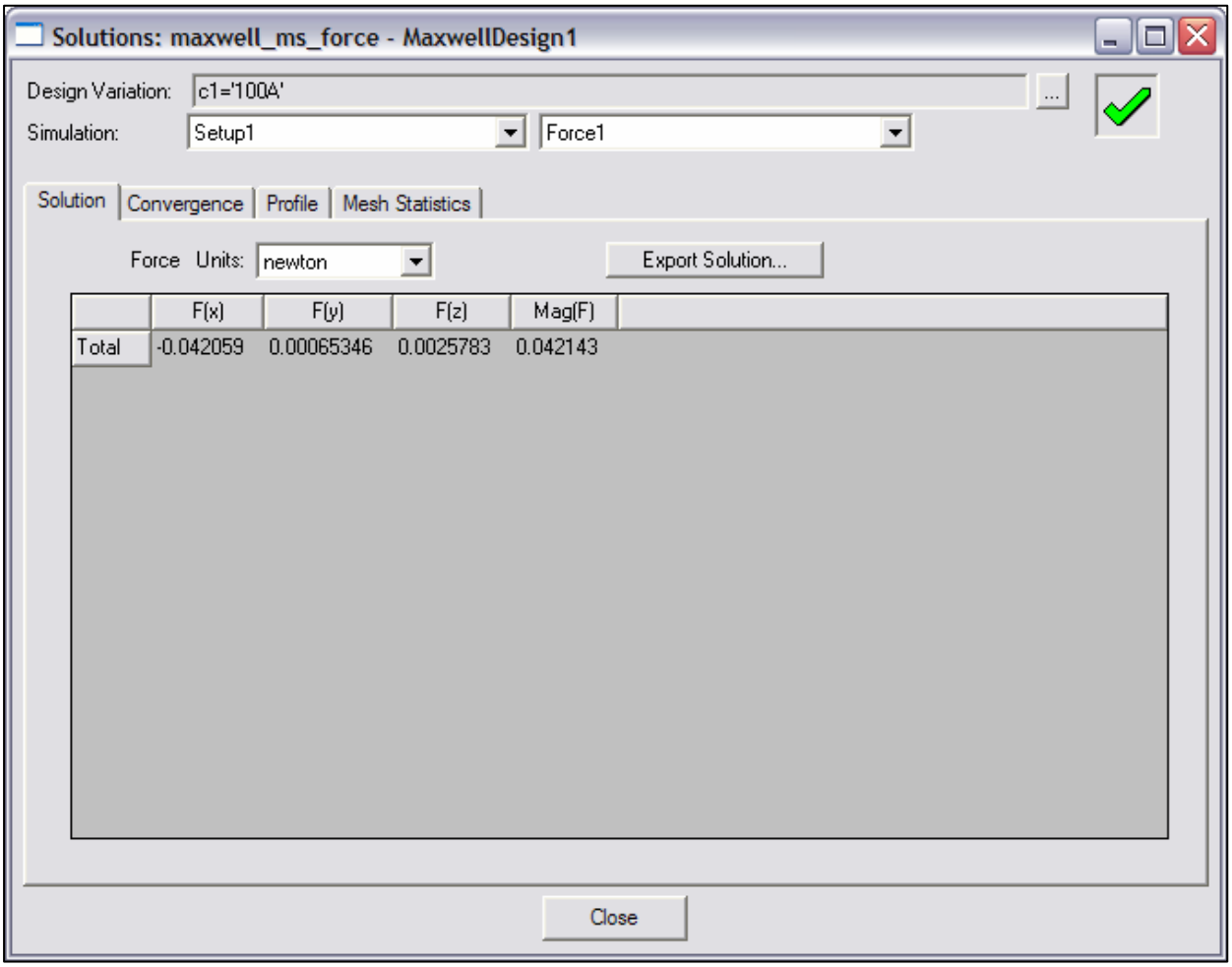
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Solution Data

To view the Solution Data:

1. Select the menu item *Maxwell > Results > Solution Data*
   - **To view the Profile:**
     1. Click the **Profile** Tab.
   - **To view the Convergence:**
     1. Click the **Convergence** Tab
     - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
   - To view the Solutions:
     1. Click the **Solutions** Tab
2. Click the **Close** button



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Optimetrics Setup - Parametric Sweep

- During the design of a device, it is common practice to develop design trends based on swept parameters. Ansoft Maxwell 3D with Optimetrics Parametric Sweep can automatically create these design curves.

Add a Parametric Sweep

1. Select the menu item *Maxwell > Optimetrics Analysis > Add Parametric*
2. Setup **Sweep Analysis** Window:
    1. Click the **Sweep Definitions** tab:
        1. Click the **Add** button
        2. Add/Edit Sweep Dialog
            1. Select Variable: **c1**
            2. Select **Linear Count**
            3. Start: **0A**
            4. Stop: **500A**
            5. Step: **100**
            6. Click the **Add** button
            7. Click the **OK** button
    2. Click the **Options** tab:
        1. Save Fields and Mesh: ☑ **Checked**
    3. Click the **OK** button

Analyze Parametric Sweep

- **To start the solution process:**
    1. Expand the Project Tree to display the items listed under **Optimetrics**
    2. Right-click the mouse on **ParametricSetup1** and choose **Analyze**

Optimetrics Results

- **To view the Optimetrics Results:**
    1. Select the menu item *Maxwell> Optimetrics Analysis > Optimetrics Results*
    2. Select the **Profile** Tab to view the solution progress for each setup.
    3. Click the **Close** button when you are finished viewing the results

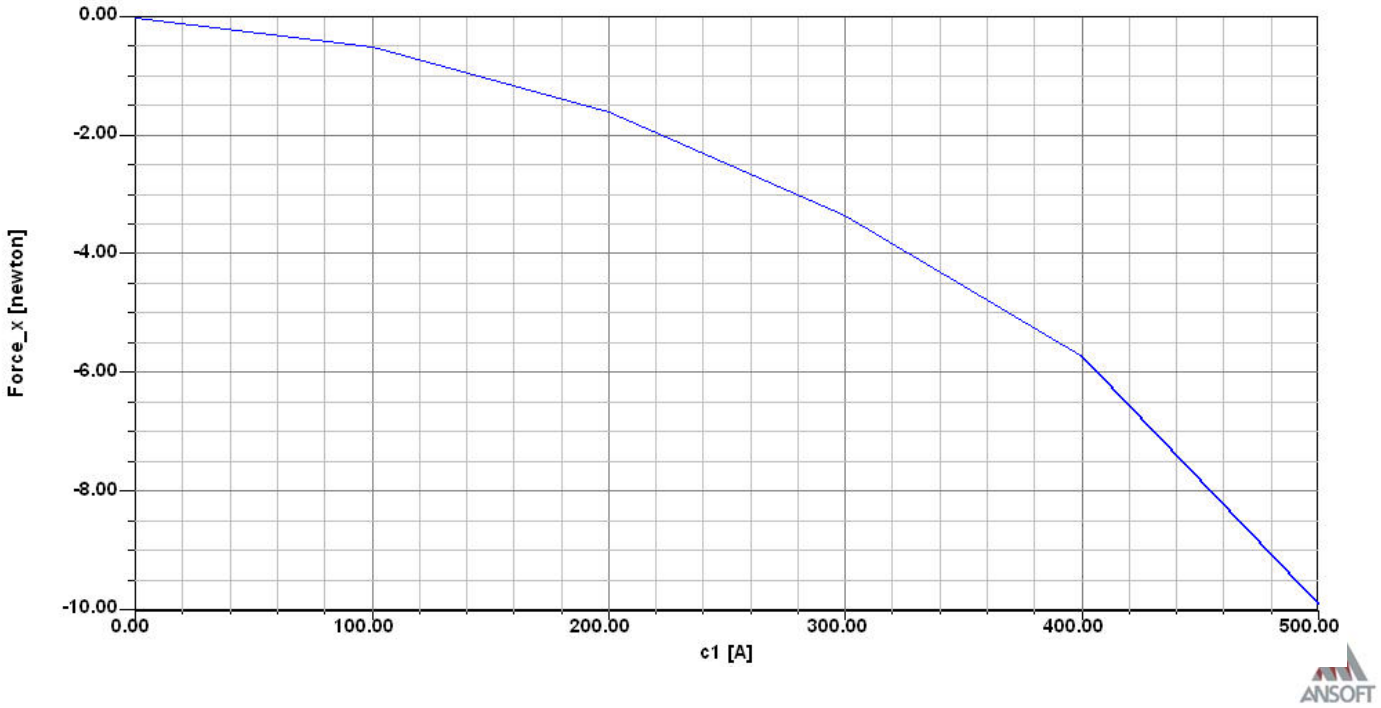
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Magnetic Force

Create Plot of Force at each Current

To create a report:

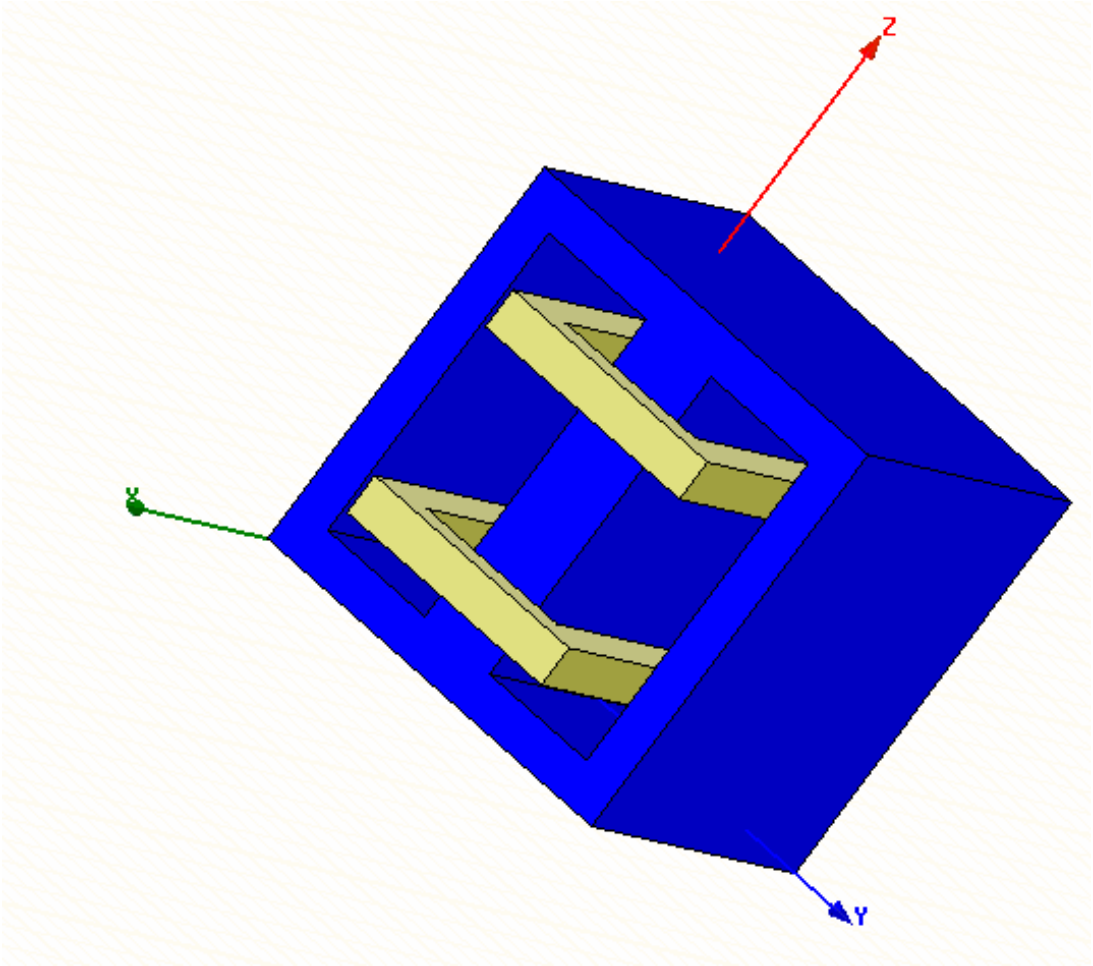
1. Select the menu item *Maxwell > Results > Create Report*
2. Create Report Window:
    1. Report Type: **Magnetostatic**
    2. Display Type: **Rectangular Plot**
    3. Click the **OK** button
3. Traces Window:
    1. Solution: **Setup1: Force**
    2. Click the **Sweeps** tab
        1. Select **Sweep Design and Project variable values**
        2. Make sure that **c1** is selected as the primary sweep
    3. Click the **Y** tab
        1. Category: **Force**
        2. Quantity: **Force_x**
        3. Function: **<none>**
        4. Click the **Add Trace** button
    4. Click the **Done** button



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

- The implementation and application of Nonlinear Inductor when using the Magnetostatic Solver
  - The nonlinear inductance is calculated by Maxwell.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Rectangular Box**
    - Surface Operations: **Draw Rectangle**
    - Boolean Operations: **Subtract**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Inductance Matrix**
  - Field Overlays:
    - **Flux Density Mapping**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

Setting Tool Options

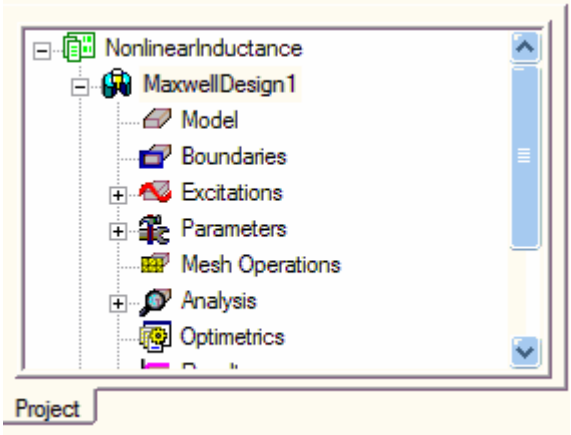
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options*.
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Opening a New Project

- **To open a new project:**
  1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  2. Select the menu item *Project > Insert Maxwell Design*, or click on the icon



Set Solution Type

- Select the menu item: *Maxwell > Solution Type > Magnetostatic*, or right mouse click on MaxwellDesign1 and select *Solution Type …*

Creating the 3D Model of a Nonlinear Inductor

- The example that will be used to demonstrate how to model the Nonlinear Inductor. By using 2 independent windings, we can observe the self inductance and mutual inductance. The inductance varies regard to the saturation level of the operation.

Set Model Units

- Select the menu item *3D Modeler > Units >* Select Units*: mm (millimeters)*

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the Core:

- **Box is used to create the core:**
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 0, Z: 0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 10, dZ: 10**, Press the **Enter** key
     - OK
     - The name of box is box1
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 1, Z: 1**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 3, dZ: 8**, Press the **Enter** key
     - OK
     - The name of box is box2
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 6, Z: 1**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 3, dZ: 8**, Press the **Enter** key
     - OK
     - The name of box is box3
  - *Hold Ctrl key and Select: Box1, Box2, Box3*
  - *3D Modeler>Boolean>Subtract...*
  - *OK*
  - The object name is Box1 since Box1 was selected first.
  - Click on the just created object in the drawing window and in the panel on the left change its name from *Box1* to *Core*.
  - Change the Material from **vacuum** to **Steel_1010**.
  - Change the Color from **Gray** to **Blue**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the Coils and Terminals:

- **Box is used to create the coil:**
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: -2, Y: 2, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 14, dY: 6, dZ: 1**, Press the **Enter** key
     - OK
     - The name of box is box4
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: -1, Y: 3, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 12, dY: 4, dZ: 1**, Press the **Enter** key
     - OK
     - The name of box is box5
  - *Hold Ctrl key and Select: Box4, Box5*
  - *3D Modeler>Boolean₩>Subtract*
  - *OK*
  - The object name is Box4 since Box4 was selected first.
  - Click on the just created object in the drawing window and in the panel on the left change its name from *Box4* to *CoilBottom*.
  - Change the Material from **vacuum** to **Copper**.
  - Change the Color from **Gray** to **Yellow.**
  - Change the Transparent **0** to **0.1**
  - *Draw > Rectangle*
  - Change the Draw plane from *XY* to *XZ*
  1. Using the coordinate entry field, enter the corner of the rectangle
     - **X: 11, Y: 5, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the rectangle
     - **dX: 1, dY: 0, dZ: 1**, Press the **Enter** key
     - OK
     - The name of rectangle is Rectangle1
     - Click on the just created object in the drawing window and in the panel on the left change its name from *Rectangle1* to *TerminalBottom*..

WWW.MOHANDESYAR.COM

Create the Coils and Terminals:

- *Select : CoilBottom, TerminalBottom*
- *Edit>Duplicate>Along Line*

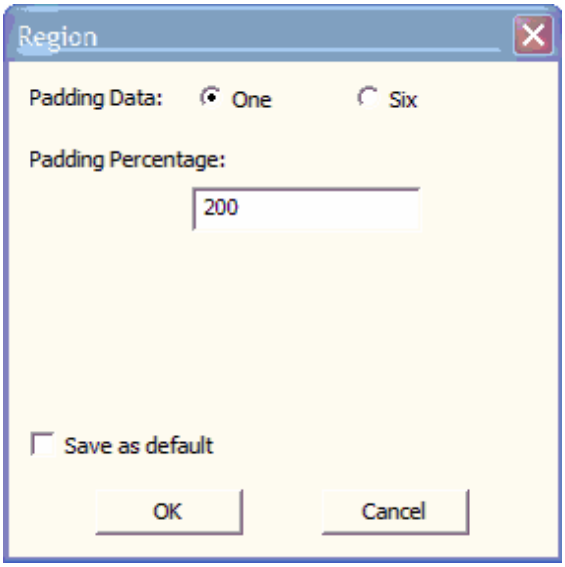
1. Using the coordinate entry field, enter the first point of duplicate vector.
   - **X: 0, Y: 0, Z: 0**, Press the **Enter** key
2. Using the coordinate entry field, enter the second point of duplicate vector.
   - **dX: 0, dY: 0, dZ: 5**, Press the **Enter** key
   - **OK**
   - Click on the just created object in the drawing window and in the panel on the left change its name from *TerminalBottom_1* to *TerminalTop*.
   - Click on the just created object in the drawing window and in the panel on the left change its name from *CoilBottom_1* to *CoilTop*.

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the problem Region

- One of the main differences between Maxwell V10 and V11 is that a Background Region is not automatically created when a project is started. A separate object needs to be specifically created.
- Create a rectangular region that has the same shape as the Nonlinear Inductor by selecting the menu item ***Draw > Region***



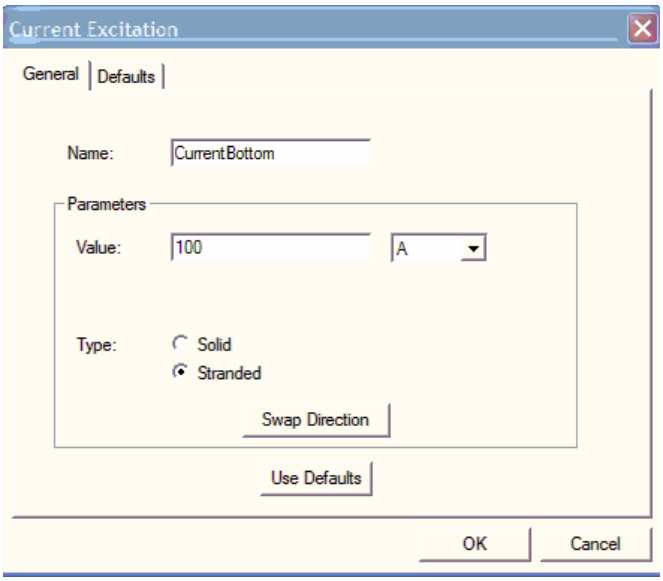
- Padding Percentage: **200**
- **Ok**
- **Ok**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Assign Current Source to the Terminals

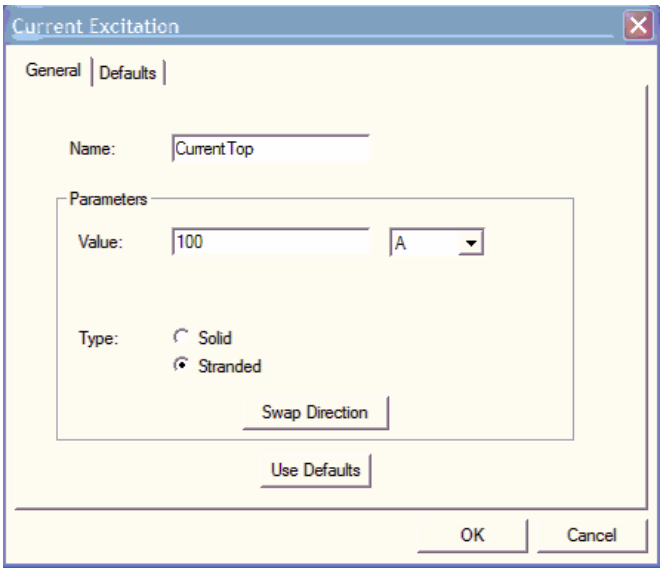
- Select **TerminalBottom.**
- Select the menu item *Maxwell > Excitations > Assign > Current*
  1. Change the Name to **CurrentBottom**
  2. Change the value to **100** Amps
  3. Change the type to **Stranded**
  4. **Ok**



WWW.MOHANDESYAR.COM

Assign Current Source to the Terminals

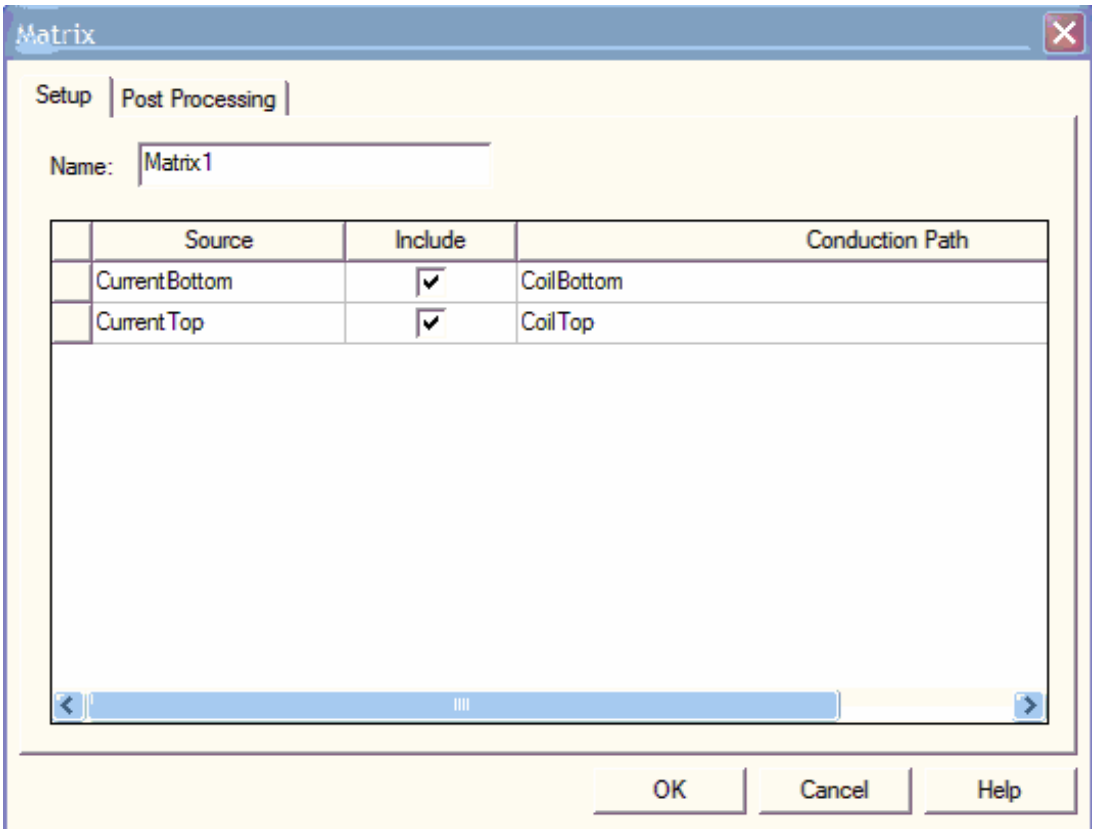
- Select **TerminalTop.**
- Select the menu item *Maxwell > Excitations > Assign > Current*
  1. Change the Name to **CurrentTop**
  2. Change the value to **100** Amps
  3. Change the type to **Stranded**
  4. **Ok**



WWW.MOHANDESYAR.COM

Add Inductance Matrix Calculation

- On the Project Window, Right Click **Parameters.**
- ***Assign > Matrix...***
  1. Check **CurrentTop and CurrentBottom**

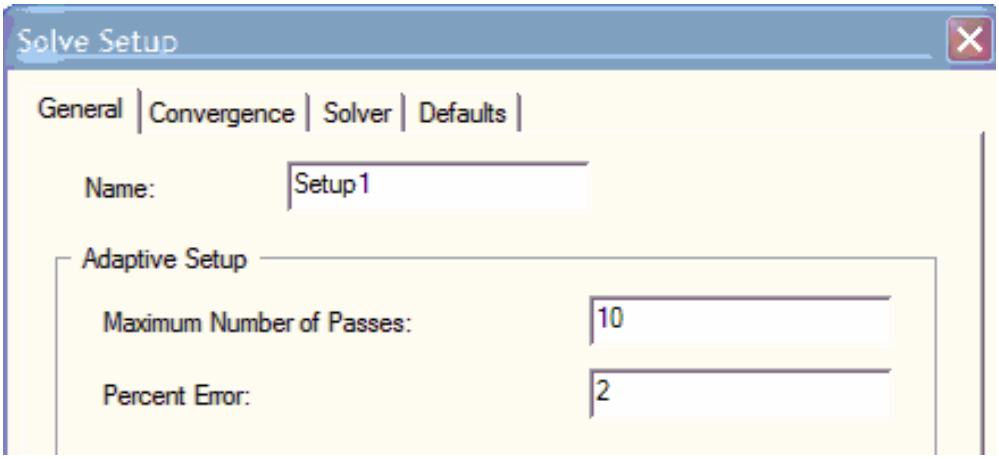


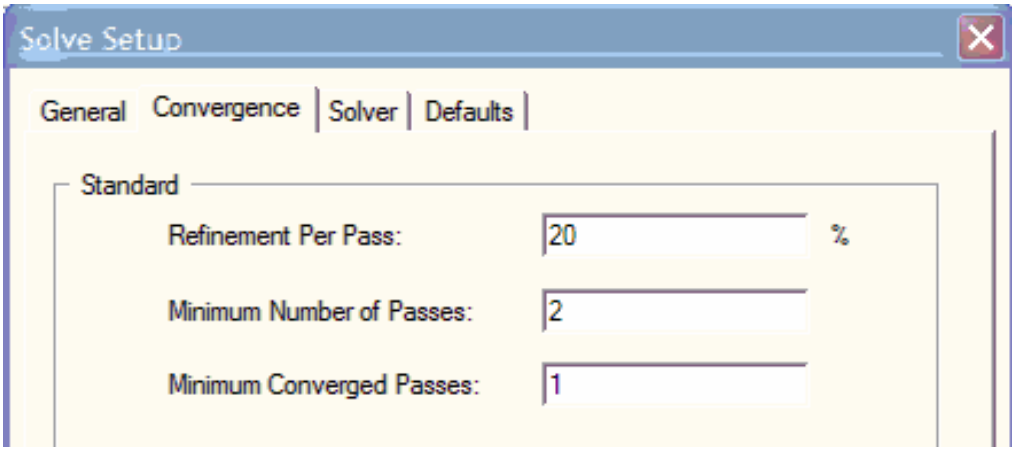
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create an Analysis Setup

- Select the menu item ***Maxwell > Analysis Setup > Add Solution Setup***
- Select **General** and change the Percent Error from **1** to **2**
- Select **Convergence** and change the Refinement Per Pass from **30** to **20**





WWW.MOHANDESYAR.COM

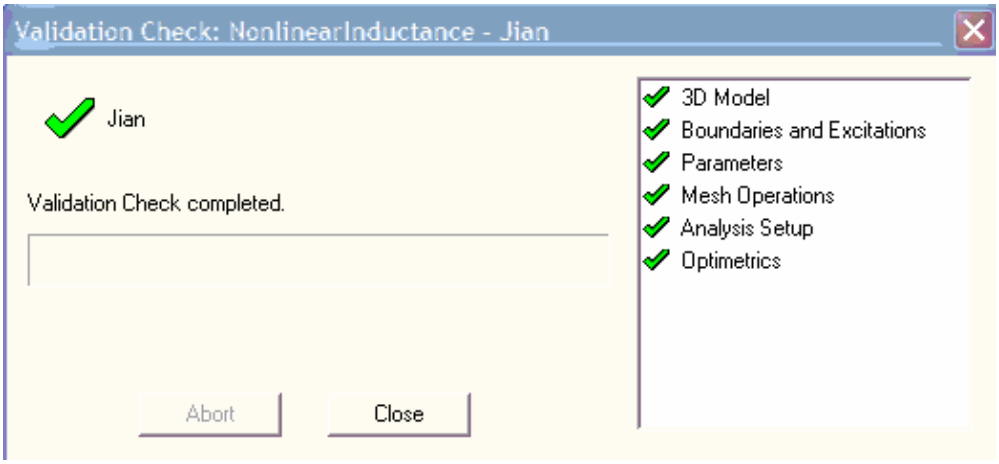
Example (Magnetostatic) - Nonlinear Inductor

Save the Project

- Select the menu item **File > Save As**
- From the Save As window, type in **NonlinearInductance.**
- Click on the **Save** button

Check the Validity of the Model

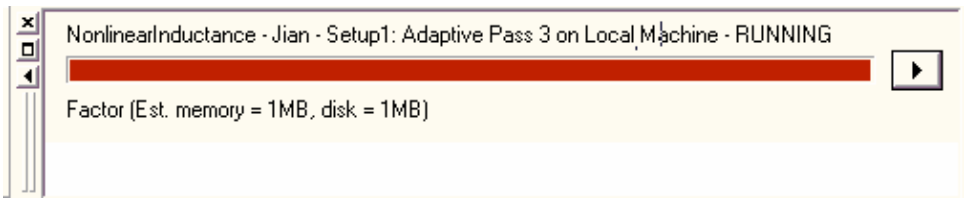
- Select the menu item ***Maxwell > Validation Check***, or click on the icon



- The problem won't solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the icon

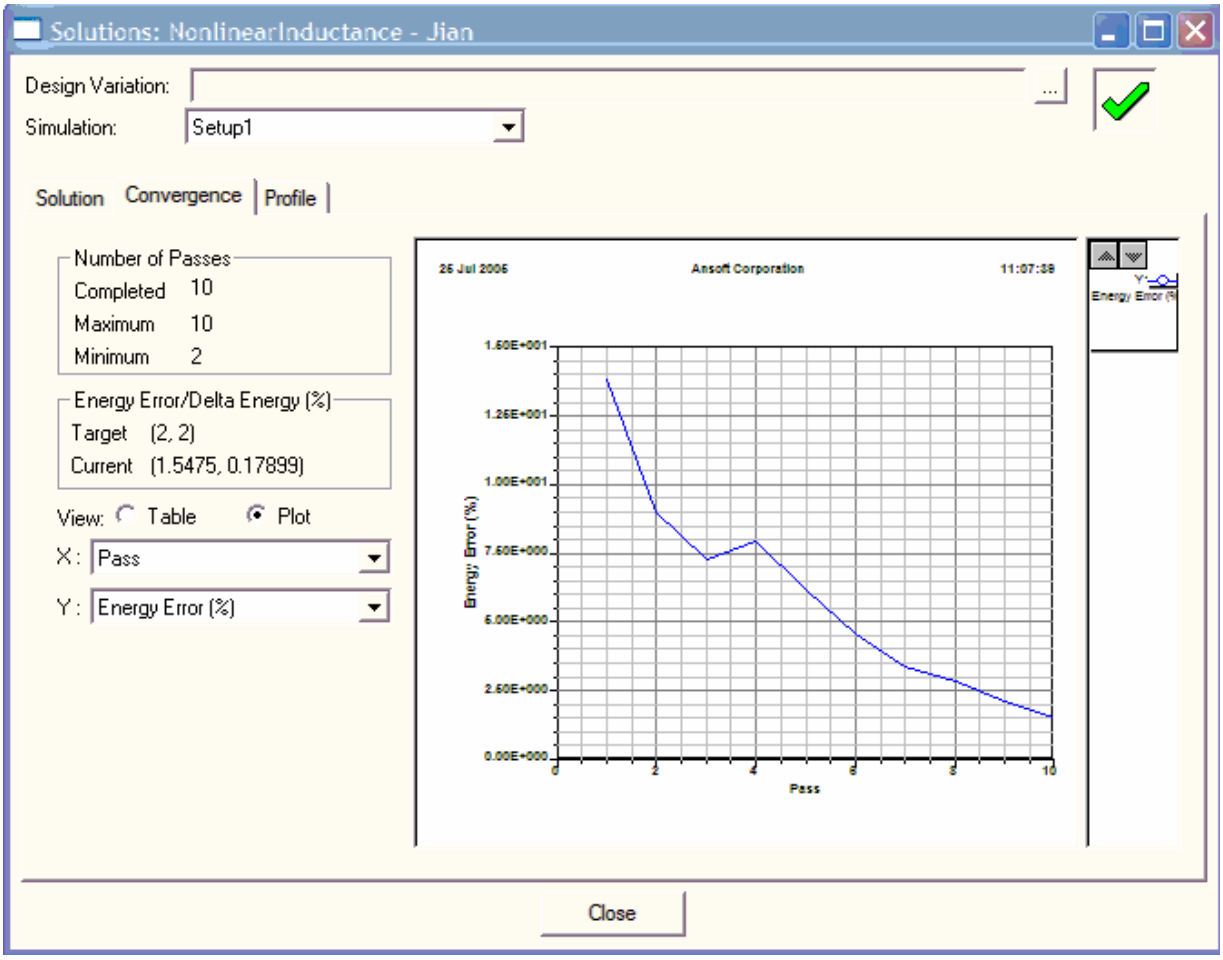


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Solution Data

- To view the Solution Data, select the menu item ***Maxwell > Results > Solution Data***
- Here you can view the **Profile** and the **Convergence.** Note: The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

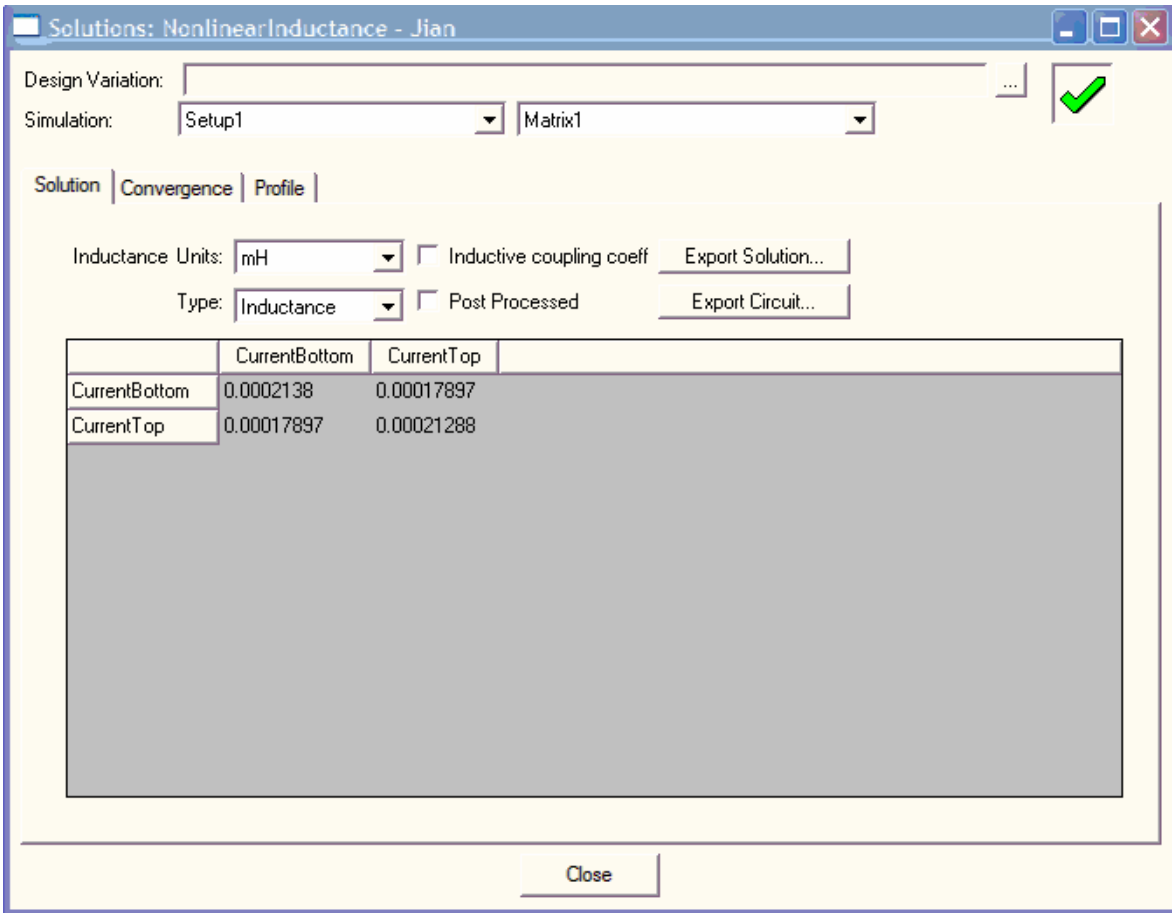


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Observe the Inductance Matrix

- *Maxwell > Results> Solution Data...*



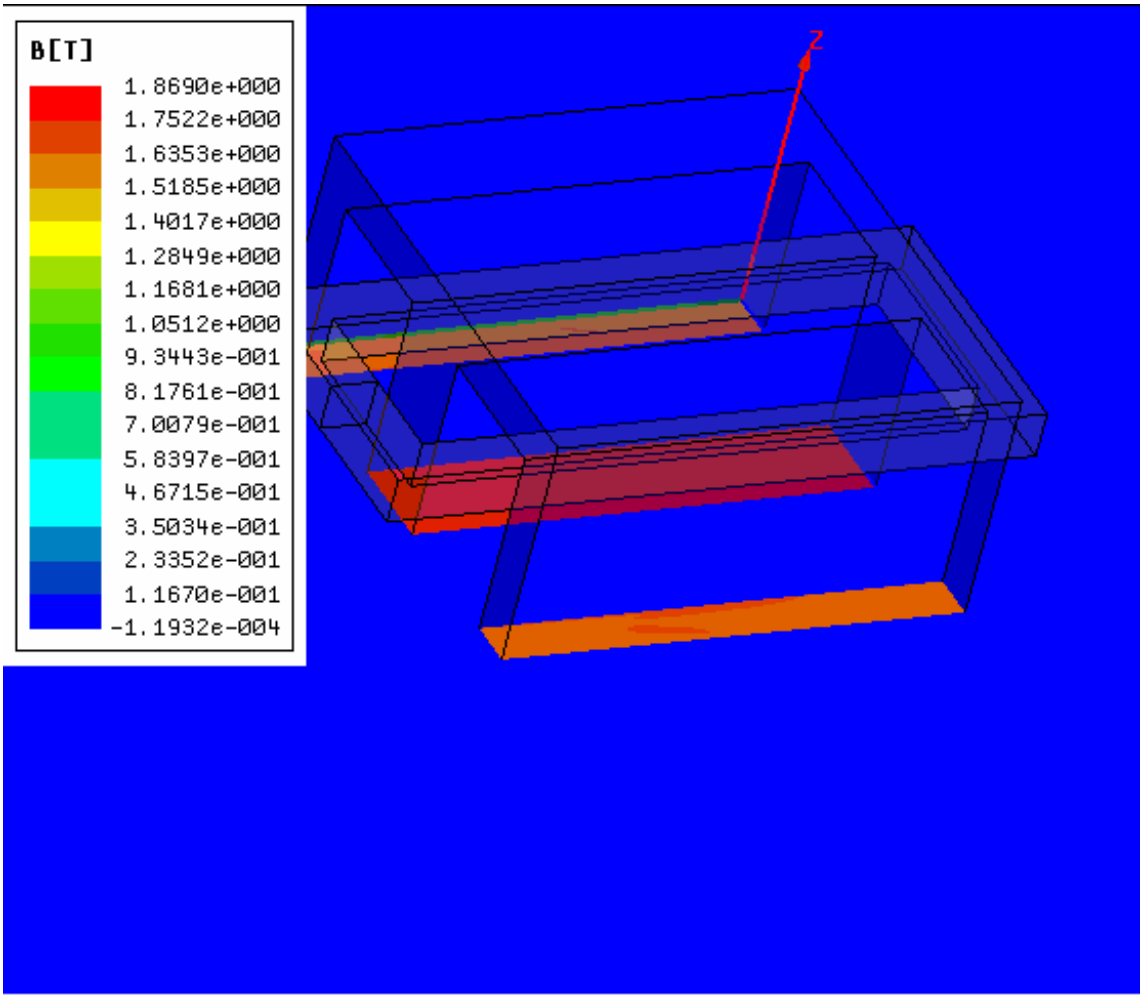
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Plot Flux Density on Core Cross section

- To plot the flux density on the core cross section, a new coordinate system is needed.
- Here are the steps to create:
- **3D Modeler>Coordinate System>Create>Relative CS>Offset**

  Using the coordinate entry field, enter the position of the new coordinates:
  - **X: 0, Y: 0, Z: 5**, Press the **Enter** key
  - The name of new Coordinate is **RelativeCS1**
- Select the plot plane: **Planes>RelativeCS1:XY**
- Right mouse click: *Field Overlays > Fields > B> Mag_B.*
- From the plot, we can see the flux density on core is around **1.86** Telsla. It is close to the saturation point of Steel_1010.
- If we drive the inductor with much lower current, the operating point will be in the linear area. Then the inductance will be much higher than the inductance with operating point close to saturation point.

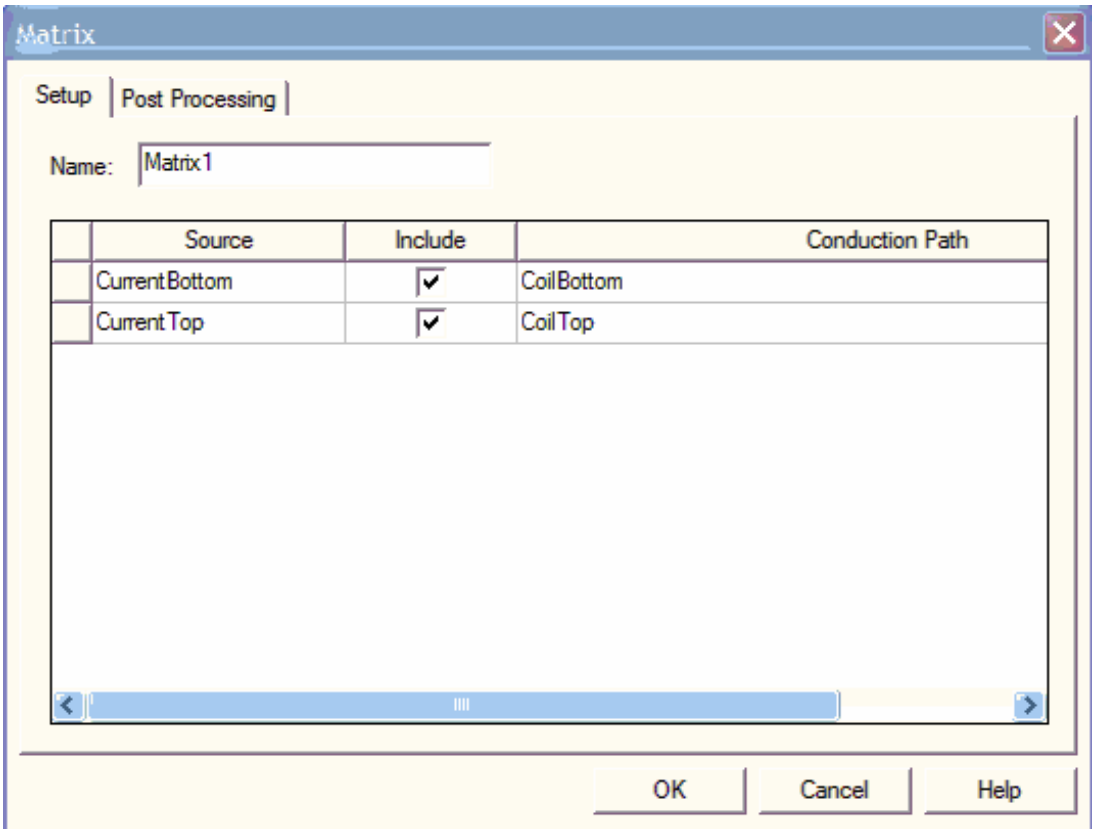


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create a new project

- Right Click on Project **MaxwellDesign1> Copy**
- Click on **NonlinearInductance>Paste.**
- A new project named **MaxwellDesign2** is created.
- Click **Excitations: CurrentBottom**, Change Current from 100 A to 1 A.
- Click **Excitations: CurrentTop**, Change Current from 100 A to 1 A.
- Right Click on **Parameters>Assign>Matrix...**
- Check **CurrentTop and CurrentBottom**

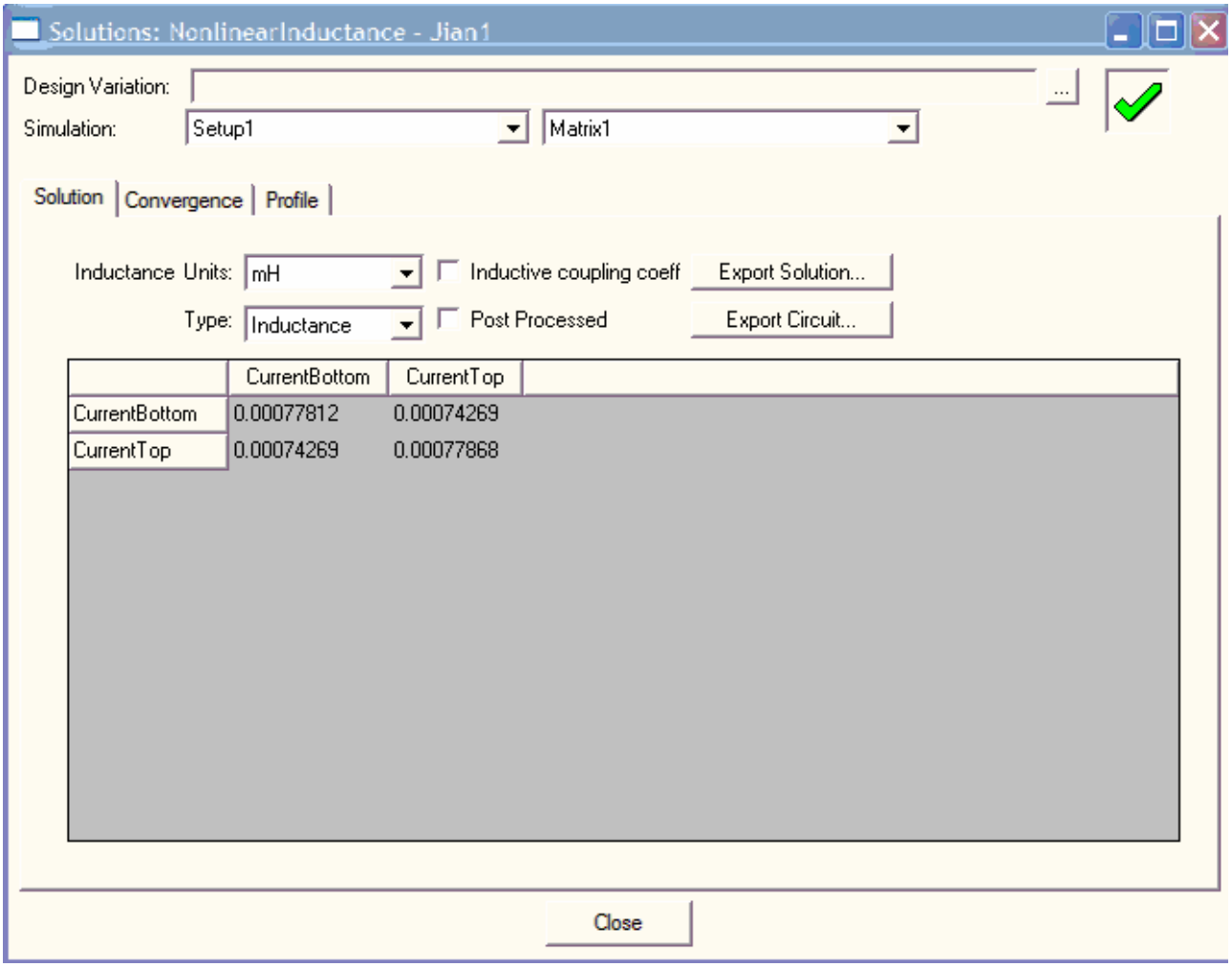


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Run the new project and Observe the Inductance Matrix

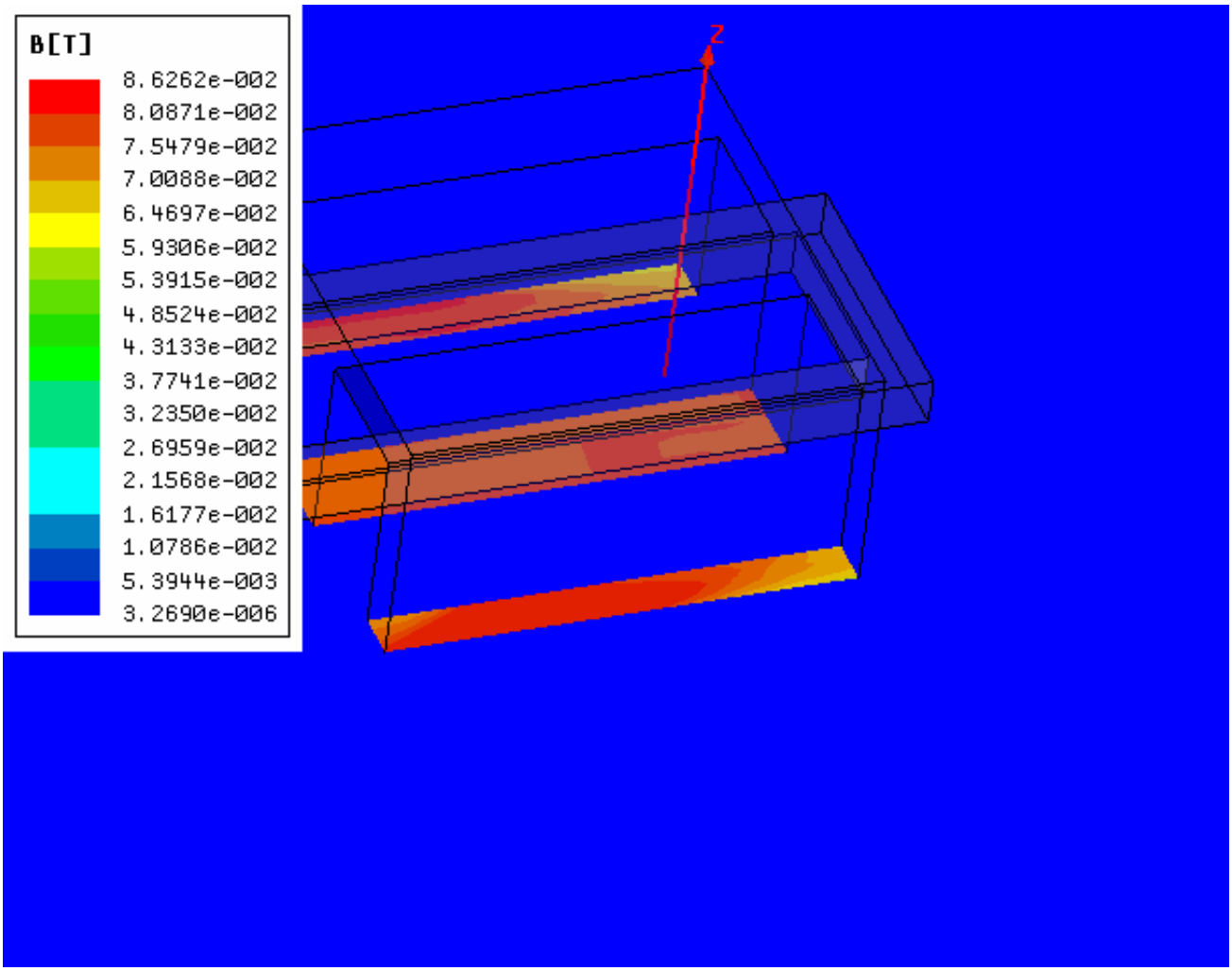
- Click on ***Analysis>Setup1***
- Right Click on ***Setup1: Analyze.***
- The simulation run and stop.
- ***Maxwell > Results> Solution Data…***
- We can see the inductance is much higher than the previous simulation.



WWW.MOHANDESYAR.COM

Plot Flux Density on Core Cross section

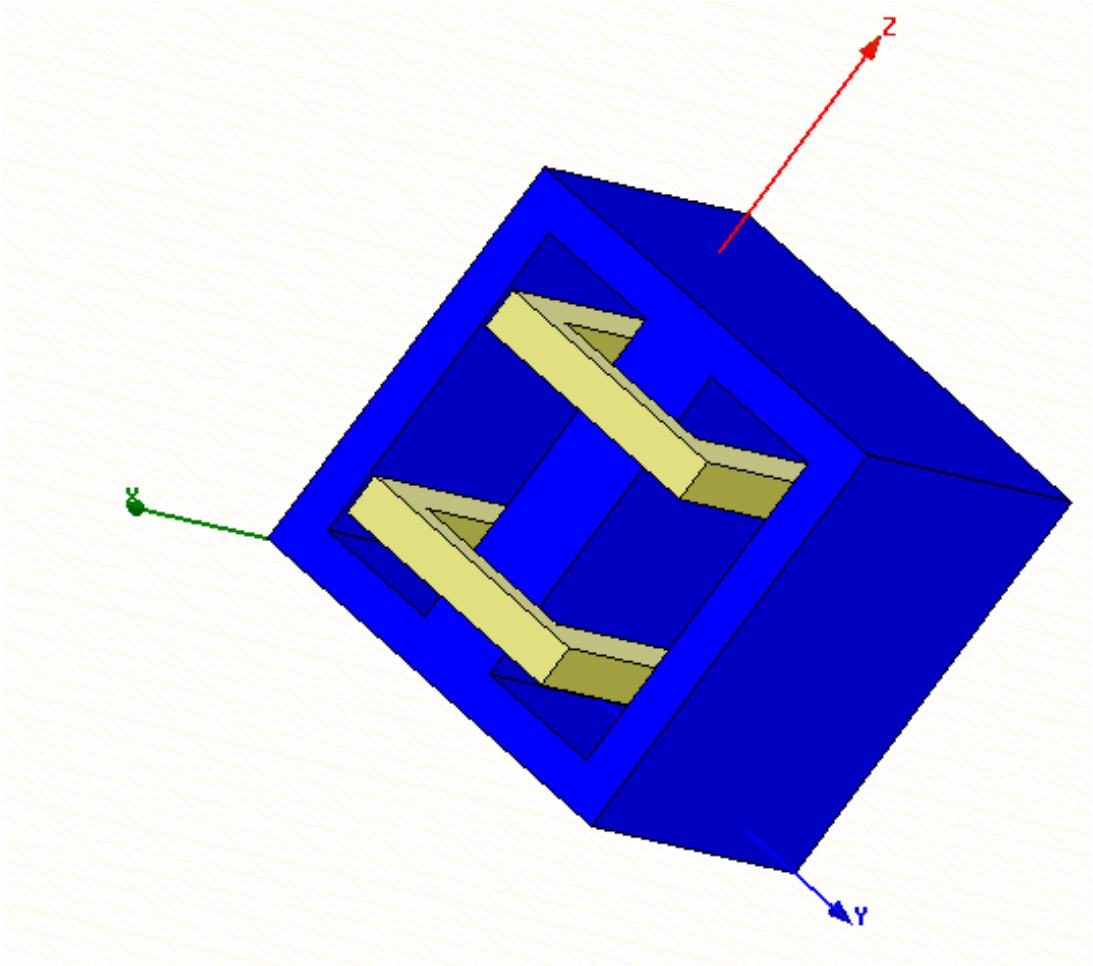
- Select the plot plane: **Planes>RelativeCS1:XY**
- Right mouse click: ***Field Overlays > Fields > B> Mag _B.***
- From the plot, we can see the flux density on core is around **0.086** Telsla. It is working on the linear area of Steel_1010.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

- The implementation and application of Nonlinear Inductor when using the Magnetostatic Solver
  - The nonlinear inductance is calculated by Maxwell.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Rectangular Box**
    - Surface Operations: **Draw Rectangle**
    - Boolean Operations: **Subtract**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Inductance Matrix**
  - Field Overlays:
    - **Flux Density Mapping**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

Setting Tool Options

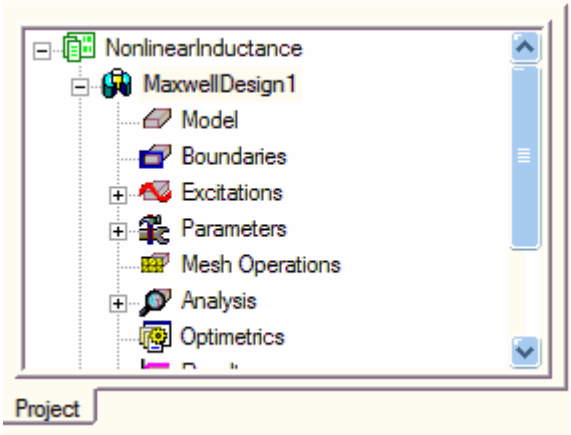
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options*.
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Opening a New Project

- **To open a new project:**
  1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  2. Select the menu item *Project > Insert Maxwell Design*, or click on the icon



Set Solution Type

- Select the menu item: *Maxwell > Solution Type > Magnetostatic*, or right mouse click on MaxwellDesign1 and select *Solution Type …*

Creating the 3D Model of a Nonlinear Inductor

- The example that will be used to demonstrate how to model the Nonlinear Inductor. By using 2 independent windings, we can observe the self inductance and mutual inductance. The inductance varies regard to the saturation level of the operation.

Set Model Units

- Select the menu item *3D Modeler > Units >* Select Units*: mm (millimeters)*

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the Core:

- **Box is used to create the core:**
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 0, Z: 0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 10, dZ: 10**, Press the **Enter** key
     - OK
     - The name of box is box1
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 1, Z: 1**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 3, dZ: 8**, Press the **Enter** key
     - OK
     - The name of box is box2
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: 0, Y: 6, Z: 1**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 10, dY: 3, dZ: 8**, Press the **Enter** key
     - OK
     - The name of box is box3
  - *Hold Ctrl key and Select: Box1, Box2, Box3*
  - *3D Modeler>Boolean>Subtract...*
  - *OK*
  - The object name is Box1 since Box1 was selected first.
  - Click on the just created object in the drawing window and in the panel on the left change its name from *Box1* to *Core*.
  - Change the Material from **vacuum** to **Steel_1010**.
  - Change the Color from **Gray** to **Blue**

WWW.MOHANDESYAR.COM

Create the Coils and Terminals:

- **Box is used to create the coil:**
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: -2, Y: 2, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 14, dY: 6, dZ: 1**, Press the **Enter** key
     - OK
     - The name of box is box4
  - *Draw > Box*
  1. Using the coordinate entry field, enter the corner of the box
     - **X: -1, Y: 3, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the box
     - **dX: 12, dY: 4, dZ: 1**, Press the **Enter** key
     - OK
     - The name of box is box5
  - *Hold Ctrl key and Select: Box4, Box5*
  - *3D Modeler>Boolean₩>Subtract*
  - *OK*
  - The object name is Box4 since Box4 was selected first.
  - Click on the just created object in the drawing window and in the panel on the left change its name from *Box4* to *CoilBottom*.
  - Change the Material from **vacuum** to **Copper**.
  - Change the Color from **Gray** to **Yellow.**
  - Change the Transparent **0** to **0.1**
  - *Draw > Rectangle*
  - Change the Draw plane from *XY* to *XZ*
  1. Using the coordinate entry field, enter the corner of the rectangle
     - **X: 11, Y: 5, Z: 2**, Press the **Enter** key
  2. Using the coordinate entry field, enter the size of the rectangle
     - **dX: 1, dY: 0, dZ: 1**, Press the **Enter** key
     - OK
     - The name of rectangle is Rectangle1
     - Click on the just created object in the drawing window and in the panel on the left change its name from *Rectangle1* to *TerminalBottom*..

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the Coils and Terminals:

- *Select : CoilBottom, TerminalBottom*
- *Edit>Duplicate>Along Line*

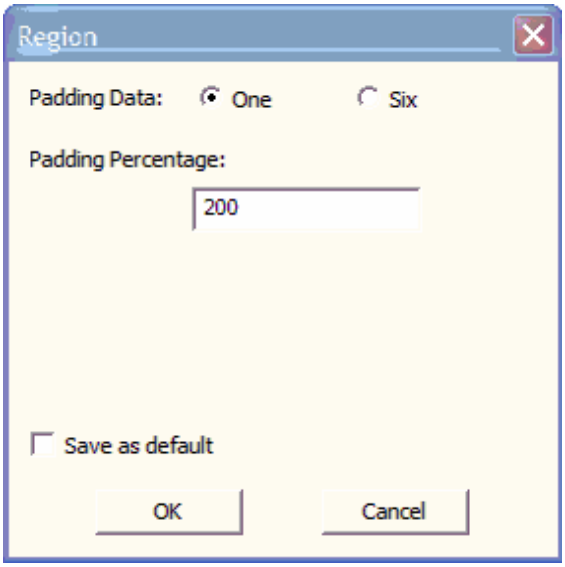
1. Using the coordinate entry field, enter the first point of duplicate vector.
   - **X: 0, Y: 0, Z: 0**, Press the **Enter** key
2. Using the coordinate entry field, enter the second point of duplicate vector.
   - **dX: 0, dY: 0, dZ: 5**, Press the **Enter** key
   - **OK**
   - Click on the just created object in the drawing window and in the panel on the left change its name from *TerminalBottom_1* to *TerminalTop*.
   - Click on the just created object in the drawing window and in the panel on the left change its name from *CoilBottom_1* to *CoilTop*.

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create the problem Region

- One of the main differences between Maxwell V10 and V11 is that a Background Region is not automatically created when a project is started. A separate object needs to be specifically created.
- Create a rectangular region that has the same shape as the Nonlinear Inductor by selecting the menu item ***Draw > Region***

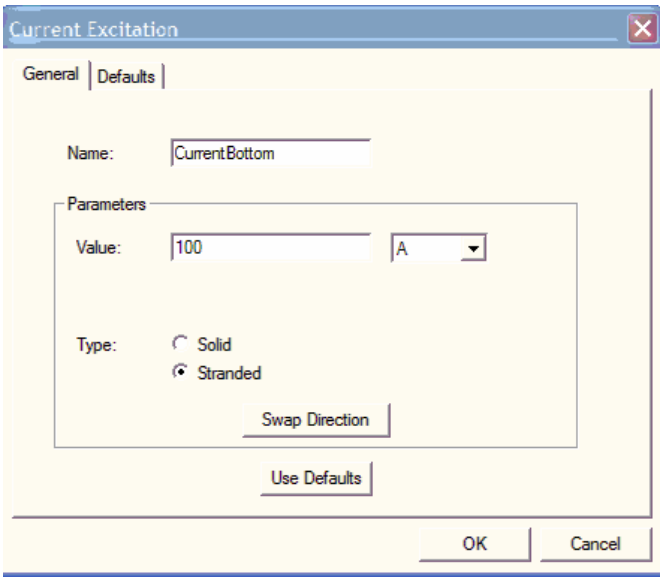


- Padding Percentage: **200**
- **Ok**
- **Ok**

WWW.MOHANDESYAR.COM

Assign Current Source to the Terminals

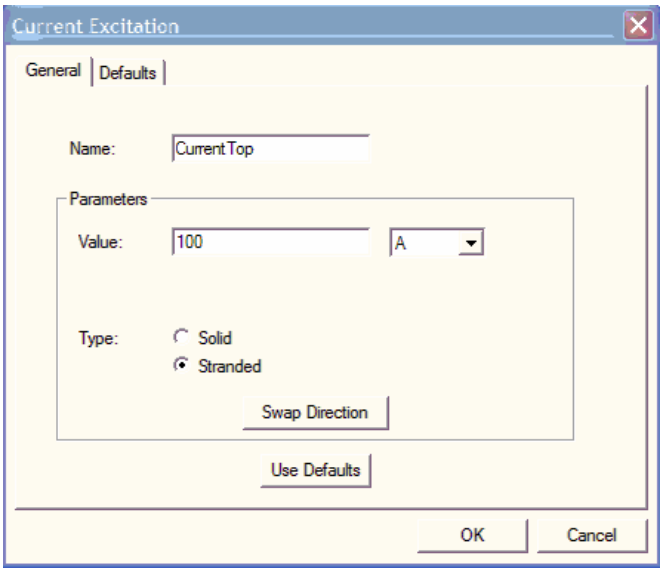
- Select **TerminalBottom.**
- Select the menu item *Maxwell > Excitations > Assign > Current*
  1. Change the Name to **CurrentBottom**
  2. Change the value to **100** Amps
  3. Change the type to **Stranded**
  4. **Ok**



WWW.MOHANDESYAR.COM

Assign Current Source to the Terminals

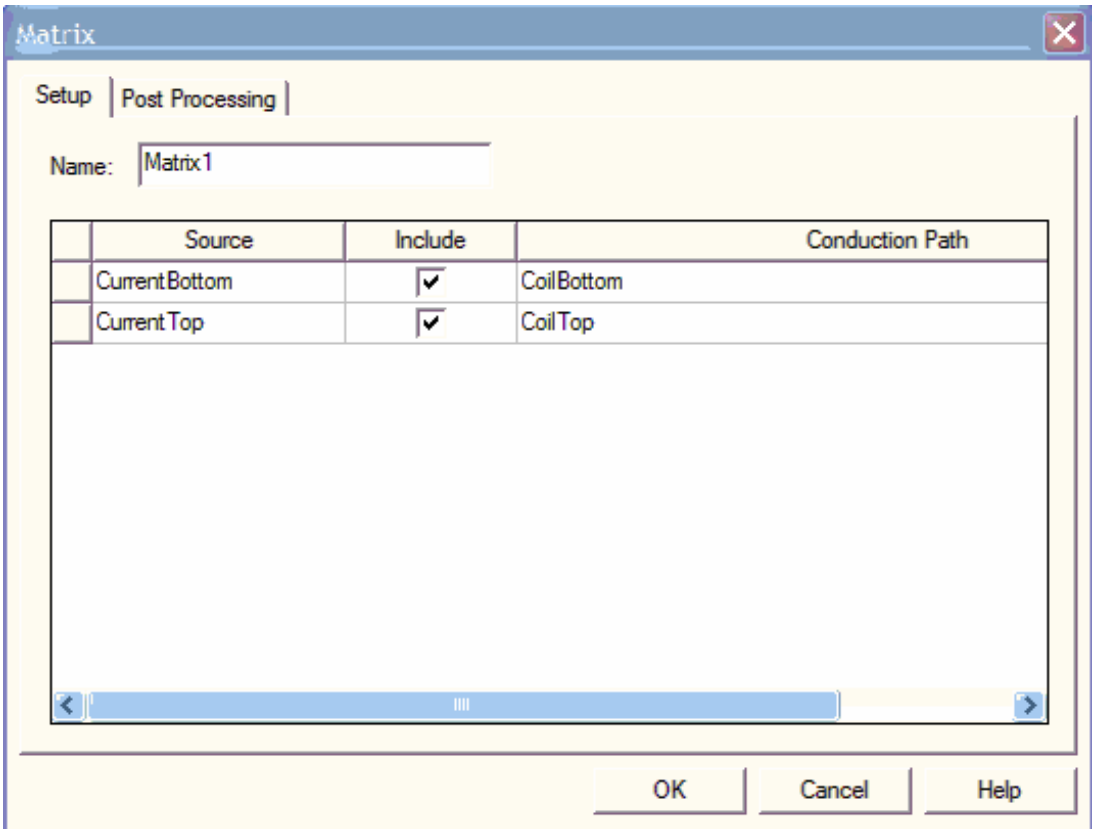
- Select **TerminalTop.**
- Select the menu item *Maxwell > Excitations > Assign > Current*
  1. Change the Name to **CurrentTop**
  2. Change the value to **100** Amps
  3. Change the type to **Stranded**
  4. **Ok**



WWW.MOHANDESYAR.COM

Add Inductance Matrix Calculation

- On the Project Window, Right Click **Parameters.**
- ***Assign > Matrix...***
  1. Check **CurrentTop and CurrentBottom**

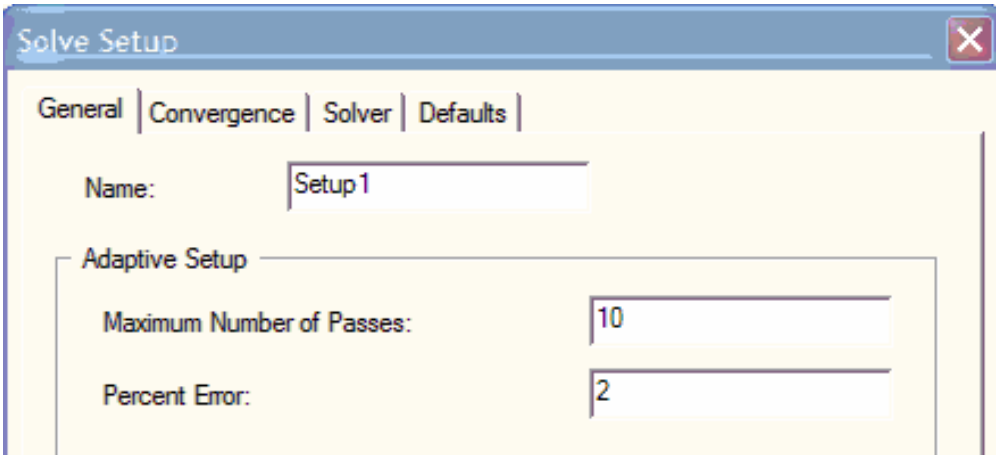


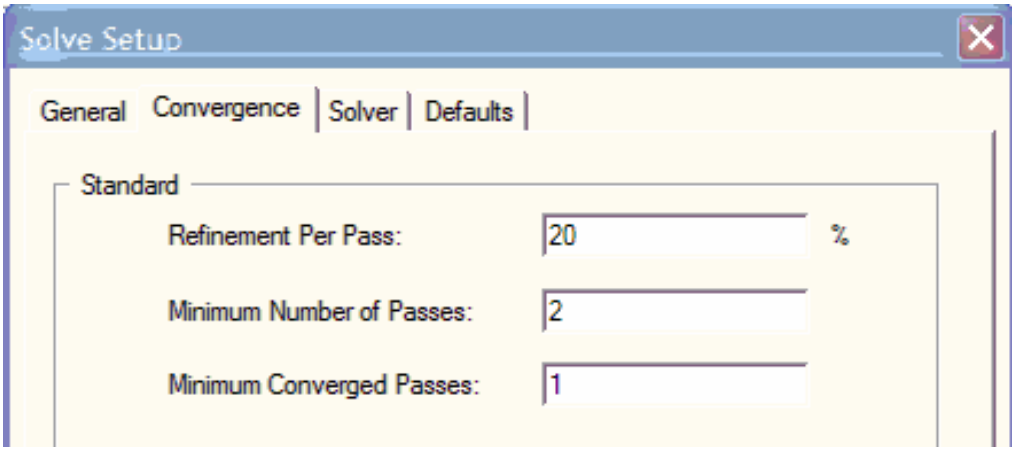
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create an Analysis Setup

- Select the menu item ***Maxwell > Analysis Setup > Add Solution Setup***
- Select **General** and change the Percent Error from **1** to **2**
- Select **Convergence** and change the Refinement Per Pass from **30** to **20**





WWW.MOHANDESYAR.COM

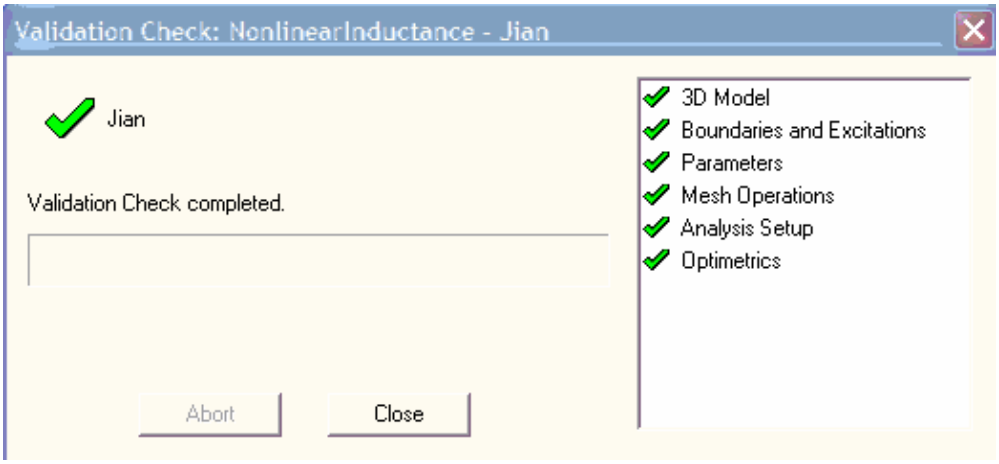
Example (Magnetostatic) - Nonlinear Inductor

Save the Project

- Select the menu item **File > Save As**
- From the Save As window, type in **NonlinearInductance.**
- Click on the **Save** button

Check the Validity of the Model

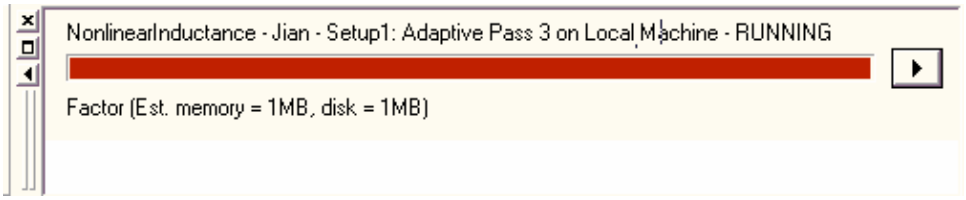
- Select the menu item ***Maxwell > Validation Check***, or click on the icon



- The problem won't solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the icon

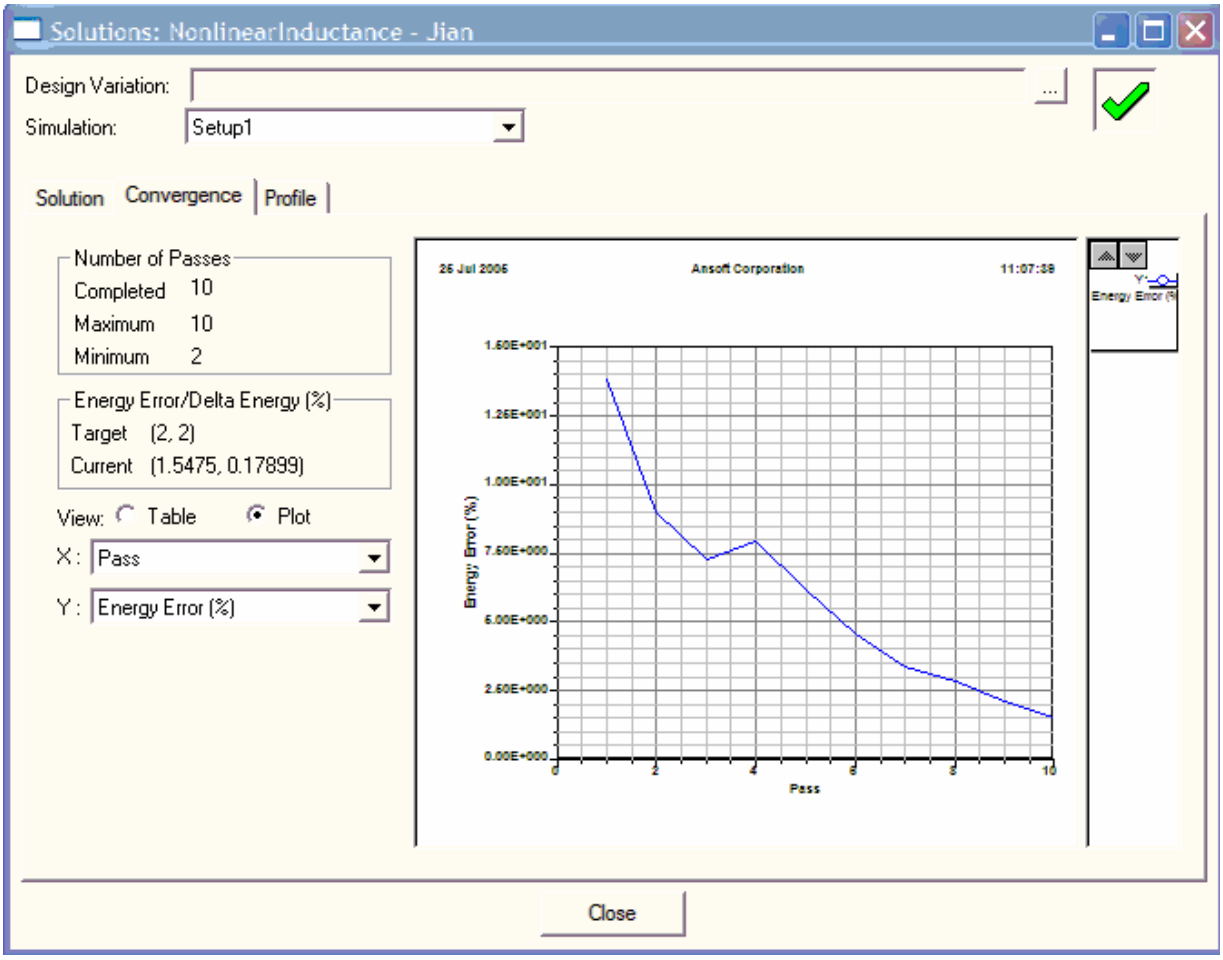


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Solution Data

- To view the Solution Data, select the menu item ***Maxwell > Results > Solution Data***
- Here you can view the **Profile** and the **Convergence.** Note: The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

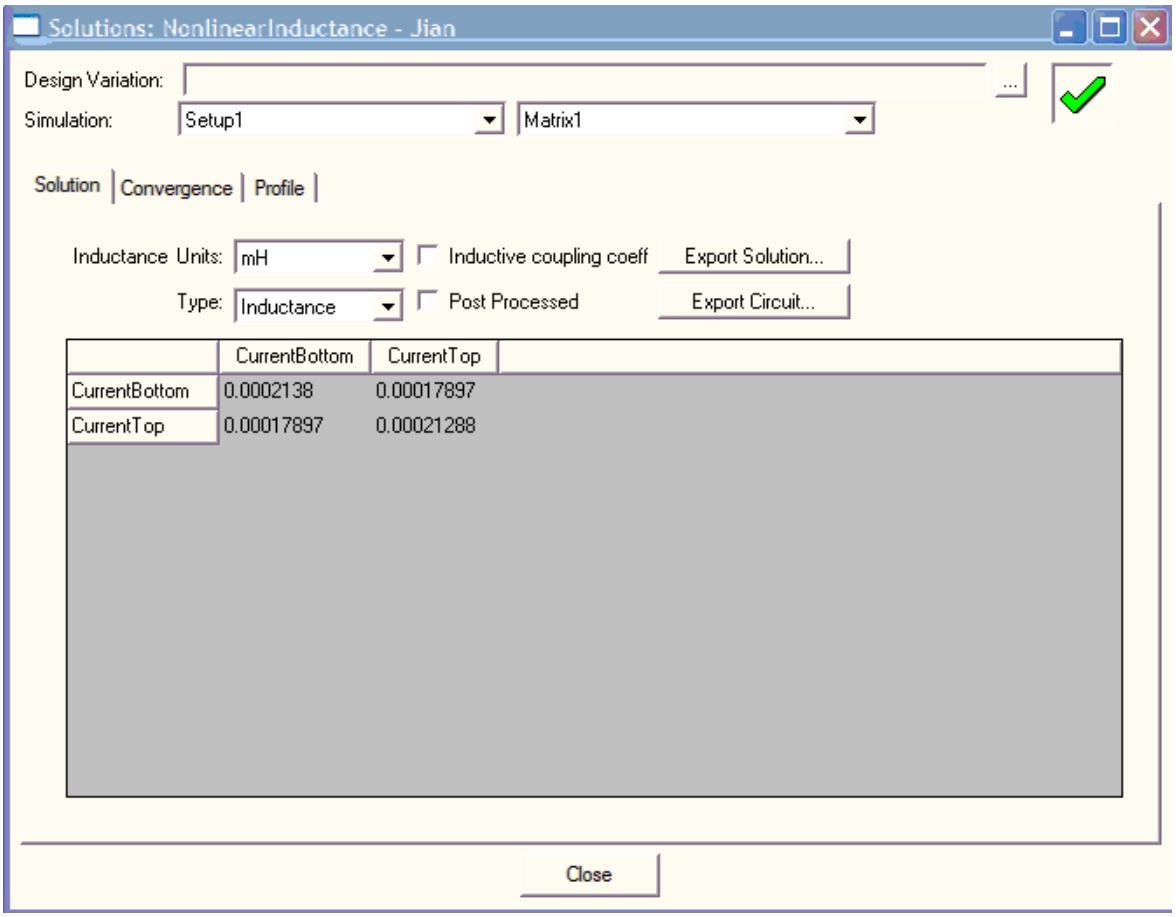


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Observe the Inductance Matrix

- *Maxwell > Results> Solution Data...*



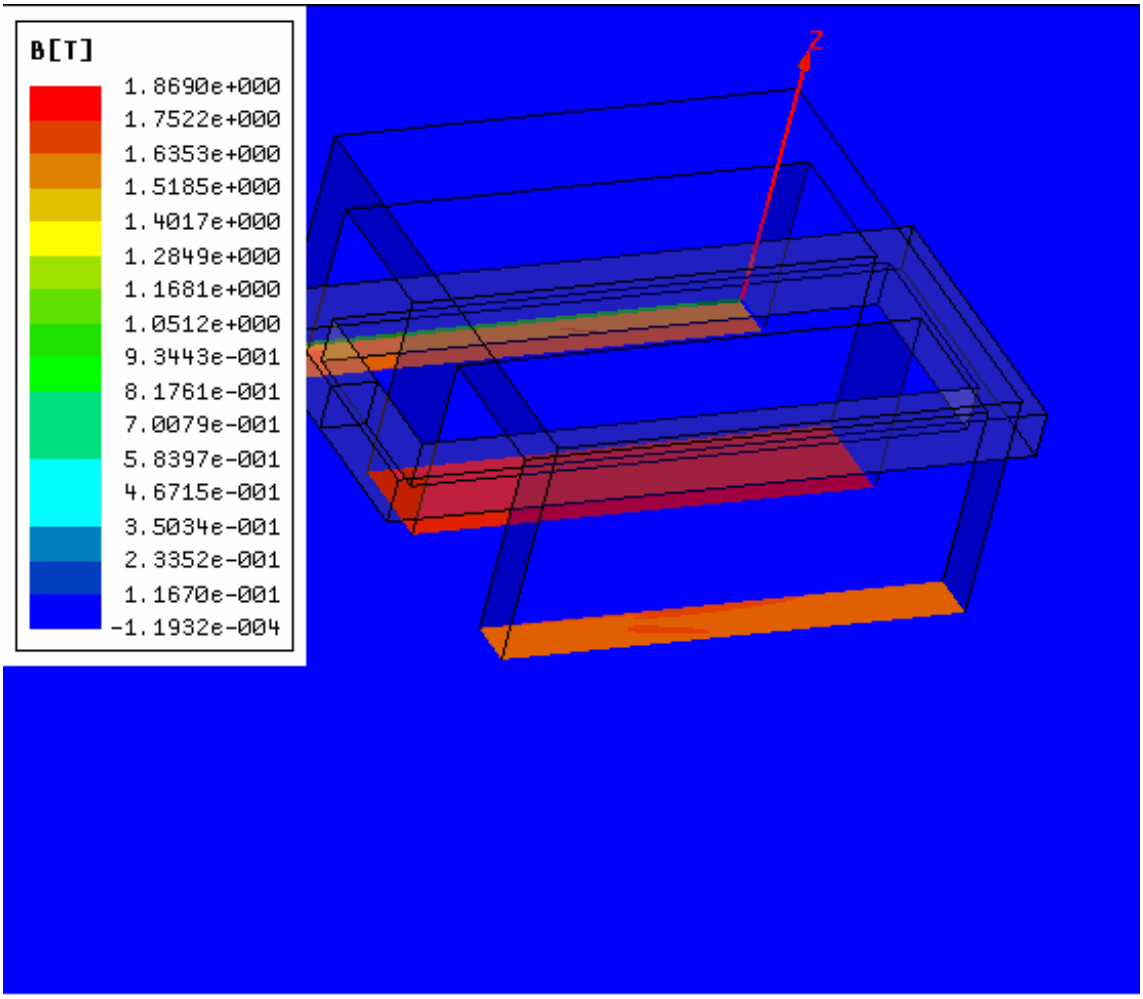
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Plot Flux Density on Core Cross section

- To plot the flux density on the core cross section, a new coordinate system is needed.
- Here are the steps to create:
- **3D Modeler>Coordinate System>Create>Relative CS>Offset**

  Using the coordinate entry field, enter the position of the new coordinates:
  - **X: 0, Y: 0, Z: 5**, Press the **Enter** key
  - The name of new Coordinate is **RelativeCS1**
- Select the plot plane: **Planes>RelativeCS1:XY**
- Right mouse click: ***Field Overlays > Fields > B> Mag_B.***
- From the plot, we can see the flux density on core is around **1.86** Telsla. It is close to the saturation point of Steel_1010.
- If we drive the inductor with much lower current, the operating point will be in the linear area. Then the inductance will be much higher than the inductance with operating point close to saturation point.

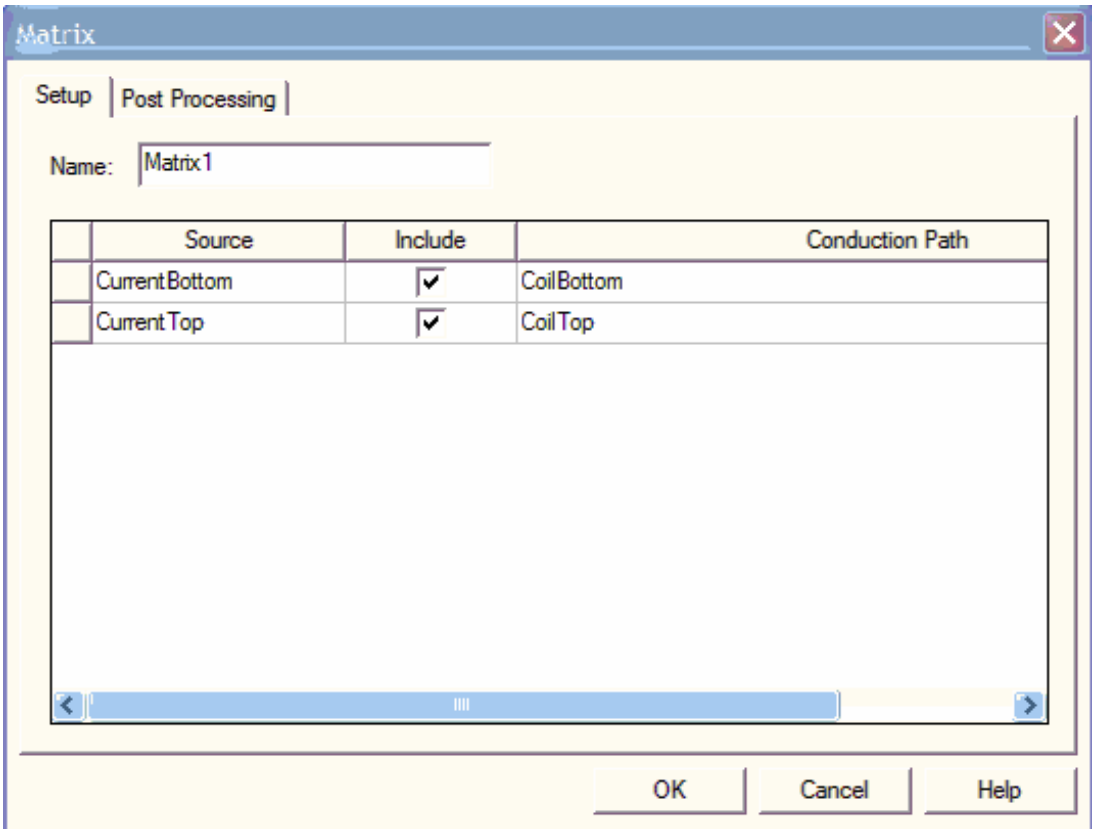


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Create a new project

- Right Click on Project **MaxwellDesign1> Copy**
- Click on **NonlinearInductance>Paste.**
- A new project named **MaxwellDesign2** is created.
- Click **Excitations: CurrentBottom**, Change Current from 100 A to 1 A.
- Click **Excitations: CurrentTop**, Change Current from 100 A to 1 A.
- Right Click on **Parameters>Assign>Matrix...**
- Check **CurrentTop and CurrentBottom**

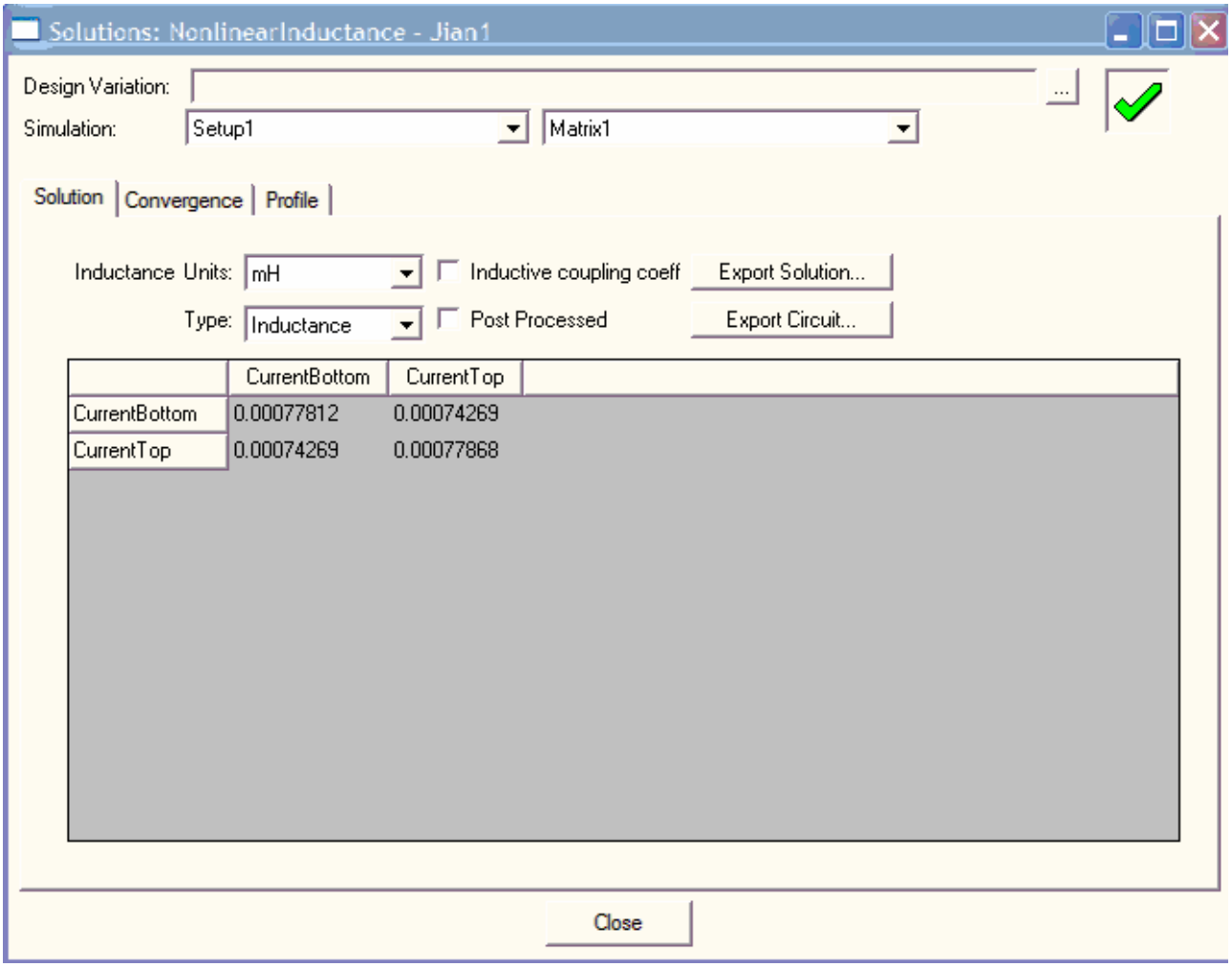


WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Nonlinear Inductor

Run the new project and Observe the Inductance Matrix

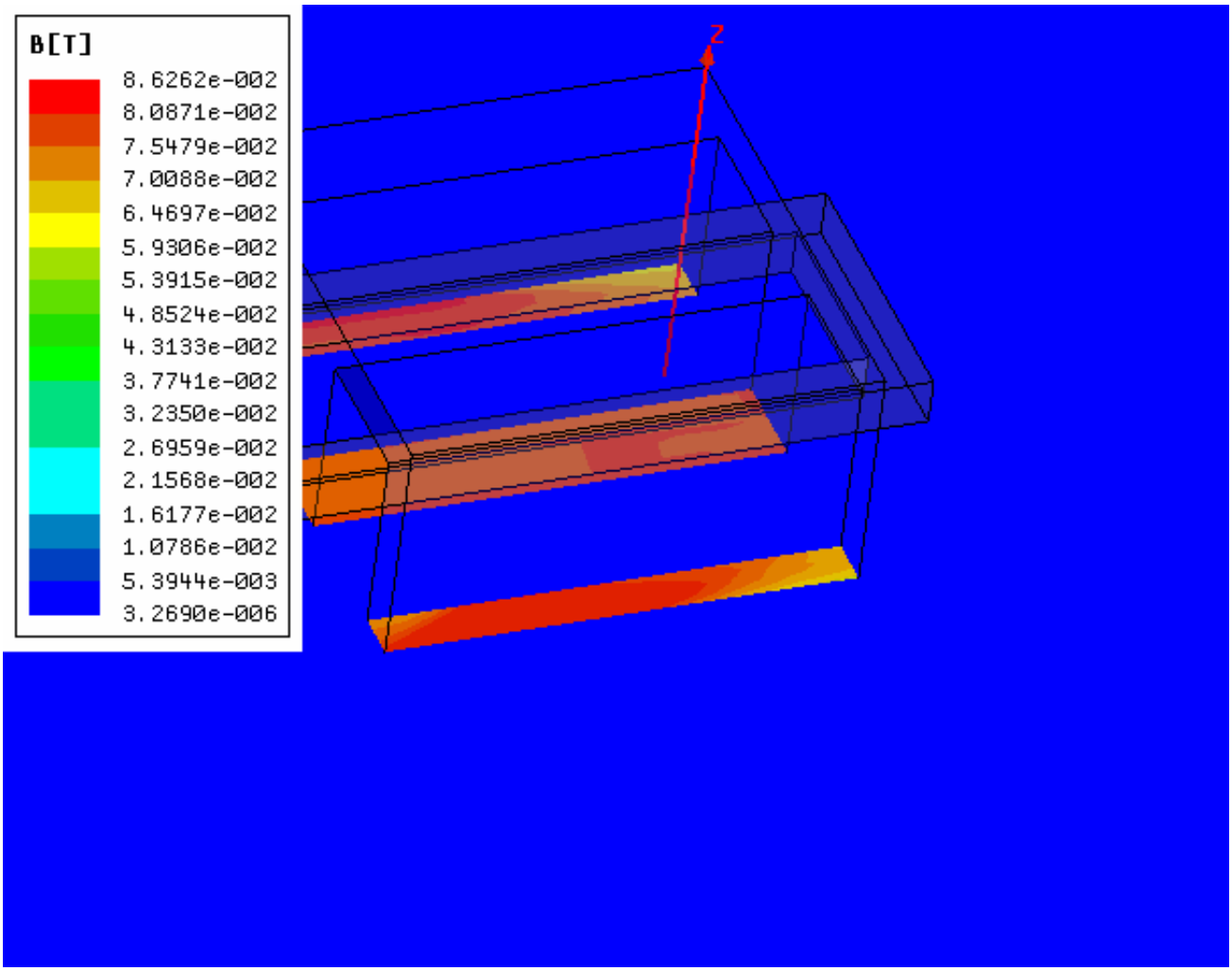
- Click on ***Analysis>Setup1***
- Right Click on ***Setup1: Analyze.***
- The simulation run and stop.
- ***Maxwell > Results> Solution Data…***
- We can see the inductance is much higher than the previous simulation.



WWW.MOHANDESYAR.COM

Plot Flux Density on Core Cross section

- Select the plot plane: **Planes>RelativeCS1:XY**
- Right mouse click: ***Field Overlays > Fields > B> Mag _B.***
- From the plot, we can see the flux density on core is around **0.086** Telsla. It is working on the linear area of Steel_1010.



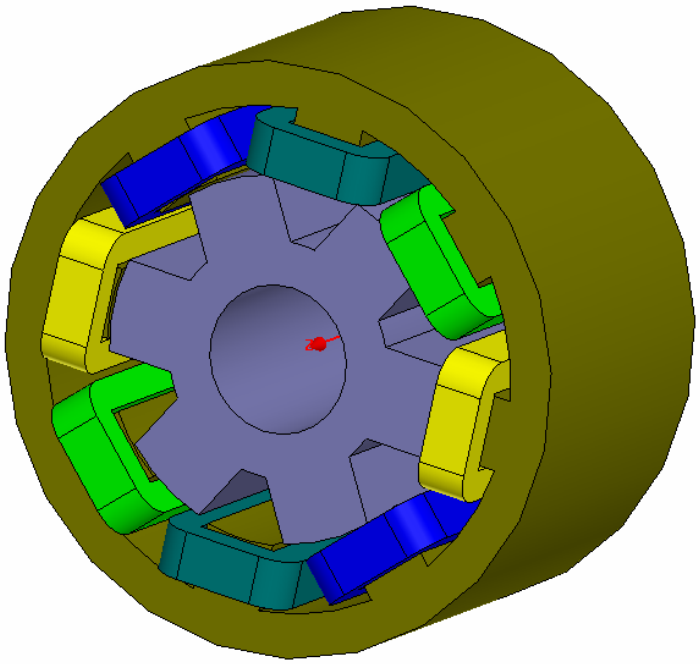
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Switched Reluctance Motor

- This example is intended to show you how to create and analyze a magnetostatic problem with stranded conductors on a Switched Reluctance Motor geometry using Magnetostatic solver in the Ansoft Maxwell 3D Design Environment.
- Within the Maxwell 3D Design Environment, solid coils can be modeled as Stranded Conductors. There are many advantages to using Stranded Conductors when modeling coils that have multiple turns. The first obvious advantage is that a coil with multiple wires, say 2500, can be modeled as a single object as opposed to modeling each wire which would be impracticable. Defining a Stranded Conductor means that the current density will be uniform throughout the cross section of the conductor.
- The example that will be used to demonstrate how Stranded Conductors are implemented in a switched Reluctance Motor. This switched reluctance motor will have four phases and two coils per phase..

- **Eddy Current Solver:** There is no fundamental difference between how stranded conductors are treated in the Eddy Current solver as compared to the Magnetostatic Solver. When using the Eddy Current Solver and specifying Stranded for the terminal excitation the Eddy Effect is automatically turned off.
- **Transient Solver:** See **Example 7.1** for setup instructions for this design.



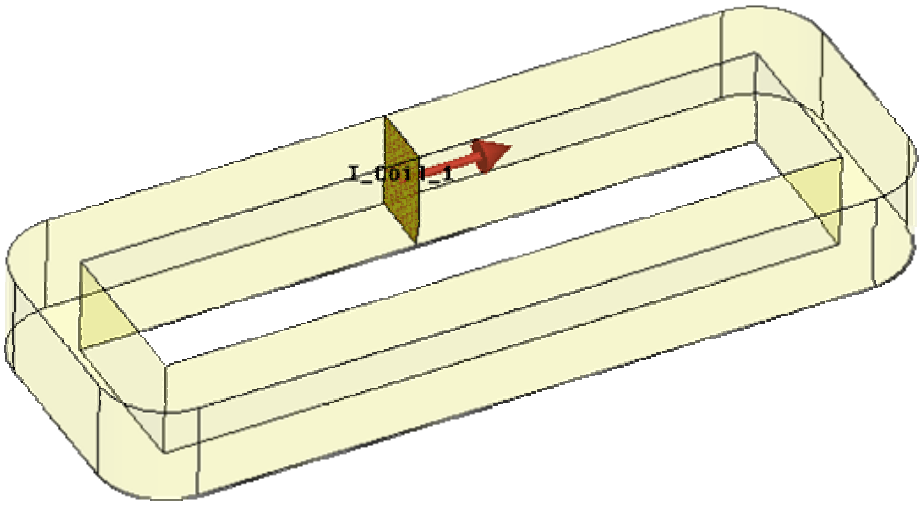
WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Theory - Magnetostatic Solver

When using the Magnetostatic Solver, with a Stranded Current Source, the solver assumes the following conditions:

1. The current density is uniformly distributed over the cross section of the terminal
2. The direction of the current is indicated by the arrow as seen on the terminal. Change the direction of the current by clicking on ***Swap Direction*** in the Current Excitation window to reverse the flow.



3. The solver does not know how many turns are represented by the coil, thus the value of current that is being applied represents the total ampere-turns. For this example, the value 3570 could represent any of the following:
    1. 25 amps at 150 turns
    2. 5 amps at 750 turns
    3. 15 amps at 250 turns

    or any such combination that produces a value of 3750 ampere-turns. The ration of amp to turns does not impact the magnetics, although it does impact the inductance. Please refer to the overview for details of the inductance calculation.

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Theory - Magnetostatic Solver (Continued)

- The ampere-turn value used to define the current source is used to calculate the current density which is the initial condition used by the solver according to the equation:

$$\vec{J} = \frac{I}{S}\hat{n}$$

Where: I is the total current in ampere-turns
S is the area of the terminal in m^2
$\hat{n}$ is the unit normal direction
$\vec{J}$ is the Current Density vector in A/m^2

- The path of the current is determined by the conduction path. Select the menu item ***Maxwell > Excitations > Conduction Paths*** to view the conduction path, and verify that the conduction path is correct. In this example, if any of the coils touched the stator, then this would constitute a separate conduction path since the material properties of the stator has a conductivity value 2e6 siemens/meter; to solve this problem an insulation boundary will need to be applied to the coils or stator; please refer to the overview section for details on insulation boundaries.
- Although the Magnetostatic solver uses the current density vector **J** as the initial condition, it does not solve for **J** directly as part of the output solution matrix. The quantity that the Magnetostatic solver directly calculates is the magnetic field intensity **H**. From **H**, the current density **J** in any conductor is derived by:

$$\vec{J} = \nabla \times \vec{H}$$

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - User Defined Primitives (UDP): **Switched Reluctance Motor**
    - Primitives: **Regular Polyhedron**
    - Surface Operations: **Section**
    - Boolean Operations: **Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Field Calculator**
  - Field Overlays:
    - **Magnitude B**

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

Setting Tool Options

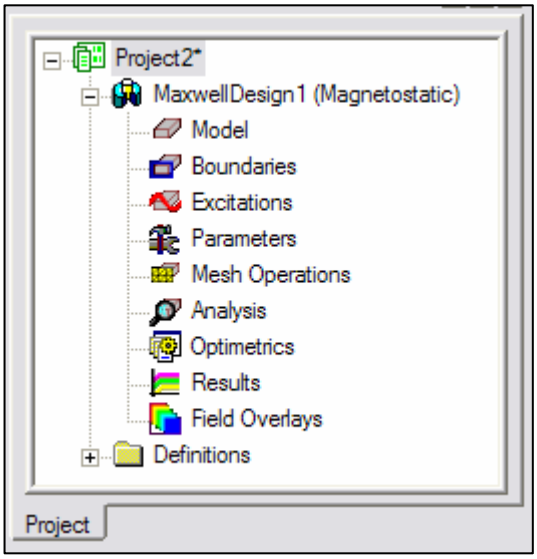
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

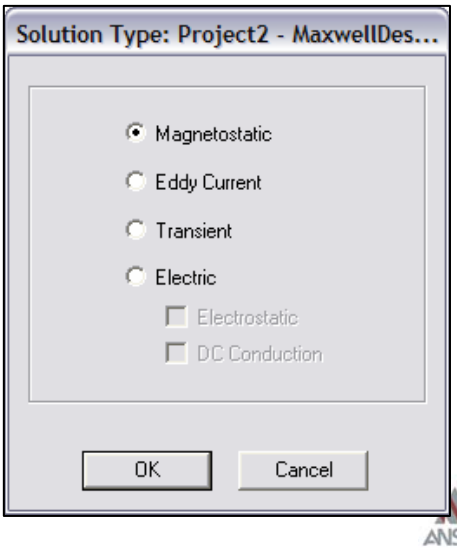
Opening a New Project

- **To open a new project:**
  - In an Ansoft Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - From the *Project* menu, select *Insert Maxwell Design*.



Set Solution Type

- **To set the solution type:**
  - Select the menu item *Maxwell > Solution Type*
  - Solution Type Window:
    - Choose **Magnetostatic**
    - Click the **OK** button



WWW.MOHANDESYAR.COM

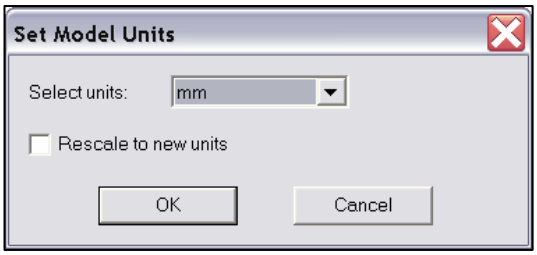
Example (Magnetostatic) - Switched Reluctance Motor

Creating the 3D Model

Set Model Units

To set the units:

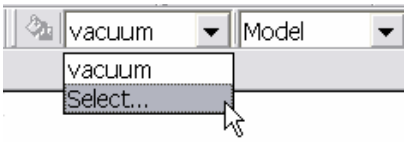
1. Select the menu item *3D Modeler > Units*
2. Set Model Units:
   1. Select Units: **mm**
   2. Click the **OK** button

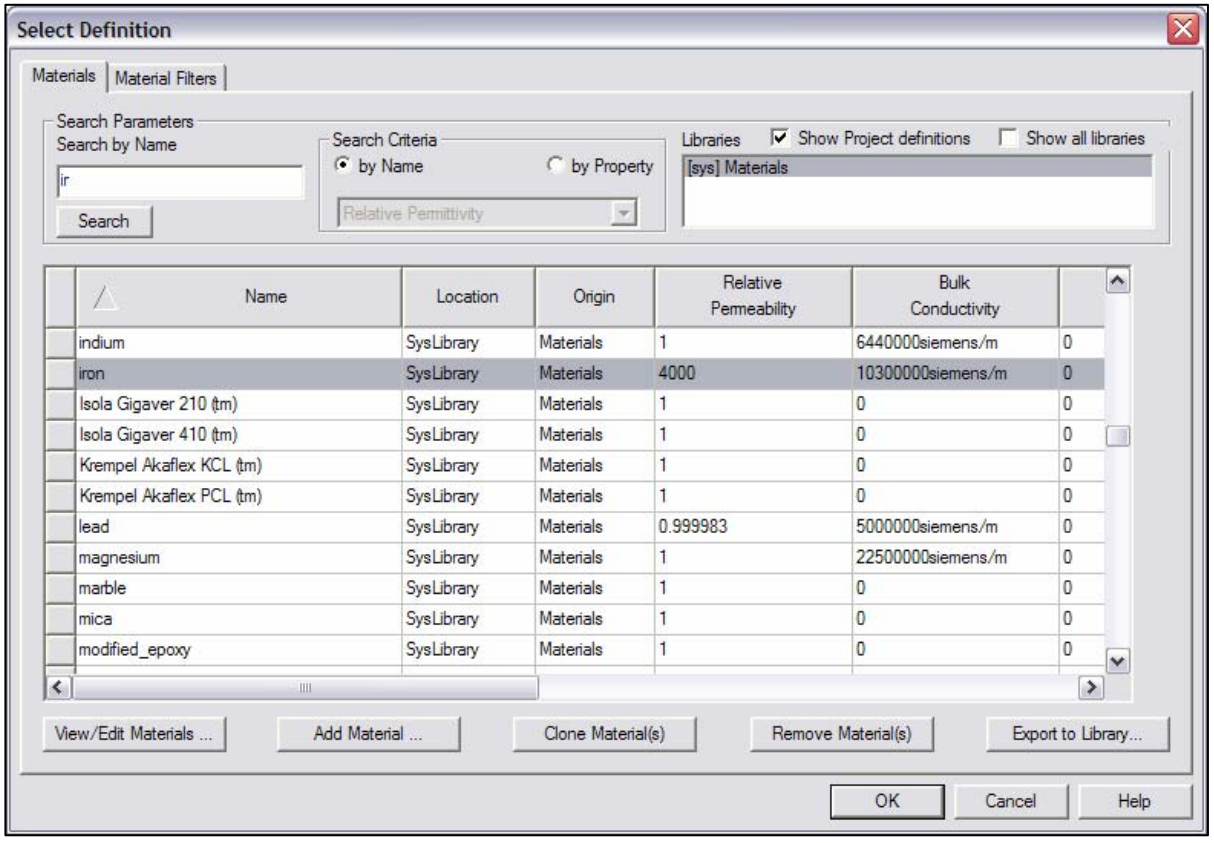


Set Default Material

To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **iron** in the **Search by Name** field
   2. Click the **OK** button





WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Create Rotor

To create the rotor:

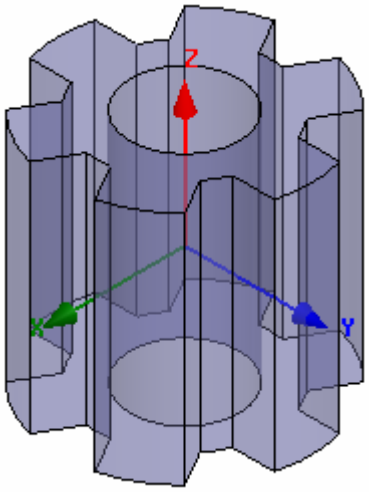
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
    1. For the value of **DiaGap,** type: **70**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **30**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **65**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **6**, Click the **Tab** key to accept
    5. For the value of **ThkYoke**, type: **9**, Click the **Tab** key to accept
    6. For the value of **Embrace**, type: **0.5**, Click the **Tab** key to accept
    7. For the value of **EndExt**, type: **0**, Click the **Tab** key to accept
    8. For the value of **InfoCore**, type: **0**, Click the **Tab** key to accept
    9. Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Rotor**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View*.



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Create Stator and Coils:

To create the Stator and Coils

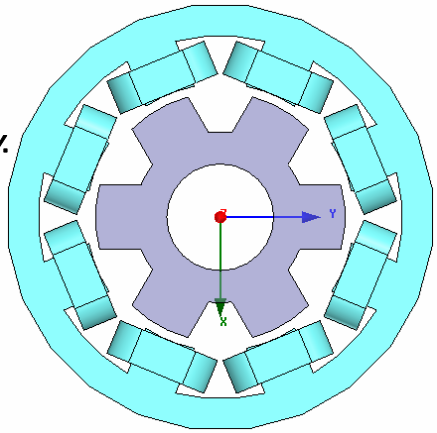
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
    1. For the value of **DiaGap,** type: **75**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **120**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **65**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **8**, Click the **Tab** key to accept
    5. For the value of **ThkYoke**, type: **9**, Click the **Tab** key to accept
    6. For the value of **Embrace**, type: **0.5**, Click the **Tab** key to accept
    7. For the value of **EndExt**, type: **1**, Click the **Tab** key to accept
    8. For the value of **InfoCore**, type: **1**, Click the **Tab** key to accept
    9. Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Stator**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View.*



Ungroup the Stator and Coils

To separate bodies

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Stator**
    2. Click the **OK** button
3. Select the menu item, *3D Modeler > Boolean > Separate Bodies*

Process the Coils:

Select Coils

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Stator_2, Stator_3, Stator_4, Stator_5, Stator_6, Stator_7, Stator_8**
    2. Click the **OK** button

Delete Coils

1. Select the menu item *Edit > Delete*

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Create Terminals

- **Section Coils**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Stator_1**
     2. Click the **OK** button
- **Section Object**
  1. Select the menu item *Edit > Surface > Section*
     1. Section Plane: *XY*
     2. Click the **OK** button
- **Separate Bodies**
  1. Select the menu item *Edit > Boolean > Separate Bodies*
- **Delete**
  1. Select the menu item *Edit > Delete*
- **Section Coils**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Section1**
     2. Click the **OK** button
- **Assign Excitation**
  1. Select the menu item *Maxwell > Excitations > Assign > Current*
  2. Current Excitation : General
     1. Name: **Current1**
     2. Value: **3750 A**
     3. Type: **Stranded**
  3. Click the **OK** button

Verify Duplicate Boundary Option is Set

- **Section To set the tool options:**
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Change Properties

Change Properties

1. Select the menu item *Maxwell > List*
2. Design List Window
    1. From the list, select row: **Stator_1**
    2. Click the **Properties** button
    3. Properties Window
        1. Name: **Coil**
        2. Material: **copper**
        3. Click the **OK** button
    4. From the list, select row: **Section1**
    5. Click the **Properties** button
    6. Properties Window
        1. Name: **Terminal**
        2. Click the **OK** button
    7. Click the **Done** button

Duplicate Coil and Terminal

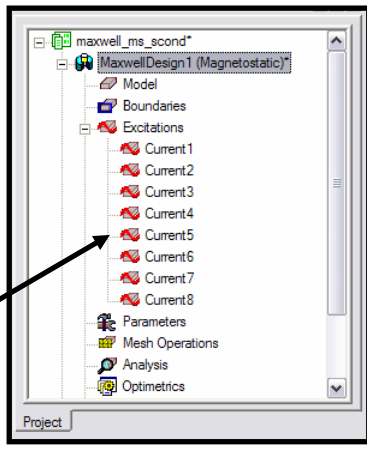
To select objects

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Coil, Terminal**
    2. Click the **OK** button

To duplicate the objects:

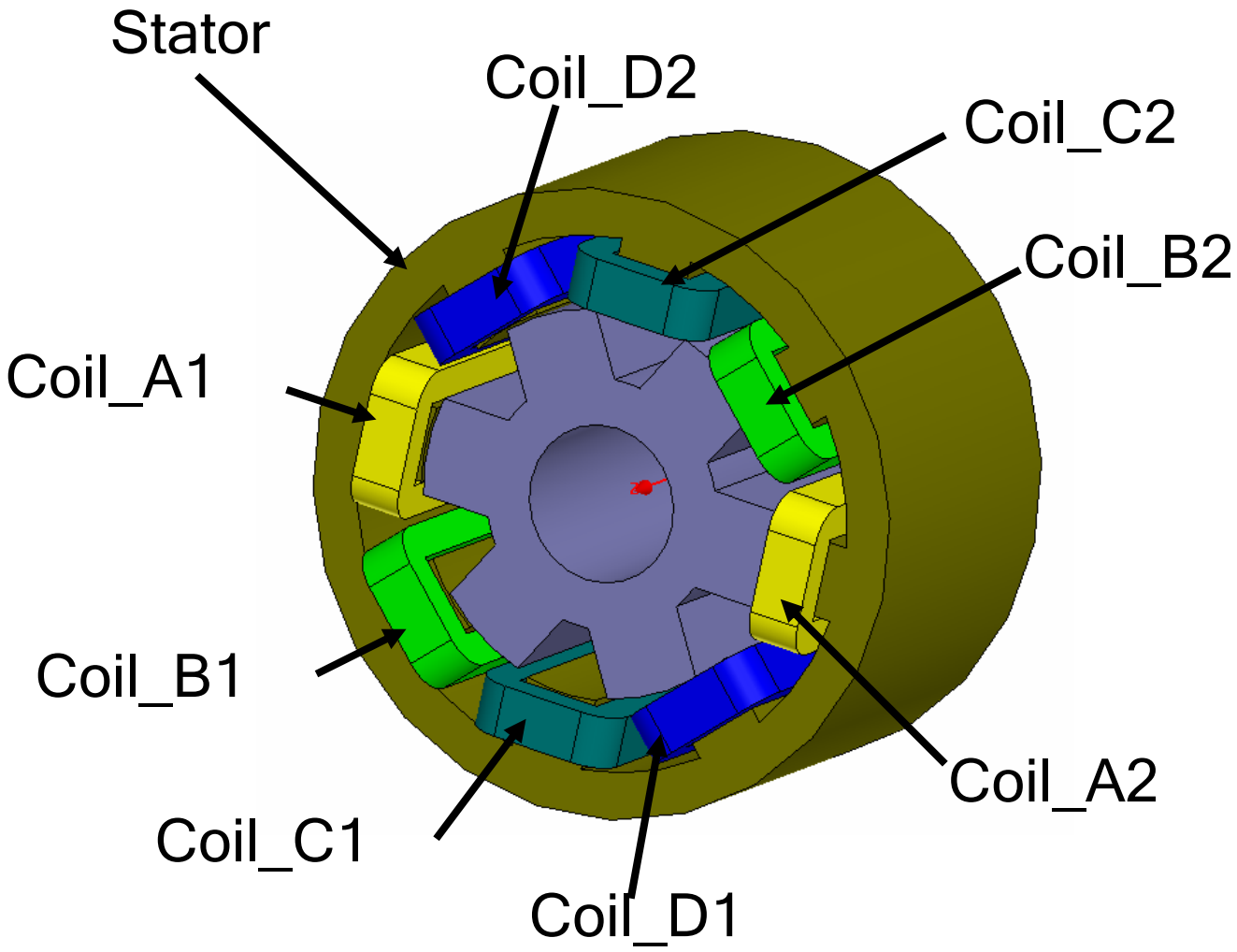
1. Select the menu item, *Edit > Duplicate > Around Axis*
2. Duplicate Around Axis Window
    1. Axis: **Z**
    2. Angle: **45deg**
    3. Total Number: **8**
    4. Click the **OK** button
3. Click the **OK** button

Project Tree: Eight Excitations



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Set Default Material

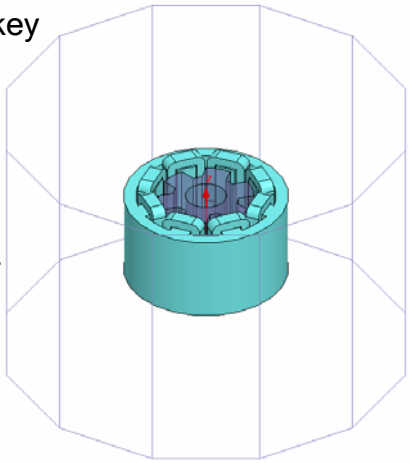
To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **vacuum**

Create Region

To create the region:

1. Select the menu item *Draw > Regular Polyhedron*
2. Using the coordinate entry fields, enter the center position
   - X: **0.0**, Y: **0.0**, Z: **-100.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the radius:
   - dX: **150.0** dY: **0.0**, dZ: **0.0**, Press the **Enter** key
4. Using the coordinate entry fields, enter the height:
   - dX: **0.0**, dY: **0.0**, dZ: **200.0**, Press the **Enter** key
   - Segment Number Window
     - Number of Segments: **12**
     - Click the **OK** button



To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Region**
3. **Display Wireframe**: ☑ **Checked**
4. Click the **OK** button

To turn off the visibility:

1. Select the menu item *View > Hide Selection > Active View*

To fit the view:

1. Select the menu item *View > Fit All > Active View*.

WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Analysis Setup

Creating an Analysis Setup

- **To create an analysis setup:**
  1. Select the menu item *Maxwell > Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **General** tab:
        - Percent Error: **10**
     2. Click the **Convergence** tab:
        - Refinement Per Pass: **10 %**
     3. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item *File > Save As*.
  2. From the **Save As** window, type the Filename: **maxwell_ms_relmotor**
  3. Click the **Save** button

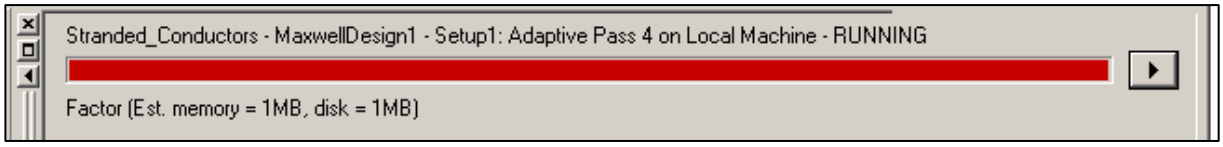
Analyze

Model Validation

- **To validate the model:**
  1. Select the menu item *Maxwell > Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item *Maxwell > Analyze All*



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Solution Data

To view the Solution Data:

1. Select the menu item *Maxwell > Results > Solution Data*

To view the Profile:

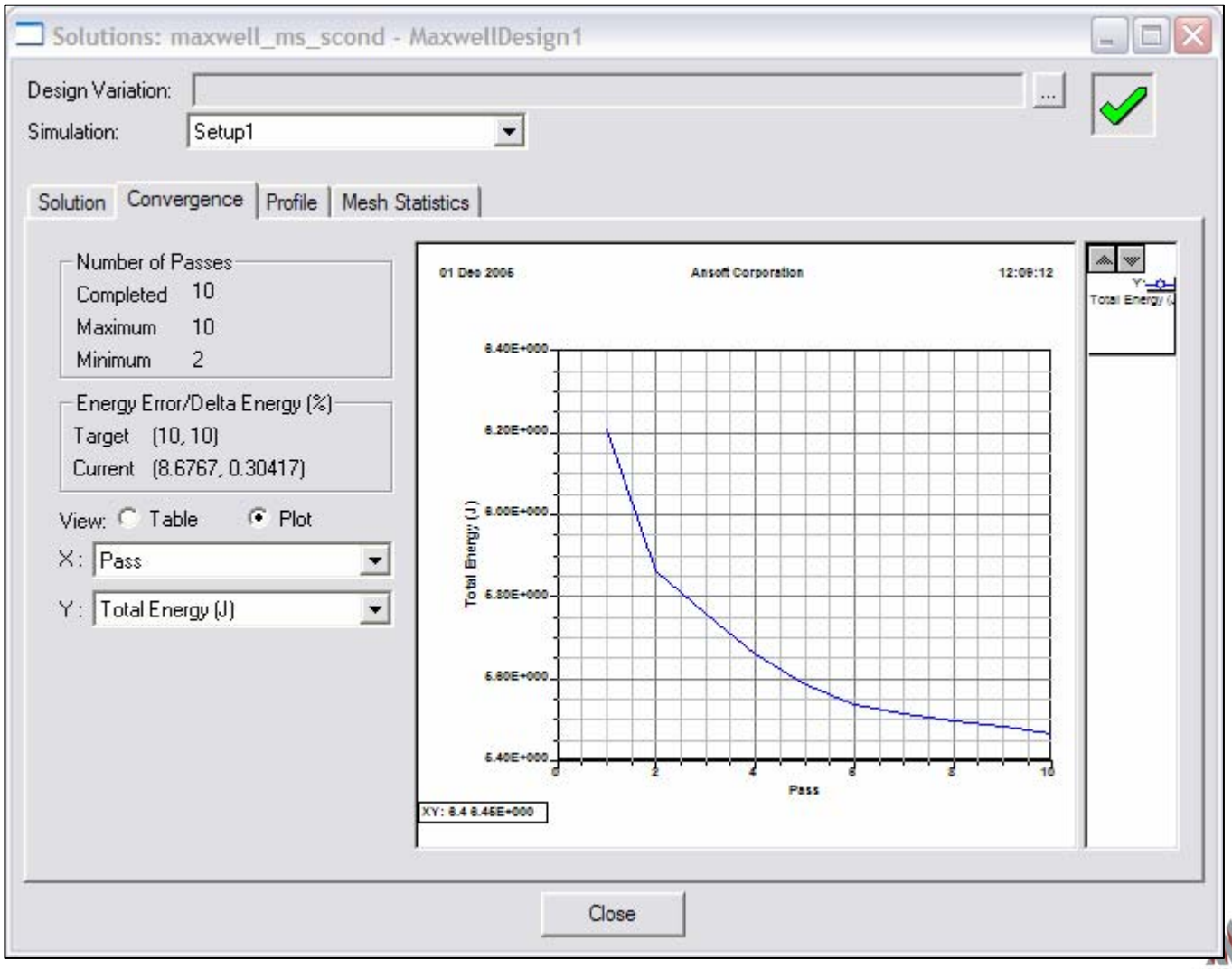
1. Click the **Profile** Tab.

To view the Convergence:

1. Click the **Convergence** Tab

Note: The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

2. Click the **Close** button



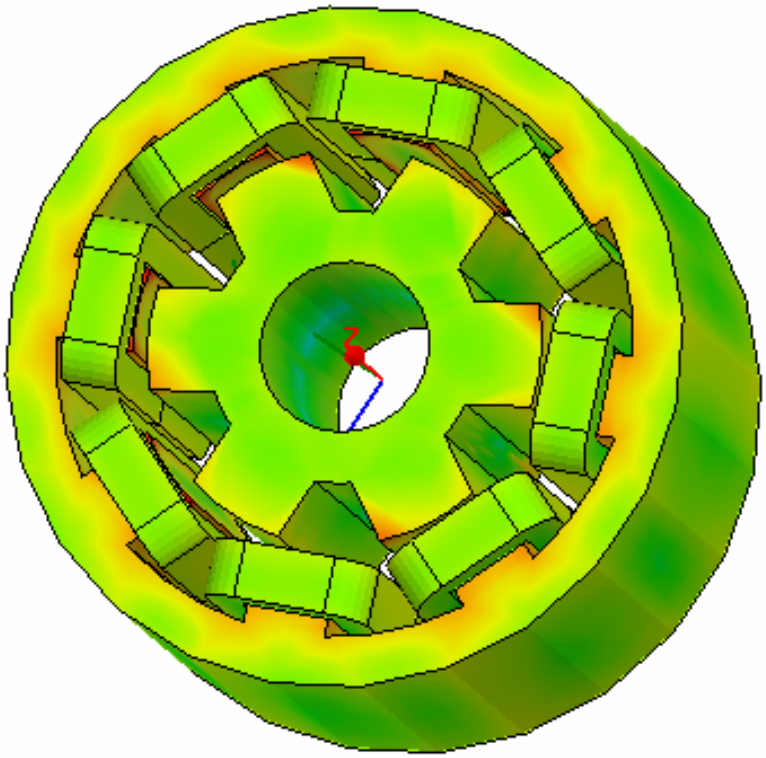
WWW.MOHANDESYAR.COM

Field Overlays

Create Field Overlay

To create a field plot:

1. Select **stator** and **rotor**
    1. Select the menu item *Edit > Select > By Name*
    2. Press and hold the **CTRL** key and click on **stator** and **rotor**
2. Select the menu item *Maxwell > Fields > Fields > B > Mag_B*
3. Create Field Plot Window
    1. Solution: **Setup1 : LastAdaptive**
    2. Quantity: **Mag_B**
    3. In Volume: **All**
    4. Plot on Surface Only: ☑ **Checked**
    5. Click the **Done** button



WWW.MOHANDESYAR.COM

Example (Magnetostatic) - Switched Reluctance Motor

Calculate Current

To use the field calculator

1. Select the menu item **Maxwell > Fields > Calculator**
2. Select **Quantity: J**
3. Select **Vector**: **Scal?** > **Scalar Z**

Vec : <Jx,Jy,Jz>

Scl : ScalarZ(<Jx,Jy,Jz>)

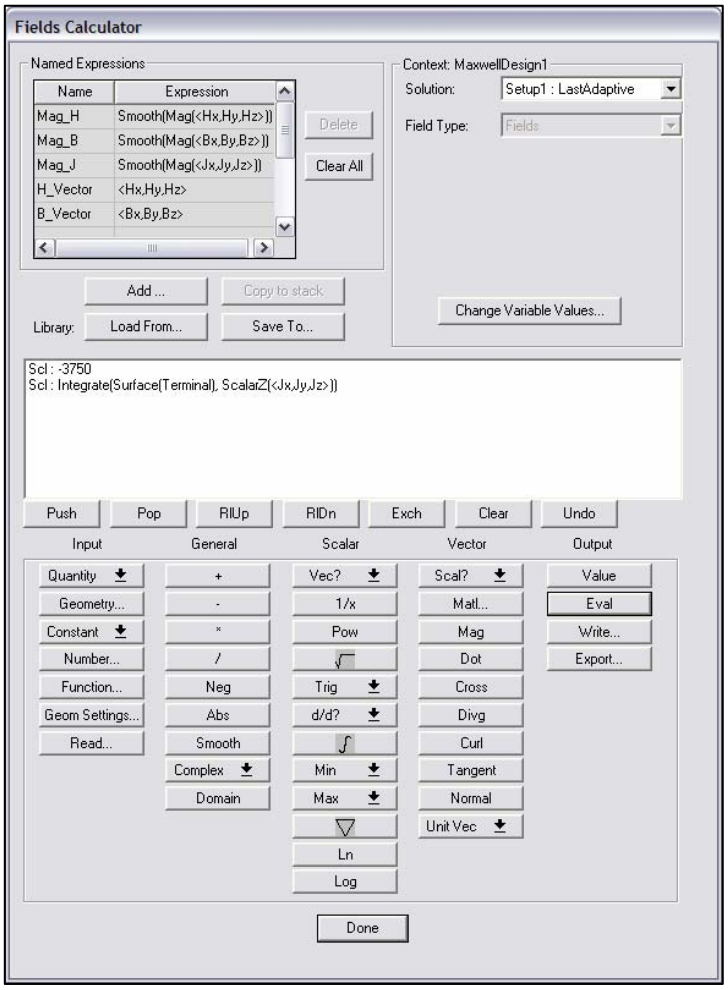
4. Click the button **Geometry**
5. Geometry Window
    1. Select the radio button **Surface**
    2. From the list select: **Terminal**
    3. Click the **OK** button

Srf : Surface(Terminal)
Scl : ScalarZ(<Jx,Jy,Jz>)

6. Click the ∫ (Integrate) button
7. Click the **Eval** button

Scl : -3750
Scl : Integrate(Surface(Terminal), ScalarZ(<Jx,Jy,Jz>))

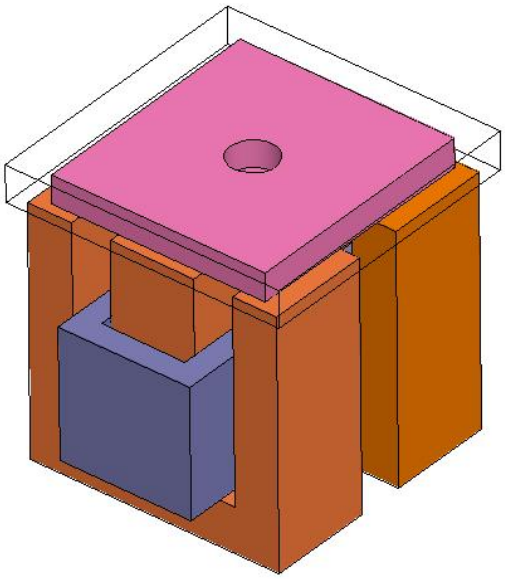
8. Click the **Done** button to close the calculator



WWW.MOHANDESYAR.COM

Realization of an equivalent circuit extraction (ECE) of a problem with linear movement.

- The Electro Mechanical software package provided by Ansoft enables system simulation as well as component simulation. Often, the results of an in depth study of a magnetic component are needed in a top level, system analysis.
- With Maxwell, the results of a parametric analysis can be used to generated an Equivalent Circuit that will be used into Simplorer. This Equivalent Circuit is transmitted under the format of a Look up table containing the sweep variables as well as the output variables.
- This application note presents the extraction of an equivalent circuit of a Linear Actuator. We will vary the air gap of the Armature (with the stators) and the Input current in the Coil. The outputs will be the Force and the inductance of the Coil. Our component in Simplorer will all 4 Terminals :
  - 2 Electrical Terminals with the current and the EMF (the Inductance of the Coil) as Through and Across quantities.
  - 2 Mechanical Terminals with the Force and the air gap as Through and Across quantities.



WWW.MOHANDESYAR.COM

Topic - ECE: Linear movement

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic:
  - 3D Solid Modeling
    - Primitives: **Box, Cylinder, Lines**
    - Surface Operations: **Section, Sweep Along Vector, Sweep along Path**
    - Boolean Operations: **Separate Bodies, Subtract, Duplicate (Mirror)**
  - Design Properties
    - Parameters: **Design Parameters**
  - Boundaries/Excitations
    - Current: **Stranded**
    - Boundary: **Insulating**
  - Mesh Operations
    - Volume: **Length Based**
  - Executive Parameters
    - Force: **Virtual Force**
    - Matrix: **Inductance**
  - Output variables
    - From Executive Parameters
  - Analysis
    - Magnetostatic
  - Optimetrics Analysis
    - Parametric Sweep
    - Distributive Analysis
  - Results
    - Solutions Data
  - Equivalent Circuit Extraction
    - From Parametric: **Linear Movement**

WWW.MOHANDESYAR.COM

Overview of the Study:

- Getting Started
  - Launching Maxwell
  - Setting Tool Options
  - Opening a new project
  - Set Solution Type

- Creating the 3D Model of a Solenoid
  - Set Model Units
  - Create the Armature
  - Create the Stators
  - Create the Coil
  - Create the Band
  - Create the Coil Terminal
  - Separate the Grouped Terminals
  - Create the Problem Region
  - Assign Current Source to the Terminal
  - Assign Boundaries Conditions
  - Assign Executive Parameters
  - Assign Mesh Operations

- Solve the Nominal Problem
  - Create an Analysis Set up
  - Save the Project
  - Check the Validity of the Model
  - Analyze
  - Solution Data of the Nominal Problem

WWW.MOHANDESYAR.COM

Overview of the Study (Continued)

- Create Parametric Analysis
  - Assign Problem Parameters
  - Create Output Variables
  - Create Parametric Analysis Setup
- Solve the Parametric Sweep
  - Solution Data of the Parametric Sweep
- ECE from the Parametric Analysis
- Create the circuit in Simplorer
  - Create the Schematic sheet in Simplorer
  - Create the graphical Displays in Simplorer
  - Create a Transient Analysis
  - Run the Analysis

WWW.MOHANDESYAR.COM

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

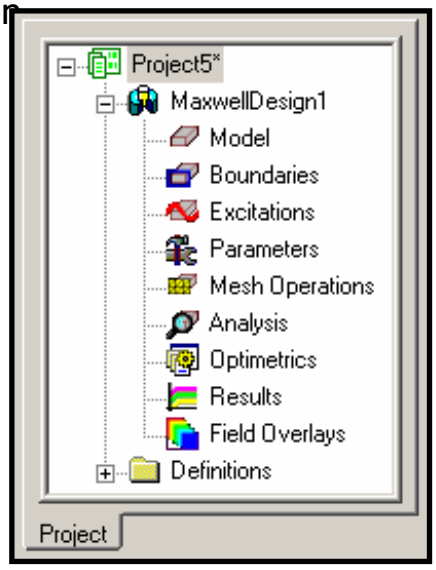
Setting Tool Options

- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Opening a New Project

- **To open a new project:**
  1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  2. Select the menu item *Project > Insert Maxwell Design*, or click on the icon



Set Solution Type

- Select the menu item: *Maxwell > Solution Type > Magnetostatic*, or right mouse click on MaxwellDesign1 and select *Solution Type …*

Creating the 3D Model of a solenoid

- The example that will be used to demonstrate how to generate an equivalent circuit extraction with Maxwell. Then the importation into SIMPLORER of the circuit will be described

Set Model Units

- Select the menu item *3D Modeler > Units >* Select Units*: in (inches)*

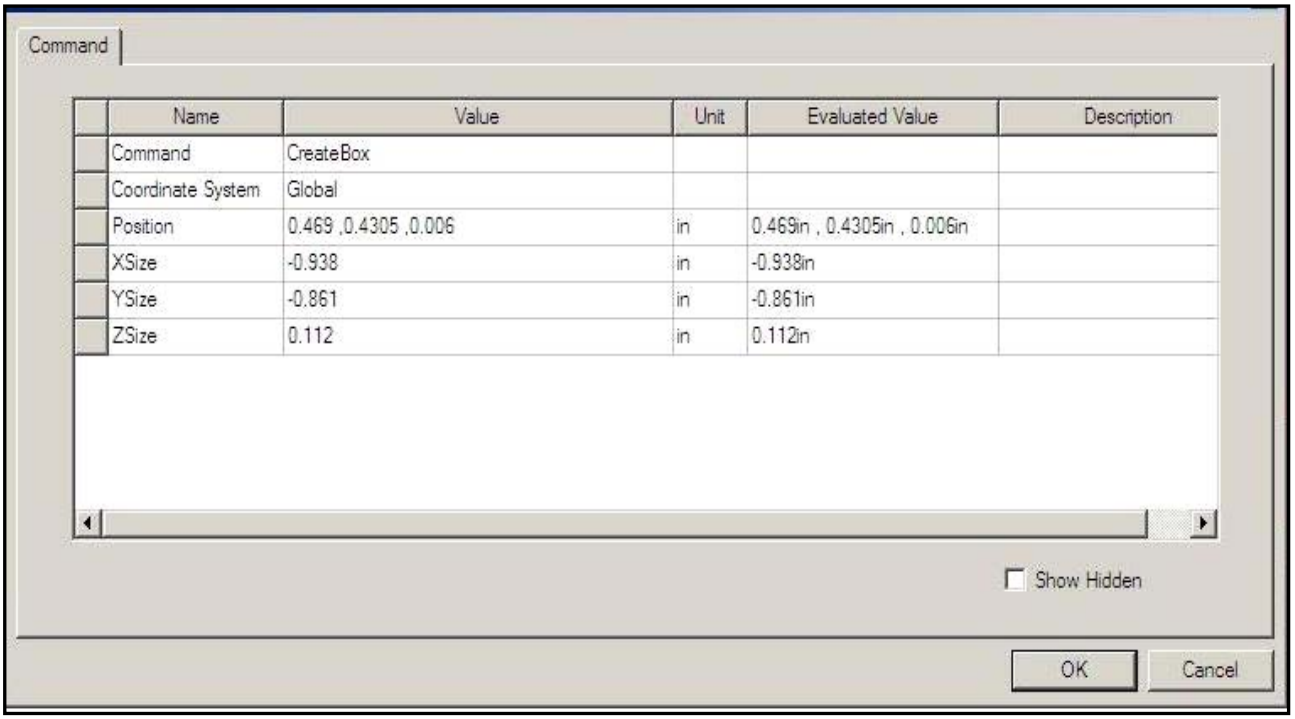
WWW.MOHANDESYAR.COM

Create the Armature:

- The armature is made of a box with a cylinder hole.
- Create the armature . Select the Menu item *Draw > Box* or click on the icon.

1. Using the coordinate entry field, enter the box position
   - **X: 0.469, Y: 0.4305, Z: 0.006**, Press the **Enter** key
2. Using the coordinate entry field, enter the relative size of the box
   - **dX: -0.938, dY: -0.861, dZ: 0.112**, Press the **Enter** key

- **Note**: To create the box with the GUI, use the mouse to draw the box.



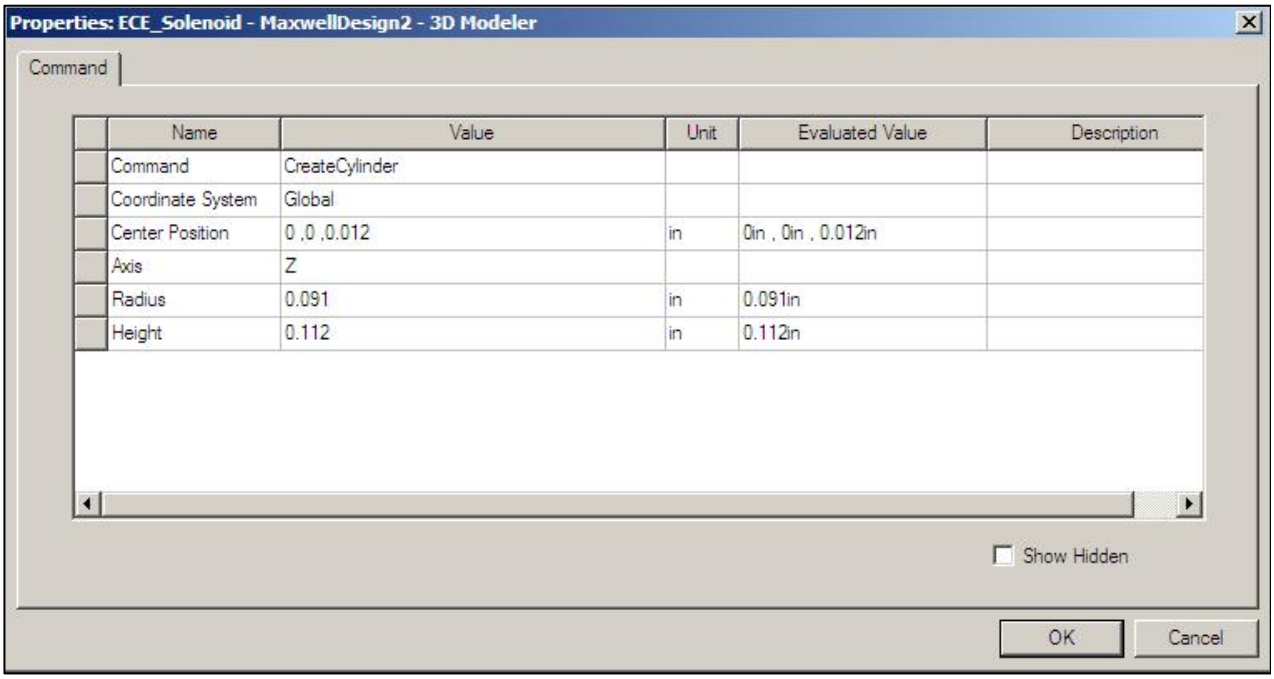
- **Note**: In the case of a typo, the modification of the coordinates can be made within the box.
- Click on the created object in the panel on the left representing the object tree. Change its name from *Box1* to *Armature*.
- Change the Material from **vacuum** to **Steel_1010**.

WWW.MOHANDESYAR.COM

- Create the hole *Draw > Cylinder* or click on the icon.

  1. Using the coordinate entry field, enter the center of base
     - **X: 0.0, Y: 0.0, Z: 0.012**, Press the **Enter** key
  2. Using the coordinate entry field, enter the coordinates of a external point of the cylinder base
     - **X: 0.091, Y: 0.0, Z:0.0**, Press the **Enter** key
  3. Using the coordinate entry field, enter the height
     - **X: 0.0, Y: 0.0, Z: 0.112**, Press the **Enter** key

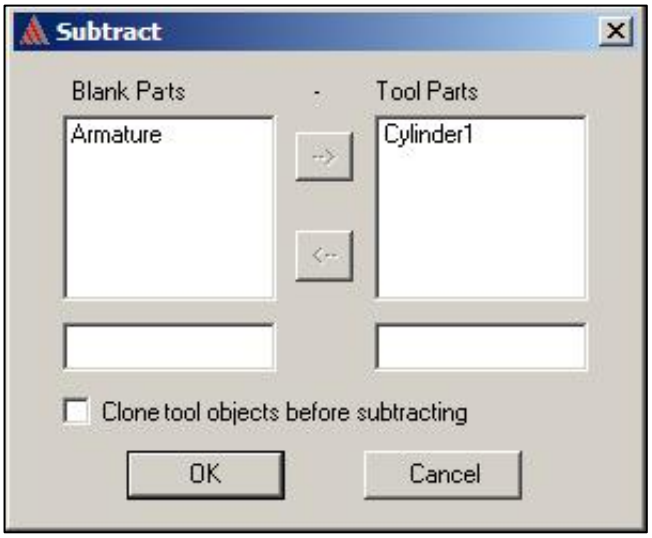
- **Note**: To create the cylinder graphically, use the mouse to draw the cylinder.



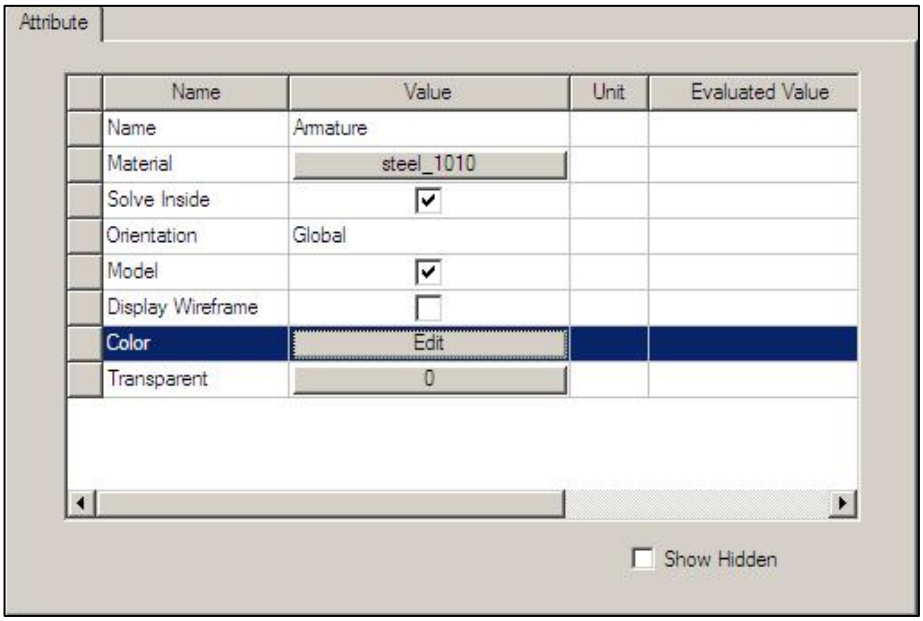
- Select the objects *Armature* and *Cylinder1* from the objects tree (maintain the Ctrl key to select both objects).

WWW.MOHANDESYAR.COM

- Select the menu item *3D Modeler > Boolean > Subtract* or click on the icon.



- Make sure to have *Armature* in the box Blank Parts and *Cylinder1* in the box Tool Parts. Use the arrows to exchange the objects if necessary.
- Change the Material from **vacuum** to **steel_1010**.
- Change the color to purple.



WWW.MOHANDESYAR.COM

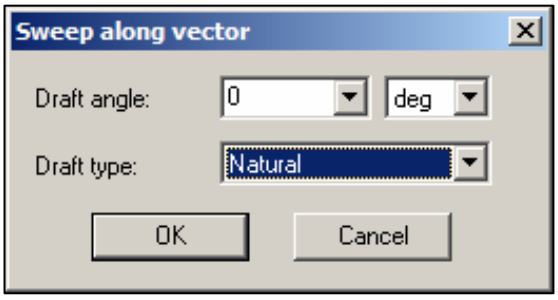
Create the Stators

- The two stators are created using a polyline.
- Create the polyline *Draw > Line* or click on the icon.
  1. Using the coordinate entry field, enter the first point of the line
     - **X: 0.5025, Y: 0.0, Z: 0.0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the second point of the line
     - **X: 0.5025, Y: 0.168, Z:0.0**, Press the **Enter** key
  3. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.168, Z:-0.803**, Press the **Enter** key
  4. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.324, Z:-0.803**, Press the **Enter** key
  5. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.324, Z:0.0**, Press the **Enter** key
  6. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.5, Z:0.0**, Press the **Enter** key
  7. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.5, Z:-0.98**, Press the **Enter** key
  8. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.5, Z:-0.98**, Press the **Enter** key
  9. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.5, Z:0.0**, Press the **Enter** key
  10. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.324, Z:0.0**, Press the **Enter** key
  11. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.168, Z:-0.803**, Press the **Enter** key
  12. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.168, Z:0.0**, Press the **Enter** key
  13. Using the coordinate entry field, enter the last point of the line
     - **X: 0.5025, Y: 0.0, Z:0.0**, Press the **Enter** key *twice*.
- Change the name of the polyline from *Polyline1* to *base* and leave the other default parameters.
- **Note**: If you happen to have an incorrect coordinate, you can modify any entry from the model object tree, **Lines > Base > CreatePolyline**.

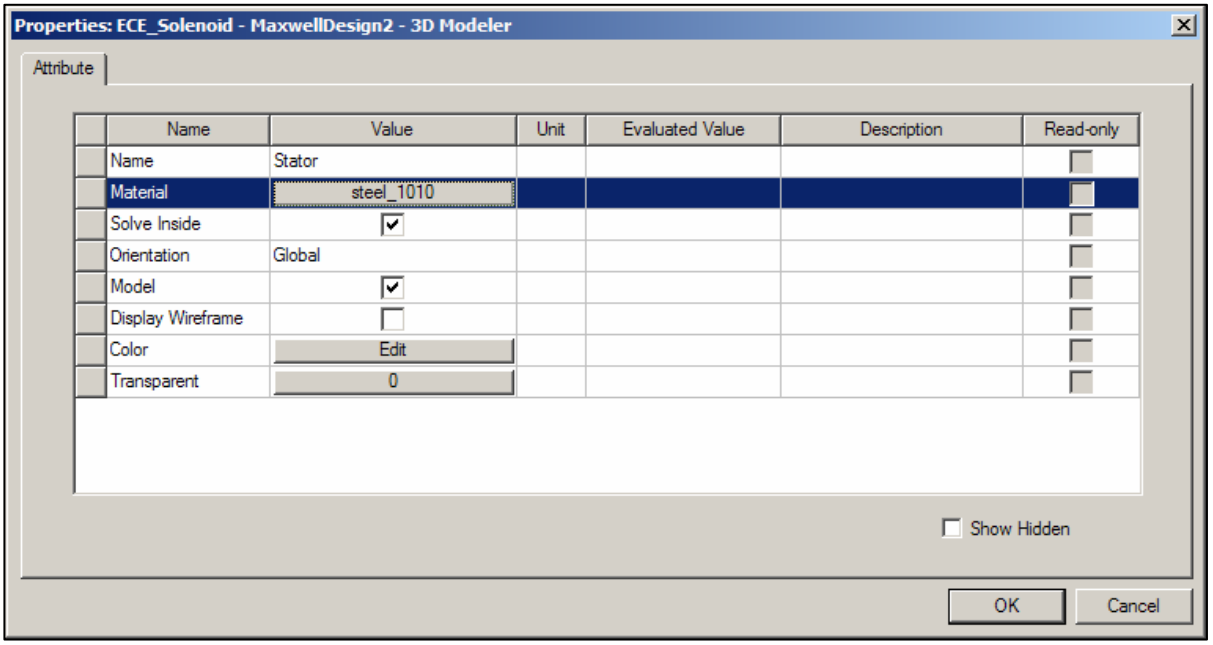
WWW.MOHANDESYAR.COM

Create the Stators (Continued)

- This base will be swept along a vector.
- Select the object ***base*** from the objects tree.
- Select the menu item ***Draw > Sweep > Along vector***
  1. Using the coordinate entry field, enter the first point of the vector
     - **X: 0.5025, Y: 0.0, Z:0.0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the last point of the vector
     - **X: 0.091, Y: 0.0, Z:0.0**, Press the **Enter** key.



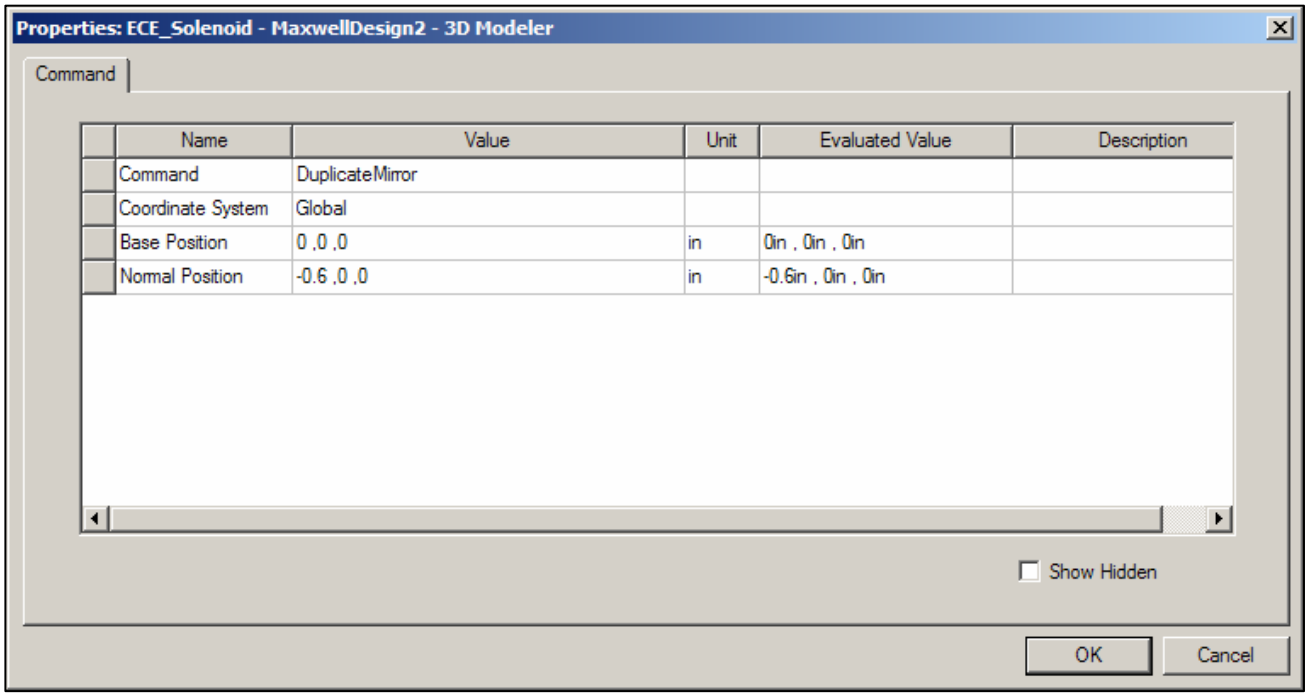
- Leave the Draft angle to 0 deg and change Draft type to Natural.



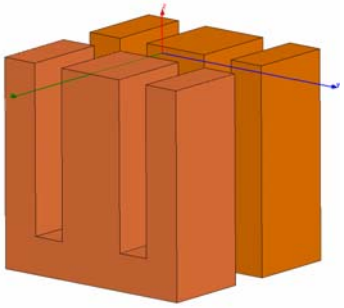
- Change the name of the object to ***stator***. Change the Material from **vacuum** to **steel_1010** and change the color to dark orange.

WWW.MOHANDESYAR.COM

- The second stator will be duplicated from ***Stator***.
- Select ***Stator*** from the objects tree.
- Select the menu item ***Edit > Duplicate > Mirror*** or click on the icon
  1. Using the coordinate entry field, enter the anchor point of mirror plane
     - **X: 0.0, Y: 0.0, Z:0.0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the target point of vector normal to the mirror plane
     - **X: -0.6, Y: 0.0, Z:0.0**, Press the **Enter** key



- After the operation, a window appears and recall the coordinates.
- A new object ***Stator_1*** has been created with the same material property and the same color as ***Stator***.



WWW.MOHANDESYAR.COM

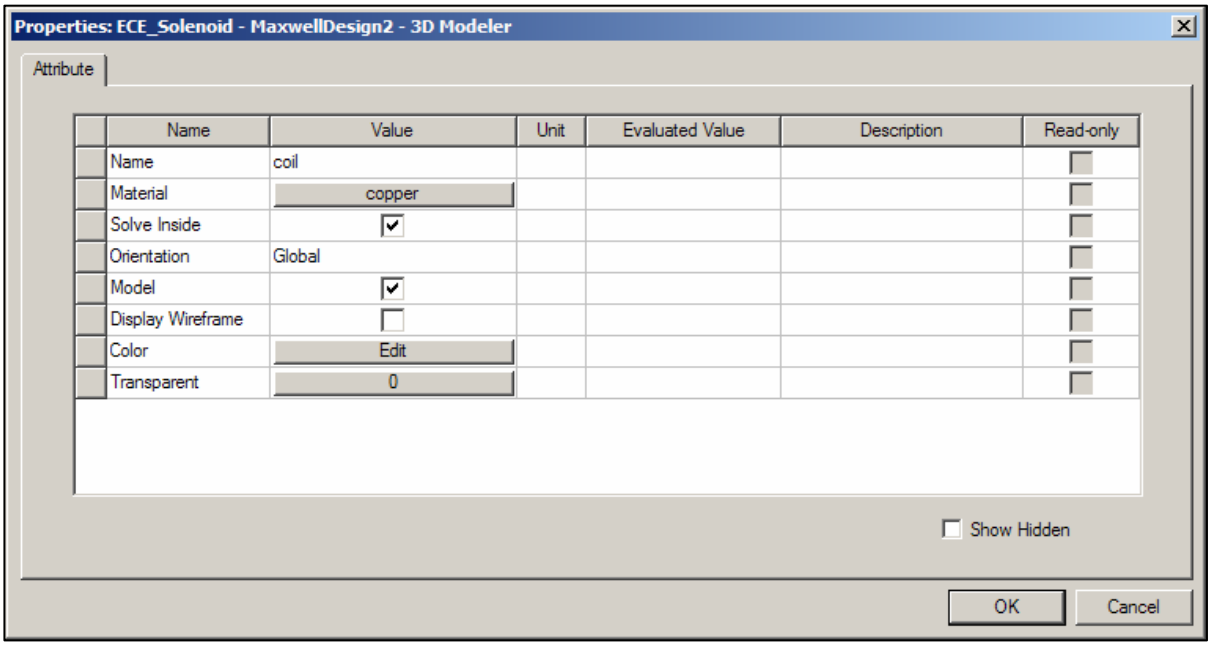
Create the Coil:

- The coil is created using a base surface and sweeping the surface along a path.
- Create the polyline ***Draw > Line*** or click on the icon.
  1. Using the coordinate entry field, enter the first point of the line
     - **X: 0.5025, Y: 0.168, Z: -0.803**, Press the **Enter** key
  2. Using the coordinate entry field, enter the second point of the line
     - **X: 0.5025, Y: 0.264, Z:-0.803**, Press the **Enter** key
  3. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.264, Z:-0.292**, Press the **Enter** key
  4. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: 0.168, Z:-0.292**, Press the **Enter** key
  5. Using the coordinate entry field, enter the last point of the line
     - **X: 0.5025, Y: 0.168, Z:-0.803**, Press the **Enter** key ***twice***
- Change the name of the polyline from ***Polyline1*** to ***Basecoil*** and leave the other default parameters.
- **Note**: If you happen to have an incorrect coordinate, you can modify any entry from the model's Lines tree, *Lines > Basecoil > CreatePolyline*.
- Create the path ***Draw > Line*** or click on the icon.
  1. Using the coordinate entry field, enter the first point of the line
     - **X: 0.5025, Y: 0.168, Z: -0.292**, Press the **Enter** key
  2. Using the coordinate entry field, enter the second point of the line
     - **X: -0.5025, Y: 0.168, Z: -0.292**, Press the **Enter** key
  3. Using the coordinate entry field, enter the next point of the line
     - **X: -0.55, Y: 0.168, Z: -0.292**, Press the **Enter** key
  4. Using the coordinate entry field, enter the next point of the line
     - **X: -0.55, Y: -0.168, Z: -0.292**, Press the **Enter** key
  5. Using the coordinate entry field, enter the next point of the line
     - **X: -0.5025, Y: -0.168, Z: -0.292**, Press the **Enter** key
  6. Using the coordinate entry field, enter the next point of the line
     - **X: 0.5025, Y: -0.168, Z: -0.292**, Press the **Enter** key
  7. Using the coordinate entry field, enter the next point of the line
     - **X: 0.55, Y: -0.168, Z: -0.292**, Press the **Enter** key
  8. Using the coordinate entry field, enter the next point of the line
     - **X: 0.55, Y: 0.168, Z: -0.292**, Press the **Enter** key
  9. Using the coordinate entry field, enter thenext point of the line
     - **X: 0.5025, Y: 0.168, Z:-0.292**, Press the **Enter** key

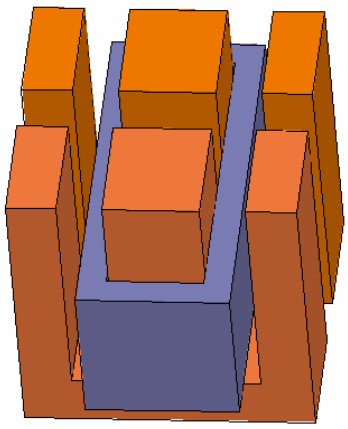
WWW.MOHANDESYAR.COM

10. Using the coordinate entry field, enter the last point of the line **X: 0.5025, Y: 0.168, Z:-0.803**, Press the **Enter** key *twice*

- Change the name of the polyline from *Polyline1* to *path* and leave the other default parameters.
- Select the objects *Basecoil* and *path* from the objects tree.
- Select the menu item *Draw > Sweep > Along Path.* Leave Twist Angle and Draft Angle to 0. Make sure Draft type is selected to Natural.



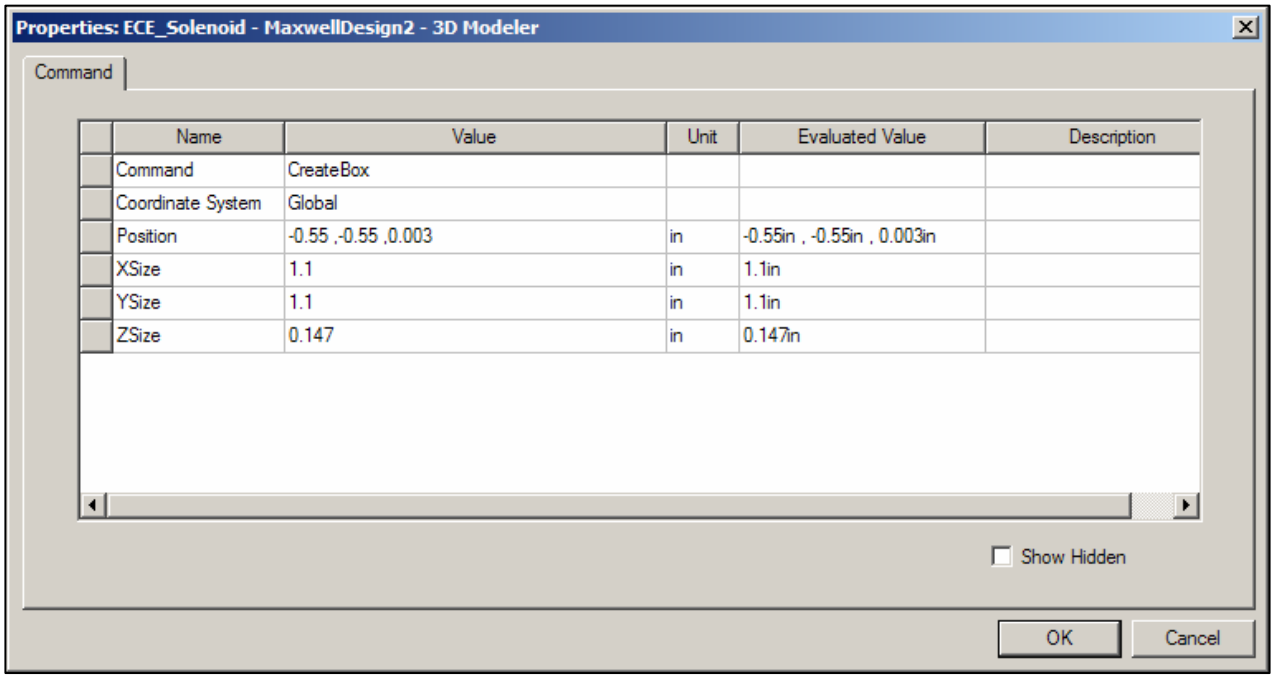
- Rename the object to *Coil* and assign **Copper** as material. Change the color in blue.



WWW.MOHANDESYAR.COM

Create the Band

- In our study, the *Armature* will be moved up and down along the Z axis. It is always a good idea to help the mesh creation by creating an air box object around a object that will be moved (even if the project will be performed with the Magnetostatic solver).
- Create the air object *Draw > Box* or click on the icon. Use the values below for the base coordinates and the size of the box.

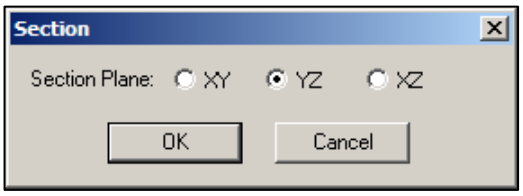


- Change the name of the object to *Band*. Also, change the Transparency from 0 to 1; this will keep the Region object displayed as a wire frame. Keep the material as vacuum.

WWW.MOHANDESYAR.COM

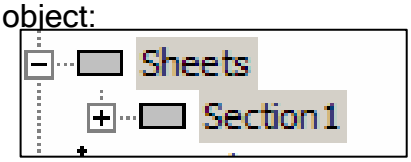
Create the Coil Terminal

- A coil terminal needs to be created in order to assign the current excitation.
- Select *Coil* from the objects tree.
- Select the menu item *3D Modeler > Surface > Section > YZ Plane .*

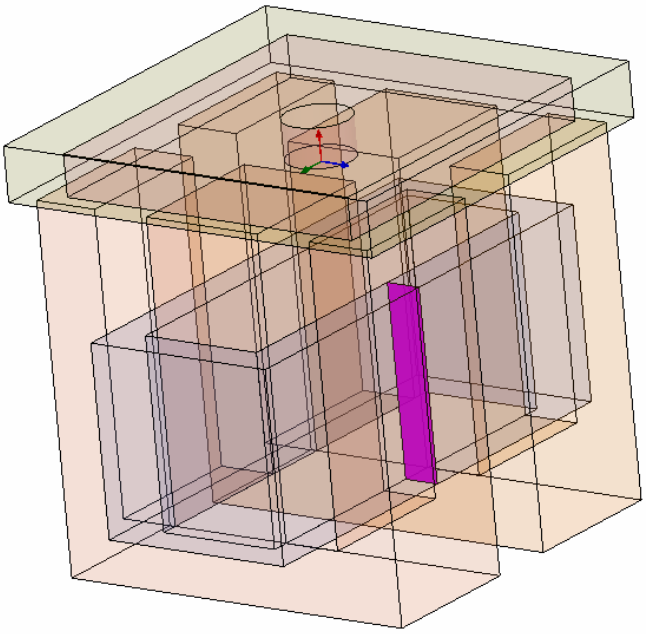


Separate the Grouped Terminals

- The result of the Surface > Section command, yields two terminals that are united. Since only one terminal is needed, the other terminal needs to be deleted, but first they need to be separated.
- Select the sheet object:



- Select the menu item *3D Modeler > Boolean > Separate Bodies*
- There are now 2 separate terminals. Delete the second terminal and rename the remaining object to *Terminal*.

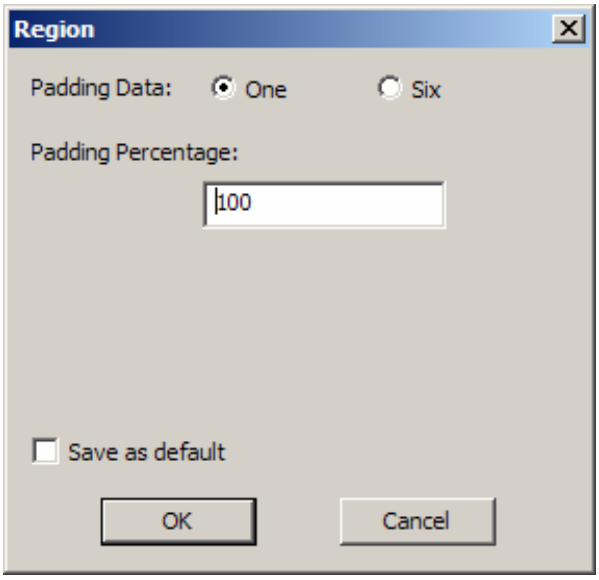


WWW.MOHANDESYAR.COM

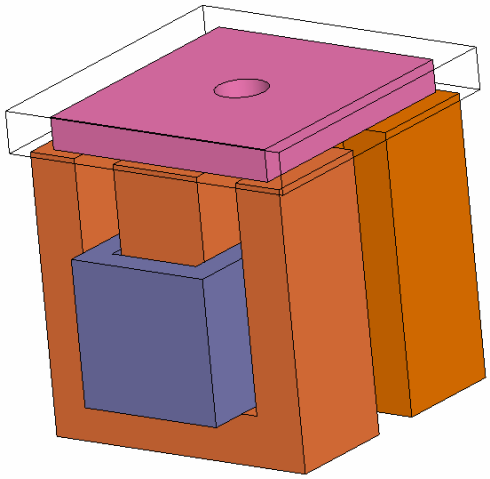
Create the Problem Region

- One of the main differences between Maxwell V10 and V11 is that a Background Region is not automatically created when a project is started. A separate object needs to be specifically created.

- Create the region ***Draw > Region*** or select the icon



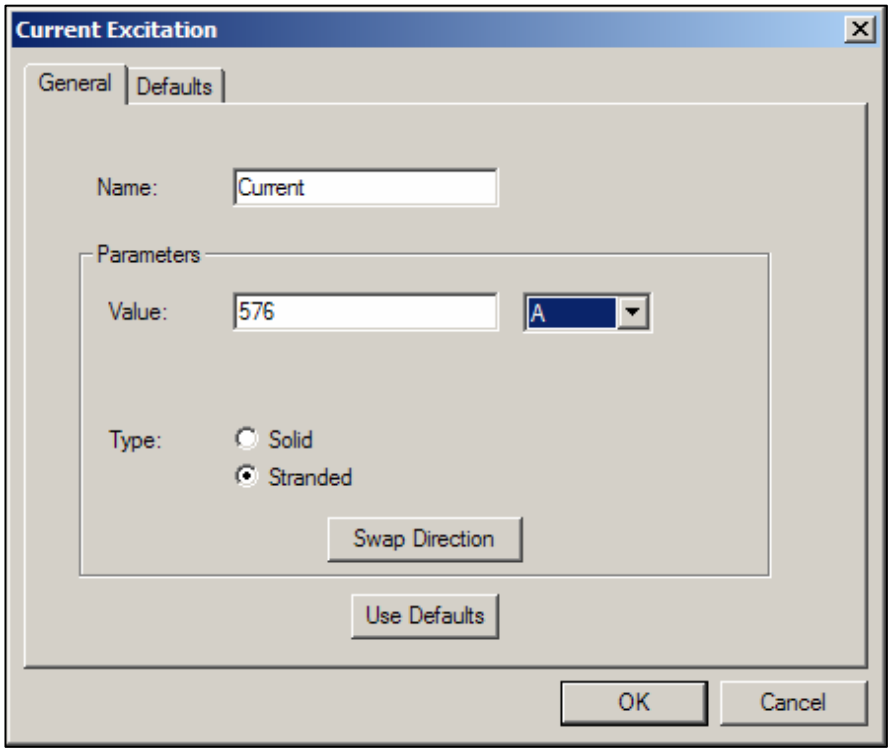
- The Padding determines how far will the limits of the region be from the objects. In this case, select 100 as Padding Percentage to have plenty of room around the solenoid.



WWW.MOHANDESYAR.COM

Assign Current Source to the Terminal

- Select the coil *Terminal* by using your mouse or selecting from tree on the left hand side of the GUI.
- Select the menu item *Maxwell > Excitations > Assign > Current*
  1. Change the Base Name to Current
  2. Change the value to 576 Amps
  3. Change the type to Stranded



Assign Boundaries Conditions

- We assume that no current goes through the stators.

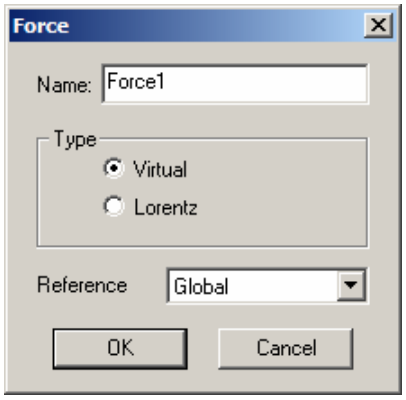
WWW.MOHANDESYAR.COM

Assign Boundaries Conditions (Continued)

- Select the two stators by holding down the CTRL key and using your mouse or selecting from tree on the left hand side of the GUI.
- Select the menu item ***Maxwell > Excitations > Assign > Insulating.*** Leave the default name Insulating1 et select OK.

Assign Executive Parameters

- The force applied on the armature as well as the self inductance of the coil are needed . These outputs will be used during the Equivalent Circuit Extraction process.
- Select the ***Armature*** from the objects tree or by using the mouse.
- Select the menu item ***Maxwell > Parameters > Assign > Force***

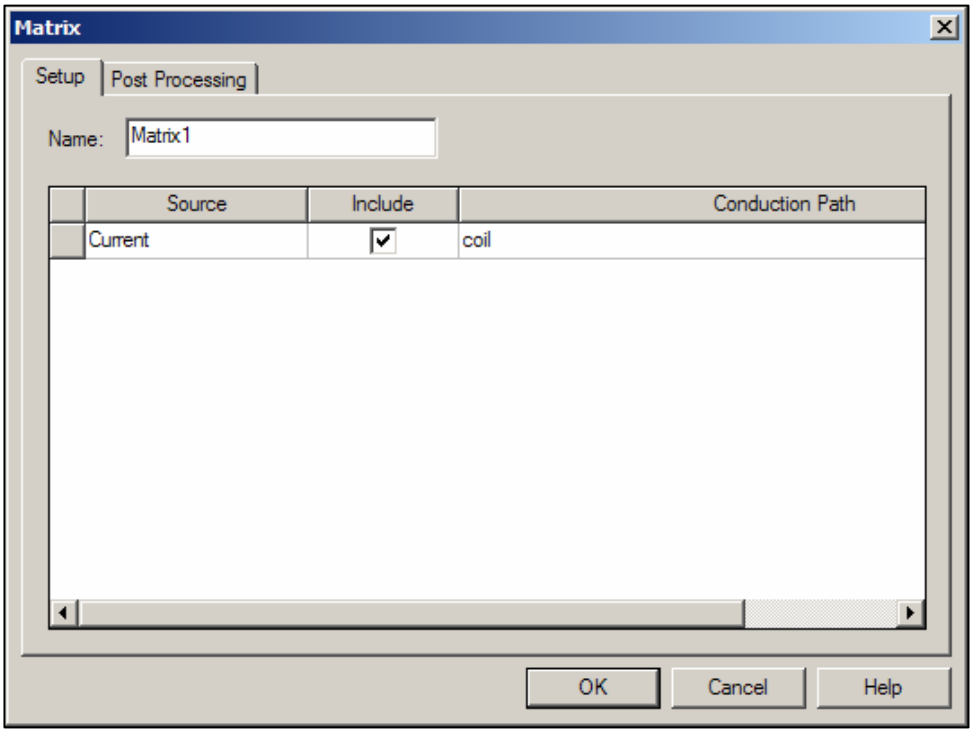


- Select Virtual and leave the default name. Make sure the Reference is set to Global.
- **Note**: with Maxwell 11, The Virtual and the Lorentz force are available. The virtual force, based on the Virtual Work Principle is more general and more accurate. The force computation is very mesh dependent, so a fine mesh is advisable whenever the Force is needed.
- The inductance computation leads to an Inductance Matrix (size NxN, with N the number of selected conductors), where the M(i,i) entry corresponds to the self inductance of the i^{th} conductor and the M(i,j) = M(j,i) entries correspond to the mutual inductance terms. In our case, we only have one conductor, therefore we will only obtain the self inductance of the Coil (in a 1 by 1 matrix).

WWW.MOHANDESYAR.COM

Assign Executive Parameters (Continued)

- Select the menu item *Maxwell > Parameters > Assign > Matrix*
- Include the term current in the matrix computation (see below).



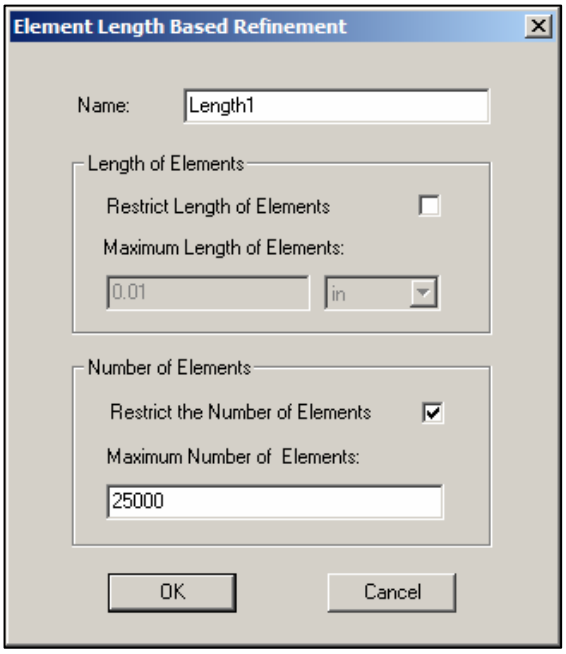
Assign Mesh Operations

- As mentioned in the previous section, the Force computation is very mesh sensitive. Therefore, to save some time computation, we will use a mesh operation.
- Select the two stators, the Band object, the Coil, the Armature by holding down the CTRL key and using your mouse or selecting from tree on the left hand side of the GUI. Do not select the Region.
- Select the menu item *Maxwell > Mesh Operations > Assign > Inside Selection > Length Based*

WWW.MOHANDESYAR.COM

Assign Mesh Operations (Continued)

- The addition of 25 000 elements will be enough to achieve a reasonable accuracy. We deselect the Restrict Length of Elements and put 25 000 as the Maximum number of Elements.



Solve the Nominal Problem

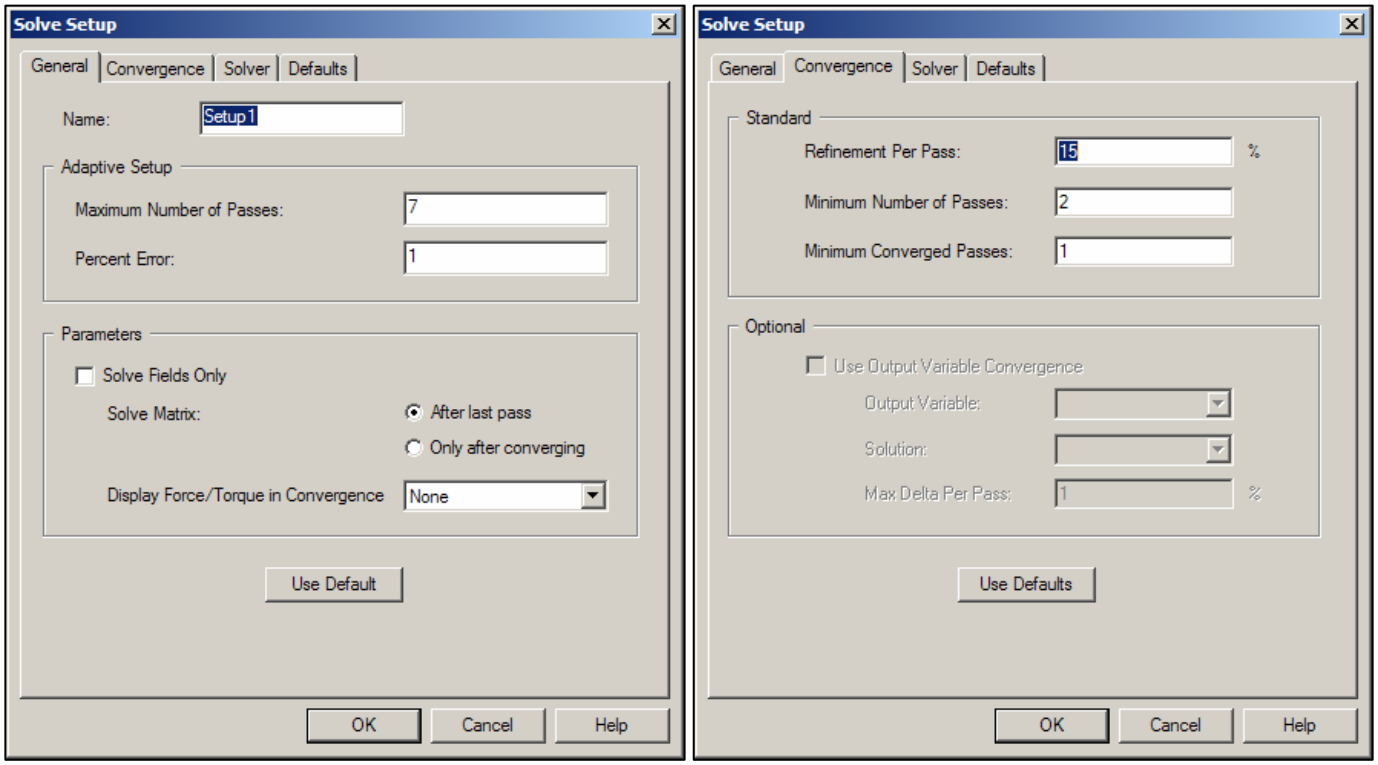
Create an Analysis Set up

- Before performing a parametric sweep on the current and the air gap, the nominal problem needs to be solved: it is always a good idea to verify that the problem is correctly set.
- **Note**: A solved project is required if output variables that will serve as entries in the Parametric table have been created in the **Calculator**.
- Select the menu item *Maxwell > Analysis Setup > Add Solution Setup*.
- Select **General** and change the Percent Error from 1to .7. Change the number of passes to 5. Leave the bottom of the window unchanged. This way, the Force and the Inductance Matrix will be calculated as we expect: the default configuration make Maxwell solve for the executive parameters after the last adaptive pass (see next page).

WWW.MOHANDESYAR.COM

Create an Analysis Set up (Continued)

- Select **Convergence** and change the Refinement Per Pass from 30 to 15 because we do not need to increase the size of the mesh too much after each pass.



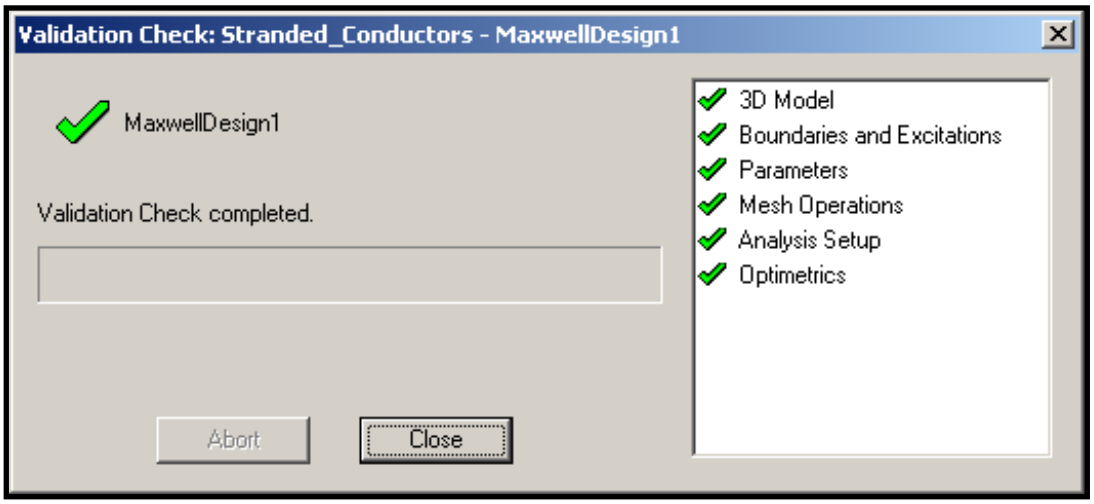
Save the Project

- Select the menu item **File > Save As**
- From the Save As window, type in **Solenoid**.
- Click on the **Save** button.

WWW.MOHANDESYAR.COM

Check the Validity of the Model

- Select the menu item ***Maxwell > Validation Check***, or click on the icon.



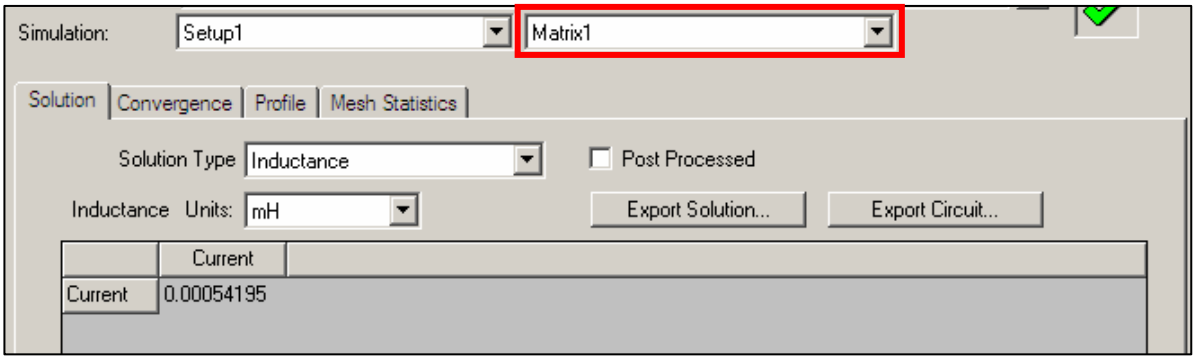
- The problem will not solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the icon.
- The progress bar shows the different stages of the ongoing computation.

Solution Data of the nominal problem

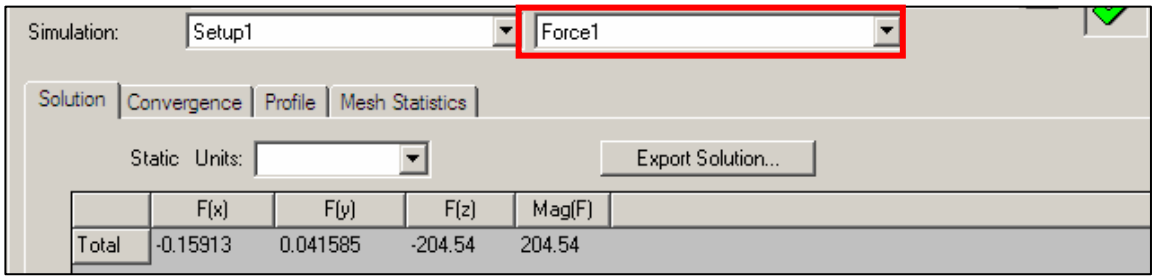
- To view the Solution Data, select the menu item ***Maxwell > Results > Solution Data.*** You have access from this window to the inductance and to the force computed as well as other information:
  - Select the **Solution** tab: the inductance is available by choosing Matrix1 in the pull-down menu. In this case the inductance matrix has only one entry.



WWW.MOHANDESYAR.COM

Solution Data of the nominal problem (Continued)

- Select the **Solution** tab: the force values are available by choosing Force1 in the pull down menu. We have access to the X,Y, and Z components of the Force as well as its magnitude. As expected, the Z component of the Force is much more larger than the other components.



- Select the **Convergence** tab: the log of the different adaptive passes is displayed with the successive sizes of the mesh.
- Select the **Profile tab**: you can find information about the CPU time, the size of the generated data.
- Select the **Mesh Statistics**: all the details about the final mesh are presented for each of the problem's region.

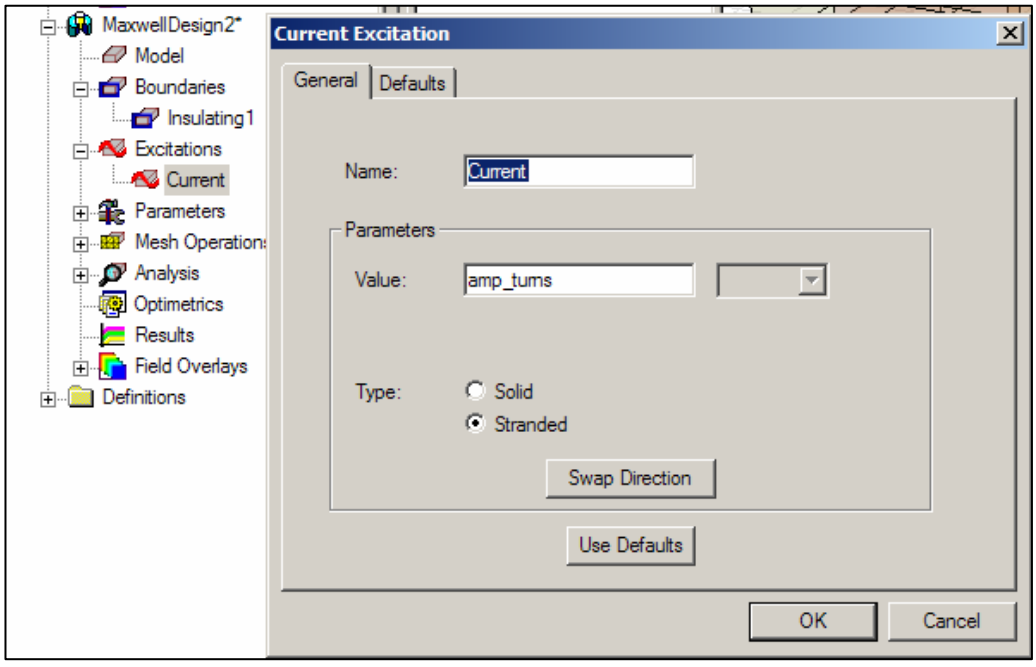
- **Note**: you can calculate any quantity using the calculator of Maxwell. Select the Menu item **Maxwell > Results > Calculator** or click on the icon.

Create Parametric Analysis

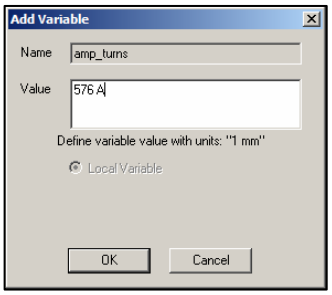
WWW.MOHANDESYAR.COM

Assign Problem Parameters

- Once the nominal problem has been successfully solved, some parameters need to be declared to perform a parametric sweep. We are interested in the inductance and the force computed in respect to the input current and the air gap between the Armature and the Stator, therefore, we create two parameters: the air gap and the input current. In Maxwell 11, any input can be set as parameter.
- **Warning**: When assigning problem parameters, the solution is invalidated. The nominal problem has to be copied if the results are needed. Also, the variables created in the calculator that will be used in the parametric sweep must be declared before the parameter declaration.
- We begin with the current source. From the model tree on the left hand side of the GUI, expand the Excitations title and double click on the Current entry. In the value case, change from 576 to *amp_turns*.



- A window pops up asking for the value of amp_turns. Leave it to 576 Amps.

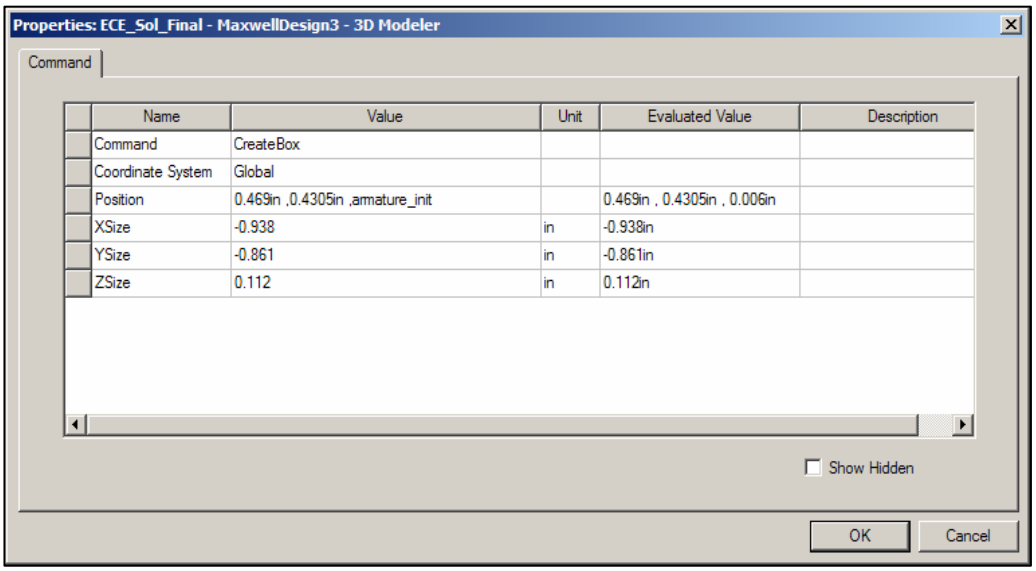


WWW.MOHANDESYAR.COM

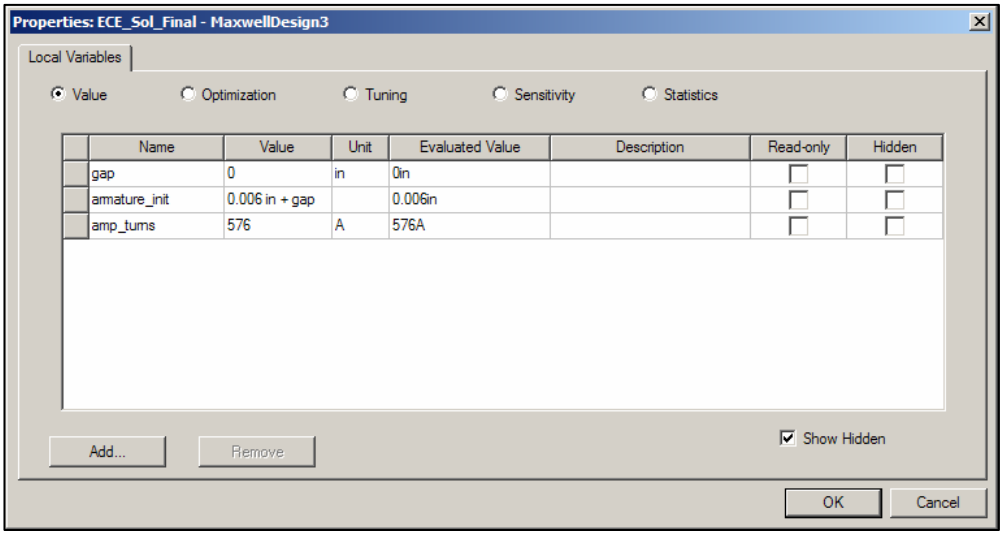
Assign Problem Parameters (Continued)

- To enter the air gap parameter, expand from the objects tree the *Armature* entry. Double click on CreateBox object.

- Change the Z- position of the corner with the parameter **armature_init**.



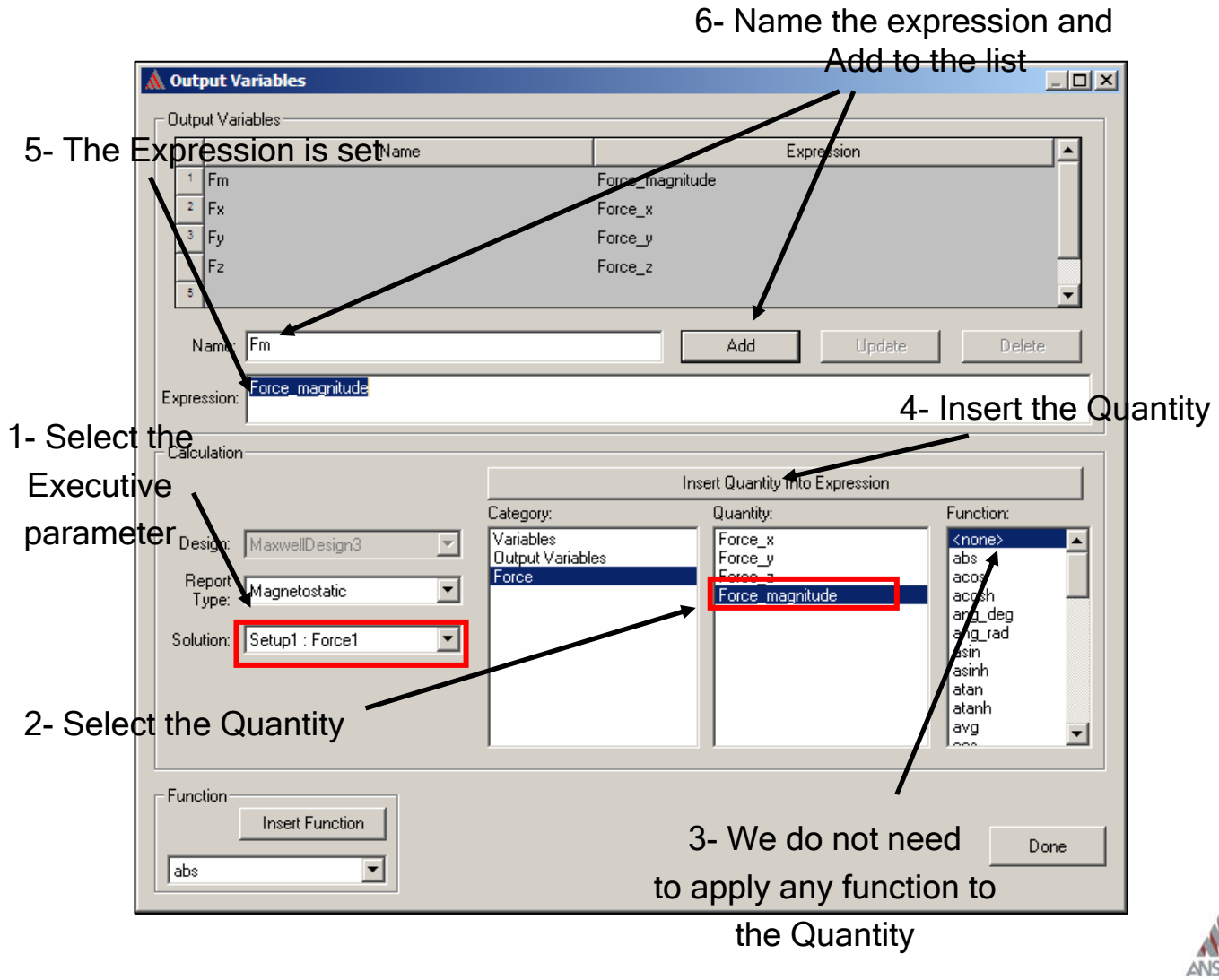
- Set the value of **armature_ini** as **.006in + gap**. Set **gap** as **0.0in** for the time being.
- At any time you can access at the list of the problem parameters. Select the menu item **Maxwell > Design Properties**. The window should look like:



WWW.MOHANDESYAR.COM

Create Output Variables

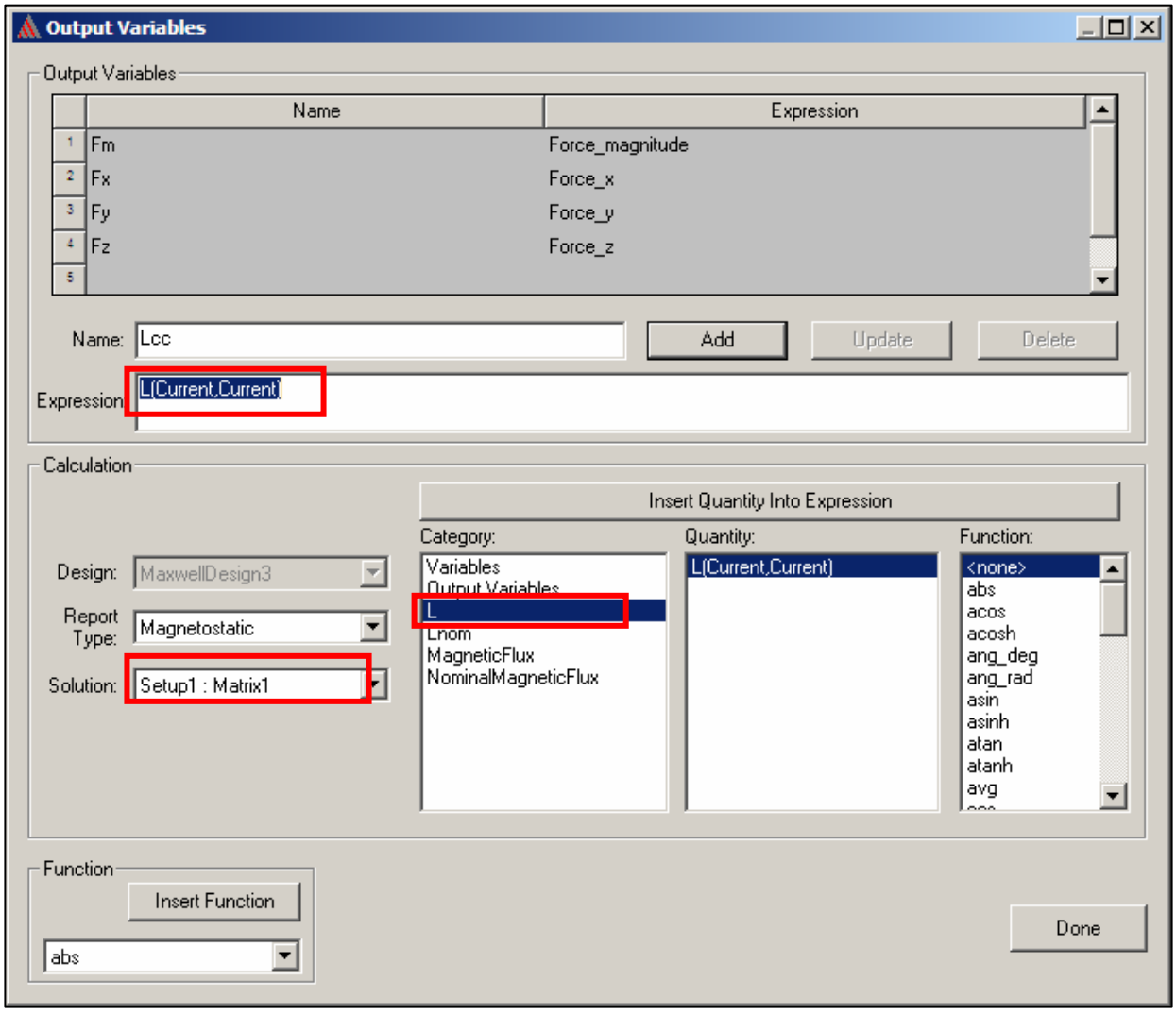
- For a parametric analysis, we are only interested in a limited number of output parameters. Our goal is to generate an equivalent circuit extraction, therefore we want to have for each row the inductance and the output force. If needed, Maxwell enables you to save the solution fields for every design of a parametric sweep.
- Select the menu item **Maxwell > Results > Output Variables**. From this window, we assign output variables.
- To create the output variable corresponding to the magnitude of the virtual force (**Fm**), follow the steps below:



WWW.MOHANDESYAR.COM

Create Output Variables (Continued)

- Proceed the same way to declare Fx, Fy, Fz the components of the Force applied to the Armature.
- We also declare the inductance as an output variable. Follow a similar procedure, but select Matrix1 from the pull-down menu (see below). **Lcc** stands for the inductance of the coil.

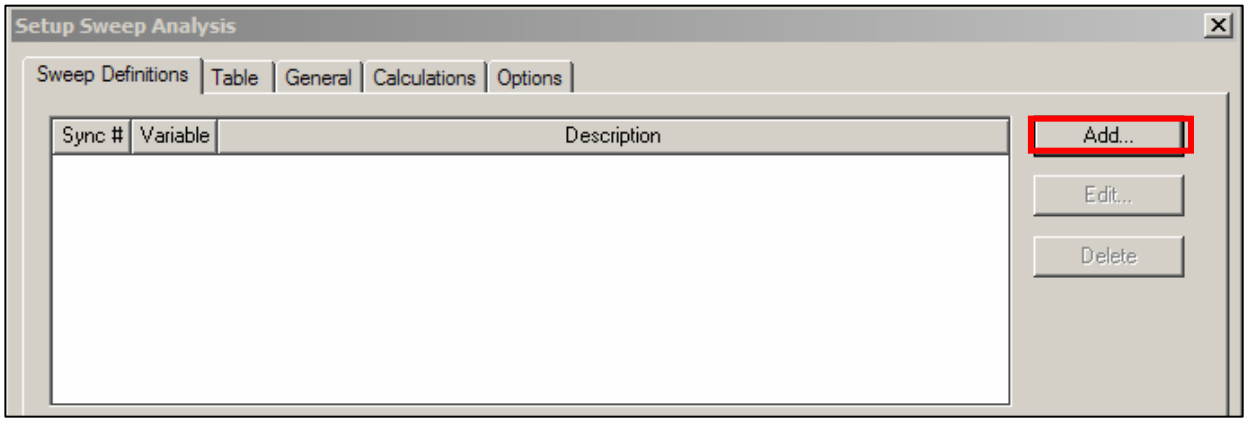


- Finally, you should have 5 output variables declared: Fm, Fx, Fy, Fz and Lcc.

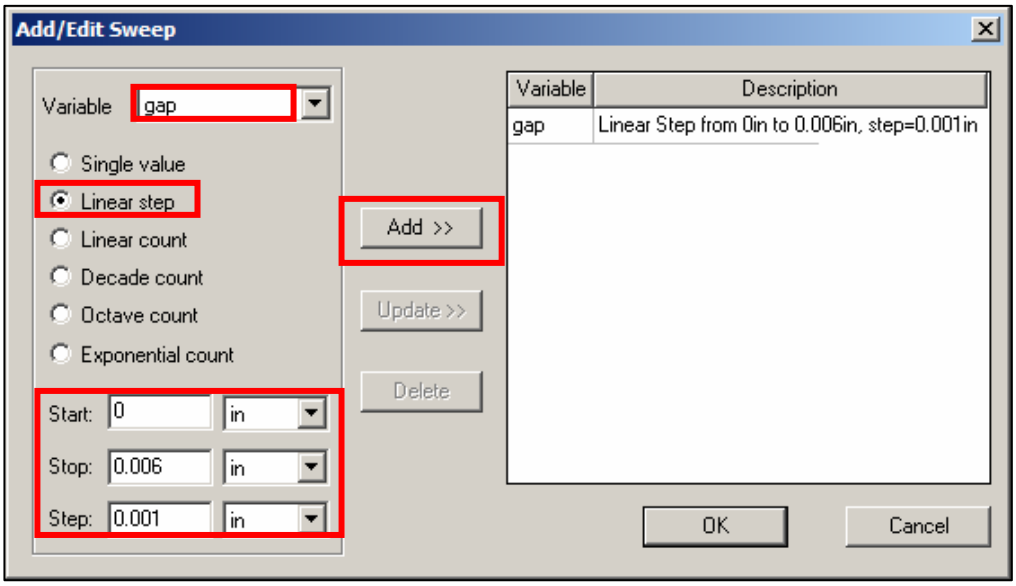
WWW.MOHANDESYAR.COM

Create Parametric Analysis

- **Note**: The module **Optimetrics** must be installed in order to be able to run a parametric sweep.
- Select the Menu item Maxwell > Optimetrics Analysis > Add Parametric. The Setup Sweep Analysis pops up. We add the different variables for each parameter that will be included in the sweep. Select Add (see Below).



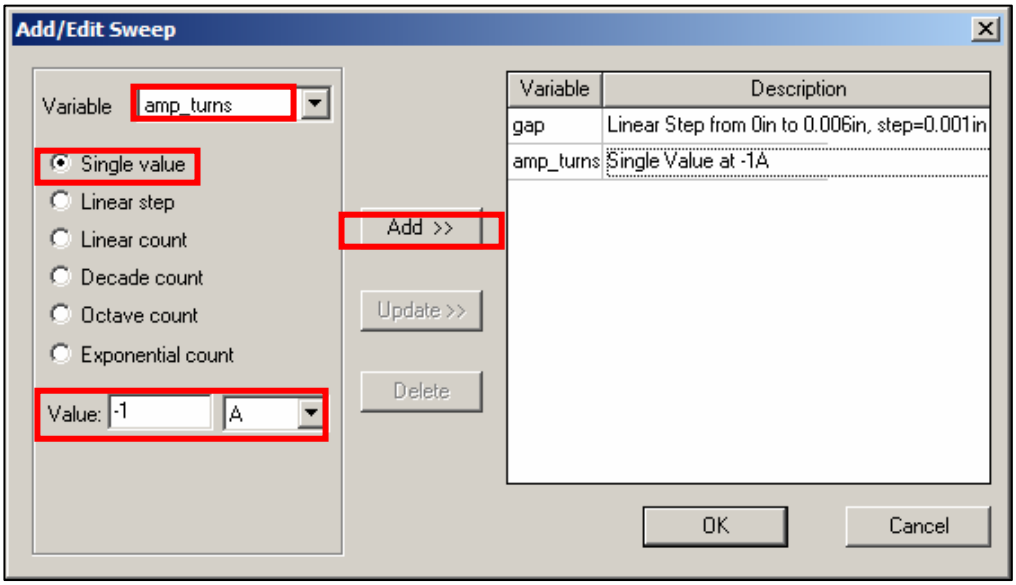
- From the Add/Edit Sweep window, select **gap** in the pull down menu. The air gap will linearly vary from 0.0in to 0.006in with a 0.001in Step. Enter these values in the corresponding boxes. Click Add to proceed.



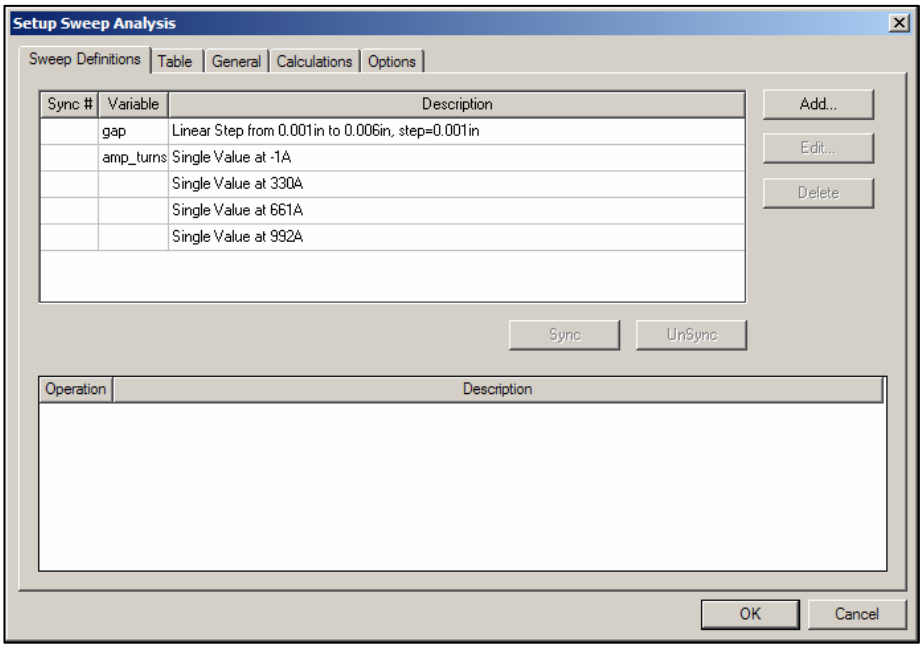
WWW.MOHANDESYAR.COM

Create Parametric Analysis (Continued)

- Select **amp_turns** from the Pull down menu. We sweep this variable with 4 single values: -1 A, 330A, 661A and 992 A. Click on the Single Value box, enter the actual value and click Add for the 4 different values. Do you forget to press Add each time. Close the Window.



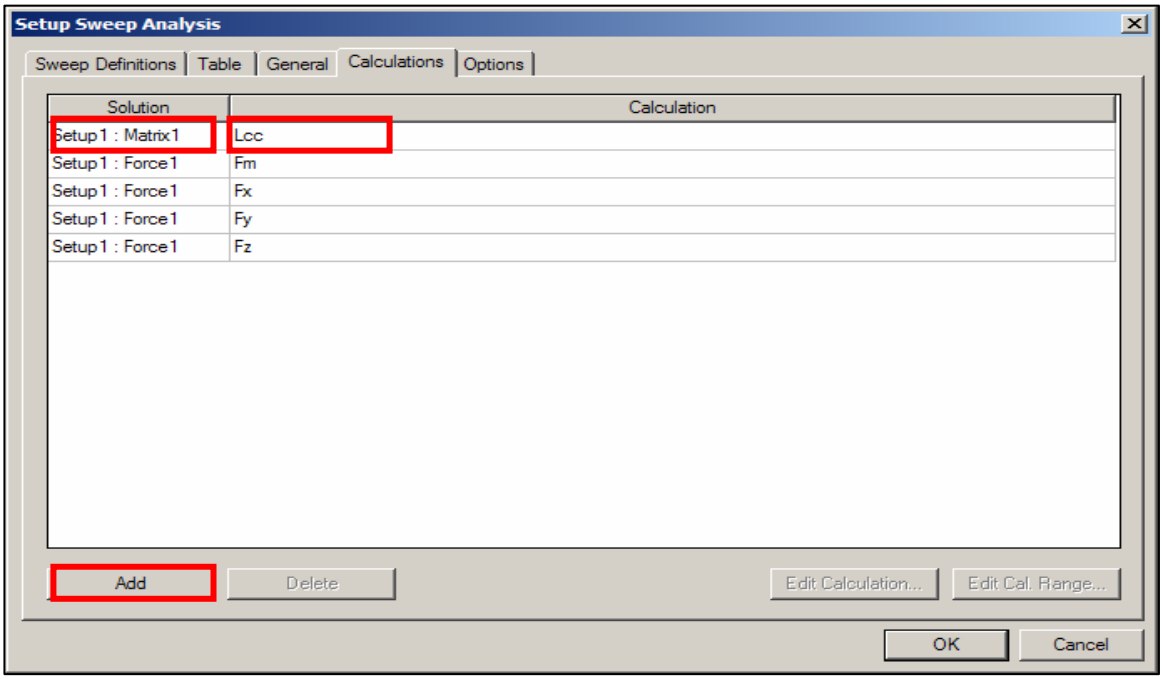
- The Setup Sweep Analysis window has the following definitions:



WWW.MOHANDESYAR.COM

Create Parametric Analysis (Continued)

- From the 'Table' tab, the 24 variations are displayed.
- Leave 'General' tab of Setup Sweep Definition unchanged and select the 'Calculations' tab. From this tab, we will define the output variables to include in the sweep.

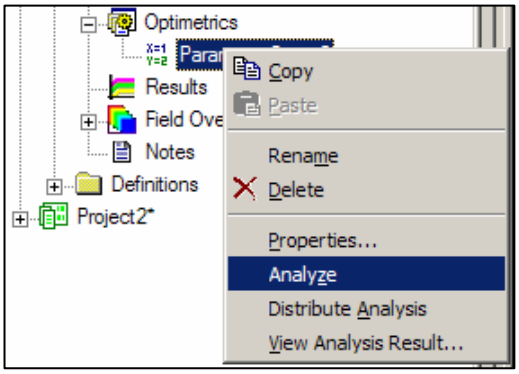


- 5 output variables are to be included. Click 5 times to the Add button. On the first line, from the pull down menu on the left, select **Setup: Matrix1**, then from the pull down menu on the right, select **Lcc**. On the second line select **Setup: Force1** , then select **Fm.** Repeat the procedure for the other force variables: **Fx**, **Fy**, **Fz**.
- In the 'Options' tab, leave everything unchecked.
- The parametric sweep is set up .
- **Note**: the number of adaptive passes, the target error and all the analysis parameters of the nominal problem will be used. If you happen to have several analysis setups, select the setup to use in the 'General' tab.

WWW.MOHANDESYAR.COM

Solve the Parametric Sweep

- Right click on the ParametricSetup from the Design tree and run the analysis.



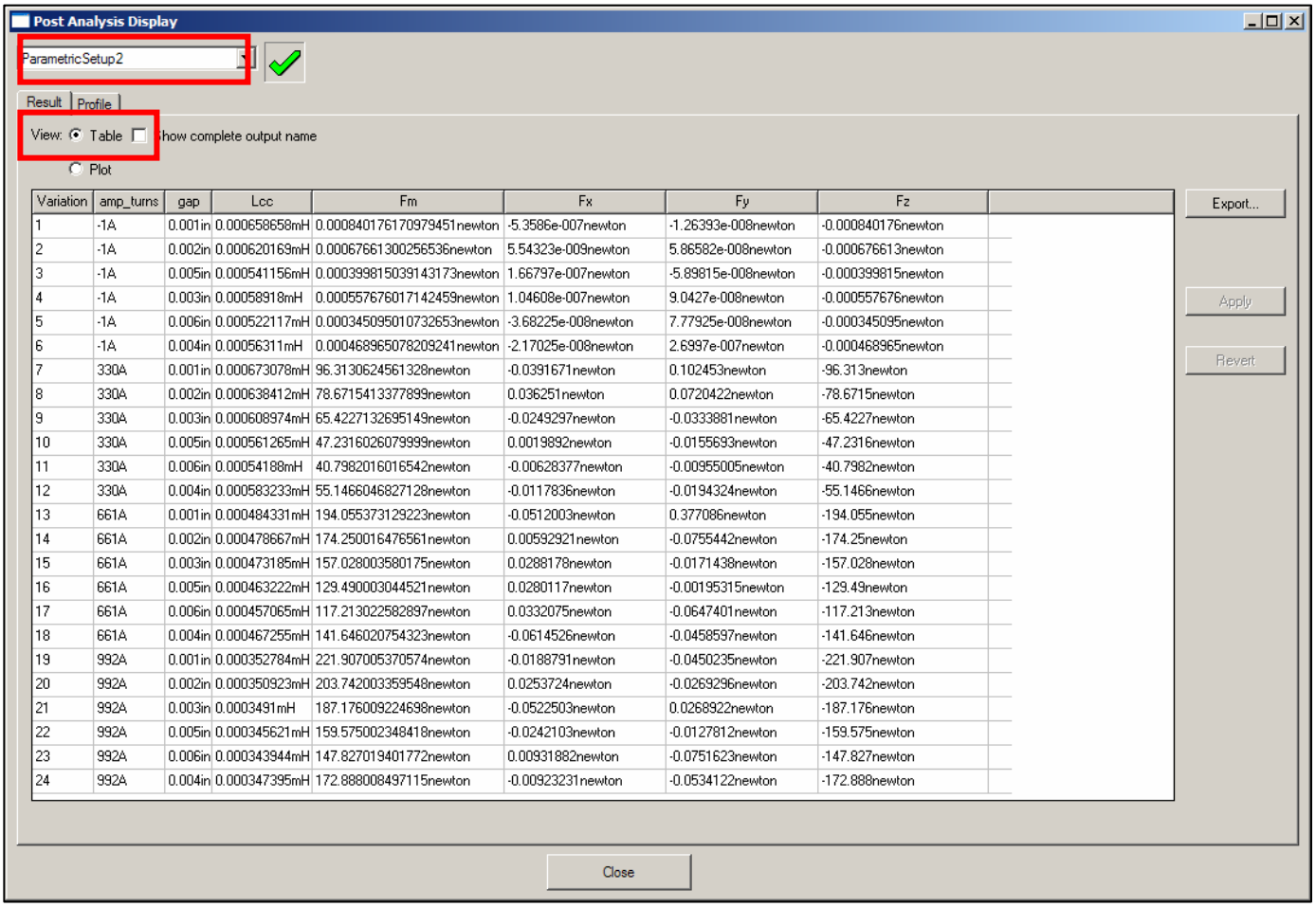
- Maxwell will start an separated analysis for every row of the table.
- If you have a Distributive Analysis license, you can run the parametric sweep on up to 10 machines per Distributive Analysis license. Use the **Distributive Analysis** button in this case.

- **Note**: If you cannot click on the **Distributive Analysis** button, it means that the list of machines you want to use is not correctly set. Select the menu item **Tool > Options > General Options**. Go to the Distributed Analysis Options tab. Make sure you have at least two machines in the Machine for Distributed Analysis box. The local machine (the machine where Maxwell has been running) is not included by default. Also, for a two processor machine, you can enter the machine's name twice.

WWW.MOHANDESYAR.COM

Solution Data of the Parametric Sweep

- Once the analysis is completed, Select the Menu item Maxwell > Optimetrics Analysis > Optimetrics Results.



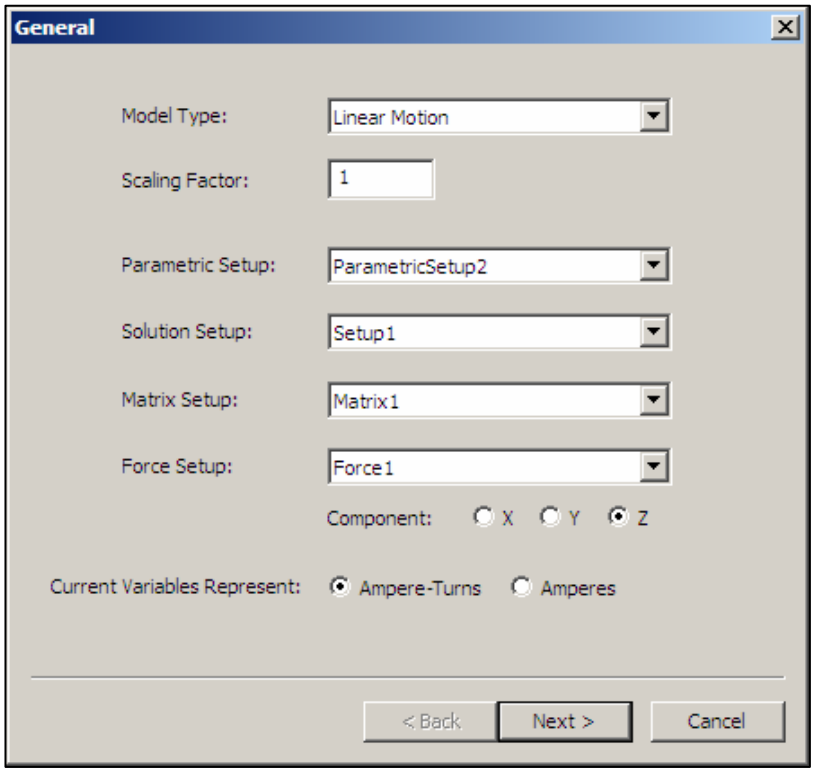
Variation	amp_turns	gap	Lcc	Fm	Fx	Fy	Fz
1	-1A	0.001in	0.000658658mH	0.00084017617097945 1newton	-5.3586e-007newton	-1.26393e-008newton	-0.000840176newton
2	-1A	0.002in	0.000620169mH	0.00067661300256536newton	5.54323e-009newton	5.86582e-008newton	-0.000676613newton
3	-1A	0.005in	0.000541156mH	0.00039981503914 3173newton	1.66797e-007newton	-5.89815e-008newton	-0.000399815newton
4	-1A	0.003in	0.00058918mH	0.00055767601714 2459newton	1.04608e-007newton	9.0427e-008newton	-0.000557676newton
5	-1A	0.006in	0.000522117mH	0.00034509501073 2653newton	-3.68225e-008newton	7.77925e-008newton	-0.000345095newton
6	-1A	0.004in	0.00056311mH	0.00046896507820 9241newton	-2.17025e-008newton	2.6997e-007newton	-0.000468965newton
7	330A	0.001in	0.000673078mH	96.3130624561328newton	-0.0391671newton	0.102453newton	-96.313newton
8	330A	0.002in	0.000638412mH	78.6715413377899newton	0.036251newton	0.0720422newton	-78.6715newton
9	330A	0.003in	0.000608974mH	65.422713269514 9newton	-0.0249297newton	-0.0333881newton	-65.4227newton
10	330A	0.005in	0.000561265mH	47.2316026079999newton	0.0019892newton	-0.0155693newton	-47.2316newton
11	330A	0.006in	0.00054188mH	40.7982016016542newton	-0.00628377newton	-0.00955005newton	-40.7982newton
12	330A	0.004in	0.000583233mH	55.1466046827128newton	-0.0117836newton	-0.0194324newton	-55.1466newton
13	661A	0.001in	0.000484331mH	194.055373129223newton	-0.0512003newton	0.377086newton	-194.055newton
14	661A	0.002in	0.000478667mH	174.250016476561newton	0.00592921newton	-0.0755442newton	-174.25newton
15	661A	0.003in	0.000473185mH	157.028003580175newton	0.0288178newton	-0.0171438newton	-157.028newton
16	661A	0.005in	0.000463222mH	129.490003044521newton	0.0280117newton	-0.00195315newton	-129.49newton
17	661A	0.006in	0.000457065mH	117.213022582897newton	0.0332075newton	-0.0647401newton	-117.213newton
18	661A	0.004in	0.000467255mH	141.646020754323newton	-0.0614526newton	-0.0458597newton	-141.646newton
19	992A	0.001in	0.000352784mH	221.907005370574newton	-0.0188791newton	-0.0450235newton	-221.907newton
20	992A	0.002in	0.000350923mH	203.742003359548newton	0.0253724newton	-0.0269296newton	-203.742newton
21	992A	0.003in	0.0003491mH	187.176009224698newton	-0.0522503newton	0.0268922newton	-187.176newton
22	992A	0.005in	0.000345621mH	159.575002348418newton	-0.0242103newton	-0.0127812newton	-159.575newton
23	992A	0.006in	0.000343944mH	147.827019401772newton	0.00931882newton	-0.0751623newton	-147.827newton
24	992A	0.004in	0.000347395mH	172.888008497115newton	-0.00923231newton	-0.0534122newton	-172.888newton

- Make sure to select the correct optimetrics setup from the pull down menu, then select the Table radio button to view the results table.

WWW.MOHANDESYAR.COM

ECE from the Parametric Analysis

- Select the menu item **Maxwell > Equivalent Circuit Extraction > From Parametric Solutions**.
- Maxwell fills the ECE window with some parameters that correspond to the project:
  - Make sure that the Model type is set to **Linear Motion**.
  - Leave the scaling factor to **1** (this factor is used for models with symmetry).
  - Choose the Parametric Setup corresponding to your analysis with the according solution Setup, Matrix Setup and Force Setup.
  - Only the **Z** component of the force will be used.
  - Amperes-Turns have been entered in the model (we selected **Stranded Conductor**): select the corresponding radio button.

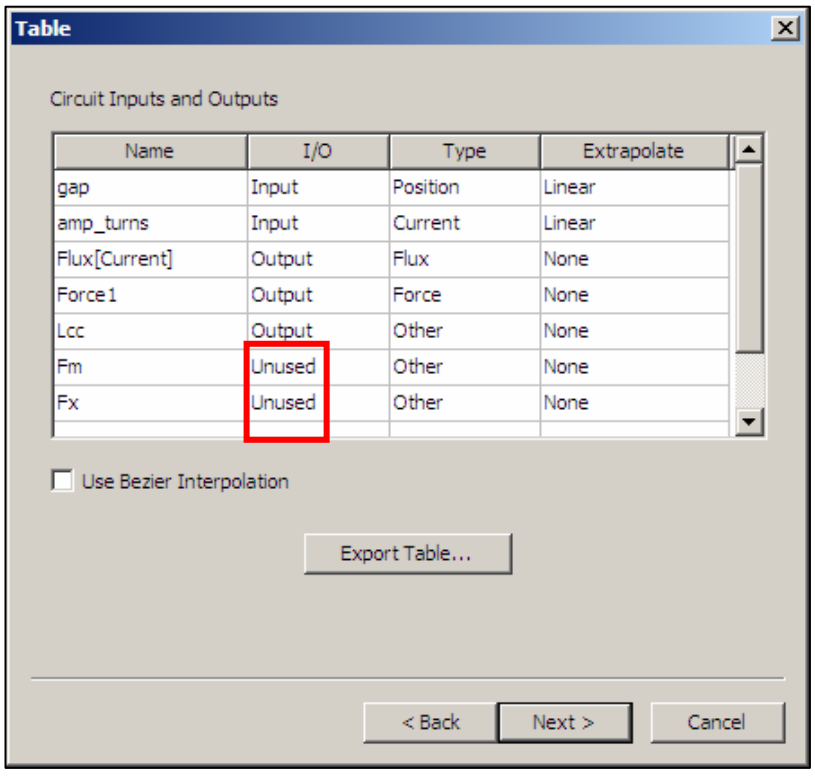


- Click on Next to continue.

WWW.MOHANDESYAR.COM

ECE from the Parametric Analysis (Continued)

- The next window enables you to select or deselect the entries from the parametric tables.
- Fm, Fx and Fy computations will not be used in Simplorer. From the pull-down menu in the I/O column, select **Unused**. We leave the other parameters to their default values.
- Make sure that the data types are correct. gap defines a Position, amp_turns a current.

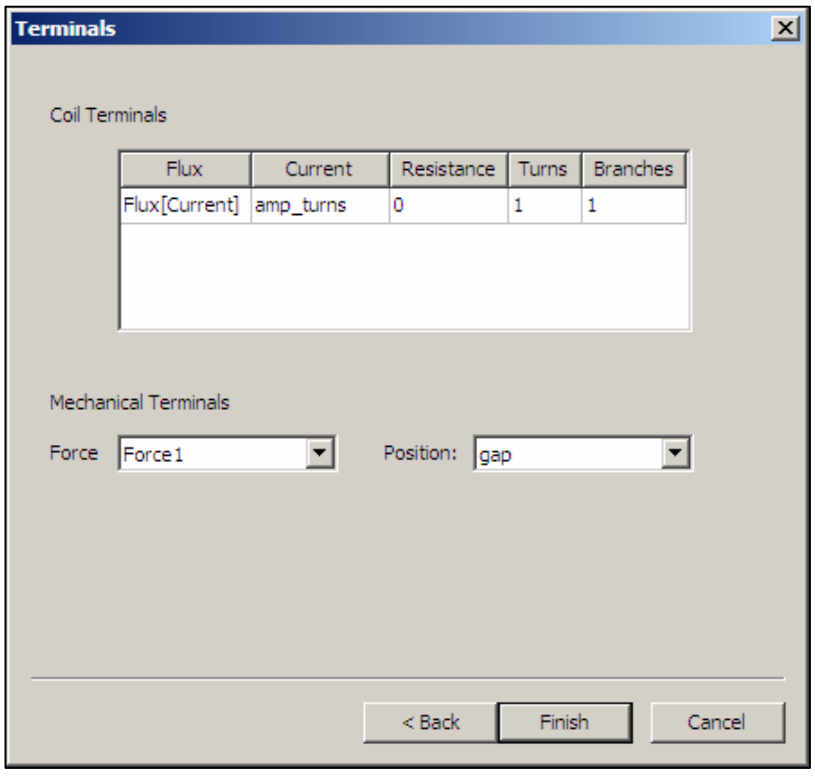


- Click on Next to continue.

WWW.MOHANDESYAR.COM

ECE from the Parametric Analysis (Continued)

- This window defines the Terminals of the conservative nodes in Simplorer. The conservative nodes will have their Across and Through quantities solved by Simplorer, ensuring the physical meaning of the simulation.
- The Flux (and therefore the EMF) is the electrical Across quantity in this case. The current is the Through quantity.
- In the Mechanical domain, gap is the Across quantity whereas Force1 is the Through quantity.

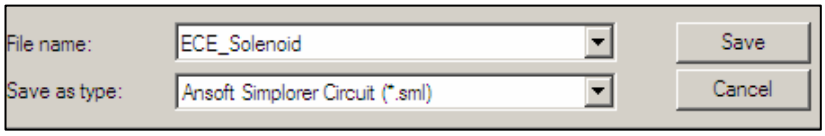


- Maxwell has defined the correct values for the Terminals. Click on Finish.

WWW.MOHANDESYAR.COM

ECE from the Parametric Analysis (Continued)

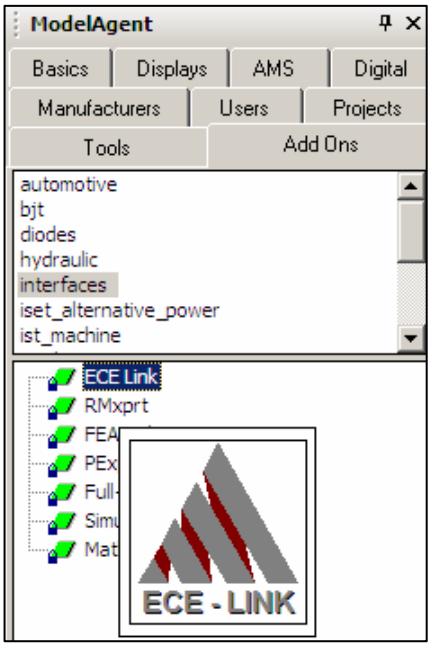
- Give a File name and save the output file.



- Maxwell creates the SML file containing the model of the component to be used in Simplorer. SML stands for Simplorer Modeling Language, the internal language of Simplorer. This script file will be imported into Simplorer.
- To improve visibility of the component in Simplorer, it is advised to capture the image of the Maxwell 3D design. Select the menu item **Edit > Copy To Clipboard**. You can paste this image in any software (like Paint, …)and save it under a bmp format.

Create the circuit in Simplorer

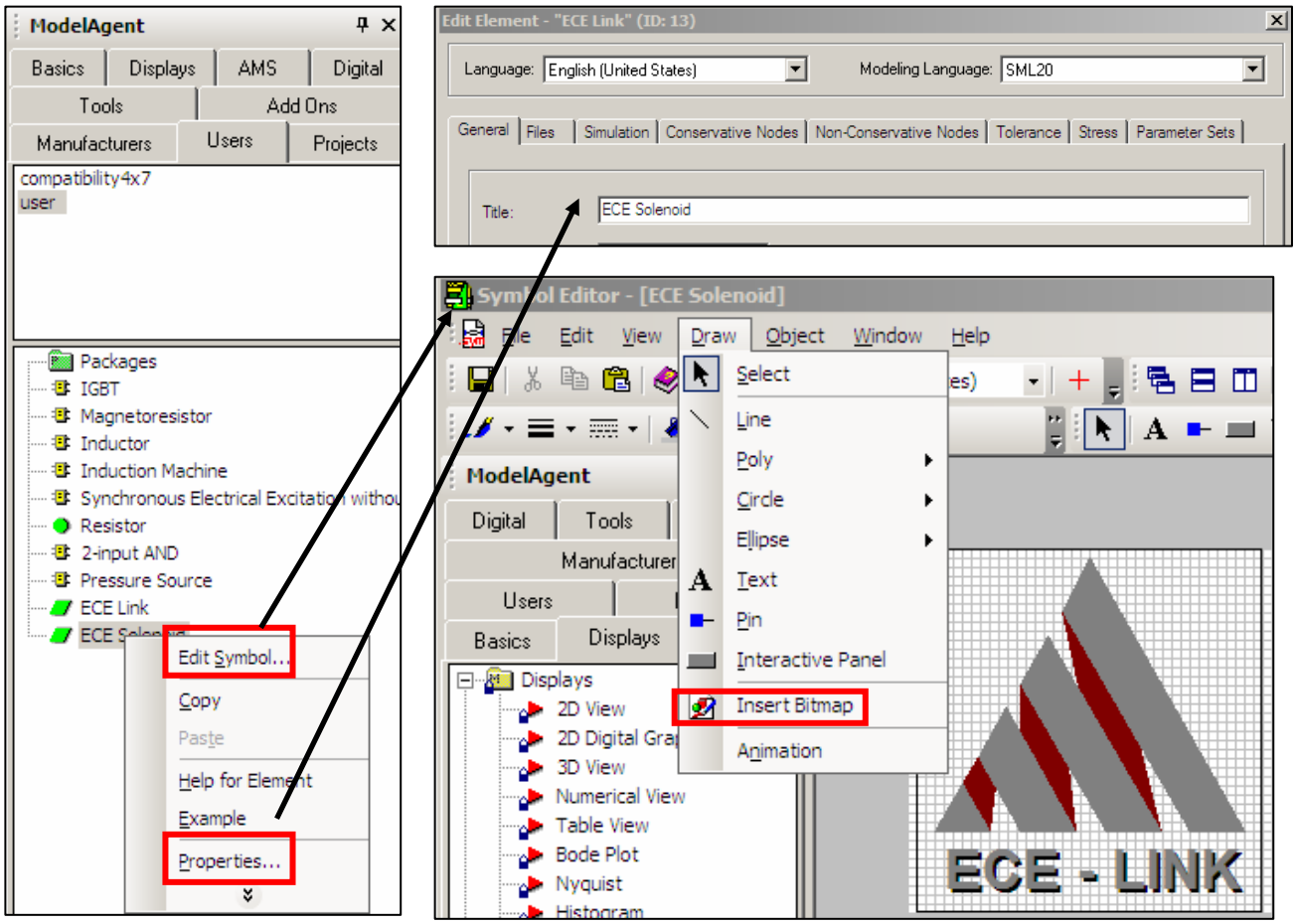
- **Note**: The software package **Simplorer 7** must be installed in order to be able to continue.
- Open Simplorer version 7, create a project then a Schematic.
- From the ModelAgent bar, Select the **AddOns** tab, then click on **interfaces** in the folder. Right click on ECE Link and select Copy.



WWW.MOHANDESYAR.COM

Create the Component in Simplorer (Continued)

- Select the tab Users (or the Project tab is some libraries have been created in the Project). Select user and select the menu item **Edit > Paste**. The ECE component has been copied to your user section. It is now possible to change the name of the component and to modify the component symbol.

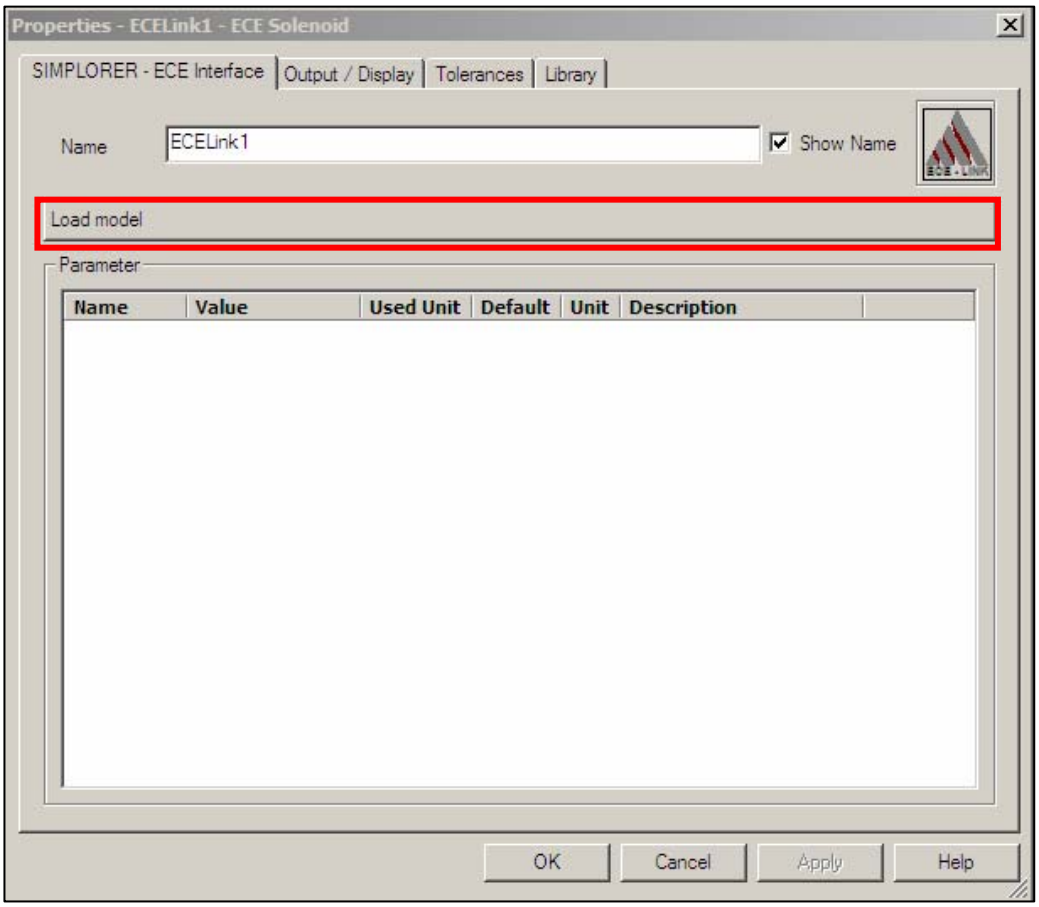


- Right click after having selecting the ECE link component. Choose properties. In the window, select the General tab and give the component name **ECE Solenoid**. Leave the other parameters to their default values and Select OK.
- Right click and select Edit Symbol. From this window, you can draw your own symbol or import it from a bmp image (Select the menu item **Draw > Insert Bitmap**).

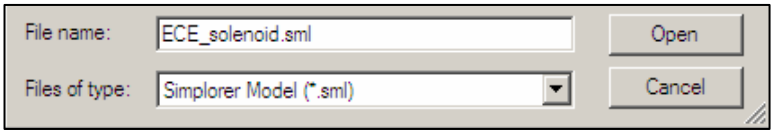
WWW.MOHANDESYAR.COM

Create the Component in Simplorer (Continued)

- Drag and drop the ECE Solenoid component on the sheet. Open the component dialogue box. Click on the **Load model** button.



- Browse your disk to recover the output file generated by Maxwell. This file has the extension SML.

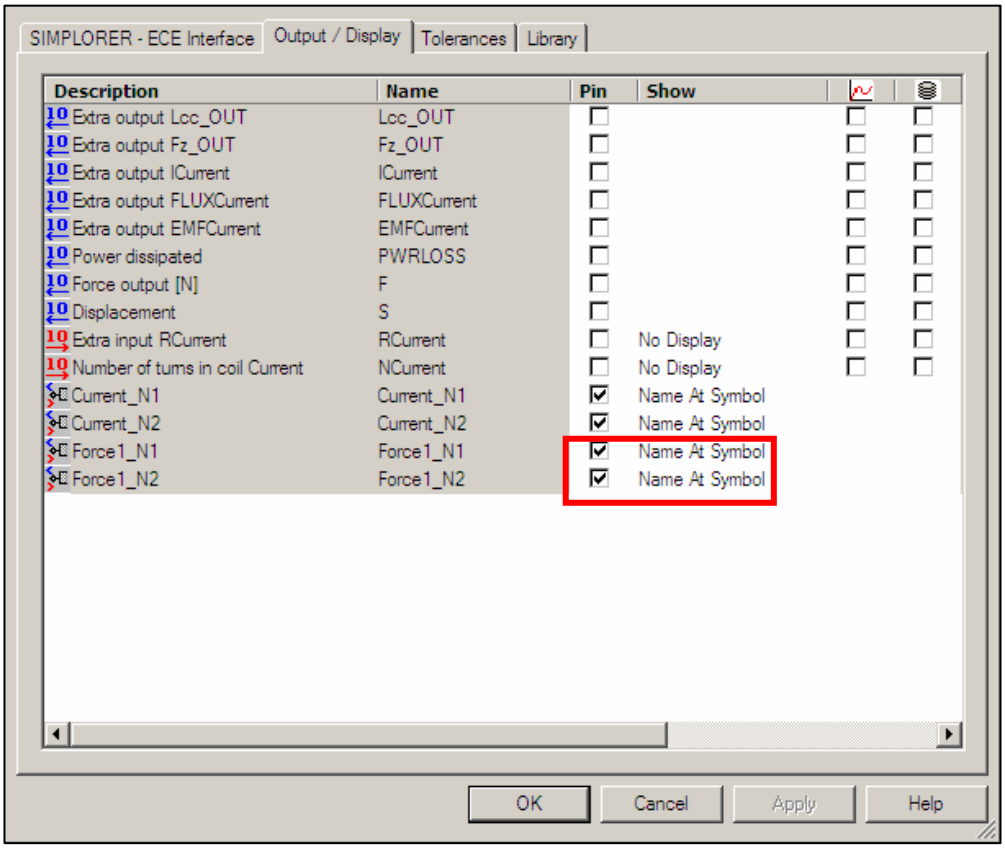


- Click on OK. The model is loaded.

WWW.MOHANDESYAR.COM

Create the Component in Simplorer (Continued)

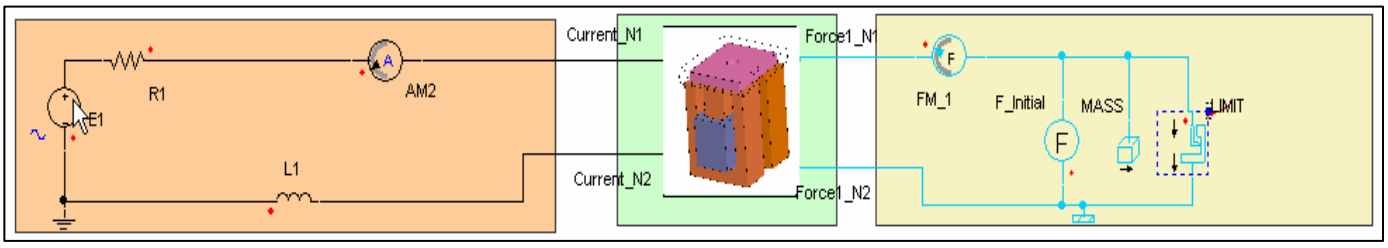
- In the component dialogue box, select the **Output/Display** tab. In the **Pin** column, 4 boxes are checked: Current_N1, Current_N2, Force1_N1, Force1_N2. These terminals correspond to the electrical and mechanical terminals we defined during the ECE process in Maxwell. Verify that in the **Show** column, the **Name at Symbol** is chosen to ease the wire connection on the sheet.



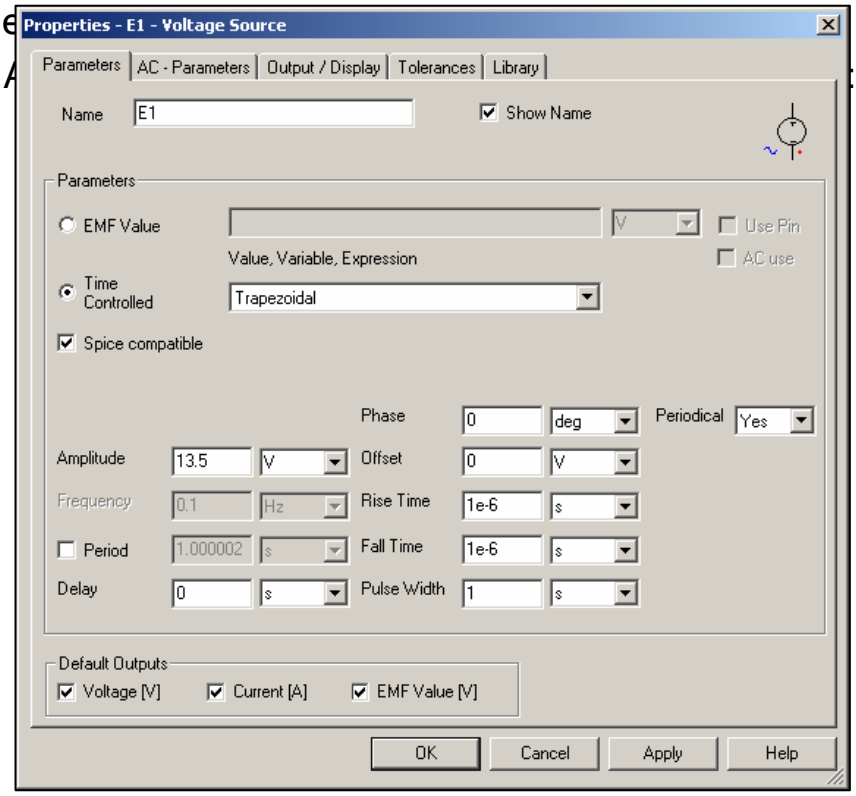
- As additional input (non conservative node), you can specify the number of turns in the Coil (in Maxwell, we just gave the overall value of the current) as well as give an additional resistance corresponding to the coil in the Solenoid.
- The component is ready to be used in a Simplorer project.

WWW.MOHANDESYAR.COM

Create the Schematic sheet in Simplorer



- In this application, the system surrounding the actuator has an electric part and a mechanical part.
- For the electrical part, the following components have been dragged on the

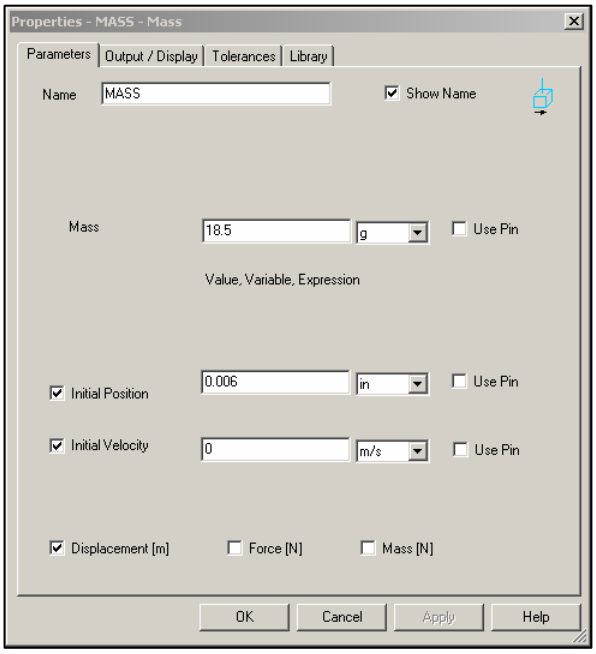


- A resistance R1 with R= 215 mΩ

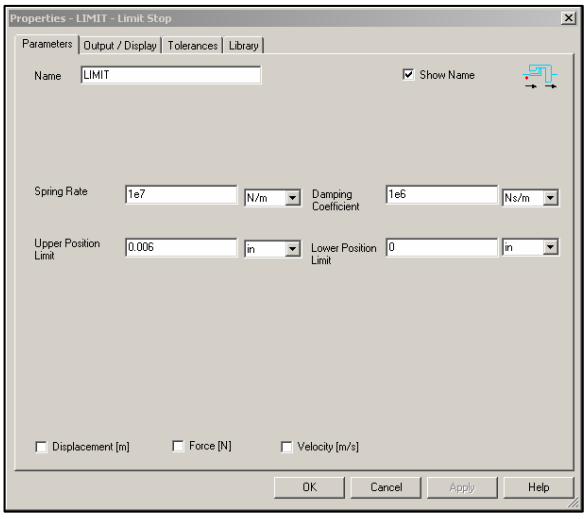
WWW.MOHANDESYAR.COM

Create the Schematic sheet in Simplorer (Continued)

- For the electrical part, the following components have been dragged on the sheet:
  - A Force-meter **FM1** to measure the actual force coming from the actuator
  - A Force Source **F_initial** corresponding to the initial force with F=125 N
  - A mass **MASS** that describes the mass of the plunger of the actuator:



  - A Limit stop **LIMIT** that represents a parallel damper and spring combination:



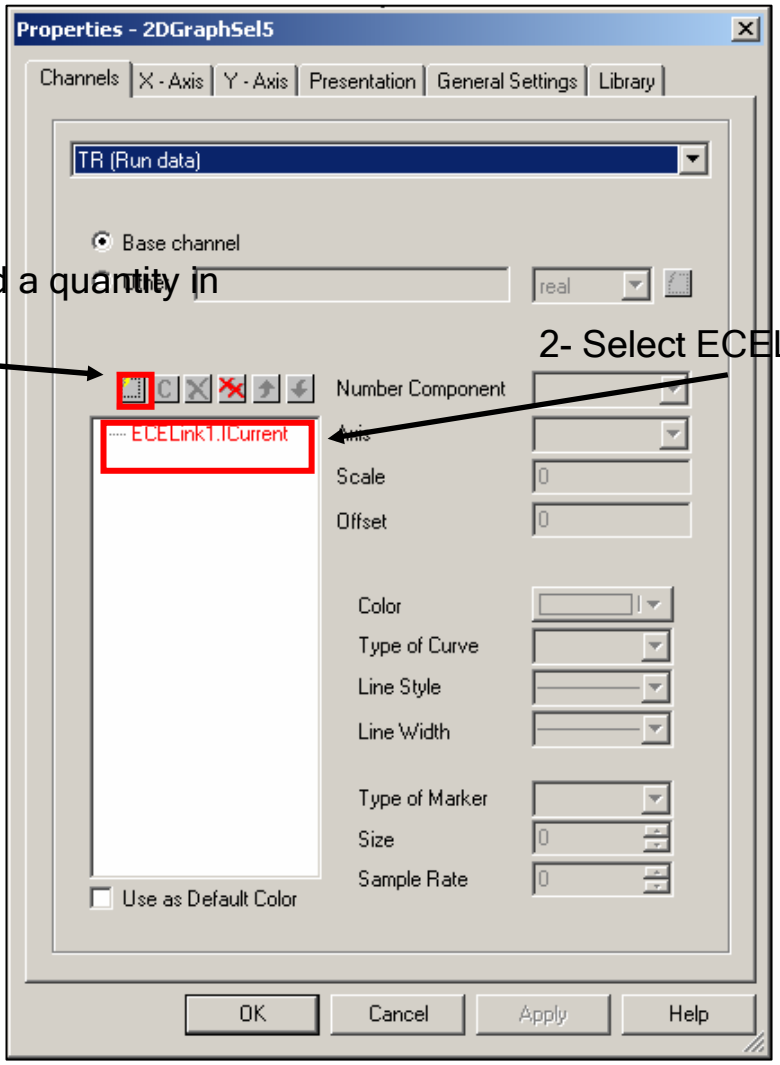
WWW.MOHANDESYAR.COM

Create the Schematic sheet in Simplorer (Continued)

- The mechanical ground and the electrical ground must be included.

Create the graphical Displays in Simplorer

- We want to display the Coil Current, the Actuator Force and the Stoke for this system.
- Drag a **2D view** element (from the **Displays** tab).



1- Click to add a quantity in the display

2- Select ECELink1.ICurrent

- Select the yellow button to add the the current of the ECE component.

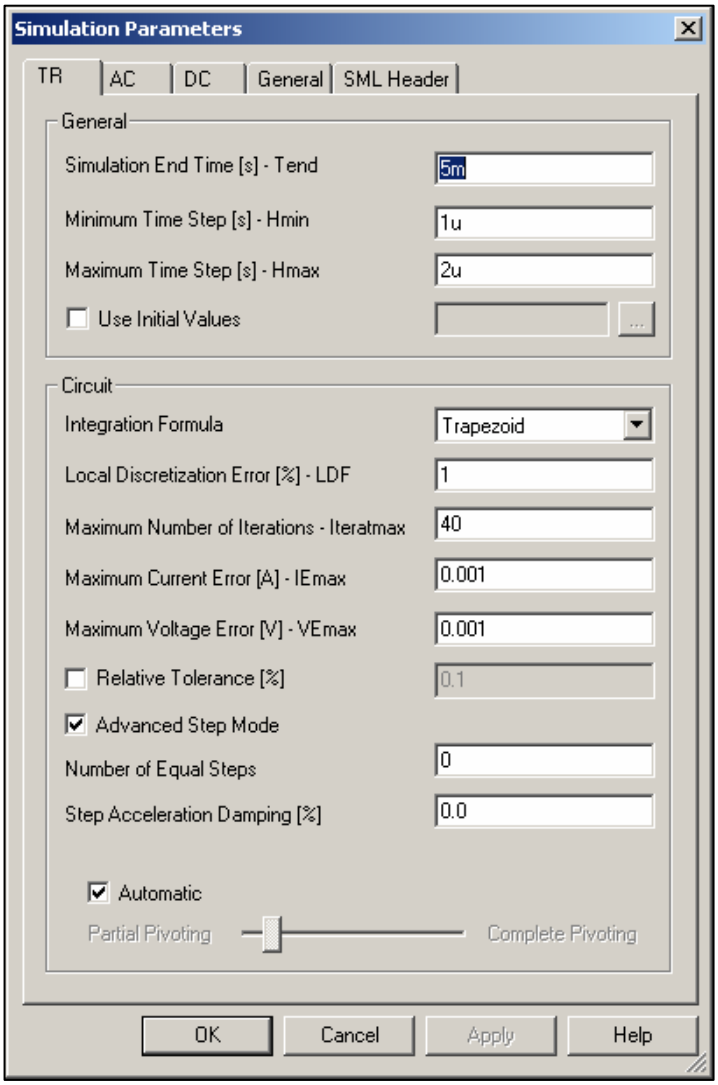
WWW.MOHANDESYAR.COM

Create the graphical Displays in Simplorer

- Repeat this operation with 2 new **2D view** elements. Add the two following quantities:
  - The force of the plunger: ECELink1.Fz_OUT
  - The stroke : 39.79 * MASS.S

Create a Transient Analysis

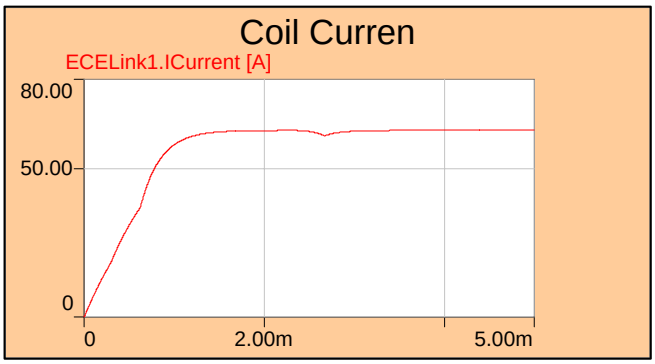
- Select the menu item Simulation > Parameters
- Fill the window with these parameters:

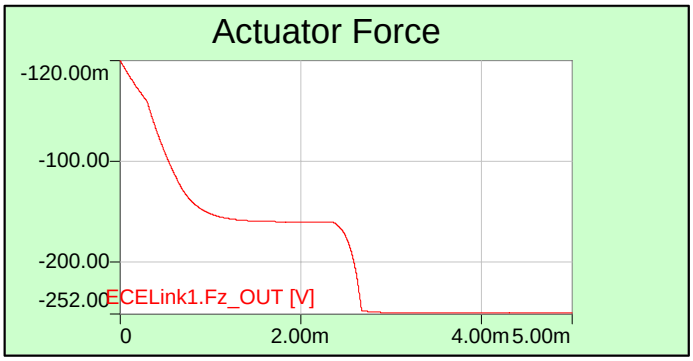


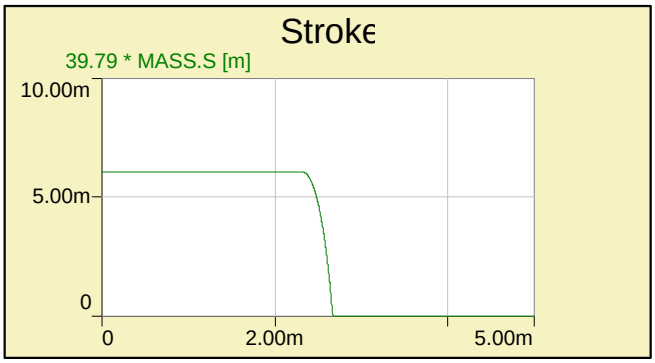
WWW.MOHANDESYAR.COM

Run the Analysis

- The simulation can be started by selecting the menu item **Simulation > Start**
- The 2D displays look like:

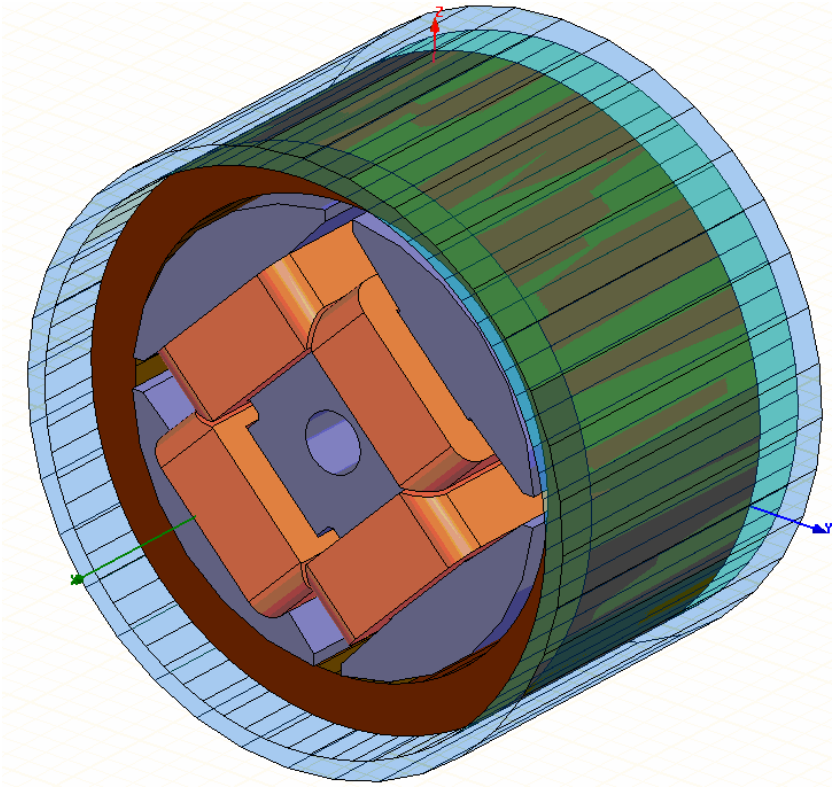






WWW.MOHANDESYAR.COM

- The implementation and application of Nonlinear Anisotropic Material when using the Magnetostatic, and Transient Solvers
  - In v11 of Maxwell 3D a new function of Nonlinear Anisotropic Material can be used with the Magnetostatic, and Transient Solvers for simulating soft magnetic materials. The obvious advantage is significance because
    - *Non-linear lamination* is extensively used in low frequency electromagnetic devices for having high induction but with significant reduction of eddy current loss in the rolling direction.
    - *Non-linear anisotropy* is widely used magnetization, magnetic recording, power transformers and large size electrical machines. Oriented materials will have higher magnetic field like magnetic flux density in special directions as users want but lower magnetic field in other directions.
  - In this note a magnetostatic model as one step of magnetization will be introduced including how to set up Non-linear lamination and Non-linear anisotropic materials. Same set up will be used for Transient model.



WWW.MOHANDESYAR.COM

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - User Defined Primitives (UDP): **RMxprt/SlotCore**
    - Primitives: **Regular Polyhedron Box Cylinder**
    - Surface Operations: **Section Edge Filet**
    - Boolean Operations: **Split Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
    - Windings
  - Analysis
    - Magnetostatic
    - (Transient)
  - Results
    - **Field Calculator**
    - **Group objects with Objectlist**
    - **2d plot with Create Report**
  - Field Overlays:
    - **Flux Density Plots**

WWW.MOHANDESYAR.COM

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

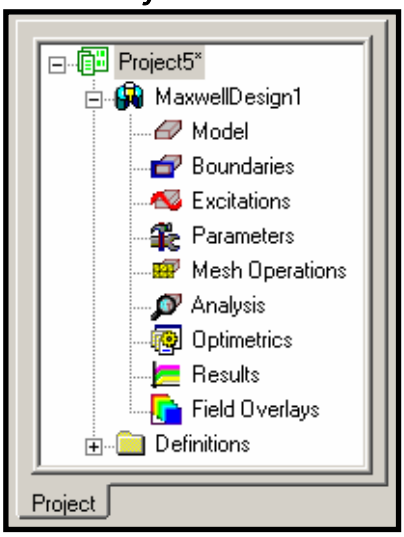
Setting Tool Options

- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options*.
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Display** tab
        - Default transparency = 0.8
     3. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     4. Click the **OK** button

WWW.MOHANDESYAR.COM

Opening a New Project

- **To open a new project:**
  1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  2. Select the menu item ***Project > Insert Maxwell Design***, or click on the icon



Set Solution Type

- Select the menu item: ***Maxwell > Solution Type > Magnetostatic***, or right mouse click on MaxwellDesign1 and select ***Solution Type …***

Creating the 3D Model of a magnetization model of a four- poles out-rotor

- The example that will be used to demonstrate how the anisotropic non-linear material & non-linear lamination are implemented in a Magnetization model of a four-poles out-rotor. The intent of this write up is not how to simulate the out-rotor motors, it is rather to demonstrate how anisotropic materials are implemented within the Magnetostatic (or Transient) solver. This magnet ring is an non-linear anisotropic one (nonlinear in R & Z direction with low permeability in Theta direction) and the field core is a laminated one.（lamination direction is Global X) Thus we can show how these materials can be set up for create an four poles out-rotor.

Set Model Units

- Select the menu item ***3D Modeler > Units >*** Select Units***: mm (millimeters)***

WWW.MOHANDESYAR.COM

Create the problem Region

- One of the main differences between Maxwell V10 and V11 is that a Background Region is not automatically created when a project is started. A separate object needs to be specifically created.
- Create a Box region by selecting the menu item ***Draw > Box*** or click on the icon.
  1. Using the coordinate entry field, enter the box position
     - **X: -50, Y: -50, Z: -50**, Press the **Enter** key
  2. Then enter the opposite corner for the base rectangle with the height
     - **X: 100.0, Y: 100.0, Z: 100.0**, Press the **Enter** key

Properties: Project1 - MaxwellDesign1 - 3D Modeler

Command | Attribute

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate System	Global		
Position	-50 ,-50 ,-50	mm	-50mm , -50mm , -50...
XSize	100	mm	100mm
YSize	100	mm	100mm
ZSize	100	mm	100mm

- Click OK to finish it. Change the name from Box1 to Region by editing the Attribute panal on the left hand side of the interface. Keep the material as vacuum.

Properties: Project1 - MaxwellDesign1 - 3D Modeler

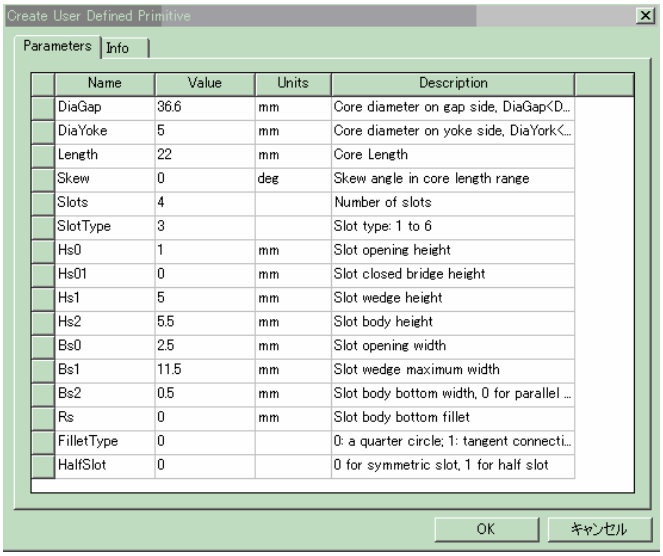
Attribute

Name	Value	Unit
Name	Region	
Material	vacuum	
Solve Inside	☑	
Orientation	Global	
Model	☑	
Display Wireframe	☐	
Color	Edit	
Transparent	0.8	

WWW.MOHANDESYAR.COM

Create the Field Core:

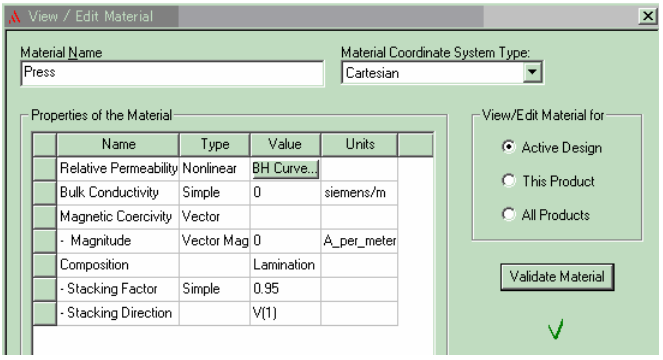
- **A User Defined Primitive will be used to create the Field Core**
  - ***Draw > User Defined Primitive > Syslib > Rmxprt > SlotCore***
  - Use the values given in the panel below to create the rotor



Name	Value	Units	Description
DiaGap	36.6	mm	Core diameter on gap side, DiaGap<D...
DiaYoke	5	mm	Core diameter on yoke side, DiaYork<...
Length	22	mm	Core Length
Skew	0	deg	Skew angle in core length range
Slots	4		Number of slots
SlotType	3		Slot type: 1 to 6
Hs0	1	mm	Slot opening height
Hs01	0	mm	Slot closed bridge height
Hs1	5	mm	Slot wedge height
Hs2	5.5	mm	Slot body height
Bs0	2.5	mm	Slot opening width
Bs1	11.5	mm	Slot wedge maximum width
Bs2	0.5	mm	Slot body bottom width, 0 for parallel ...
Rs	0	mm	Slot body bottom fillet
FilletType	0		0: a quarter circle; 1: tangent connecti...
HalfSlot	0		0 for symmetric slot, 1 for half slot

  - Click ***SlotCore1*** with right of mouse in History Tree ***Edit>Arrange>Rotate>Y >90*** degree>***OK***
  - Click on the just created object in the drawing window and in the panel on the left change its name from ***SlotCore1*** to ***FieldCore***

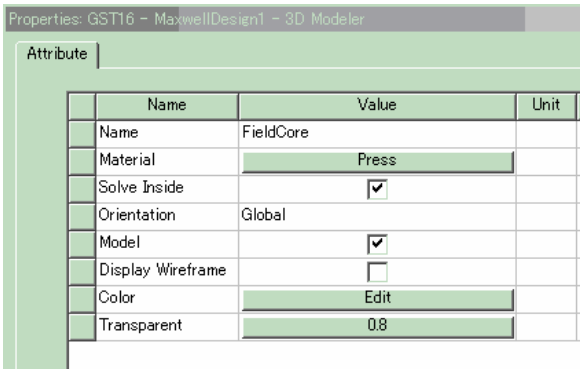
- **Create material *Press* of non-linear laminate one for *FieldCore***
  - Click ***Material*** **vacuum** to get in the Material ***Select Definition*** window, Pick ***Add Material...***
  - Change name ***Material1*** to ***Press***
  - Pick Type of ***Relative Permeability*** and change it from ***Simple*** to ***Nonlinear***, Pick ***BH curve*** in the right of it, then click ***Import from file***..., select Press.bh for it with OK to finish it.
  - Pick ***Solid*** in ***Composition Line*** and Change it to ***Lamination***
  - Change the Value of ***-Stacking Factor*** to 0.95 and keep the ***Stacking Direction*** as ***V(1)*** (X direction). Pick ***Validate Material*** , the green tick will be appeared. Click OK to exit.



Name	Type	Value	Units
Relative Permeability	Nonlinear	BH Curve...	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
Composition		Lamination	
- Stacking Factor	Simple	0.95	
- Stacking Direction		V(1)	

WWW.MOHANDESYAR.COM

- Select *OK* in Select Definition window, Then change SlotCore1 as FieldCore, make sure the Material is *Press* and the Orientation is *Global*.



Create the Housing and Magnet Ring of Out-rotor:

Create Housing

- *Draw* > *RegularPolyhedron* then using the coordinate entry field, enter the center position
  - **X: -15.3, Y: 0, Z: 0**, Press the **Enter** key
- Then enter the height and the radio
  - **dZ: 24.4** for radius, Press the **Enter** key
  - **dX: 30.8** for height, Press the **Enter** key
  - Number of segments = **36, OK**
- Change the name from RegularPolyhedron1 to *Housing*

Name	
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-15.3 ,0 ,0
Start Position	-15.3 ,0 ,24.4
Axis	X
Height	30.8
Number of Segme...	36

- *Draw* > *RegularPolyhedron* then using the coordinate entry field, enter the center position
  - **X: -14.5, Y: 0, Z: 0**, Press the **Enter** key
- Then enter the height and the radio
  - **dZ: 22.8** for radius, Press the **Enter** key
  - **dX: 30.8** for height, Press the **Enter** key
  - Number of segments = **72, OK**
- Change the name from RegularPolyhedron1 to *Housing1*

Name	Value
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-14.5 ,0 ,0
Start Position	-14.5 ,22.8 ,0
Axis	X
Height	30.8
Number of Segme...	72

- Pick Housing and Housing1 in History Tree, Use right mouse *Edit > Boolean > Subtract* to create new *Housing*
- *Draw* > *RegularPolyhedron* then using the coordinate entry field, enter the center position
  - **X: -15.3, Y: 0, Z: 0**, Press the **Enter** key
- Then enter the height and the radio
  - **dZ: 3.25** for radius, Press the **Enter** key
  - **dX: -1.7** for height, Press the **Enter** key
  - Number of segments = **16**, **OK**
- Change the name from RegularPolyhedron1 to *Housing2*

Name	Value
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-15.3 ,0 ,0
Start Position	-15.3 ,0 ,3.325
Axis	X
Height	-1.7
Number of Segme...	16

WWW.MOHANDESYAR.COM

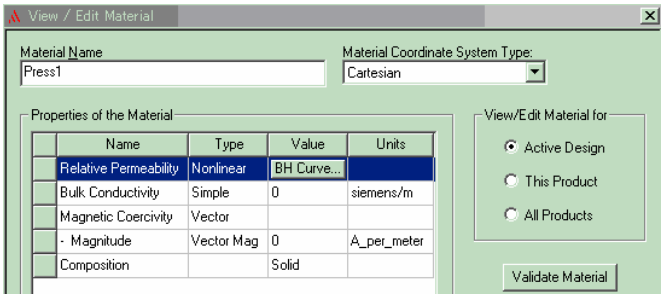
- Pick Housing and Housing2 in History Tree, Use right mouse ***Edit > Boolean > Unit*** to create new ***Housing***

- ***Draw*** > ***RegularPolyhedron*** then using the coordinate entry field, enter the center position
  - **X: 0, Y: 0, Z: 0**, Press the **Enter** key
- Then enter the height and the radio
  - **dZ: 2.45** for radius, Press the **Enter** key
  - **dX: -20** for height, Press the **Enter** key
  - Number of segments = **16**, **OK**
- Change the name from RegularPolyhedron1 to ***Housing3***

	Name	Value
	Command	CreateRegularPolyhedron
	Coordinate System	Global
	Center Position	0 ,0 ,0
	Start Position	0 ,0 ,2.45
	Axis	X
	Height	-20
	Number of Segme...	16

- Pick Housing and Housing3 in History Tree, Use right mouse ***Edit > Boolean > Substract*** to create final ***Housing***

- **Create material *Press1* of non-linear one for *Housing***
  - Click ***Housing*** in History Tree, then pick ***Material*** **vacuum** in it's Attribute to get in the Material ***Select Definition*** window, Search & pick ***Press***
  - Select ***Clone Materials***
  - Pick ***Lamination*** in ***Composition Line*** and Change it to ***Solid***
  - Click OK to exit. ***Press1*** will be appeared in Material Name Listing
  - Click OK to exit ***Select Definition***. Make sure ***Press1*** will be the Material of ***Housing*** and change the color as light blue



- **Create Magnet Ring named as *Mag***
  - ***Draw*** > ***RegularPolyhedron*** then using the coordinate entry field, enter the center position
    - **X: -10, Y: 0, Z: 0**, Press the **Enter** key
  - Then enter the height and the radio
    - **dZ: 22.8** for radius, Press the **Enter** key
    - **dX: 20** for height, Press the **Enter** key
    - Number of segments = **72**, **OK**
  - Change the name from RegularPolyhedron1 to ***Mag***

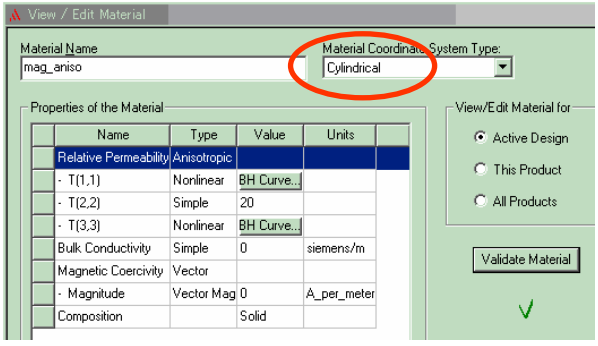
	Name	Value
	Command	CreateRegularPolyhedron
	Coordinate System	Global
	Center Position	-10 ,0 ,0
	Start Position	-10 ,0 ,22.8
	Axis	X
	Height	20
	Number of Segme...	72

WWW.MOHANDESYAR.COM

- *Draw* > *RegularPolyhedron* then using the coordinate entry field, enter the center position
  - **X: -10, Y: 0, Z: 0**, Press the **Enter** key
- Then enter the height and the radio
  - **dZ: 18.8** for radius, Press the **Enter** key
  - **dX: 20** for height, Press the **Enter** key
  - Number of segments = **72**, **OK**
- Change the name from RegularPolyhedron1 to *Mag1*
- Pick *Mag* and *Mag1* in History Tree, Use right mouse *Edit* > *Boolean* > *Substract* to create final *Mag*

Name	Value
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-10 ,0 ,0
Start Position	-10 ,0 ,18.8
Axis	X
Height	20
Number of Segme...	72

- **Create material *Mag_aniso* of non-linear anisotropic one for *Mag***
  - Click *Mag* in History Tree, then pick Material *vacuum* in it's Attribute to get in the Material *Select Definition* window, Pick *Add Material...*
  - Change name *Material1* to *mag_aniso,* and Material Coord. System Cartesian to *Cylindrical*
  - Pick Type of *Relative Permeability* and change it from *Simple* to Anisotropic
  - Change Type of T(1,1) from *Simple* to *Nonlinear*, Click *BH curve* in the right of it, then click *Import from file...*, select *Mag.bh* for it with OK to finish it.
  - Change Value of T(2,2) from 0 to *20*
  - Change Type of T(3,3) from *Simple* to *Nonlinear*, Click *BH curve* in the right of it, then click *Import from file...*, select *Mag.bh* for it with OK to finish it.
  - Click Validate Material to get green Tick Mark. OK to exit. *mag_aniso* will be appeared in Material Name Listing
  - Click OK to exit *Select Definition*. Make sure *mag_aniso* will be the Material of *Ma*



Name	Value
Type	Relative
Name	RelativeCS0
Reference CS	Global
Origin	0 ,0 ,0
X Axis	0 ,1 ,0
Y Point	0 ,0 ,1

- Using *3d Modeler* > *Coordinate System* > *Create* > *Relative CS* > *Rotated*
  Select the x axis **X:0; Y:1; Z:0** Press **Enter**
  Select the point on xy plane **X:0; Y:0; Z:1**
  Press **Enter.** Then change Coord Name From RelativeCS1 to *RelativeCS0*
- Double click *Mag* in Histery Tree, Change the Orientation Value from Global to *RelativeCS0* with **OK** to exit.

Name	Value
Name	Mag
Material	mag_aniso
Solve Inside	☑
Orientation	RelativeCS0
Model	☑
Display Wireframe	☐
Color	Edit
Transparent	0

Note: Because in Cylindrical Coordinate System, Theta direction is fixed as T(2,2), the Global CS can't be used for Mag directly. **RelativeCS0** must be created for it.

WWW.MOHANDESYAR.COM

Create the coils

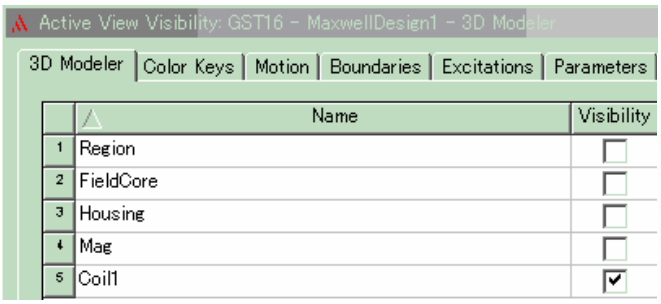
- **Create Coil named as *coil1***
  - ***Draw*** > ***Box*** then using the coordinate entry field, enter the box position
    X: 14.2, Y: 6.2, Z: -7.8, Press the **Enter** key
  - Then enter the length, width and height of the box
    dX: -28.2; dY: 6.5; dZ: 15.6, Press the **Enter** key
    Box1 will be created.

	Name	Value	Unit
	Command	CreateBox	
	Coordinate System	Global	
	Position	14.2 ,6.2 ,-7.8	mm
	XSize	-28.2	mm
	YSize	6.5	mm
	ZSize	15.6	mm

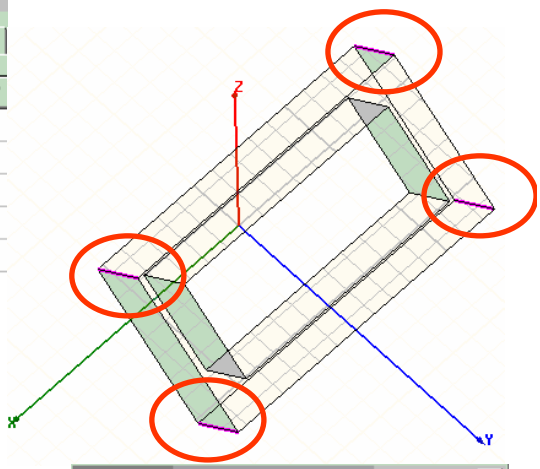
  - ***Draw*** > ***Box*** then using the coordinate entry field, enter the box position
    X: 11.2, Y: 6.2, Z: -4.8, Press the **Enter** key
  - Then enter the length, width and height of the box
    dX: -22.4; dY: 6.5; dZ: 9.6, Press the **Enter** key
    Box2 will be created.

	Name	Value	Unit
	Command	CreateBox	
	Coordinate System	Global	
	Position	11.2 ,6.2 ,-4.8	mm
	XSize	-22.4	mm
	YSize	6.5	mm
	ZSize	9.6	mm

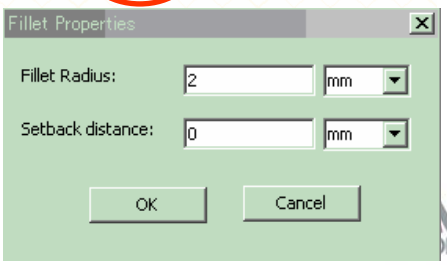
  - Pick ***Box1*** & ***Box2***, Using right mouse
    Edit > Boolean > Substract to create final ***Box1***
  - Change the name of ***Box1*** to ***Coli_1*** and set material ***Cupper*** for it
  - Select ***Coli_1*** in History Tree, Using right mouse ***Edit > Arrange > Rotate***
    select ***X*** as rotating Axis with ***45*** degree, ***OK*** to finish it.
  - ***View > Active View Visibility*** to erase the tick marks for all except ***Coil_1*** as below:



  - Change Object in Tool Bar to Edge
  - pick 4 edges of the Coil_1 as below:



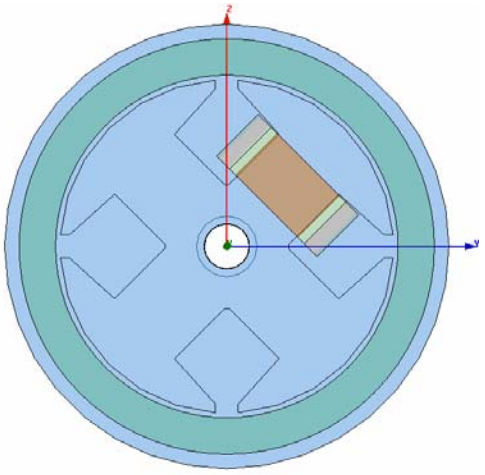
  - Select the icon of Fillet the selected edges in Tool Bar and set 2 mm as Fillet Radius



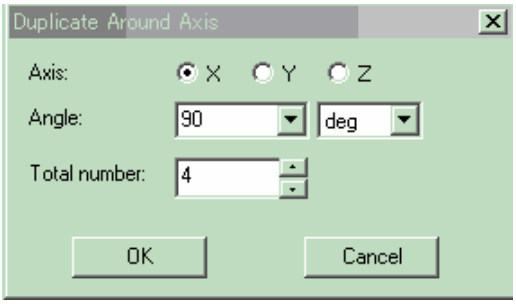
  - After OK to finished Coil_1, change the color of it as dark yellow.

WWW.MOHANDESYAR.COM

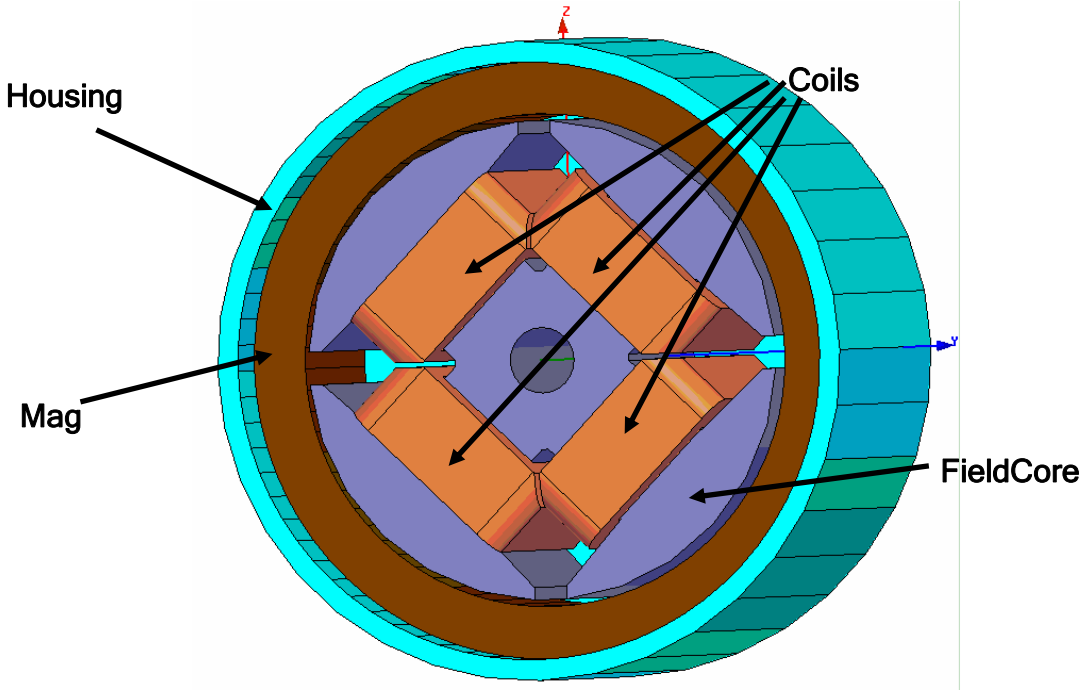
- Using Alt Key and double click the mouse on the the coordinate (0,0,0) to rotate the View as YZ plane (See right)



- Using *3d Modeler > Coordinate System > Create > Relative CS > Offiset,* input **X: 0; Y: 0.2; Z: 0.2** with **Enter** key the coordinate *CS1* will be created.
- Pick Coil_1 in History Tree, Use right mouse *Edit > Boolean > Split* and select *XY* plane with **OK**
- Pick Coil_1 in History Tree, Use right mouse *Edit > Boolean > Split* and select *XZ* plane with **OK.** The final Coil1 will be finished.
- Select *Global* coordinate in History Tree
- Pick Coil_1 in History Tree, Use right mouse *Edit > Duplicate > Around Axis* Select **X** Axis with Angle **90** & Total Number **4** All 4 coils will be finished



- Pick all Coil_1- 4 in History Tree, Use right mouse *Edit > Surface > Section* and select *YZ* plane with OK
- Pick all Section1- 4 in History Tree, Use right mouse *Edit > Boolean > Separate Bodies*
- Pick Section1_1,2_1,3_1,4_1, then Click *Delete* Key.
- Now the whole model will be as below:



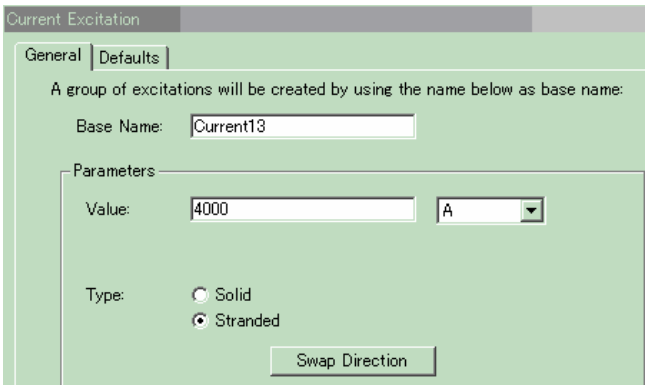
WWW.MOHANDESYAR.COM

Assign Current Source to the Terminals

- Select Coil Terminals **Section1** & **Section3** by using holding down the CTRL key and using your mouse or selecting from tree on the left hand side of the GUI

  Select the menu item *Maxwell > Excitations > Assign > Current*

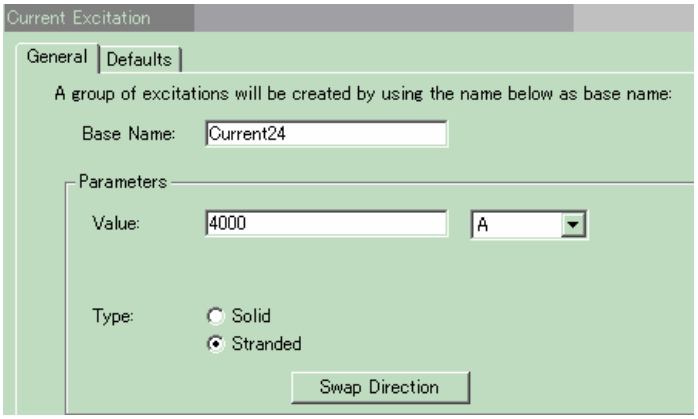
  1. Change the Base Name to **Current13**
  2. Change the value to **4000** Amps
  3. Change the type to **Stranded**



- Select Coil Terminals **Section2** & **Section4**

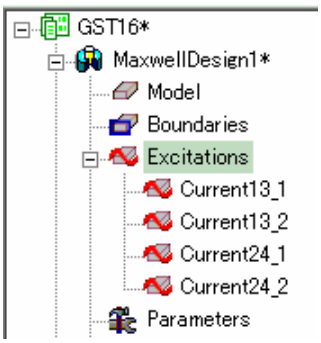
  Select the menu item *Maxwell > Excitations > Assign > Current*

  1. Change the Name to **Current24**
  2. Change the value to **4000** Amps
  3. Change the type to **Stranded**
  4. Pick **Swap Direction**



- The project tree now shows 4 separate Excitations:

  Any one of them can be selected to check the directions of the Excitation Current.



WWW.MOHANDESYAR.COM

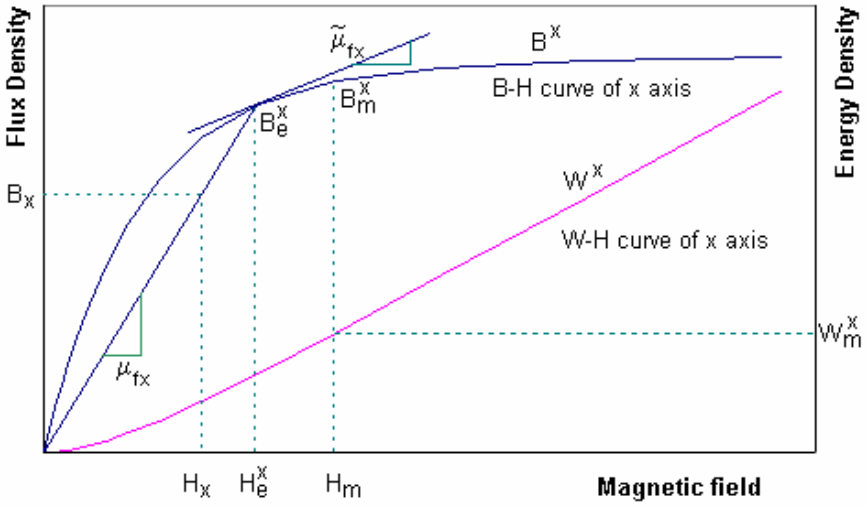
Theory of the Nonlinear Anisotropic Material for soft magnetic materials

- Basic ideas for new nonlinear anisotropic materials:
  - The cross effects of the magnetic field in the different principle directions are decoupled by introducing an equivalent magnetic field magnitude in each principle direction based on an anisotropic characterization of the energy density .
  - Only B-H curves in the principal directions of the materials – two curves in 2D or three curves in 3D – need to be measured
  - The anisotropic behavior of laminations with either isotropic or anisotropic steel is also considered.
- **Proposed approach**
  - Since applying the same value of **H** in different principle directions will cause different level of magnetic saturations, the approach refers all components of **H** to each principle direction by referring factor k to determine the saturation in that direction

$$\begin{cases} H_e^x = \sqrt{H_x{}^2 + (k_{xy}H_y)^2 + (k_{xz}k_{\mu z}H_z)^2} \\ H_e^y = \sqrt{(k_{yx}H_x)^2 + H_y{}^2 + (k_{yz}k_{\mu z}H_z)^2} \\ H_e^z = \sqrt{(k_{zx}H_x)^2 + (k_{zy}H_y)^2 + (k_{\mu z}H_z)^2} \end{cases}$$

where $k_{\mu z}$ is introduced to consider lamination effects and *k*xy is determined by magnetic energy density *w* in x and y principle directions, *k*xy = *w*x / *w*y, and similarly to other *k*'s.

  - The permeability for each principle direction is obtained in terms of the equivalent magnitude **H**e in that direction. For example, for axis **x**:



WWW.MOHANDESYAR.COM

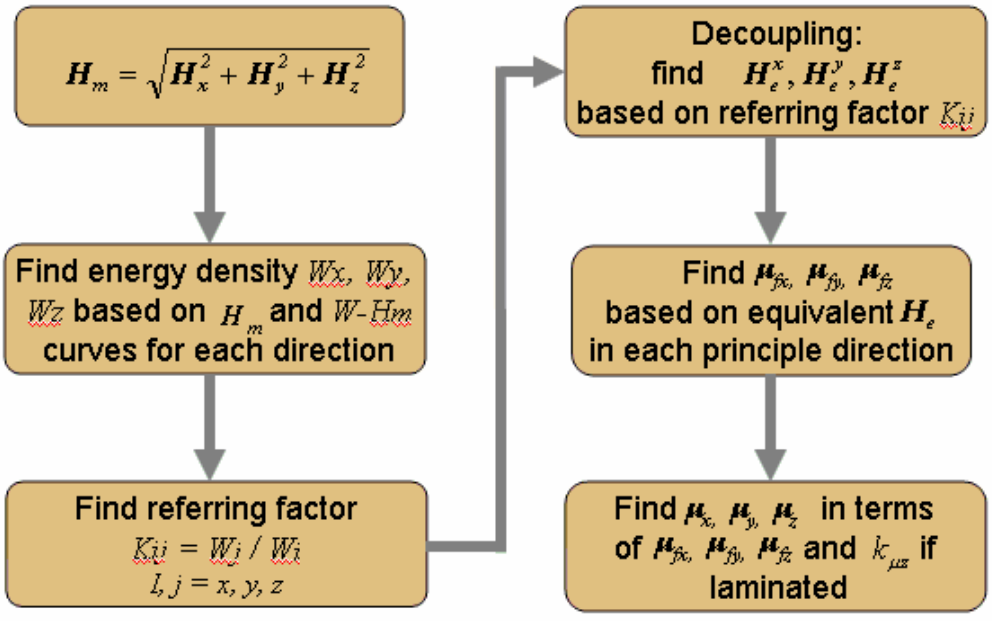
- Taking the lamination into account, the three components of permeability, based on the derived permeability (that is, μ_{fx}, μ_{fy}, μ_{fz}) in iron part, become

$$\begin{cases} \mu_x = (1 - k_{lam})\,\mu_a + k_{lam}\,\mu_{fx} \\ \mu_y = (1 - k_{lam})\,\mu_a + k_{lam}\,\mu_{fy} \\ \mu_z = k_{\mu z}\,\mu_{fz} \end{cases}$$

where k_{lam} is the stacking factor of the lamination, μ_a is the permeability of insulation part of the lamination, and

$$k_{\mu z} = \frac{\mu_a}{(1 - k_{lam})\,\mu_{fz} + k_{lam}\,\mu_a}$$

- The Modeling Sequence will be as follows:



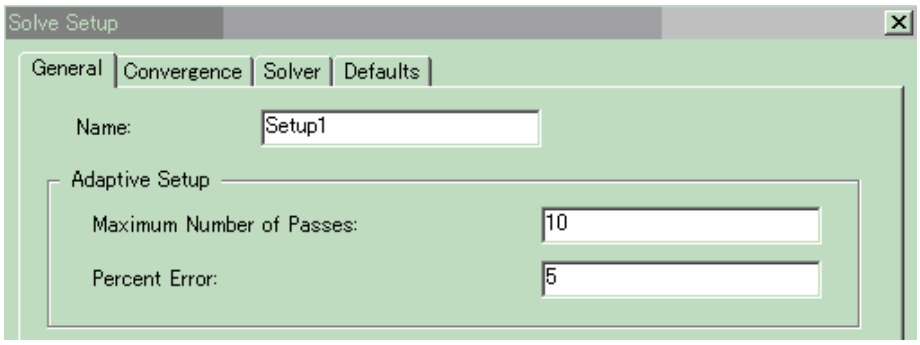
Reference:

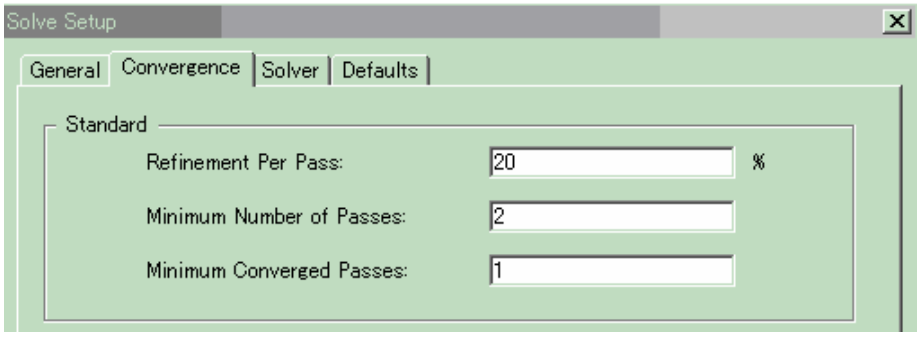
[1] D. Lin, P. Zhou, Z. Badics, W. N. Fu, Q. M. Chen and Z. J. Cendes, "A New Nonlinear Anisotropic Model for Soft Magnetic Materials" , CSY0084 of COMPUMAG'2005

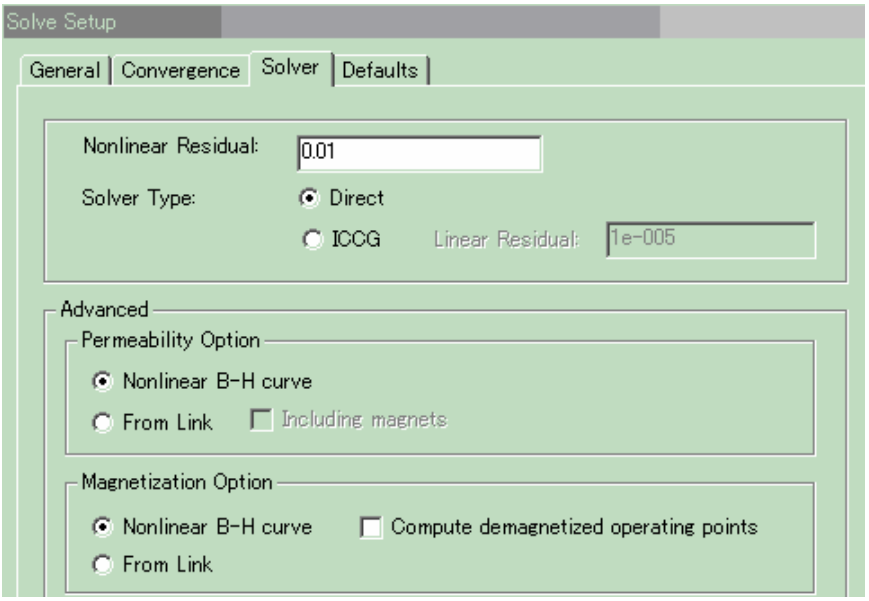
WWW.MOHANDESYAR.COM

Create an Analysis Setup

- Select the menu item ***Maxwell > Analysis Setup > Add Solution Setup***
- Select **General** and change the Percent Error from **1** to **5**
- Select **Convergence** and change the Refinement Per Pass from **30** to **20**







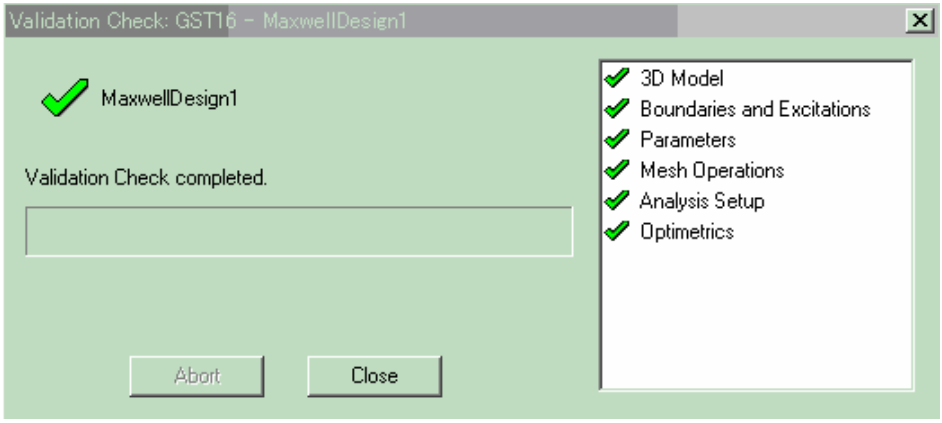
WWW.MOHANDESYAR.COM

Save the Project

- Select the menu item **File > Save As**
- From the Save As window, type in **GST16**
- Click on the **Save** button

Check the Validity of the Model

- Select the menu item ***Maxwell > Validation Check***, or click on the icon

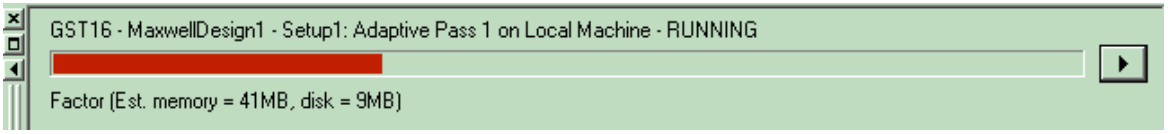


- The problem won't solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the icon

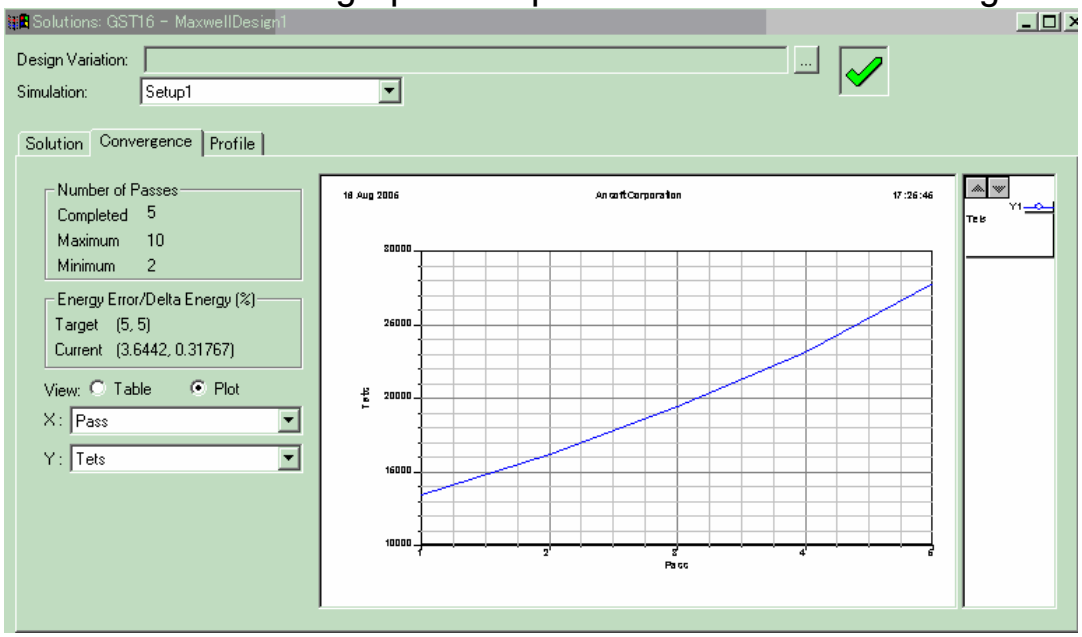
  The Analysis will be start as below:

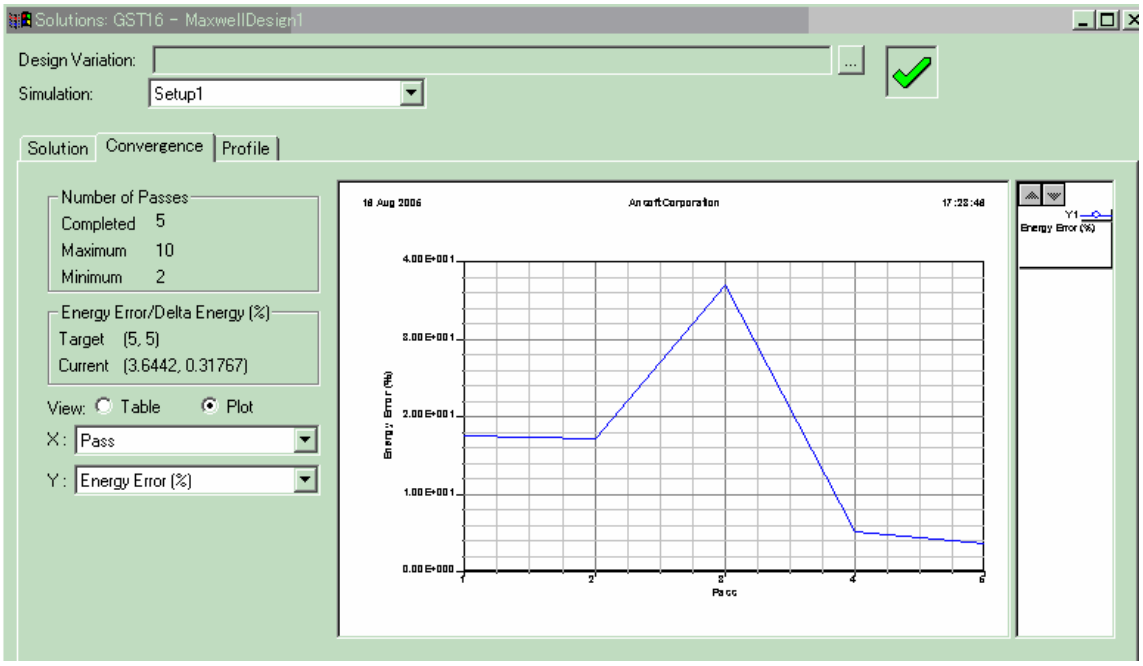


WWW.MOHANDESYAR.COM

Solution Data

- To view the Solution Data, select the menu item *Maxwell > Results > Solution Data*
- Here you can view the **Profile** and the **Convergence.** Note: The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.

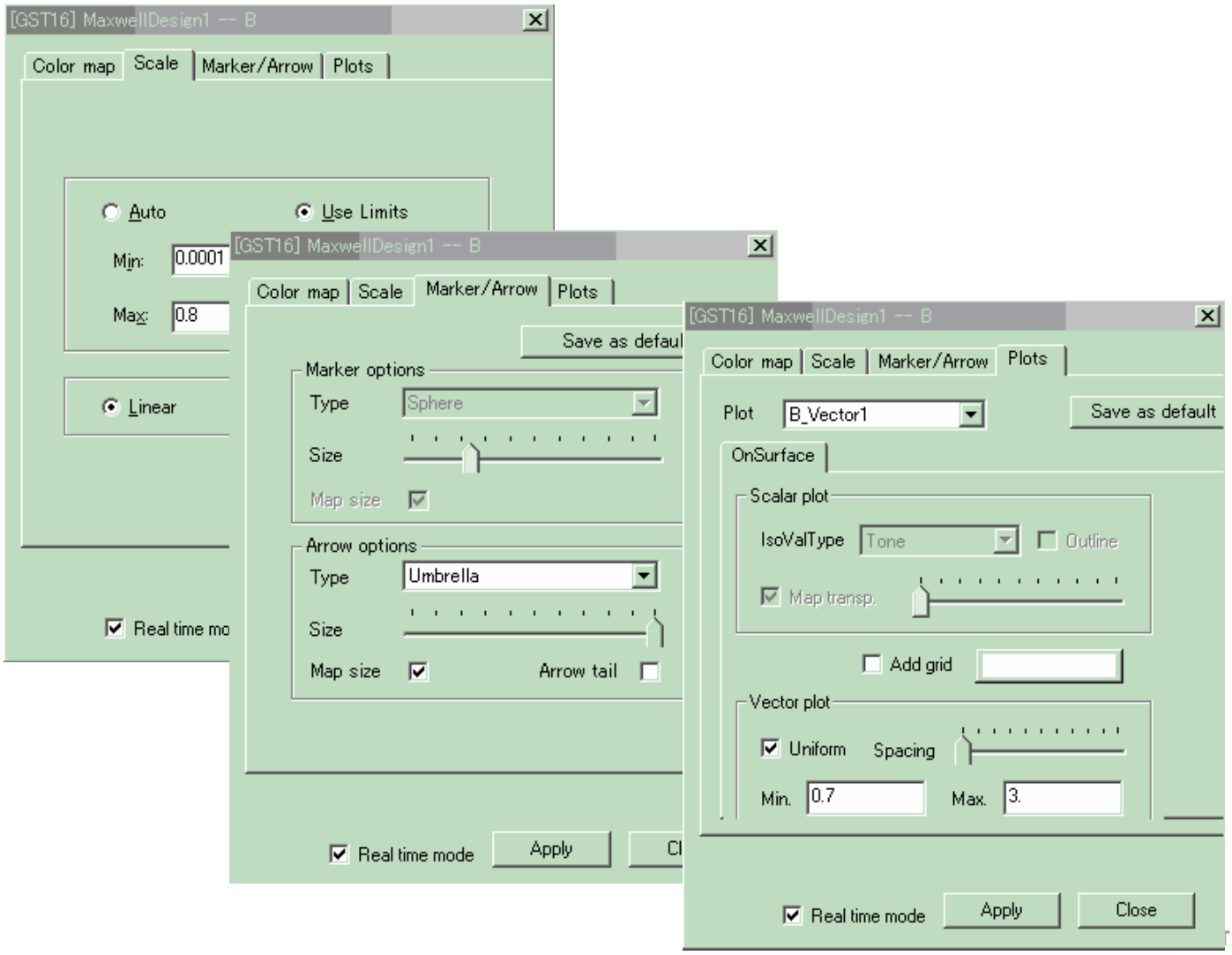




WWW.MOHANDESYAR.COM

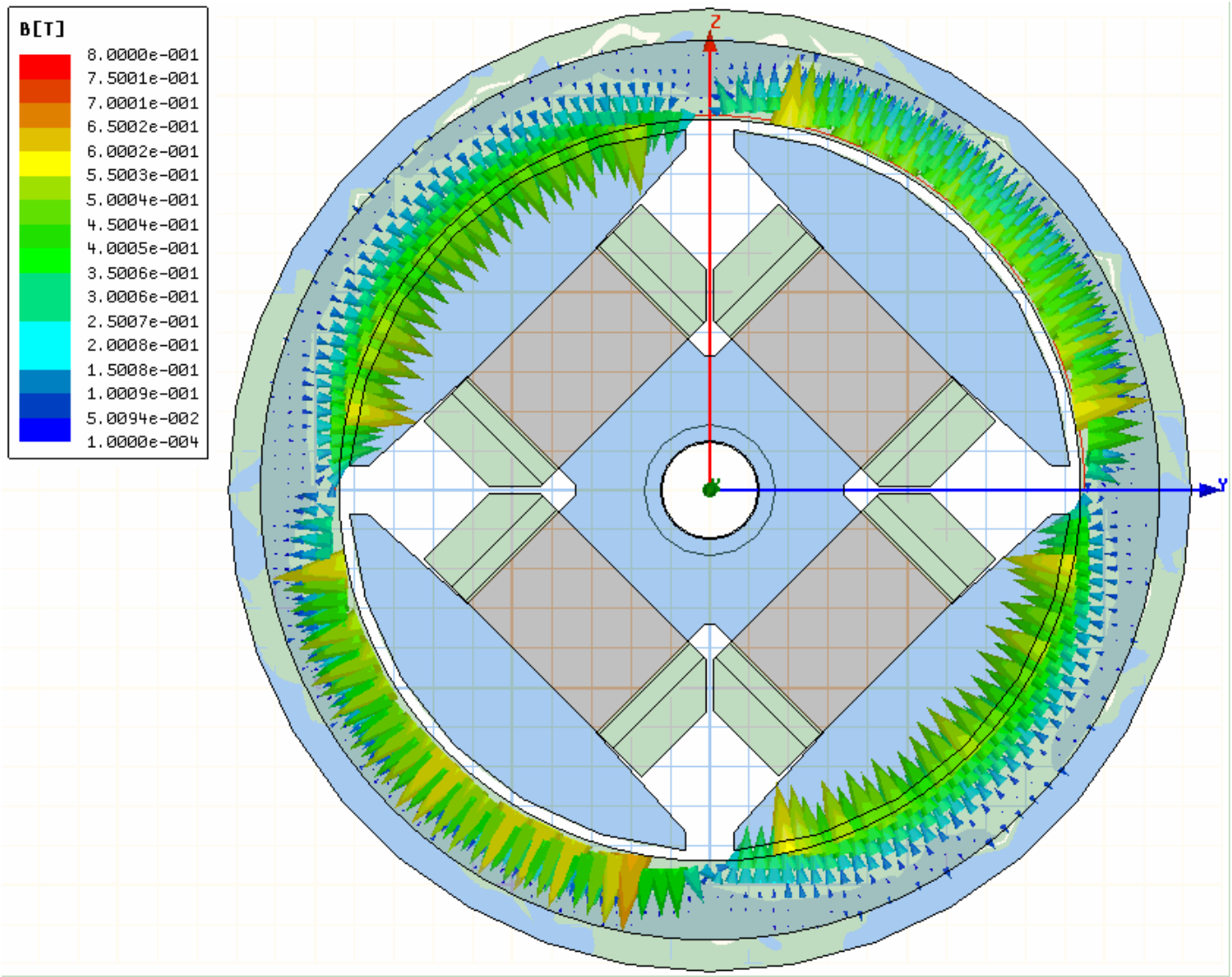
Plot Flux Density on Cross-section YZ of Mag

- To plot the flux density on YZ cut-plane of the magnetic, select the Global YZ of Planes in History Tree by clicking on it, and move the mouse to 3d Modeler Window then right mouse click and select ***Fields > B > B_Vector***, in Create Field Plot Window select ***B_Vector*** in "Quantity" and ***Mag*** in "In Volume", Press **Done**.
- The field plot is a little misleading since it show different colors across the YZ plane, but the legend displays the exact value for the full color spectrum. To change the scale, double click on the legend and then select **Scale**. Select the radio button **Use Limits** and set the **Min** to **0.0001**, the **Max** to **0.8**, click on **Apply**
- Then select **Marker/Arrow**, in Arrow options Change the Type as Umbrella and move Size Arrow to largest, click on **Apply.**
- Then select **Plots**, in Vector plot Select Uniform and move the **Spacing** size to smallest, set Min. as **0.7** and Max. as **3**, click on **Apply.**



WWW.MOHANDESYAR.COM

- The plot of the flux density on YZ cut plane of the Mag will be as below:

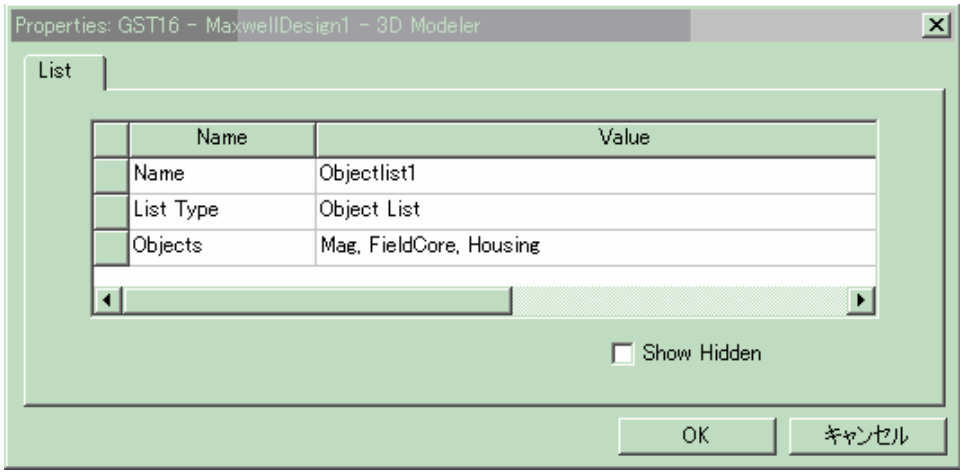


Note: Please note that if the Mag is not set up as anisotropic one (i.e. uniform Nonlinear one) the B field between two magnetic poles will be very large in the theta directions, which will be not as good as user wanted.

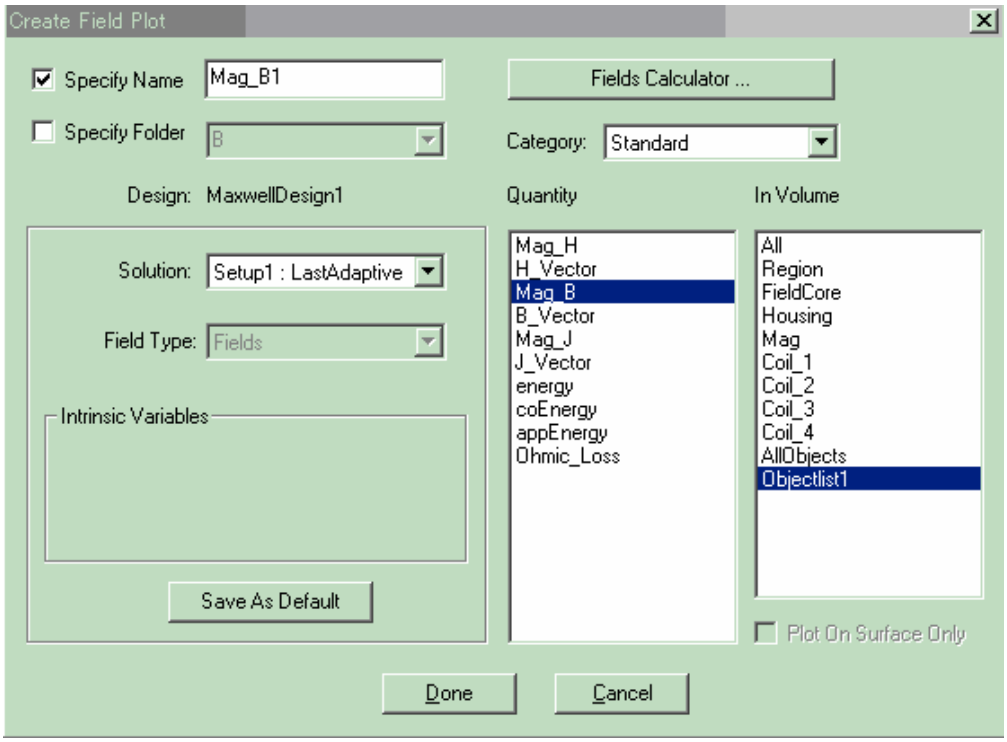
WWW.MOHANDESYAR.COM

Plot Flux Density Bmag on Cross-section YZ of Mag, FieldCore & Housing

- To plot the Field on more than one objects, the group function is needed. Pick **Mag, FieldCore** & **Housing** in History Tree, then ***3D Modeler/List/Create/Object List***, then click **OK** to create **Objectlist1**

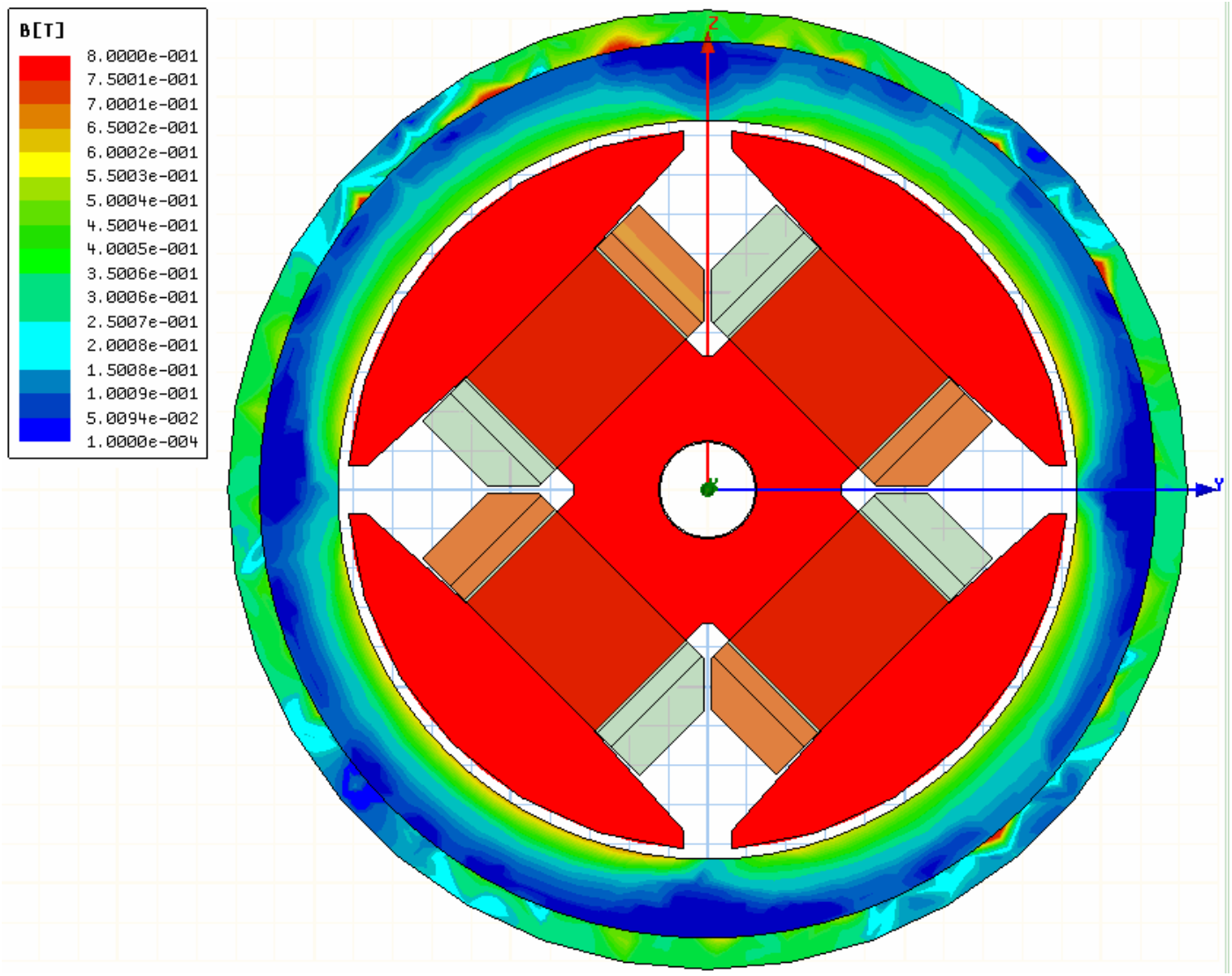


- Select the Global YZ of Planes in History Tree and move the mouse to 3d Modeler Window then right mouse click and select ***Fields > B > Mag_B,*** pick **Objectlist1** in "In Volume" then press ***Done***



WWW.MOHANDESYAR.COM

- The plot of the mag_B on YZ cut plane of the Mag, FieldCore & Housing will be as below:

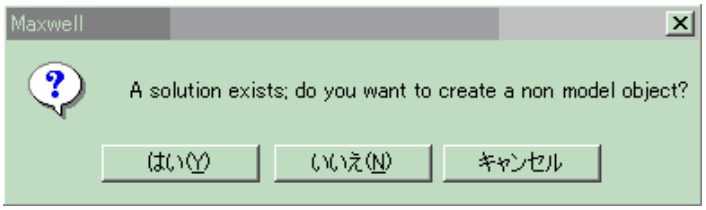


- User may double click on the legend and change **Scale** to see the detail field distributions

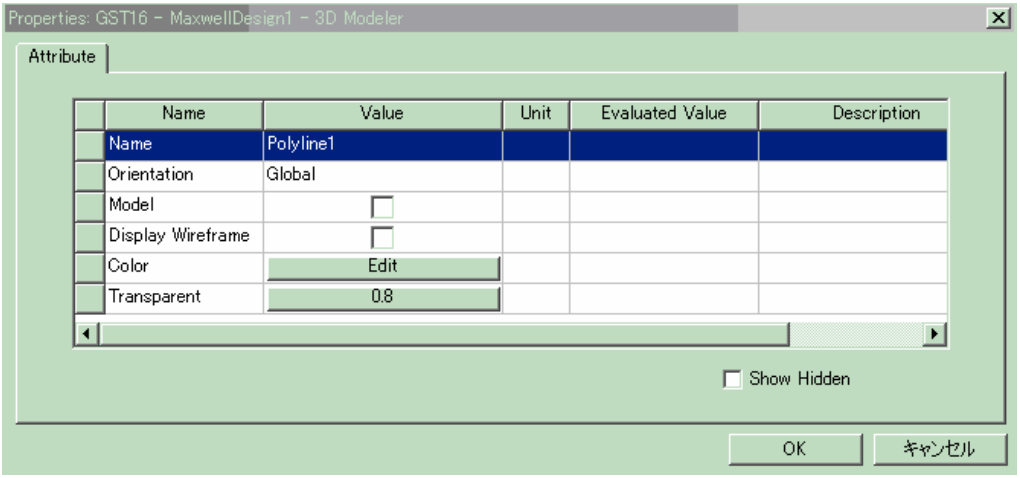
WWW.MOHANDESYAR.COM

Plot Flux Density B_radial on an arc of the Mag

- To see the detail distribution near the inner surface of the Mag, the 2d plot on a cut-line will be needed.
- Create a cut-line of arc
  - *Draw > Arc > 3 Points* and select **Yes** for Question below:



  - using the coordinate entry field, Enter first endpoint
    X: 0, Y: 19.0, Z: 0, Press the **Enter** key
  - Then enter arc intercept (middle point)
    X: 0, Y: 0, Z: 19, Press the **Enter** key
  - Enter second endpoint
    X: 0, Y: -19.0, Z: 0, Press the **Enter** key twice.

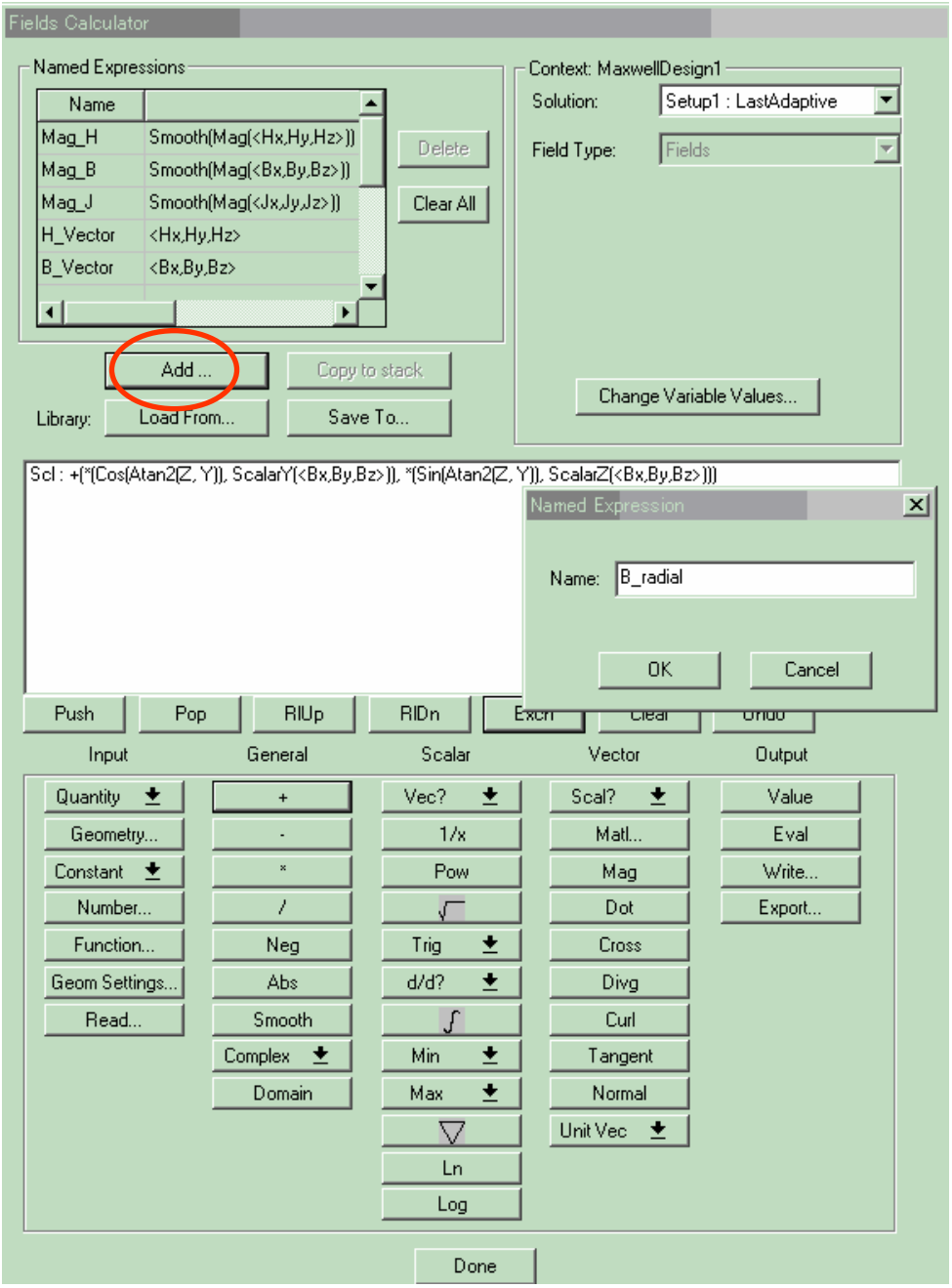


  - Press **OK** to create **Polyline1**.
- Create B_radial in Calculator
  - Select the Global YZ of Planes in History Tree and move the mouse to 3d Modeler Window then right mouse click and select *Maxwell >Fields > Calculator*
    Function > Scalar Z > OK
    Function > Scalar Y > OK
    Trig > ATan2 >Push [ATan2(Z,Y) = ATan(Z/Y)]
    Trig > Sin > Exch > Trig > Cos

WWW.MOHANDESYAR.COM

*Quantity > B > Scal? > Scalar Y > * >Exch*

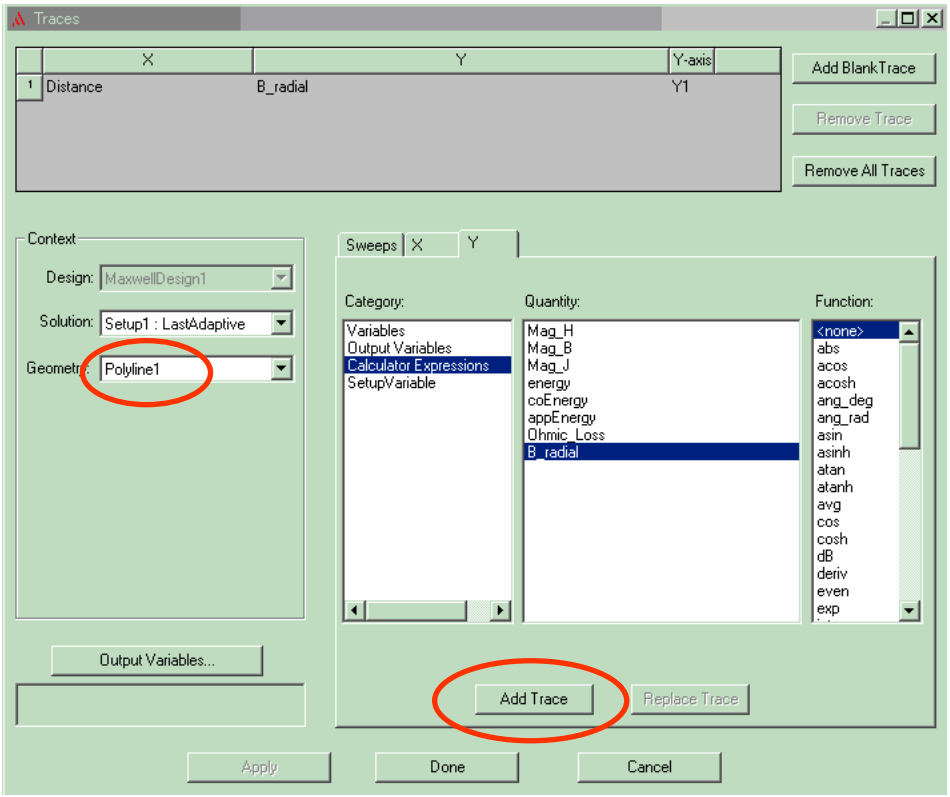
*Quantity > B > Scal? > Scalar Z > * > +* ,the Br for arc is ready.



- Pick *Add* and type the Name as *B_radial* , pick Ok to finish it.

WWW.MOHANDESYAR.COM

- In Maxwell V11 all 2d plot will be created with *Create Report*
  - *Maxwell > Result >Create Report > OK*



  - Select *Polyline1* in Geometry, pick *B_radial* in Quantity, click *Add Trace* press *Done*, the result curve on cutline will be se below:

WWW.MOHANDESYAR.COM

Chapter 6.0 - Eddy Current Examples

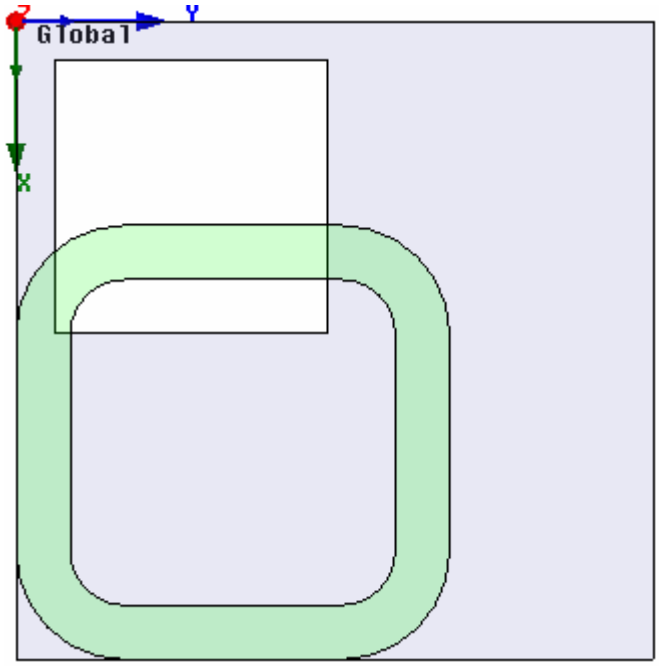
- 6.1 - Asymmetrical Conductor with a Hole

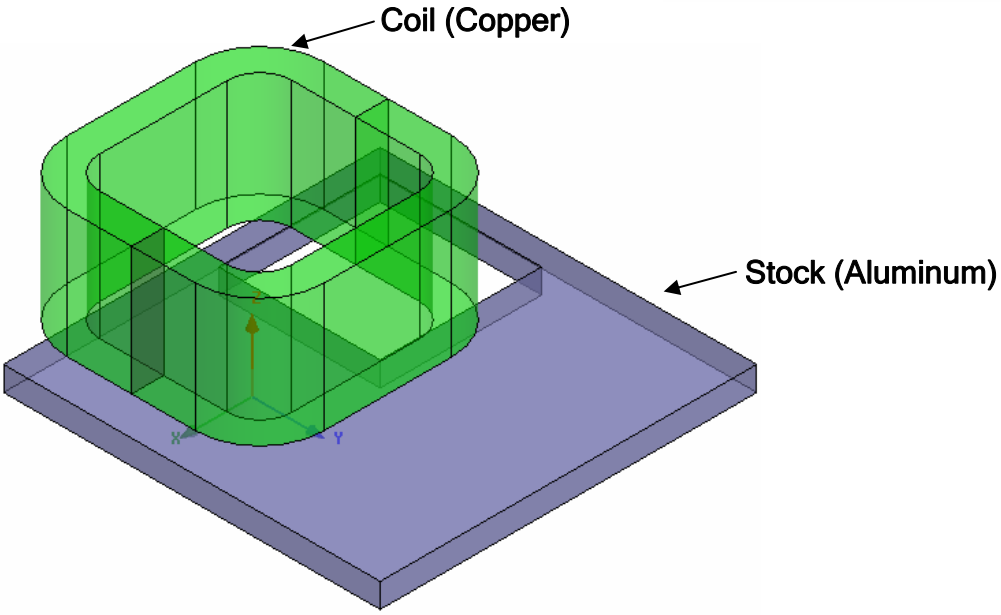
WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

The Asymmetrical Conductor with a Hole

This example is intended to show you how to create and analyze an Asymmetrical Conductor with a Hole using the Eddy Current solver in the Ansoft Maxwell 3D Design Environment.





WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Getting Started

Launching Ansoft Maxwell

1. To access Ansoft Maxwell, click the Microsoft **Start** button, select **Programs**, and select the **Ansoft > Maxwell 11** program group. **Maxwell 11.**

Setting Tool Options

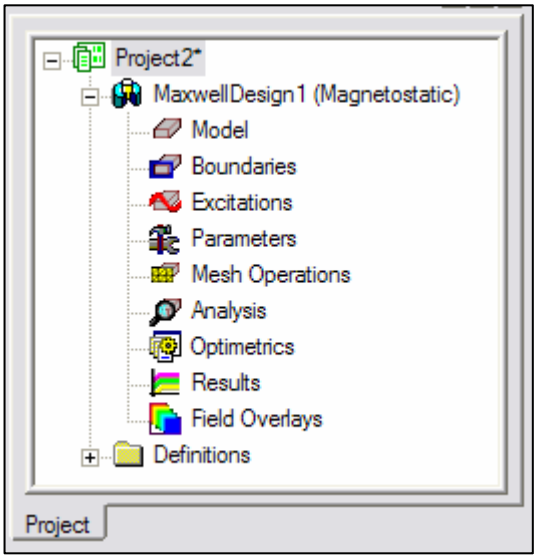
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

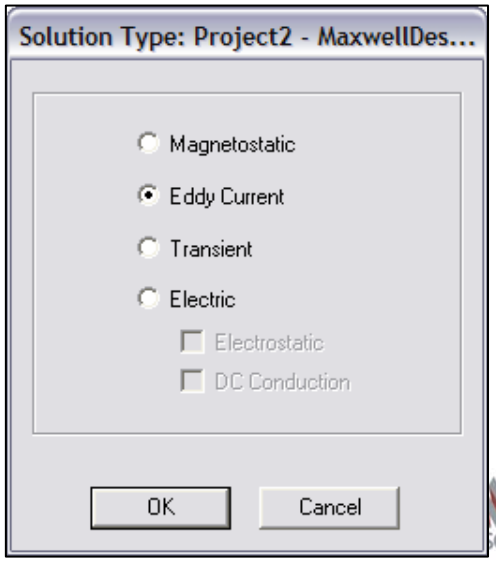
Opening a New Project

- **To open a new project:**
  - In an Ansoft Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - From the *Project* menu, select *Insert Maxwell Design*.



Set Solution Type

- **To set the solution type:**
  - Select the menu item *Maxwell > Solution Type*
  - Solution Type Window:
    - Choose **Eddy Current**
    - Click the **OK** button



WWW.MOHANDESYAR.COM

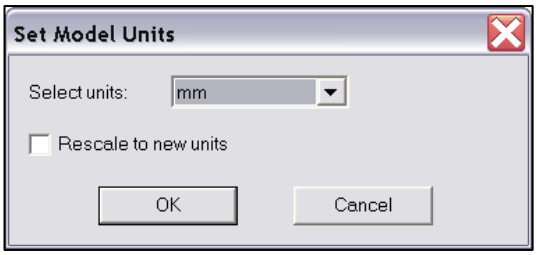
Example (Eddy Current) - Asymmetrical Conductor

Creating the 3D Model

Set Model Units

To set the units:

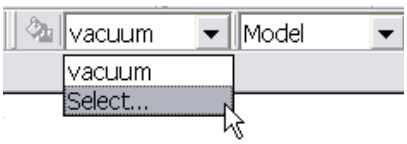
1. Select the menu item *3D Modeler > Units*
2. Set Model Units:
   1. Select Units: **mm**
   2. Click the **OK** button

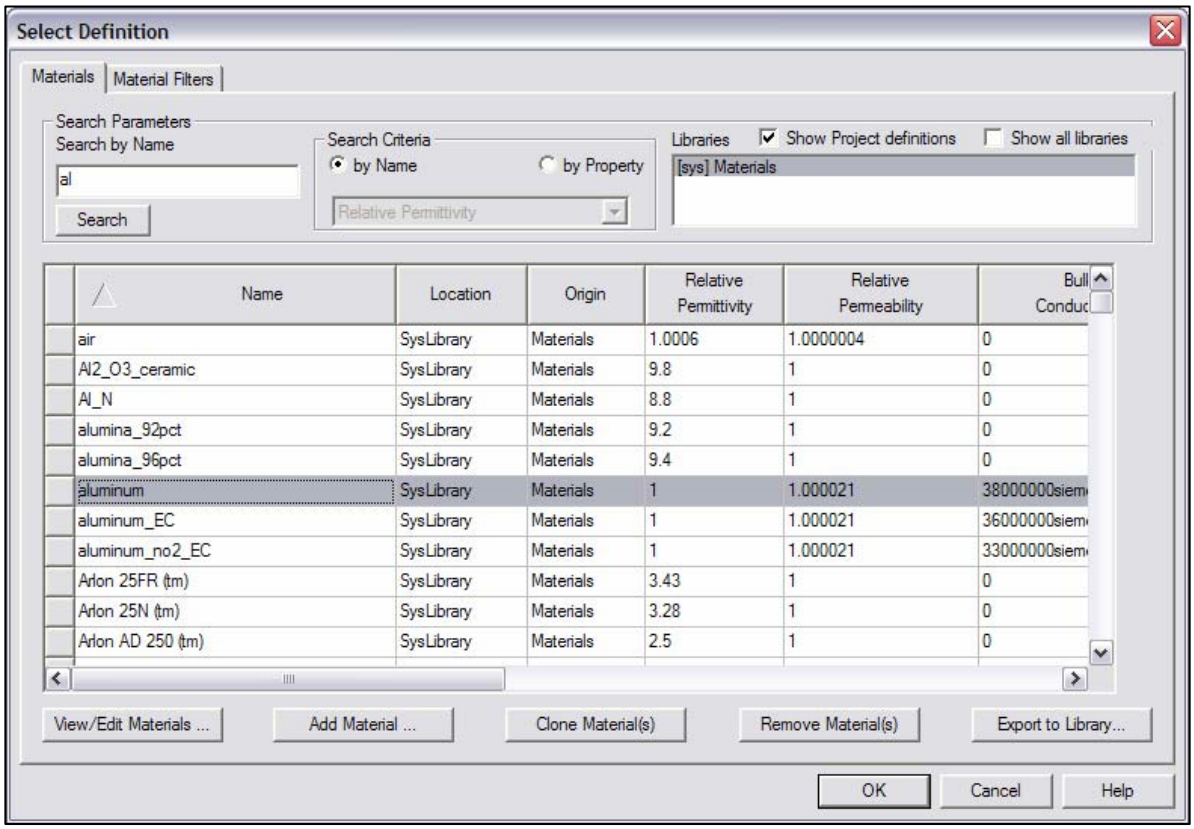


Set Default Material

To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **aluminum** in the **Search by Name** field
   2. Click the **OK** button





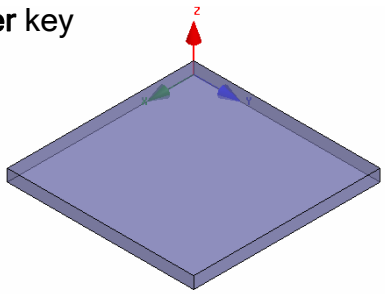
WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Create Stock

To create the stock:

1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **0.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **294.0**, dY: **294.0**, dZ: **19.0**, Press the **Enter** key



To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **stock**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View*.

Create Hole in Stock

To create the hole:

1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **18.0**, Y: **18.0**, Z: **0.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **126.0**, dY: **126.0**, dZ: **19.0**, Press the **Enter** key

To set the name:

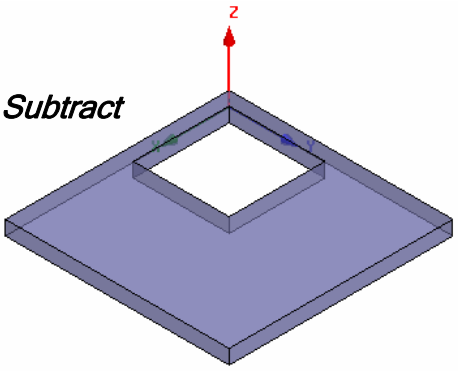
1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **hole**
3. Click the **OK** button

To select the objects:

1. Select the menu item *Edit > Select All*

To complete the stock:

1. Select the menu item *3D Modeler > Boolean > Subtract*
2. Subtract Window
   - Blank Parts: **stock**
   - Tool Parts: **hole**
   - Click the **OK** button



WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Set Default Material

1. Using the 3D Modeler Materials toolbar, choose **Select**
2. Select Definition Window:
   1. Type **copper** in the **Search by Name** field
   2. Click the **OK** button

Create Coil (Hole)

To create coil:

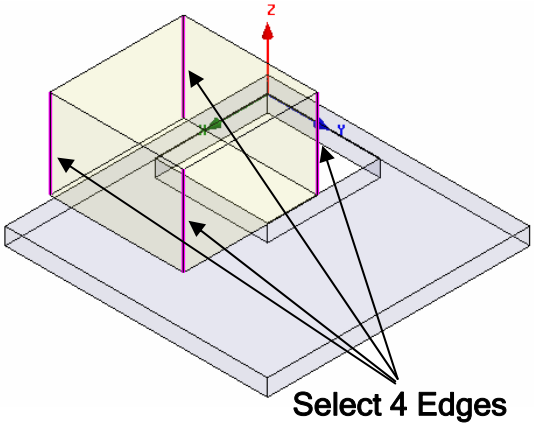
1. Select the menu item *Draw > Box*
2. Using the coordinate entry fields, enter the box position
   - X: **119.0**, Y: **25.0**, Z: **49.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
   - dX: **150.0**, dY: **150.0**, dZ: **100.0**, Press the **Enter** key

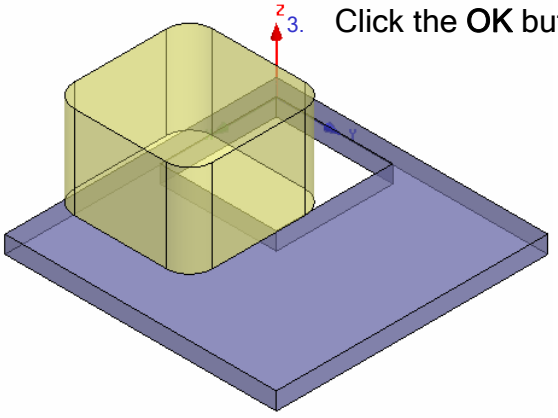
To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **coil_hole**
3. Click the **OK** button

To create the filets:

1. Select the menu item *Edit > Select > Edges*.
2. Using the mouse graphically select the 4 z-directed edges. Hold down the **CTRL** key to make multiple selections.
3. Select the menu item *3D Modeler > Fillet*
4. Fillet Properties
   1. Fillet Radius: **25mm**
   2. Setback Distance: **0mm**
   3. Click the **OK** button



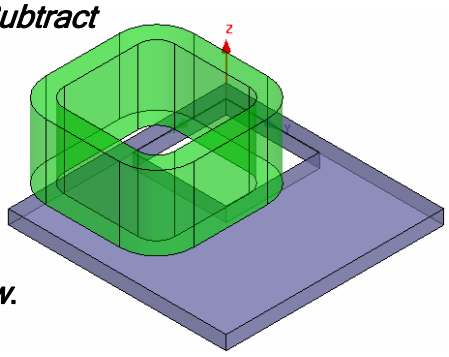


WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Create Coil

- **To create coil:**
  1. Select the menu item *Draw > Box*
  2. Using the coordinate entry fields, enter the box position
     - X: **94.0**, Y: **0.0**, Z: **49.0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the opposite corner of the box:
     - dX: **200.0**, dY: **200.0**, dZ: **100.0**, Press the **Enter** key
- **To set the name:**
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **coil**
  3. Click the **OK** button
- **To create the filets:**
  1. Select the menu item *Edit > Select > Edges*.
  2. Using the mouse graphically select the 4 z-directed edges. Hold down the **CTRL** key to make multiple selections.
  3. Select the menu item *3D Modeler > Fillet*
  4. Fillet Properties
     1. Fillet Radius: **50mm**
     2. Setback Distance: **0mm**
     3. Click the **OK** button
- **To select the object for subtract**
  1. Select the menu item *Edit > Select > Objects*
  2. Select the menu item *Edit > Select > By Name*
  3. Select Object Dialog,
     1. Select the objects named: **coil, coil_hole**
     2. Click the **OK** button
- **To complete the coil:**
  1. Select the menu item *3D Modeler > Boolean > Subtract*
  2. Subtract Window
     - Blank Parts: **coil**
     - Tool Parts: **coil_hole**
     - Click the **OK** button
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*.

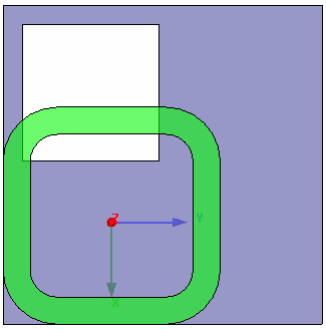


WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

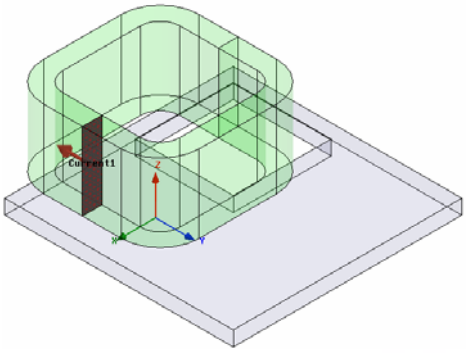
Create Offset Coordinate System

- **To create an offset Coordinate System:**
  1. Select the menu item *3D Modeler > Coordinate System > Create > Relative CS > Offset*
  2. Using the coordinate entry fields, enter the origin
     - X: **200.0**, Y: **100.0**, Z: **0.0**, Press the **Enter** key



Create Excitation

- **Object Selection**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **coil**
     2. Click the **OK** button
- **Section Object**
  1. Select the menu item *Edit > Surface > Section*
     1. Section Plane: *XZ*
     2. Click the **OK** button
- **Separate Bodies**
  1. Select the menu item *Edit > Boolean > Separate Bodies*
- **Assign Excitation**
  1. Select the menu item *Maxwell > Excitations > Assign > Current*
  2. Current Excitation : General
     1. Name: **Current1**
     2. Value: **2742 A**
     3. Type: **Stranded**
     - **Note:** The current flow should be counter-clockwise when viewing the coil from above. Use the **Swap Direction** button to change the direction.
  3. Click the **OK** button



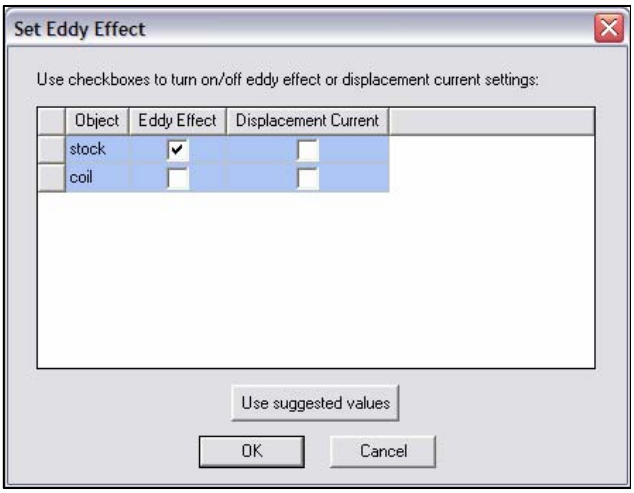
WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Set Eddy Effect

To set the eddy effect for the stock object

1. Select the menu item *Maxwell > Excitations > Set Eddy Effects*
2. Set Eddy Effect,
   1. Check the settings as shown
   2. Click the **OK** button



Show Conduction Path

Show Conduction Path

1. Select the menu item **Maxwell > Excitations > Conduction Paths > Show Conduction Paths**
2. From the Conduction Path Visualization dialog, select the rows to visualize the conduction path in on the 3D Model.
3. Click the **Close** button

Define a Region

To define a Region:

1. Select the menu item *Draw > Region*
   1. Padding Date: **One**
   2. Padding Percentage: **300**
   3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Analysis Setup

Creating an Analysis Setup

- **To create an analysis setup:**
  1. Select the menu item *Maxwell > Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **General** tab:
        - Percent Error: **2**
     2. Click the **Convergence** tab:
        - Refinement Per Pass: **50 %**
     3. Click the **Solver** tab:
        - Adaptive Frequency: **200 Hz**
     4. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item *File > Save As*.
  2. From the **Save As** window, type the Filename: **maxwell_asymcond**
  3. Click the **Save** button

Analyze

Model Validation

- **To validate the model:**
  1. Select the menu item *Maxwell > Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item *Maxwell > Analyze All*

WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Create Reports

Create z-component of B-Field (real part) vs. Distance plot on a line

- **To create a line:**
  1. Make sure that the **Global** Coordinate System is selected:
     - **3D Modeler > Coordinate System > Set Working CS**
  2. Select the menu item *Draw > Line*
  3. When the dialog appears asking to create a non-model object, click the **Yes** button.
  4. Select the menu item *Draw > Line*
  5. Using the coordinate entry fields, enter the vertex point:
     - X: **0.0**, Y: **72.0**, Z: **34.0**, Press the **Enter** key
  6. Using the coordinate entry fields, enter the vertex point:
     - X: **288.0**, Y: **72.0**, Z: **34.0**, Press the **Enter** key
  7. Using the mouse, right-click and select **Done**
- **To set the name:**
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **FieldLine**
  3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

- **To calculate real part of the z-component of B-field (use the fields calculator)**
  1. Select the menu item **Maxwell > Fields > Calculator**
  2. Select **Quantity: B**
  3. Select **Vector: Scal? > Scalar Z**
  4. Select **General: Complex > Real**
  5. Click the **Smooth** button
  6. Click the **Number** Button
     1. Type: **Scalar**
     2. Value: **10000**
     3. Click the **OK** button
  7. Click the ***** button
  8. Click the **Add** button
  9. Named Expression
     1. Name: **Bz_real**
     2. Click the **OK** button
  10. Click the **Done** button

CVc : <Bx,By,Bz>

CSc : ScalarZ(<Bx,By,Bz>)

Scl : Real(ScalarZ(<Bx,By,Bz>))

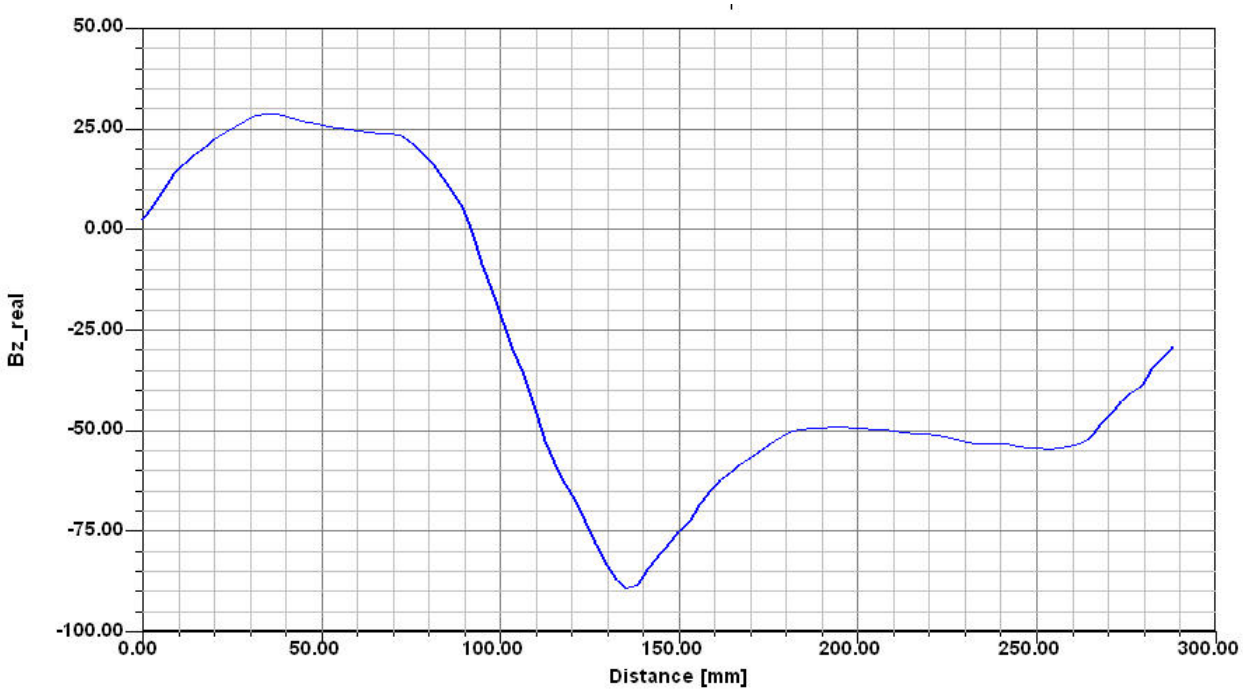
Scl : *(Smooth(ScalarZ(Real(<Bx,By,Bz>))), 10000)

Create Report

1. Select the menu item *Maxwell > Results > Create Report*
2. Create Report Window:
   1. Report Type: **Fields**
   2. Display Type: **Rectangular**
   3. Click the **OK** button
3. Traces Window:
   1. Solution: **Setup1: LastAdaptive**
   2. Domain: **FieldLine**
   3. Click the **Y** tab
      1. Category: **Calculator Expressions**
      2. Quantity: **Bz_real**
      3. Function: **<none>**
      4. Click the **Add Trace** button
   4. Click the **Done** button

WWW.MOHANDESYAR.COM

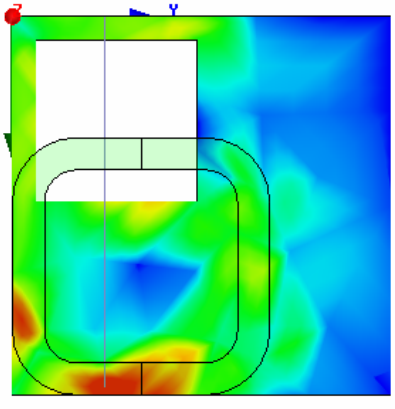
Example (Eddy Current) - Asymmetrical Conductor



Field Overlays

Create Field Overlay

- **To select an object**
  - Select the menu item *Edit > Select > By Name*
  - Select Object Dialog,
    - Select the objects named: **stock**
    - Click the **OK** button
- **To create a field plot:**
  1. Select the menu item *Maxwell > Fields > Fields > J > Mag_J*
  2. Create Field Plot Window
     1. Solution: **Setup1 : LastAdaptive**
     2. Quantity: **Mag_J**
     3. In Volume: **All**
     4. Plot on Surface Only: ☑ **Checked**
     5. Click the **Done** button

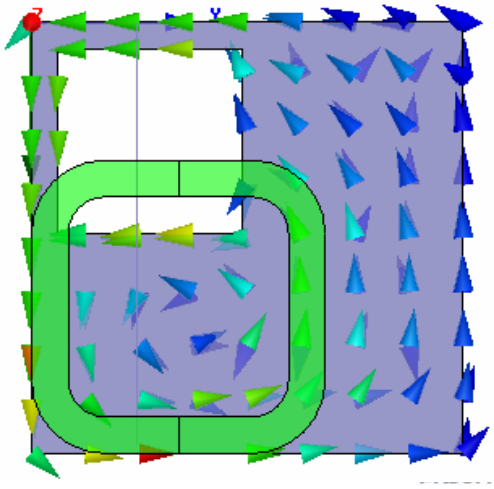


WWW.MOHANDESYAR.COM

Example (Eddy Current) - Asymmetrical Conductor

Create Field Overlay

- **To select an object**
  - Select the menu item *Edit > Select > By Name*
  - Select Object Dialog,
    - Select the objects named: **stock**
    - Click the **OK** button
- **To create a field plot:**
  1. Select the menu item *Maxwell > Fields > Fields > J > Vector_J*
  2. Create Field Plot Window
     1. Solution: **Setup1 : LastAdaptive**
     2. Quantity: **Vector_J**
     3. In Volume: **All**
     4. Plot on Surface Only: ☑ **Checked**
     5. Click the **Done** button
- **To modify a Magnitude field plot:**
  1. Select the menu item *Maxwell > Fields > Modify Plot Attributes*
  2. Select Plot Folder Window:
     1. Select: **J**
     2. Click the **OK** button
  3. J Window:
     1. Click the **Plots** tab
        1. Plot: **Vector_J1**
        2. Vector Plot
        3. Set the Spacing slider to the minimum
     2. Click the **Marker/Arrow** tab
        1. Arrow Options
        2. Map Size: ☐ **Unchecked**
        3. Arrow Tail: ☐ **Unchecked**
        4. Use the slider bar to adjust the size
     2. Click the **Close** button



WWW.MOHANDESYAR.COM

Chapter 7.0 - Transient Examples

- 7.1 - Switched Reluctance Motor (Stranded Conductors)

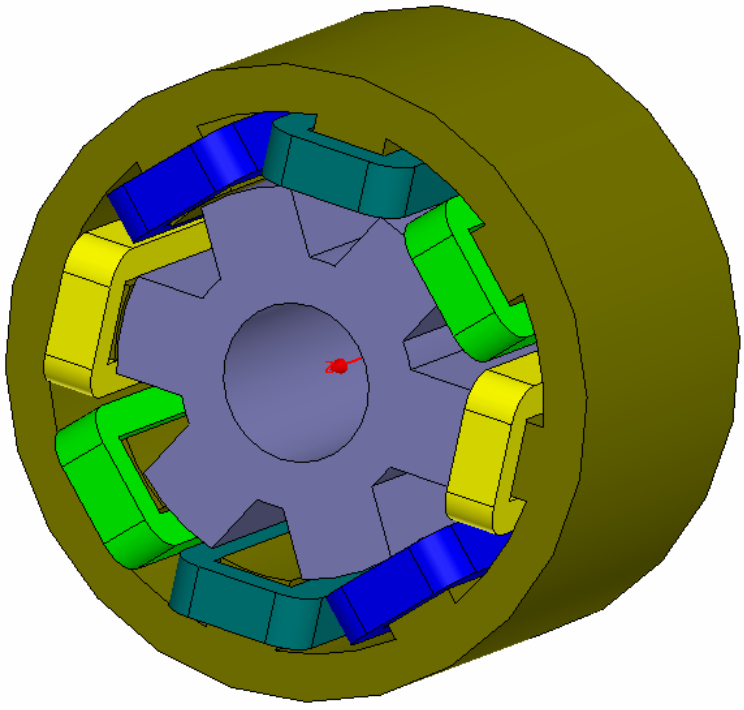
WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Stranded Conductors

- This example is intended to show you how to create and analyze a transient problem on a Switched Reluctance Motor geometry using the Transient solver in the Ansoft Maxwell 3D Design Environment.
- Within the Maxwell 3D Design Environment, solid coils can be modeled as Stranded Conductors. There are many advantages to using Stranded Conductors when modeling coils that have multiple turns. The first obvious advantage is that a coil with multiple wires, say 2500, can be modeled as a single object as opposed to modeling each wire which would be impracticable. Defining a Stranded Conductor means that the current density will be uniform throughout the cross section of the conductor.
- The example that will be used to demonstrate how Stranded Conductors are implemented is a switched Reluctance Motor. This switched reluctance motor will have four phases and two coils per phase, thus we can show how independent coils can be grouped to create windings.

- **Magnetorstatic:** See **Example 5.3** for setup instructions for this design.

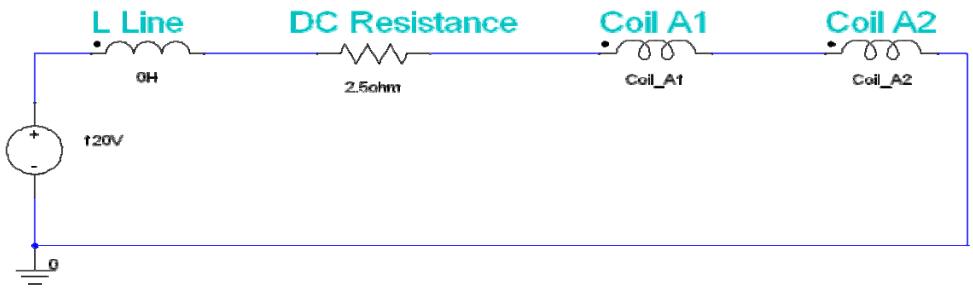


WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Theory - Transient Solver

- When creating Windings in the Transient solver, it is assumed that all of the coils used to make up that winding are connected in series.
- When creating a Winding and using voltage sources, the **Winding Panel** asks for the **Initial Current, Resistance, Inductance**, and **Voltage**.
  - **Initial Current**: This is an initial condition used by the solver
  - **Resistance**: This is the DC resistance of the total winding; for the Phase_A winding, this is the resistance of Coil_A1 and Coil_A2 in series.
  - **Inductance**: This is any extra inductance that is not modeled that needs to be added. For example, and additional line inductance or source inductance.
  - **Voltage**: This is the source voltage which can be a constant, function, or piecewise linear curve.
  - A sketch of the Phase_A Winding circuit is:

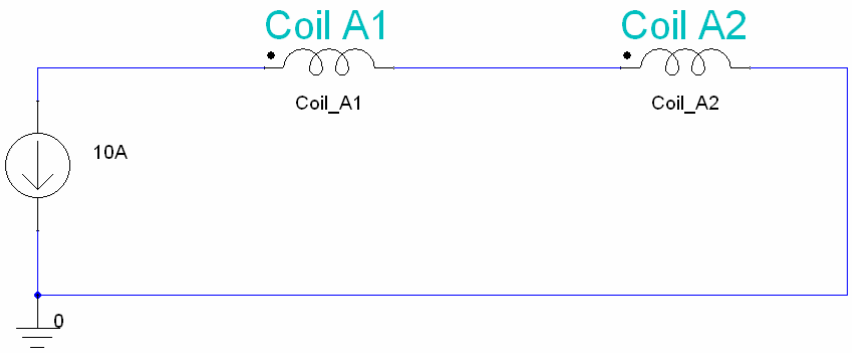


WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Theory - Transient Solver (Continued)

- If the Winding was defined as a **Current Source** instead of a **Voltage Source**, the only additional field to modify is the initial current. The circuit would look like this:



  - The DC Resistance and Extra Inductance is not needed since this is a current source and its value is guaranteed regardless of any value for the DC Resistance or Extra Inductance.
- The third option for the Winding setup is **External**. This means that there is an external circuit that is made up of arbitrary components. Please refer to the Topic paper on **External Circuits** for the details on how this is implemented.
- Please note that if the two coils that make up the **Phase_A** winding were connected in parallel instead of series, then two separate Windings would need to be created.
- In regards to the current density, the Transient solver treats stranded conductors the same as in the Magnetostatic solver; that is, the current density is uniform across the terminal and the solver calculates the magnetic field intensity **H** directly and the current density vector **J** indirectly.
- There are two options when defining the type of winding: **Solid** or **Stranded**. This write up is for Stranded Windings only. For a full description of how Solid windings are implemented, please refer to the Topic paper **Solid Conductors**.

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Ansoft Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - User Defined Primitives (UDP): **Switched Reluctance Motor**
    - Primitives: **Regular Polyhedron**
    - Surface Operations: **Section**
    - Boolean Operations: **Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Transient**
  - Results
    - **Field Calculator**
  - Field Overlays:
    - **Magnitude B**

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Getting Started

Launching Maxwell

1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 11.**

Setting Tool Options

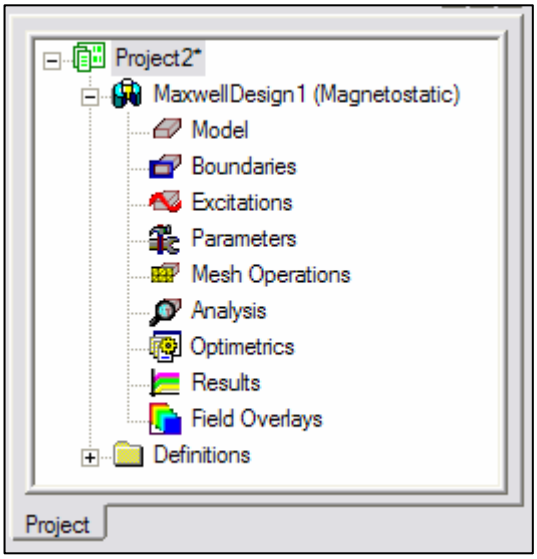
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > 3D Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

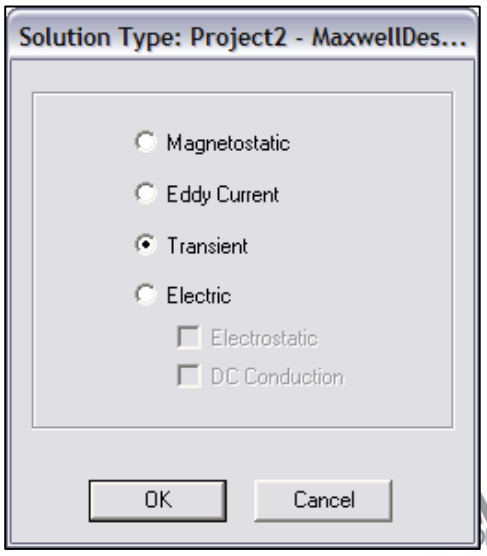
Opening a New Project

- **To open a new project:**
  - In an Ansoft Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - From the *Project* menu, select *Insert Maxwell Design*.



Set Solution Type

- **To set the solution type:**
  - Select the menu item *Maxwell > Solution Type*
  - Solution Type Window:
    - Choose **Transient**
    - Click the **OK** button



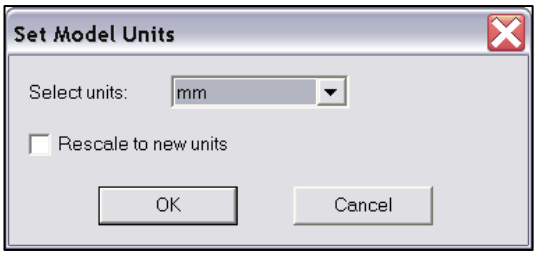
WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Creating the 3D Model

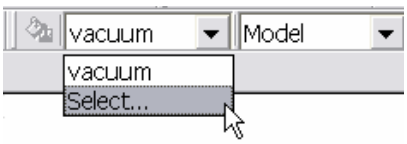
Set Model Units

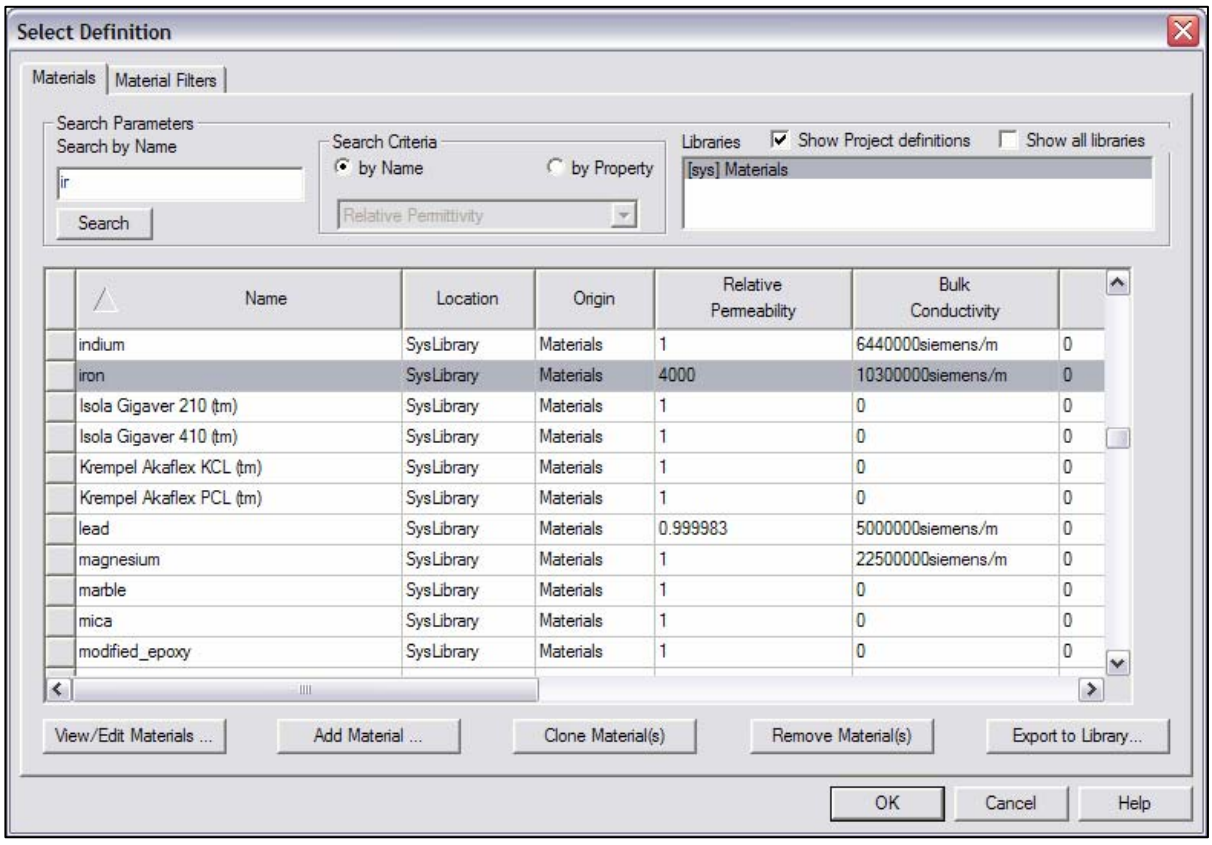
- **To set the units:**
  1. Select the menu item *3D Modeler > Units*
  2. Set Model Units:
     1. Select Units: **mm**
     2. Click the **OK** button



Set Default Material

- **To set the default material:**
  1. Using the 3D Modeler Materials toolbar, choose **Select**
  2. Select Definition Window:
     1. Type **iron** in the **Search by Name** field
     2. Click the **OK** button





WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Create Rotor

To create the rotor:

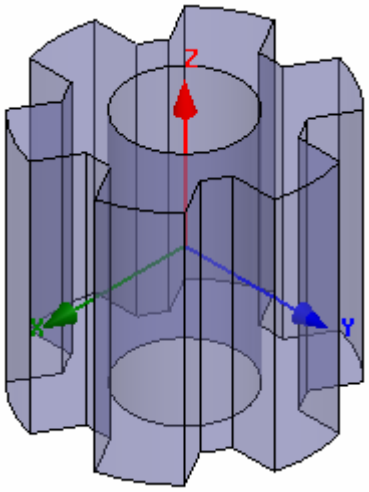
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
    1. For the value of **DiaGap,** type: **70**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **30**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **65**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **6**, Click the **Tab** key to accept
    5. For the value of **ThkYoke**, type: **9**, Click the **Tab** key to accept
    6. For the value of **Embrace**, type: **0.5**, Click the **Tab** key to accept
    7. For the value of **EndExt**, type: **0**, Click the **Tab** key to accept
    8. For the value of **InfoCore**, type: **0**, Click the **Tab** key to accept
    9. Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Rotor**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View*.



WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Create Stator and Coils:

To create the Stator and Coils

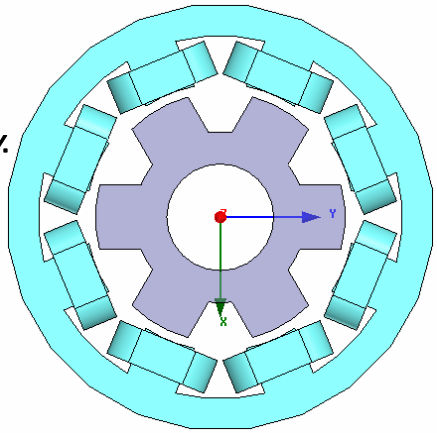
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
    1. For the value of **DiaGap,** type: **75**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **120**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **65**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **8**, Click the **Tab** key to accept
    5. For the value of **ThkYoke**, type: **9**, Click the **Tab** key to accept
    6. For the value of **Embrace**, type: **0.5**, Click the **Tab** key to accept
    7. For the value of **EndExt**, type: **1**, Click the **Tab** key to accept
    8. For the value of **InfoCore**, type: **1**, Click the **Tab** key to accept
    9. Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Stator**
3. Click the **OK** button

To fit the view:

1. Select the menu item *View > Fit All > Active View.*



Ungroup the Stator and Coils

To separate bodies

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Stator**
    2. Click the **OK** button
3. Select the menu item, *3D Modeler > Boolean > Separate Bodies*

Process the Coils:

Select Coils

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Stator_2, Stator_3, Stator_4, Stator_5, Stator_6, Stator_7, Stator_8**
    2. Click the **OK** button

Delete Coils

1. Select the menu item *Edit > Delete*

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Create Terminals

Section Coils

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Stator_1**
    2. Click the **OK** button

Section Object

1. Select the menu item *Edit > Surface > Section*
    1. Section Plane: *XY*
    2. Click the **OK** button

Separate Bodies

1. Select the menu item *Edit > Boolean > Separate Bodies*

Delete

1. Select the menu item *Edit > Delete*

Section Coils

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Section1**
    2. Click the **OK** button

Assign Excitation

1. Select the menu item *Maxwell > Excitations > Assign > Coil Terminals*
2. Coil Terminal Excitation : General
    1. Name: **CoilTerminal1**
    2. Number of Conductors: **150**
3. Click the **OK** button

Verify Duplicate Boundary Option is Set

Section To set the tool options:

1. Select the menu item *Maxwell > Options > Maxwell Options*
2. Maxwell Options Window:
    1. Click the **General** tab
        - Duplicate boundaries with geometry: ☑ **Checked**
    2. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Change Properties

Change Properties

1. Select the menu item *Maxwell > List*
2. Design List Window
    1. From the list, select row: **Stator_1**
    2. Click the **Properties** button
    3. Properties Window
        1. Name: **Coil**
        2. Material: **copper**
        3. Click the **OK** button
    4. From the list, select row: **Section1**
    5. Click the **Properties** button
    6. Properties Window
        1. Name: **Terminal**
        2. Click the **OK** button
    7. Click the **Done** button

Duplicate Coil and Terminal

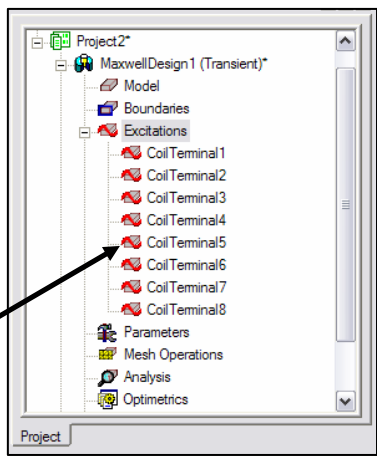
To select objects

1. Select the menu item *Edit > Select > By Name*
2. Select Object Dialog,
    1. Select the objects named: **Coil, Terminal**
    2. Click the **OK** button

To duplicate the objects:

1. Select the menu item, *Edit > Duplicate > Around Axis*
2. Duplicate Around Axis Window
    1. Axis: **Z**
    2. Angle: **45deg**
    3. Total Number: **8**
    4. Click the **OK** button
3. Click the **OK** button

Project Tree: Eight Excitations



WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Add Windings

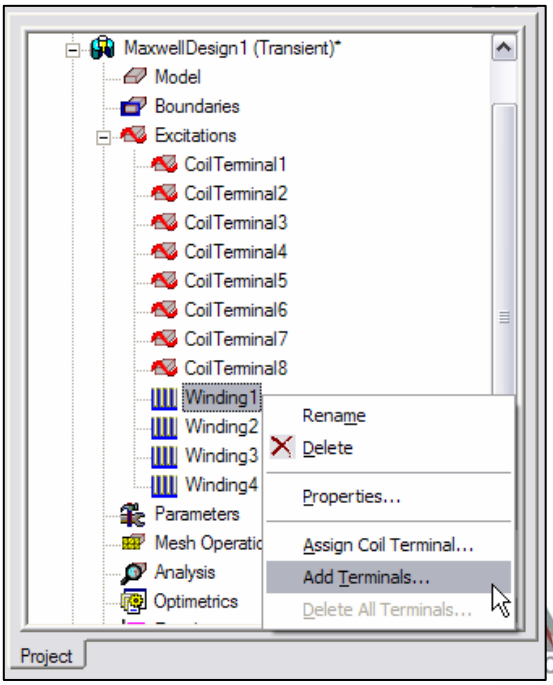
To add windings

1. Select the menu item *Maxwell > Excitations > Add Winding*
2. Winding Window
    1. Name: **Winding1**
    2. Type: **Voltage**
    3. **Stranded**
    4. Resistance: **2.5 ohm**
    5. Voltage: **120 V**
    6. Click the **OK** button
3. Repeat these steps three more times to add: **Winding2, Winding3, Winding4**

Assign Terminals to Windings

To select windings

1. Expand the Project tree to display the Excitations
2. Using the mouse, right-click on **Winding1** and select **Add Terminals**
3. Add Terminal Window
    1. From the list, select: **CoilTerminal1, CoilTerminal5**
    2. Click the **OK** button
4. Repeat these steps for:
    - **Winding2**
        - **CoilTerminal2**
        - **CoilTerminal6**
    - **Winding3**
        - **CoilTerminal3**
        - **CoilTerminal7**
    - **Winding2**
        - **CoilTerminal4**
        - **CoilTerminal8**



WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Set Default Material

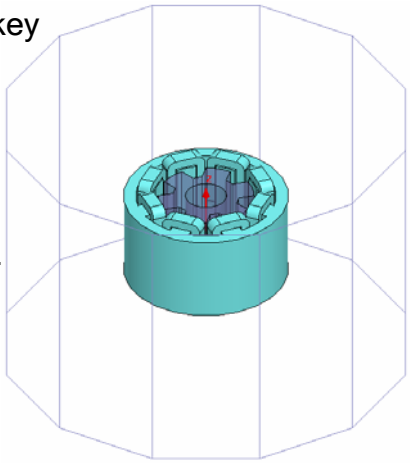
To set the default material:

1. Using the 3D Modeler Materials toolbar, choose **vacuum**

Create Region

To create the region:

1. Select the menu item *Draw > Regular Polyhedron*
2. Using the coordinate entry fields, enter the center position
   - X: **0.0**, Y: **0.0**, Z: **-100.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the radius:
   - dX: **150.0** dY: **0.0**, dZ: **0.0**, Press the **Enter** key
4. Using the coordinate entry fields, enter the height:
   - dX: **0.0**, dY: **0.0**, dZ: **200.0**, Press the **Enter** key
   - Segment Number Window
     - Number of Segments: **12**
     - Click the **OK** button



To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Region**
3. **Display Wireframe**: ☑ **Checked**
4. Click the **OK** button

To turn off the visibility:

1. Select the menu item *View > Hide Selection > Active View*

To fit the view:

1. Select the menu item *View > Fit All > Active View*.

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Create Mesh

- **Note:** The transient solver does not use automatic adaptive meshing.
- **Select Coils**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Coil, Coil_1, Coil_2, Coil_3, Coil_4, Coil_5, Coil_6, Coil_7**
     2. Click the **OK** button
- **To create a mesh:**
  1. Select the menu item *Maxwell > Mesh Operations > Assign > On Selection > Length Based*
  2. Element Length Based Assignment Window
     1. Name: **Coils**
     2. Restrict Length of Elements: ☐ **Unchecked**
     3. Restrict Number of Elements:☑ **Checked**
     4. Maximum Number of Elements: **16000** (2000/tets per coil)
     5. Click the **OK** button
- **Select Rotor and Stator**
  1. Select the menu item *Edit > Select > By Name*
  2. Select Object Dialog,
     1. Select the objects named: **Rotor, Stator**
     2. Click the **OK** button
- **To create a mesh:**
  1. Select the menu item *Maxwell > Mesh Operations > Assign > On Selection > Length Based*
  2. Element Length Based Assignment Window
     1. Name: **Rotor_Stator**
     2. Restrict Length of Elements: ☐ **Unchecked**
     3. Restrict Number of Elements:☑ **Checked**
     4. Maximum Number of Elements: **4000** (2000/tets per coil)
     5. Click the **OK** button

WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Analysis Setup

Creating an Analysis Setup

- **To create an analysis setup:**
  1. Select the menu item *Maxwell > Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **General** tab:
        - Stop time: **0.05s**
        - Time step: **0.002s**
     2. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item *File > Save As*.
  2. From the **Save As** window, type the Filename: **maxwell_trans_relmotor**
  3. Click the **Save** button

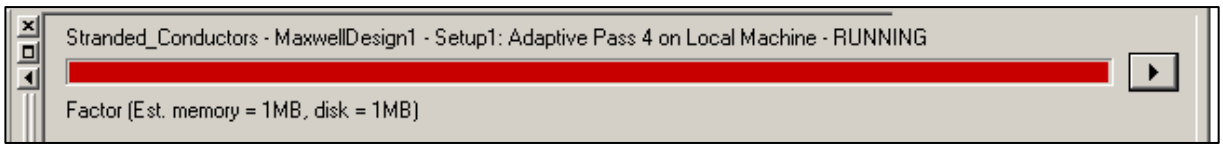
Analyze

Model Validation

- **To validate the model:**
  1. Select the menu item *Maxwell > Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item *Maxwell > Analyze All*



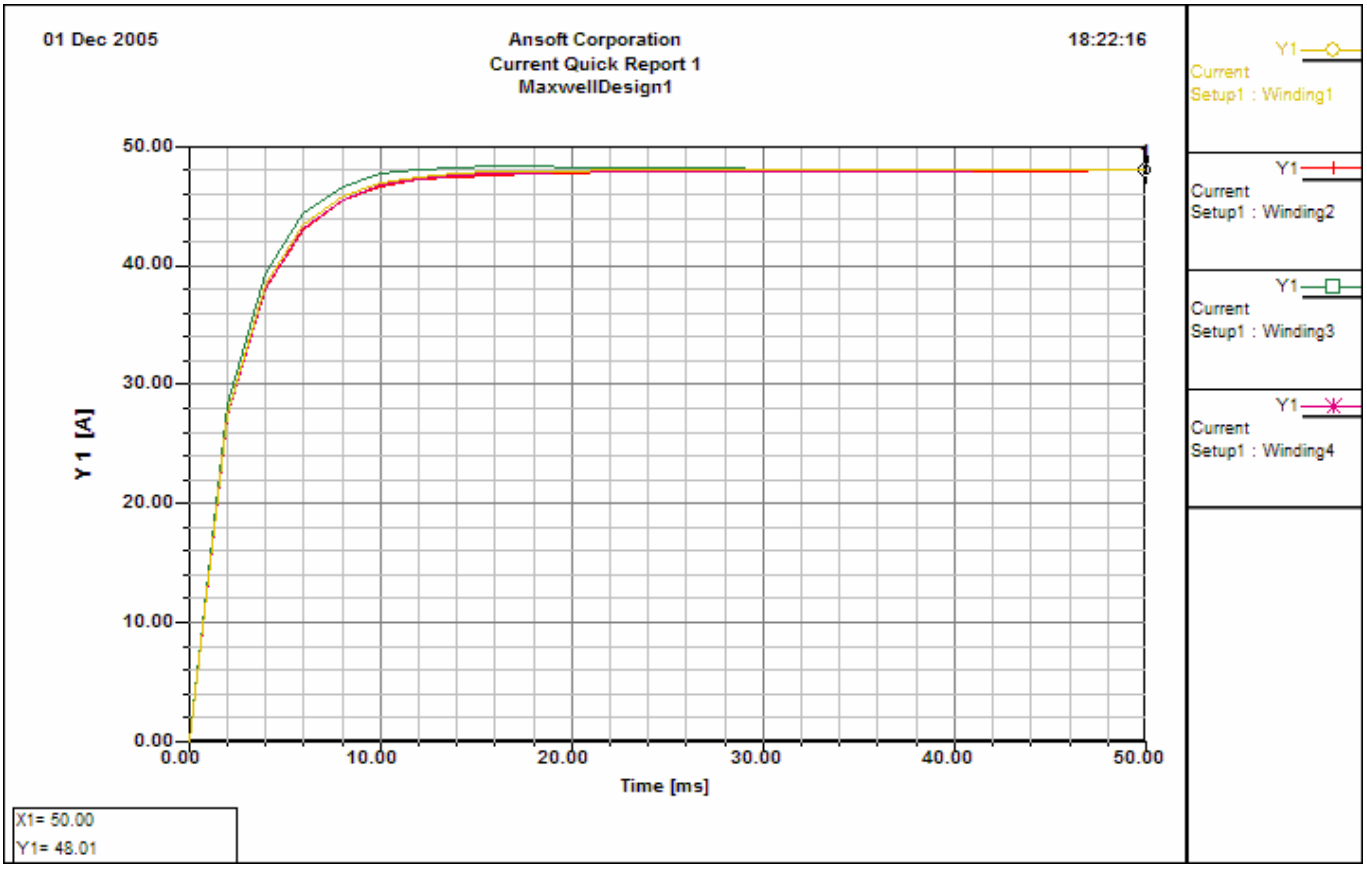
WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Create Quick Report

To create a report:

1. Select the menu item *Maxwell > Results > Create Quick Report*
2. Quick Report Window:
    1. Select: **Current**
    2. Click the **OK** button



WWW.MOHANDESYAR.COM

Example (Transient) - Switched Reluctance Motor

Calculate Current

To use the field calculator

1. Select the menu item **Maxwell > Fields > Calculator**
2. Select **Quantity: J**
3. Select **Vector**: **Scal?** > **Scalar Z**
4. Click the button **Geometry**
5. Geometry Window
    1. Select the radio button **Surface**
    2. From the list select: **Section1**
    3. Click the **OK** button
6. Click the ∫ (Integrate) button
7. Click the **Number** Button
    1. Type: **Scalar**
    2. Value: **150** (Number of Conductors)
    3. Click the **OK** button
8. Click the **/** button
9. Click the **Eval** button
10. Click the **Done** button to close the calculator

Vec : <Jx,Jy,Jz>

Scl : ScalarZ(<Jx,Jy,Jz>)

Srf : Surface(Section1)
Scl : ScalarZ(<Jx,Jy,Jz>)

Scl : -48.0069242512792
Scl : /(Integrate(Surface(Section1), ScalarZ(<Jx,Jy,Jz>)), 150)