

Ansoft

Maxwell 3D

Electromagnetic and Electromechanical Analysis

11



electronic design automation software

user's guide - Maxwell 3D

The information contained in this document is subject to change without notice. Ansoft makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Ansoft shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

© 2005 Ansoft Corporation. All rights reserved.

Ansoft Corporation
225 West Station Square Drive
Suite 200
Pittsburgh, PA 15219
USA
Phone: 412-261-3200
Fax: 412-471-9427

Maxwell, ePhysics and Optimetrics are registered trademarks or trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.

New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates which are incorporated at reprint do not cause the date to change. Update packages may be issued between editions and contain additional and/or replacement pages to be merged into the manual by the user. Note that pages which are rearranged due to changes on a previous page are not considered to be revised.

Edition: REV2.0
Date: 28 July 2006
Software Version: 11.1.1

 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 Solution Type
- 1. Parametric Model Creation
- 2. Solution Types
 - 2.1 Magnetostatic
 - 2.2 Eddy Current
 - 2.3 Transient
 - 2.4 Electrostatic
 - 2.5 DC Conduction
- 3. Mesh Overview
 - 3.1 Geometry Import & Healing
- 4. Data Reporting
 - 4.1 Field Calculator
- 5. Examples - Magnetostatic
- 6. Examples - Eddy Current
- 7. Examples - Transient
- 8. Examples - Electric

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.

Electric 3D fields:

-  Electrostatic 3D fields in dielectrics caused by a user-specified distribution of voltages and charges. Additional computed quantities you can specify include torque, force, and capacitances.
-  Electric 3D fields in conductors, characterized by a spatial distribution of voltage, electric field, and DC current density. The main additional quantity in this case is power loss.
-  A combination of the first two with conduction solutions being used as boundary conditions for an electrostatic problem.

Magnetostatic:

-  Linear and nonlinear 3D magnetostatic fields caused by a user-specified distribution of DC current density, voltage, permanent magnets, or externally applied magnetic fields. Additional computed quantities you can specify include torque, force, and inductances (self and mutual).

Eddy current:

-  Harmonic (sinusoidal variation in time) steady state 3D magnetic fields with pulsation-induced eddy currents in massive (solid) conductors caused by one of the following: by a user-specified distribution of AC currents (all with the same frequency but with possibly different initial phase angles), or by externally applied magnetic fields. The eddy solution is a full wave solution that includes electromagnetic wave radiation effects.

Transient:

-  Time domain 3D magnetic fields caused by permanent magnets and windings supplied by voltage and/or current sources with arbitrary variation as functions of time; electrical circuits will be connected with the windings. Rotational or translational motion effects can also be included in the simulation.



System Requirements (Windows)

Supported Platforms:

- Windows 2000 Professional
- Windows XP Professional
- Windows XP Professional x64 Edition
- Windows Server 2003

32-Bit System Requirements

Minimum System Requirements:

- Processor: All fully compatible 686 (or later) instruction set processors, 500 MHz
- Hard Drive Space (for Maxwell software): 200 MB
- RAM: 512 MB

Recommended Minimum Configuration (for Optimal Performance):

- Processor: All fully compatible 786 (or later) instruction set processors, 1 GHz
- Hard Drive Space (for Maxwell software and temporary files): 500 MB
- RAM: 2 GB

64-bit System Requirements

Minimum System Requirements:

- Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support
- Hard Drive Space (for Maxwell software): 200 MB
- RAM: 2 MB

Recommended Minimum Configuration (for Optimal Performance):

- Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support
- Video card: 128-bit SVGA or PCI Express video card
- Hard Drive Space (for Maxwell software and temporary files): 700 MB
- RAM: 8 GB
- Pentium -based computer
- 512 MB RAM minimum

- 8MB Video Card minimum
- Mouse or other pointing device
- CD-ROM drive



▀ System Requirements (Sun Solaris)

▀ **Supported Platforms:**

- ▀ Solaris 8
- ▀ Solaris 9

▀ **Minimum System Requirements:**

- ▀ Processor: UltraSparc v9 processor, 450 MHz
- ▀ Hard Drive Space (for Maxwell software): 550 MB
- ▀ RAM: 1 GB

▀ **Recommended Minimum Configuration:**

- ▀ Processor: UltraSparc v9 dual processor or better, 900 MHz
- ▀ Hard Drive Space (for Maxwell software and temporary files): 800 MB
- ▀ RAM: 4 GB

- ▀ **Note:** You must install Sun OpenGL libraries before installing and running Maxwell. This is available for free download at the following location:
<http://www.sun.com/software/graphics/opengl/download.html>

- ▀ **Note:** Some older graphics cards may have minor display issues (such as check marks or a “t” appearing in a report title).



▄ System Requirements (Red Hat Enterprise Linux v3)

▄ 32-Bit System Requirements

▄ Minimum System Requirements:

- ▄ Processor: All fully compatible 686 (or later) instruction set processors, 500 MHz
- ▄ Hard Drive Space (for Maxwell software): 200 MB
- ▄ RAM: 512 MB

▄ Recommended Minimum Configuration (for Optimal Performance):

- ▄ Processor: All fully compatible 786 (or later) instruction set processors, 2GHz
- ▄ Hard Drive Space (for Maxwell software and temporary files): 700 MB
- ▄ RAM: 2 GB

▄ 64-bit System Requirements

▄ Minimum System Requirements:

- ▄ Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support
- ▄ Hard Drive Space (for Maxwell software): 200 MB
- ▄ RAM: 2 MB

▄ Recommended Minimum Configuration (for Optimal Performance):

- ▄ Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support Video card: 128-bit SVGA or PCI Express video card
- ▄ Hard Drive Space (for Maxwell software and temporary files): 700 MB
- ▄ RAM: 8 GB



Installing the Ansoft Maxwell Software

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

Starting Ansoft Maxwell

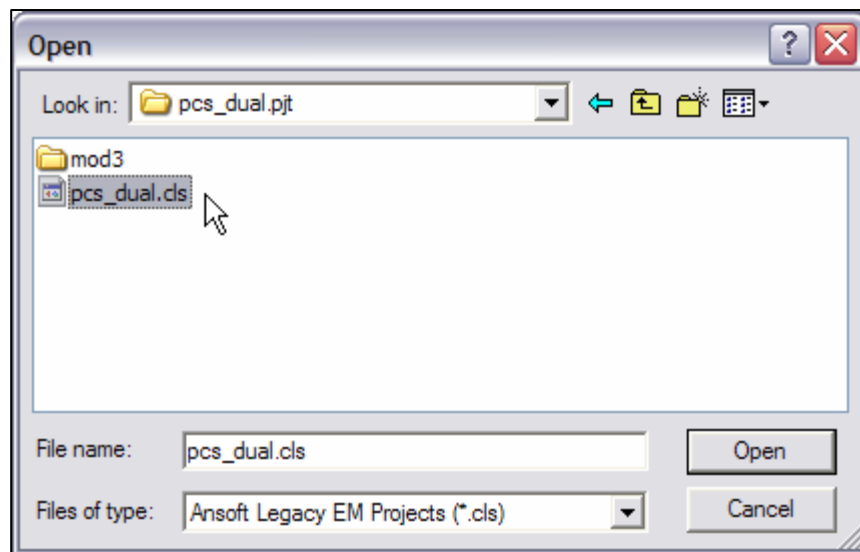
1. Click the Microsoft **Start** button, select **Programs**, and select the **Ansoft, Maxwell 11** program group. Click **Maxwell 11**.
2. Or Double click on the Maxwell 11 icon on the Windows Desktop



❖ Converting Older Maxwell file to Maxwell v11

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



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.
 -  **Tooltips** 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

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**

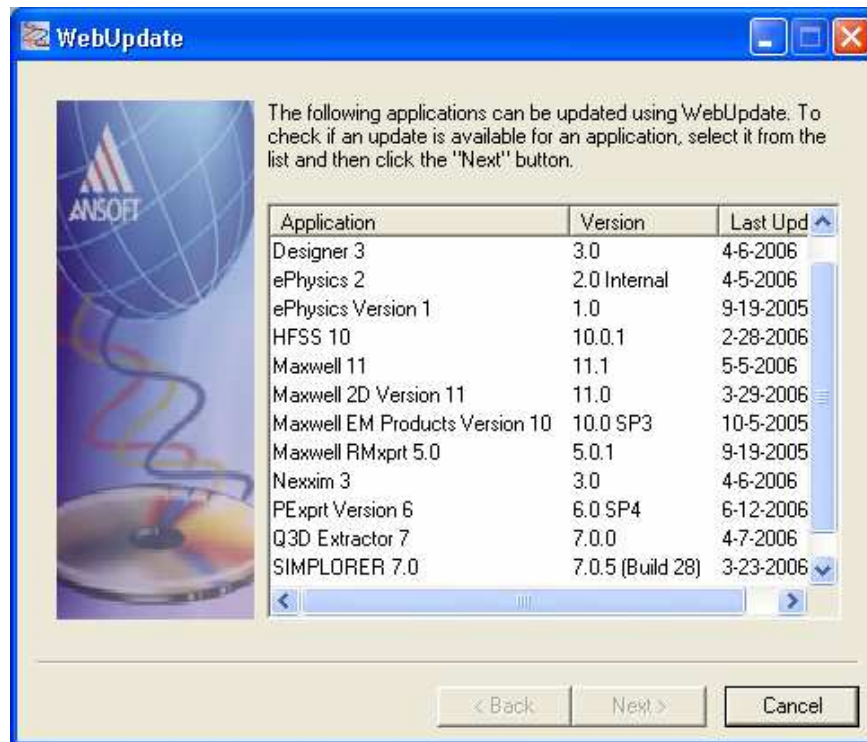
-  (503) 906-7944 or (503) 906-7947

-  **El Segundo, CA**

-  (310) 426-2287 - Ask for Technical Support

WebUpdate

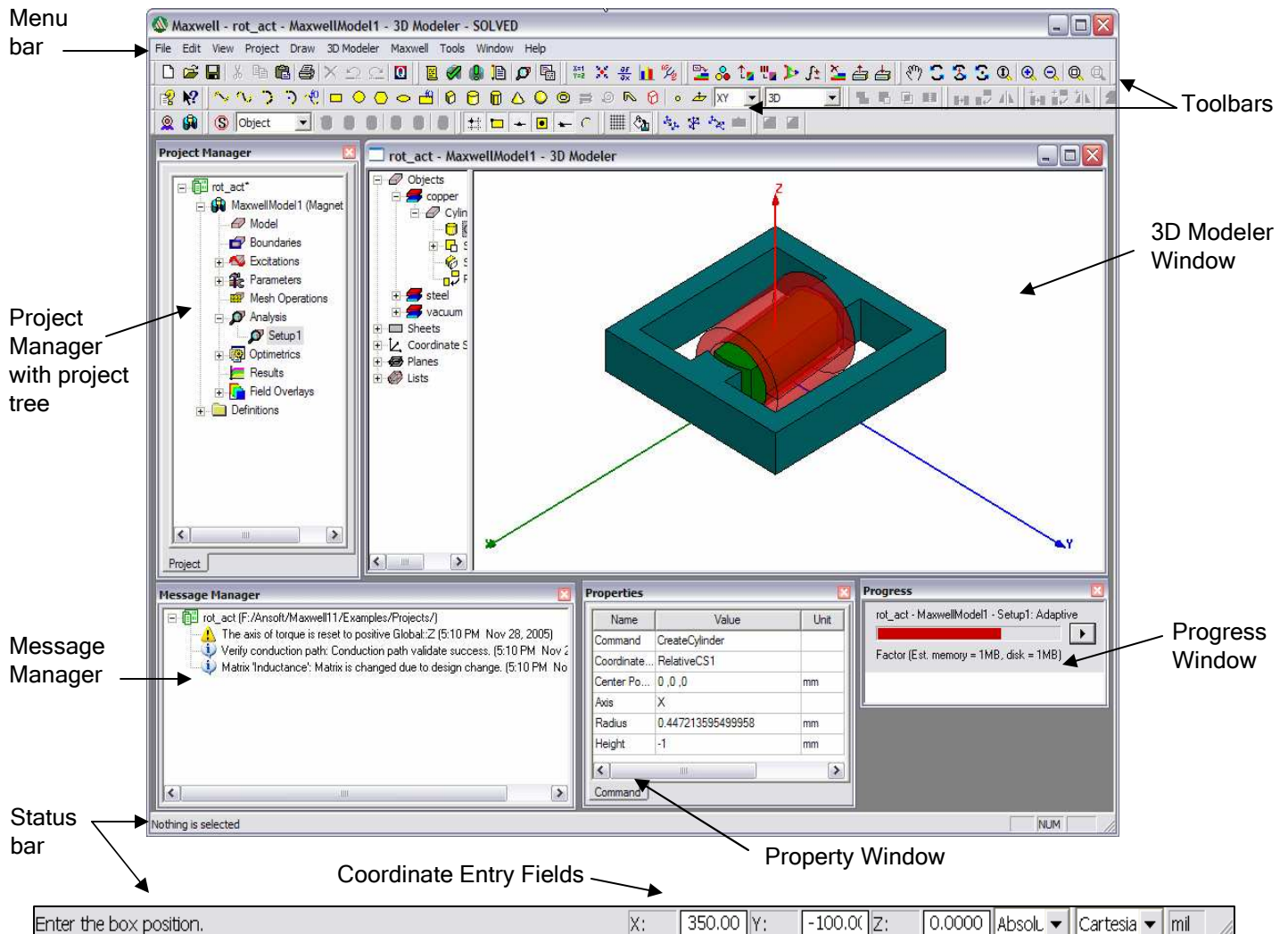
- This new feature allows you to update any existing Ansoft software from the WebUpdate window. This feature automatically scans your system to find any Ansoft software, and then allows you to download any updates if they are available.



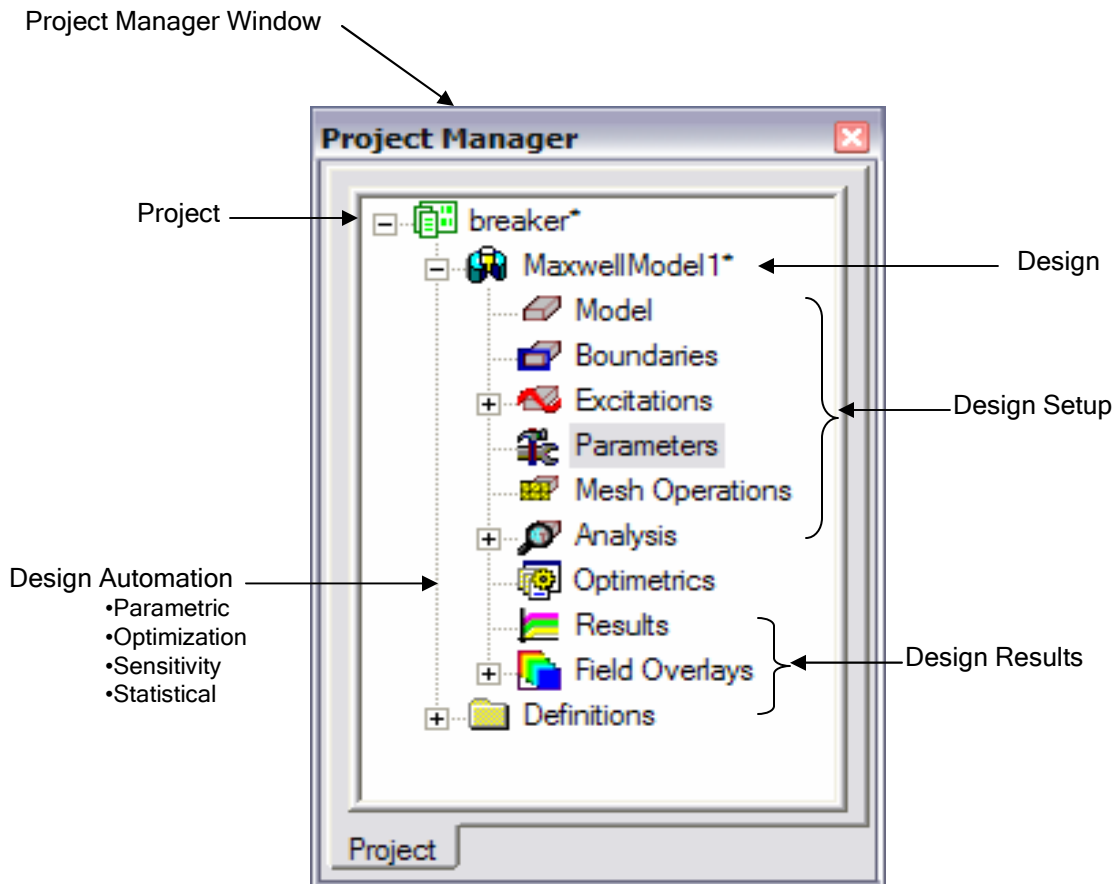
Ansoft Terms

The Ansoft Maxwell window has several optional panels:

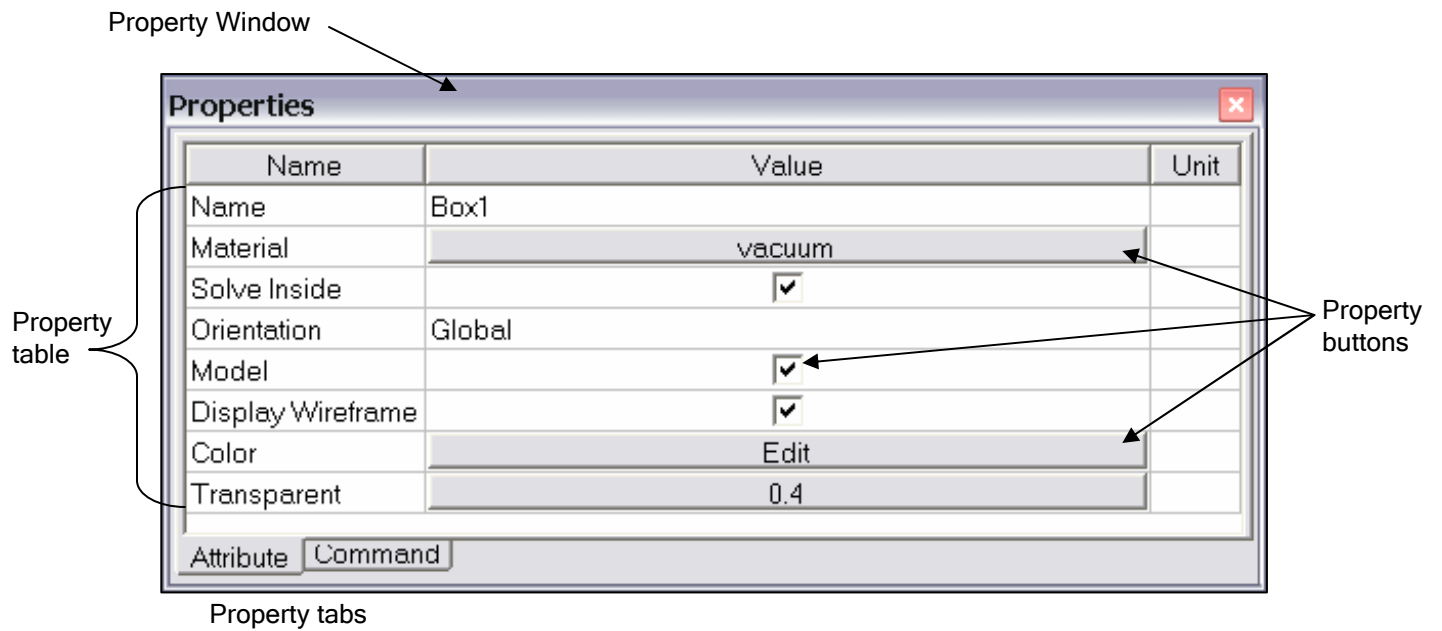
- ▲ A **Project Manager** which contains a design tree which lists the structure of the project.
- ▲ A **Message Manager** that allows you to view any errors or warnings that occur before you begin a simulation.
- ▲ A **Property Window** that displays and allows you to change model parameters or attributes.
- ▲ 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 the 3D Modeler Window, see chapter 1.



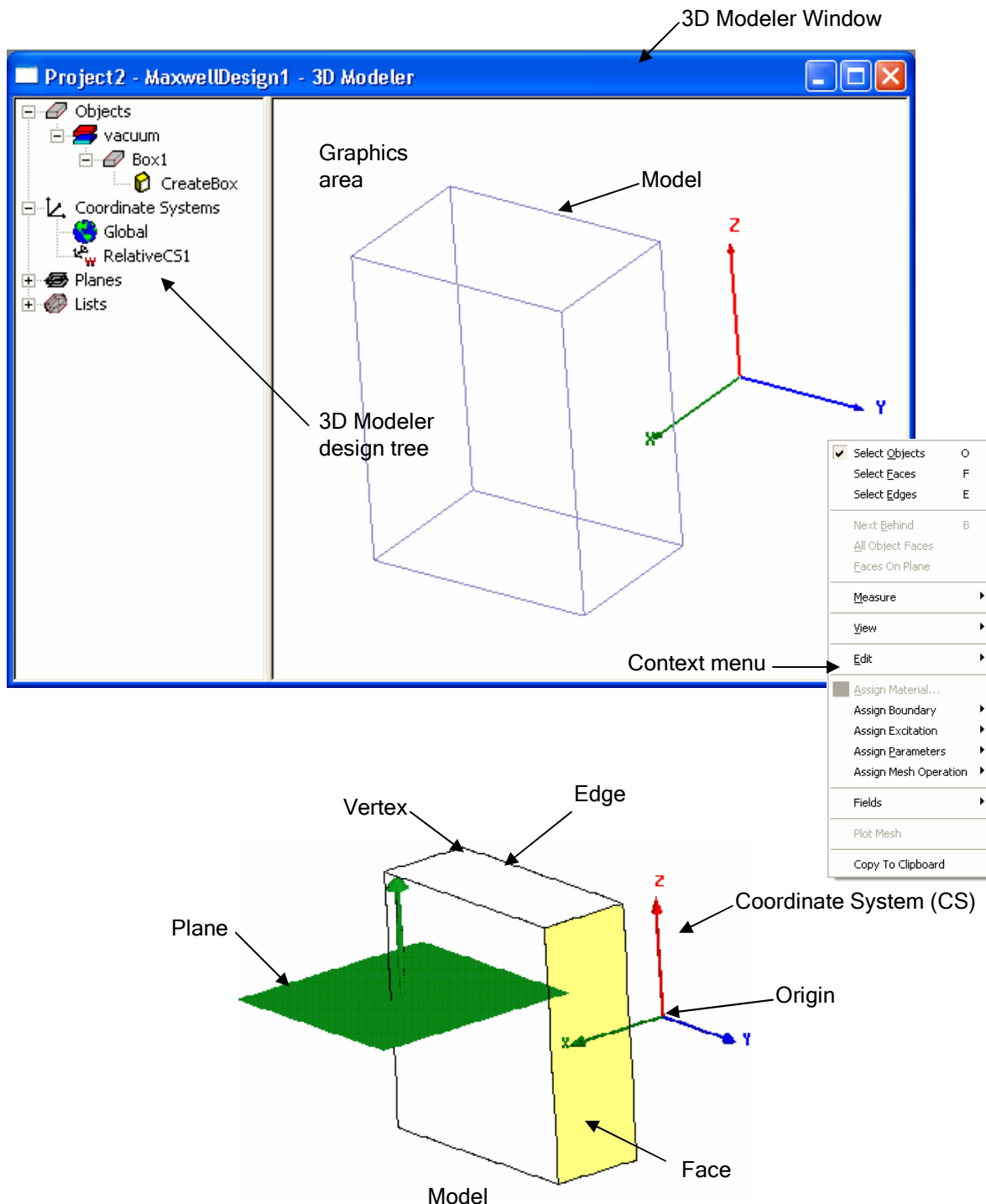
Project Manager



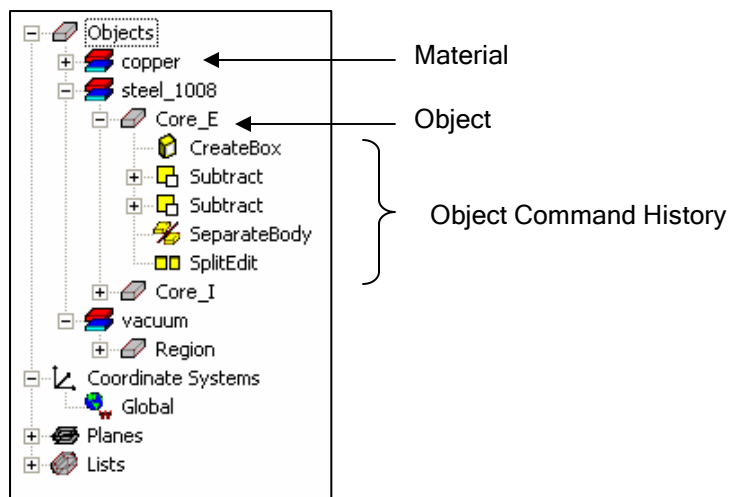
Property Window



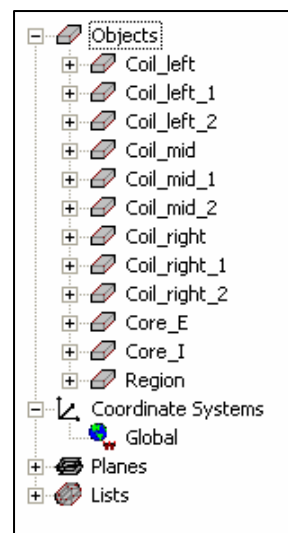
Ansoft 3D Modeler



3D Modeler Design Tree



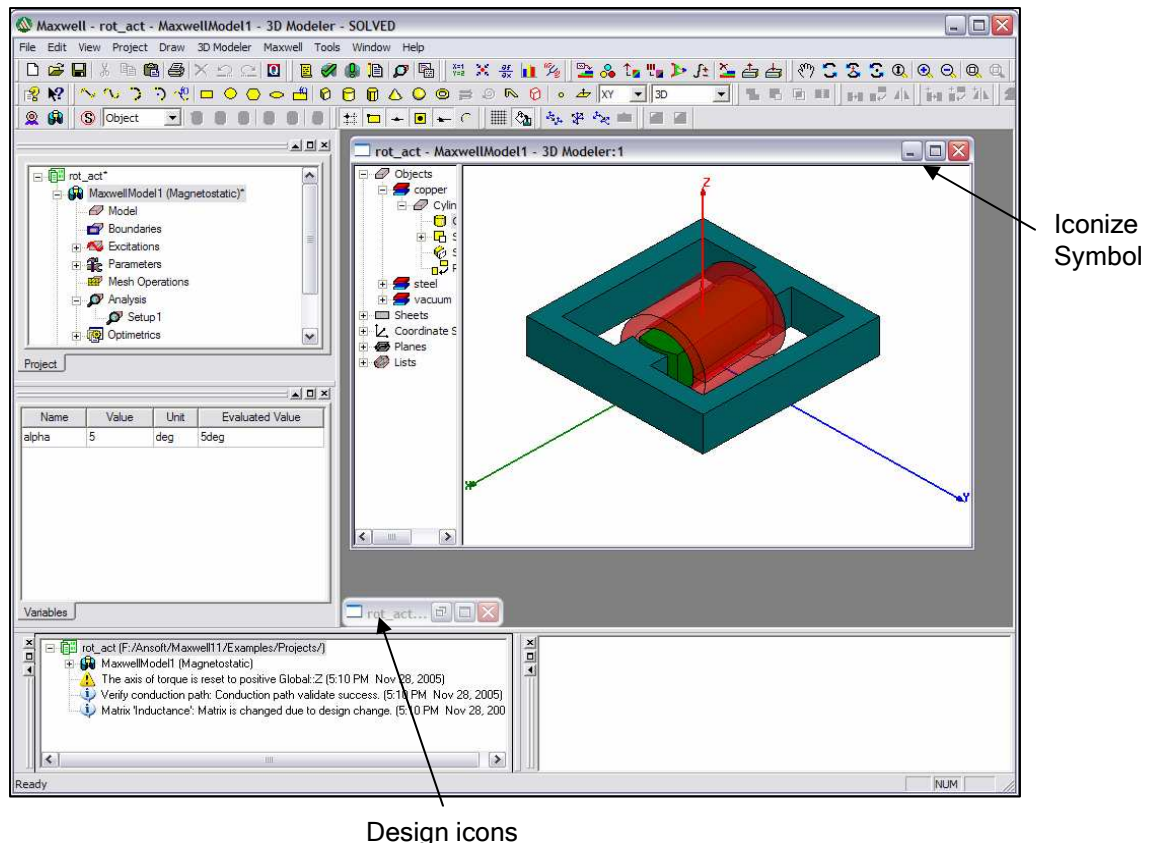
Grouped by Material



Object View

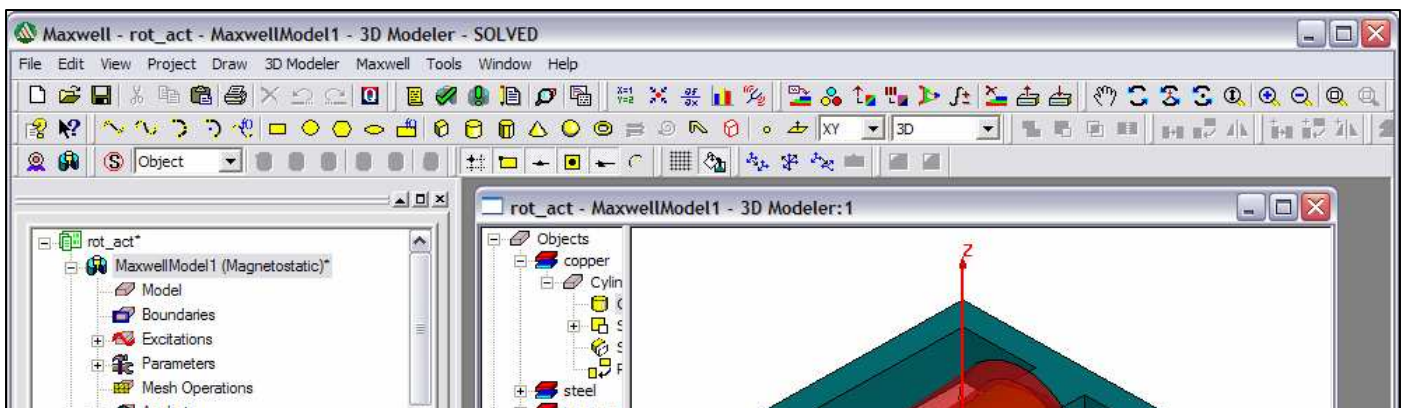
Design Windows

- ▲ 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.



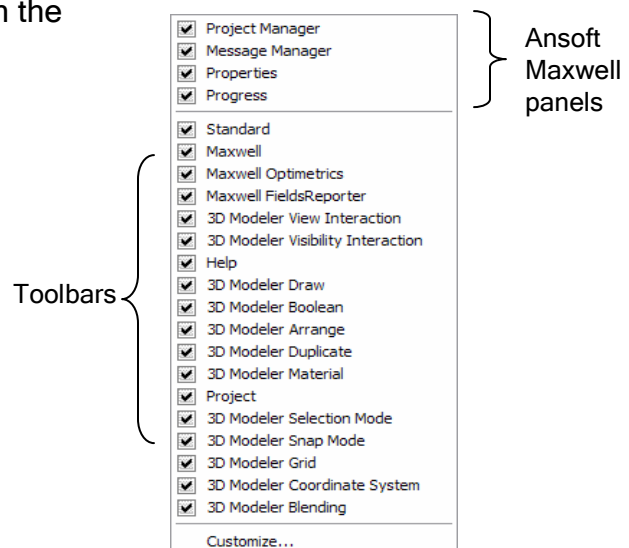
▲ Toolbars

- ▲ 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.



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.

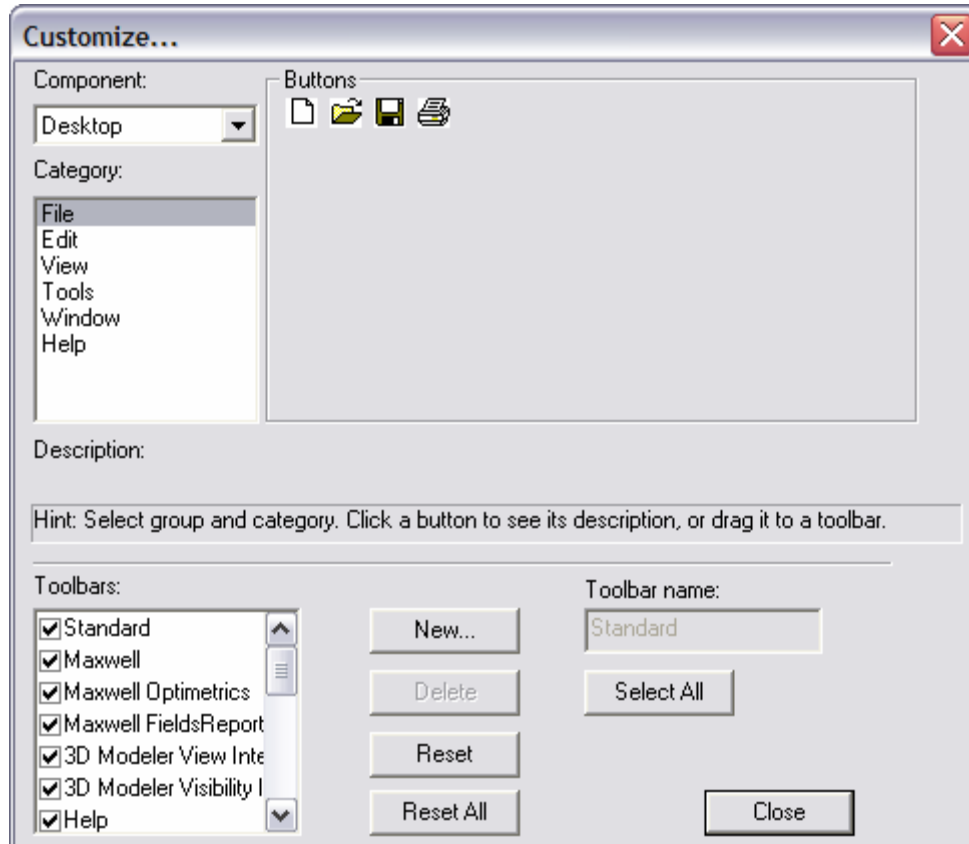
- ▲ In the Customize dialog, you can do the following:

- ▲ **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



Ansoft Maxwell Desktop

-  The complex functionality built into the Maxwell 3D solvers is accessed through the main user interface (called the desktop). With the version 11 interface and later, you can model the problem in a fairly arbitrary order (rather than following the steps in a precise order as was required in previous versions of Maxwell).
-  This flexibility allows experienced users to develop a modeling style that suits their preferences. Once the model is created, the automated Maxwell solution sequence takes over and fully controls the solution process without any interaction from the user. When the solution becomes available, the user can perform a variety of post-processing tasks as required by the design application.

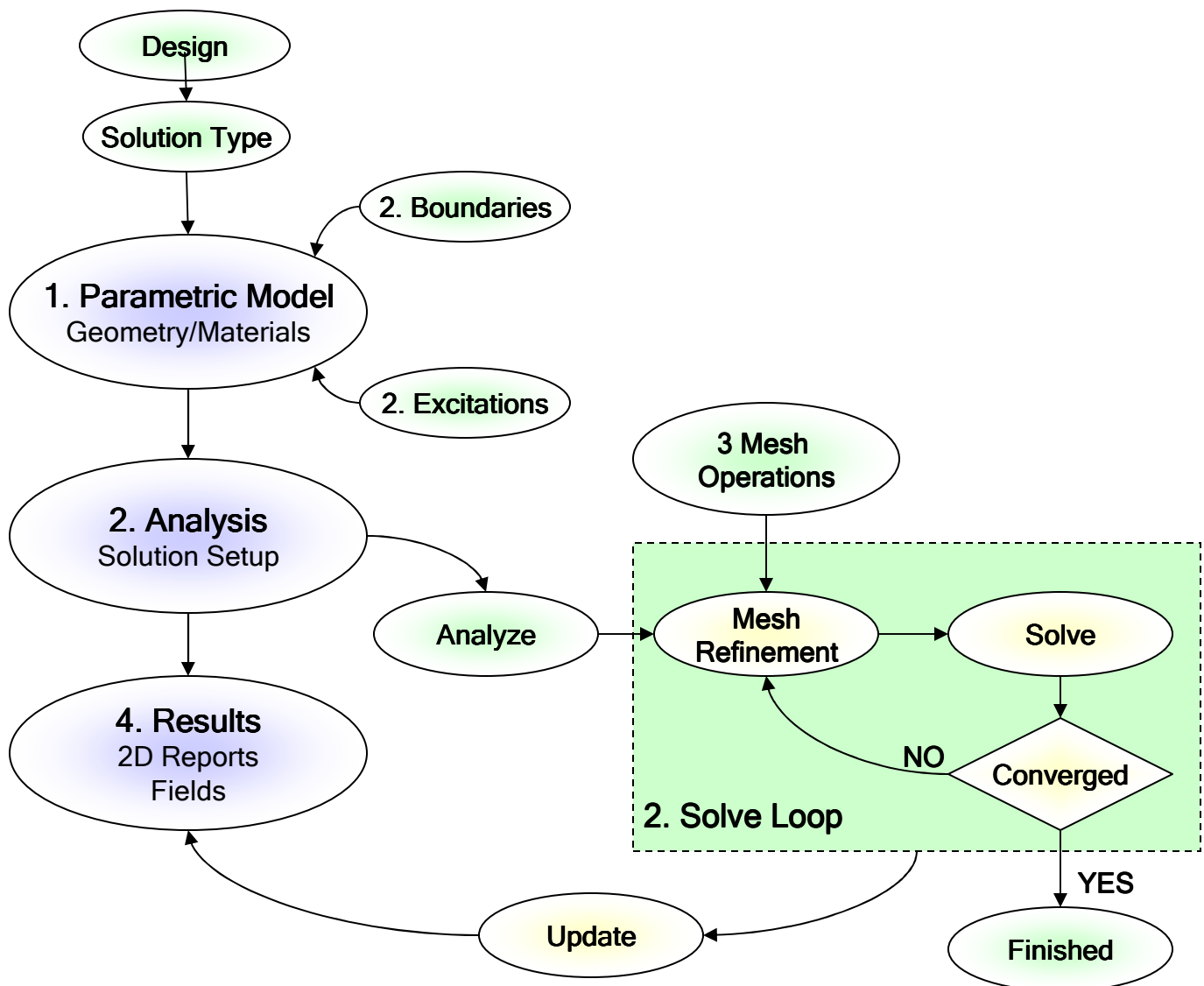
Modeling Process

-  For users new to electromagnetic field simulation, Ansoft recommends the following sequence of modeling steps:
 -  1. Based on your application, choose the type of electromagnetic analysis to be performed.
 -  2. Draw the geometry of the model using the drawing space provided by the 3D Modeler menu and Draw menu commands available through the Maxwell desktop interface.
 -  3. Assign the material properties to all solid objects in the model, and define new material properties if materials in the default library do not provide the needed material.
 -  **Note:** Always make sure the material properties assigned to an object correspond to the real properties of the materials in the electromagnetic device that is being simulated. Material properties supplied in the default library are generic properties and may not always be substituted for actual properties.
 -  4. Specify the field sources (excitations) and boundary conditions for your unique solution.
 -  5. Define additional global parameters that you want to calculate (such as force, torque, inductance/capacitance, etc.).
 -  6. Define mesh operations for special applications (such as seeding in areas/objects of interest).
 -  7. Specify solution options.
 -  8. Start the solution process.
 -  9. When the solution becomes available, perform post processing, such as plotting field quantities and calculating expressions.

Parametric Adaptive Analysis

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 (shown specifically for a Magnetostatic setup).



Opening a Maxwell project

-  The basic modeling entity in Maxwell is the design (model). The next level up is the project. A project is a collection of one or more designs (models) that is saved in a single *.mxwl file. A new project is automatically created when Maxwell is launched. As many Maxwell designs as needed can be added to a single Maxwell project.

Opening a New project

To open a new project:

1. In an Ansoft Maxwell window, select the menu item **File > New**.

To insert a Maxwell design:

1. In an Ansoft Maxwell window, select the menu item **Project > Insert MaxwellDesign**.

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.

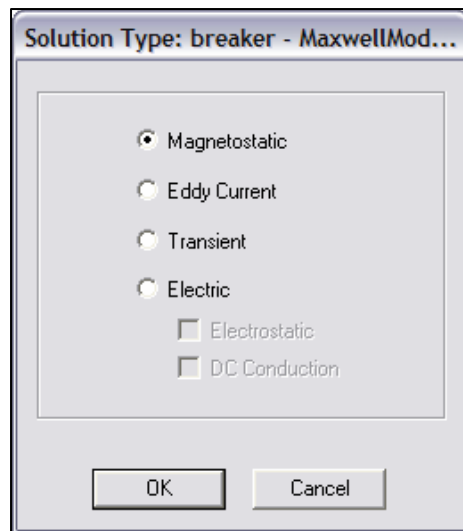
To open a project from Windows Explorer, do one of the following:

-  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.

Set Solution Type

 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 (Electrostatic, DC Conduction)**
 2. Click the **OK** button



THIS PAGE INTENTIONALLY LEFT BLANK

THIS PAGE INTENTIONALLY LEFT BLANK

Magnetostatic Analysis

Magnetostatic Analysis

-  Magnetostatic Analysis is performed by choosing the Magnetostatic solution type.
-  Applications that use Magnetostatic Analysis can be solenoids, inductors, motors, actuators, permanent magnets, stray field calculations and many others.

Overview

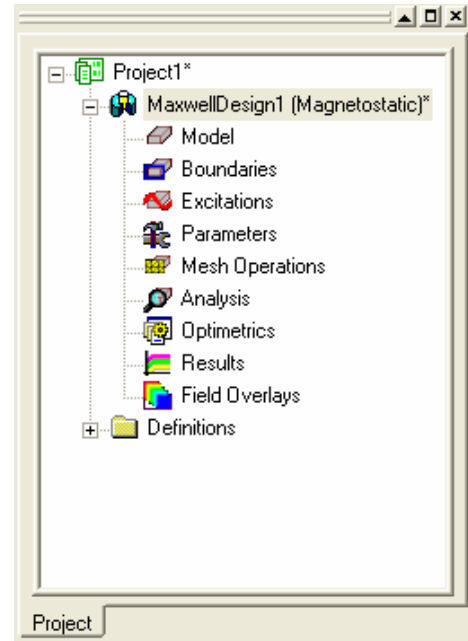
-  The magnetostatic solver computes static (DC) magnetic fields.
-  All objects are stationary.
-  The source of the static magnetic field can be:
 -  DC current in conductors
 -  Permanent magnets
 -  Static magnetic fields represented by external boundary conditions.
-  The quantity solved is the magnetic field (H).
-  Current density (J) and magnetic flux density (B) are automatically calculated from the magnetic field (H).
-  Derived quantities such as forces, torques, energy, and inductances may be calculated from these basic field quantities.
-  Material permeabilities can be nonlinear and/or anisotropic.

Magnetostatic Analysis

Setup

- ▲ The options in the project tree for a magnetostatic simulation control all the simulation setup parameters.
 - ▲ Notice that right-clicking on any of the options will open a menu with important options for each step of the simulation setup.

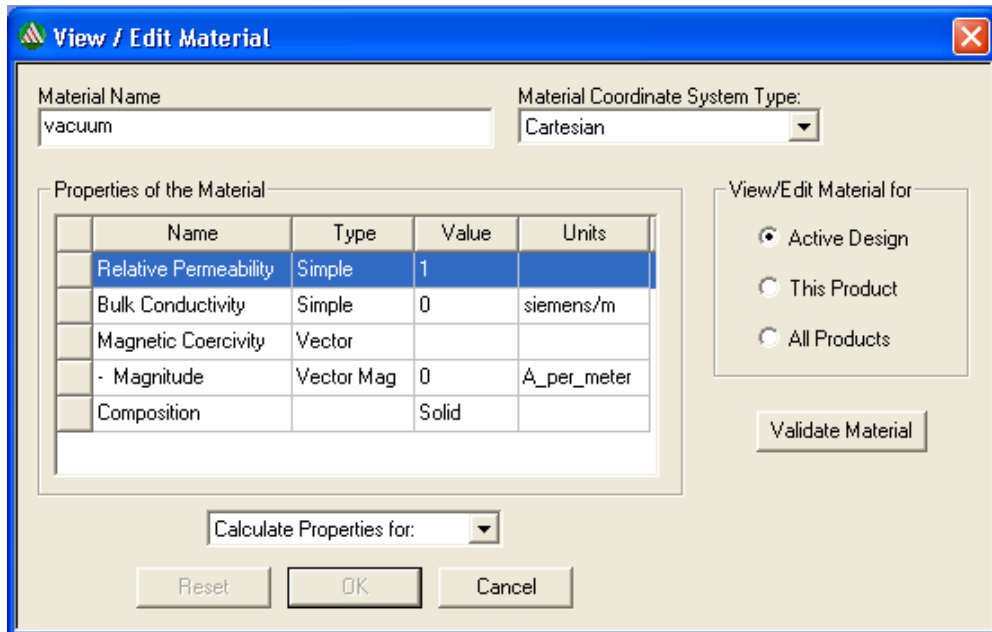
- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in a magnetostatic simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ Mesh Operations are discussed in a separate section.
- ▲ Analysis defines the solution setup.
- ▲ Optimetrics defines any automatic variational analyses.
- ▲ Results and Field overlays are discussed in a separate section.



- ▲ These options are displayed in an order that can be followed in creating a new Magnetostatic simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.
- ▲ However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.

▲ Magnetostatic Material Definition

- ▲ In a Magnetostatic simulation, the following parameters may be defined for a material:
 - ▲ **Relative Permeability** (can be Anisotropic and/or Nonlinear, or Simple)
 - ▲ Relative permeability along with the Magnetic Coercivity determine the magnetic properties of the material.
 - ▲ **Bulk Conductivity** (can be Anisotropic or Simple)
 - ▲ Bulk Conductivity is used in determining the current distribution in current carrying conductors - it has no influence in the magnetic part of the solution process.
 - ▲ **Magnetic Coercivity** (defined as a vector magnitude and direction)
 - ▲ Magnetic Coercivity is used to define the permanent magnetization of magnetic materials. When a non-zero magnitude is entered, the directional entries are visible. The direction (like all directional material properties) are determined by the coordinate system type and the object orientation.
 - ▲ **Composition** (can be Solid or Lamination)
 - ▲ Setting Composition to Lamination creates an anisotropic magnetization effect. This is discussed in the Anisotropic Material example.



View / Edit Material

Material Name: vacuum Material Coordinate System Type: Cartesian

Name	Type	Value	Units
Relative Permeability	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
Composition		Solid	

View/Edit Material for:

☒ Active Design
☐ This Product
☐ All Products

Validate Material

Calculate Properties for: [Dropdown]

Reset OK Cancel

Magnetostatic Boundary Conditions

-  **Default** - The default boundary conditions for the Magnetostatic solver are:
 -  **Natural** boundaries on the interface between objects.
 - This means that the H Field is continuous across the boundary.
 -  **Neumann** boundaries on the outer boundaries.
 - This means that the H Field is tangential to the boundary and flux cannot cross it.
-  **Zero Tangential H Field** - This boundary is often used to model an applied uniform, external field. This would model outer boundaries of the Region that are perpendicular to the applied field. In this case, the boundary should be placed far from the structure so that the simulation is not over-defined. This is equivalent to the even symmetry boundary definition.
-  **Tangential H Field** - This boundary is used primarily to model an applied uniform, external field. This would model outer boundaries of the Region that have a defined tangential magnetic field. This boundary should always be placed far from the structure so that the simulation is not over-defined. Faces must be planar and must be defined one at a time, due to the U-V field definition on each face.
 -  To apply a uniform field along any orthogonal axis of a bounding box, first, apply a Zero Tangential H Field on the top and bottom faces (with respect to the direction of the desired axis) of the box. Then apply a Tangential H Field to each side face - define each U vector to be parallel to the selected axis (the V vector does not matter because no field will exist in that direction), and assign the value of U with a constant value. The field will be in the direction of the selected axis (perpendicular to the top and bottom faces), and will have a constant value at the boundaries of the solution region.
 -  Note that it is very easy to create impossible field assignments with these boundary conditions. If your simulation does not converge when using these boundaries, try the boundary conditions without any objects included to see if the boundaries are assigned correctly (the simulation will have difficulties converging if boundaries are incorrect).

Magnetostatic Boundary Conditions (Continued)

-  **Insulating** - This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate coils from magnetic steel (defined on surfaces between the two objects), but there are many other applications.

-  **Symmetry** - There are two magnetic symmetries - odd symmetry (flux tangential) and even symmetry (flux normal). Odd symmetry defines H to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Even symmetry defines H to be normal to the boundary (this is equivalent to the Zero Tangential H Field boundary condition). Remember that geometric symmetry may not mean magnetic symmetry in all cases. 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. 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** - This boundary condition is also known as a matching boundary condition because it matches the magnetic field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement for matched geometries) and the faces need to be planar. It is required to define a U-V coordinate system along each face to properly map the matched boundary as desired. Master and Slave boundaries enable you to model only one period of a periodic structure, which will reduce the size of a design. Example applications are periodic rotational machines or infinite arrays.

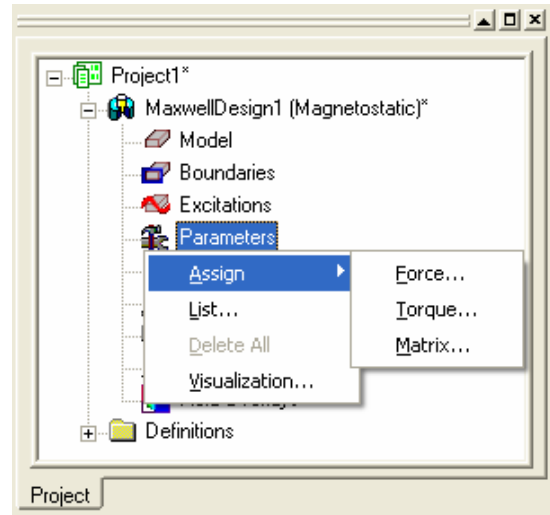
Magnetostatic Excitations

-  Typical sources for magnetostatic field problems include voltage, current, and current density. When applying the sources for the magnetic field problems, the applied current distribution must be divergence free in the entire space of the solution as it is physical for quasi-stationary conduction current density distributions. Thus, the conduction path(s) for the applied current distributions must be closed when totally contained within the solution space for the problem, or must begin and end at the boundaries.
-  Permanent Magnets and externally applied magnetic fields can also act as sources for a magnetostatic analysis, however, those are defined elsewhere.
-  **Voltage Excitations** - These are used in conjunction with the material conductivity to define the current through a solid conductor. Either multiple Voltage excitations can be used to define a voltage difference across two faces of a conductor (creating a current) or a Voltage Drop can be defined on a 2D sheet object to signify the voltage drop around a conductive loop.
-  **Current Excitations** - This excitation can be assigned on any conductor to define the total current (amp-turns) through the cross-section of a loop or to define the current into and out of the opposing, external faces of a conducting object. This is a very general purpose excitation that comes in two flavors - Solid or Stranded. More information about Stranded Magnetostatic Current excitations (along with an example and explanation) can be found in the Magnetostatic Switched Reluctance Motor example.
-  **Current Density** - These excitations are used to define a known current density throughout an object and must be used with a Current Density Terminal. The Current Density is defined on the 3D object, and the Terminal is defined on either an internal cross-sectional sheet, or on all external cross-sectional faces.

Magnetostatic Analysis

Parameters

- ▲ There are three parameters that can be automatically calculated in a Magnetostatic simulation - **Force**, **Torque**, and Inductance **Matrix**.
- ▲ All three quantities are computed directly from the magnetic field solution.
- ▲ Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).



- ▲ Inductance Matrix has many post-processing options available - this is discussed in detail in the Magnetostatic Inductance Matrix example.
- ▲ The results of any parameters can be found by selecting **Maxwell > Results > Solution Data...** or by clicking on the  icon.
- ▲ Further results can be obtained manually through the field calculator.

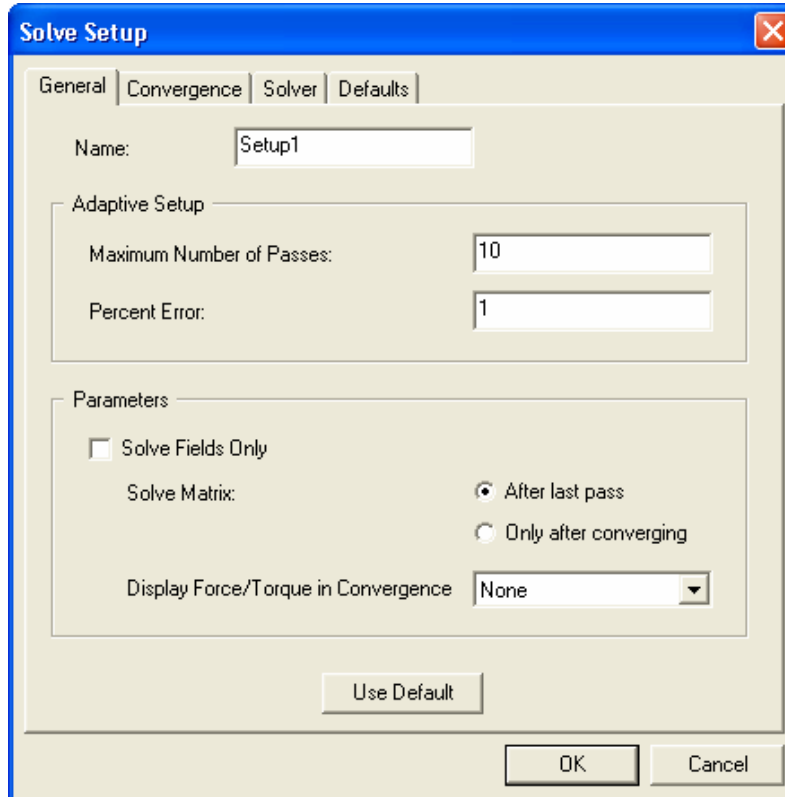
Mesh Operations

- ▲ Mesh operations are described in detail in the Mesh operations section.
- ▲ Remember that the Magnetostatic solver has an adaptive mesh solution, so excessive mesh operations are not usually required. It can often be worse to over-define the mesh than to not define mesh operations at all (it will take longer to solve, and it will be more difficult to adapt correctly).

Solution Setup

-  The solution setup defines the parameters used for solving the simulation.
-  Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.

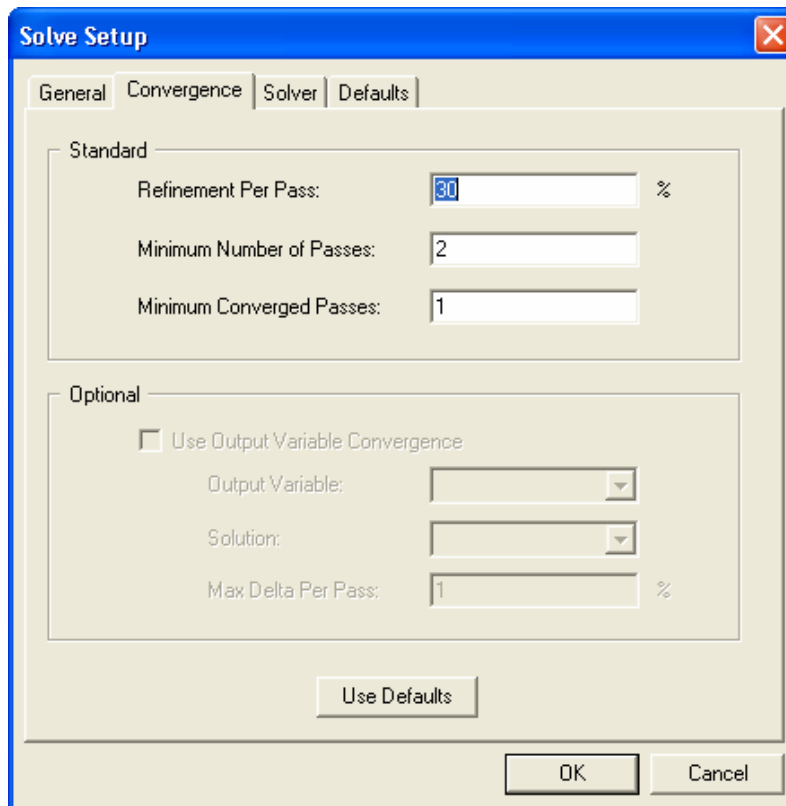
-  The following window appears with the General Setup parameters.



-  You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
-  **Maximum Number of Passes** defines a limit to the number of adaptively refined passes that the solver performs (the default value is 10).
-  **Percent Error** is the error goal for both the Error Energy and Delta Energy.
-  **Solve Fields Only** ignores any defined parameters if checked.
-  **Solve Matrix** has the options of calculating the matrix after the last solved pass or calculating the matrix only if the solution converges.
-  An option is included to display one Force or Torque parameter in the Convergence table.

Solution Setup (Continued)

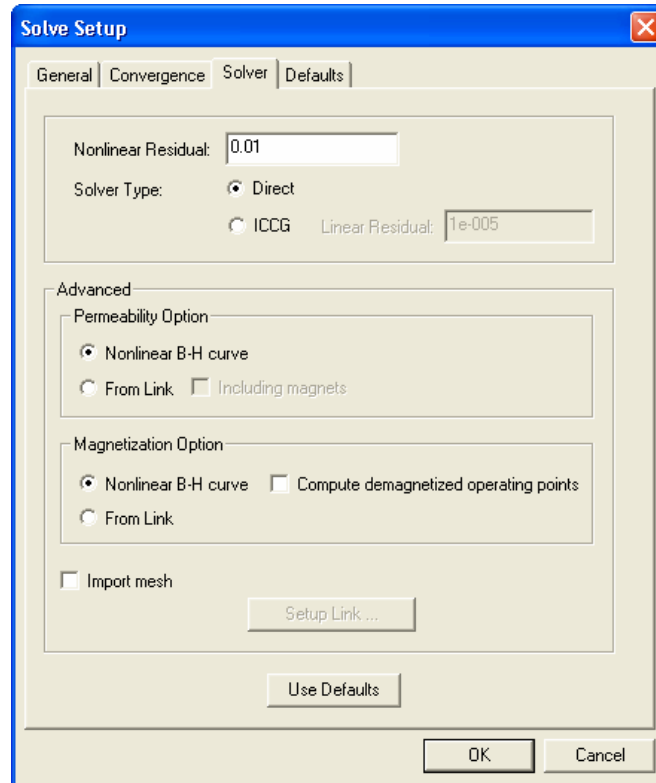
-  The second tab of the Solution Setup contains information about Convergence.



-  **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
-  **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
-  **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).
-  **Use Output Variable Convergence** is an option to include a defined output variable as an additional convergence criterion with a specified maximum percent change per pass (an output variable must be defined for this option to be available).

Solution Setup (Continued)

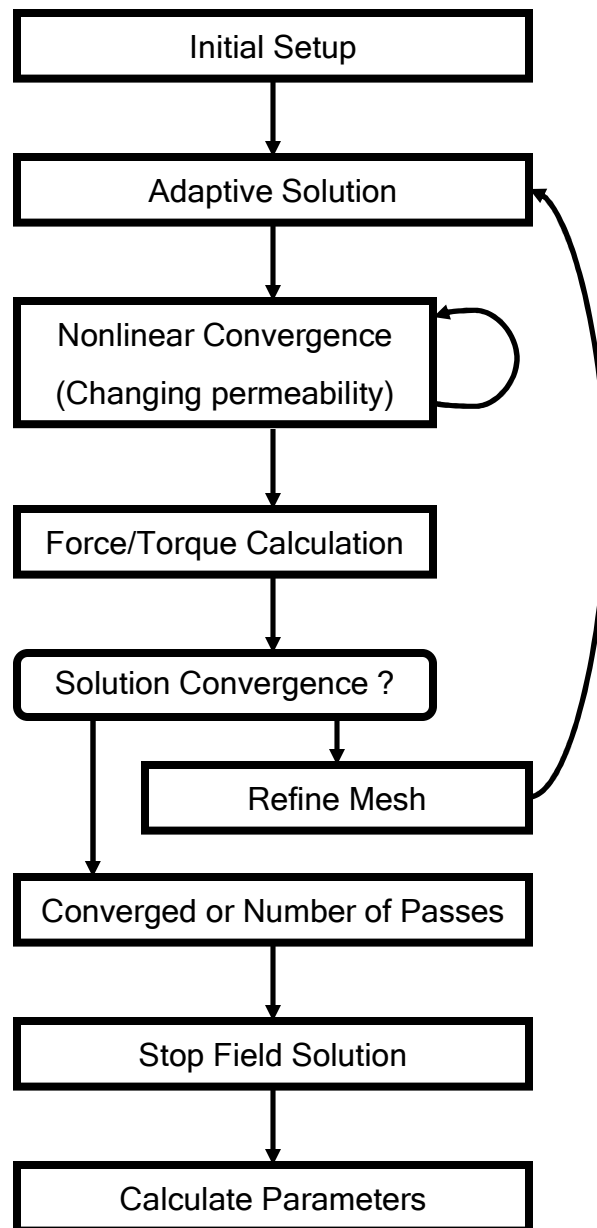
-  The third tab of the Solution Setup contains information about the Solver.



-  **Nonlinear Residual** defines how precisely the nonlinear solution must define the B-H nonlinear operating points (the default value is 0.01).
-  **Solver Type** has options for the **Direct** or **ICCG** solvers (Direct is the default).
-  **Permeability Option** allows the nonlinear B-H operating points either to be calculated by the solver from the **Nonlinear B-H curve** or to use frozen permeabilities **From Link** - the linked solution must have the exact same geometry as the current simulation (Nonlinear B-H curve is the default).
-  **Magnetization Option** allows the permanent magnetization to be determined from the **Nonlinear B-H curve** or to use demagnetized values **From Link** - where the linked solution selected **Compute demagnetized operating points** - the linked solution must have the exact same geometry (Nonlinear B-H curve is the default).
-  **Import Mesh** allows the initial mesh to be imported from another solution - the linked solution must have the exact same geometry as the current simulation.
-  **Setup Link** must be defined when selecting **From Link** or **Import Mesh**.

Magnetostatic Solution Process

-  Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



THIS PAGE INTENTIONALLY LEFT BLANK

Eddy Current Analysis

Eddy Current Analysis

-  Eddy Current Analysis is performed by choosing the Eddy Current solution type.
-  Applications that use Eddy Current Analysis can be solenoids, inductors, motors, stray field calculations and many others.

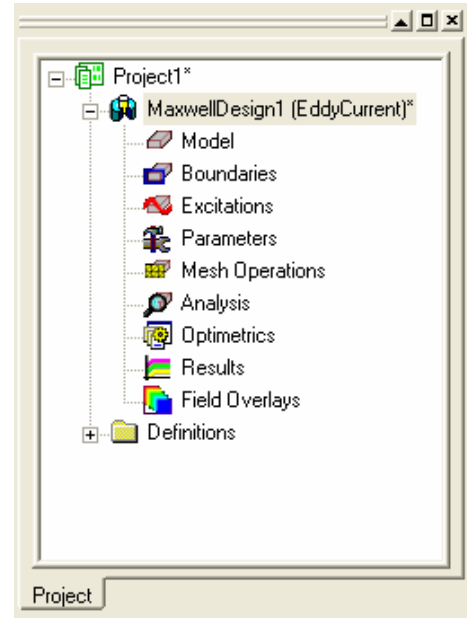
Overview

-  The eddy current solver computes steady-state, time-varying (AC) magnetic fields at a given frequency - this is a frequency domain solution.
-  All objects are stationary.
-  The source of the static magnetic field can be:
 -  Sinusoidal AC current (peak) in conductors.
 -  Time-varying external magnetic fields represented by external boundary conditions.
-  The quantities solved are the magnetic field (H) and the magnetic scalar potential (Ω).
-  Current density (J) and magnetic flux density (B) are automatically calculated from the magnetic field (H).
-  Derived quantities such as forces, torques, energy, and inductances may be calculated from these basic field quantities.
-  Material permeabilities and conductivities can be anisotropic, but must be linear.

Setup

- ▲ The options in the project tree for a eddy current simulation control all the simulation setup parameters.
 - ▲ Notice that right-clicking on any of the options will open a menu with important options for each step of the simulation setup.

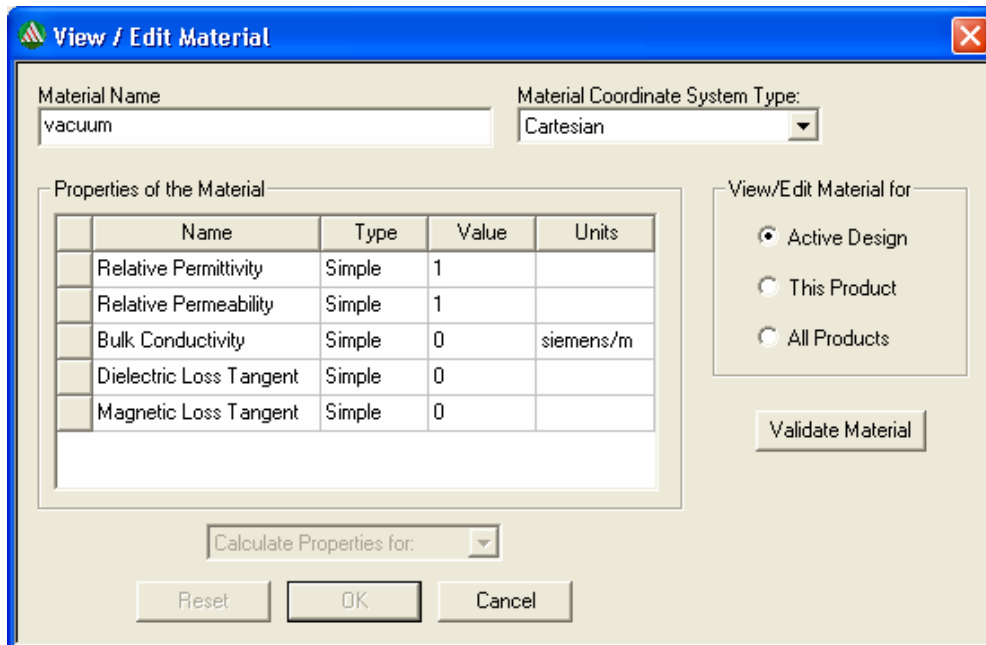
- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in a eddy current simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ Mesh Operations are discussed in a separate section.
- ▲ Analysis defines the solution setup.
- ▲ Optimetrics defines any automatic variational analyses.
- ▲ Results and Field overlays are discussed in a separate section.



- ▲ These options are displayed in an order that can be followed in creating a new Eddy Current simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.
- ▲ However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.

▲ Eddy Current Material Definition

- ▲ In an Eddy Current simulation, the following parameters may be defined for a material:
 - ▲ **Relative Permittivity** (can be Anisotropic or Simple)
 - ▲ Relative Permittivity effects the solution when displacement currents are considered in an object.
 - ▲ **Relative Permeability** (can be Anisotropic or Simple)
 - ▲ Relative Permeability along with the Bulk Conductivity determine the time-varying magnetic properties of the material.
 - ▲ **Bulk Conductivity** (can be Anisotropic or Simple)
 - ▲ Bulk Conductivity is used both in determining the current distribution in current carrying conductors and in calculating eddy currents and the resulting magnetic field solution.
 - ▲ **Dielectric Loss Tangent** (can be Anisotropic or Simple)
 - ▲ Dielectric Loss Tangent controls the ratio of imaginary and real permittivities.
 - ▲ **Magnetic Loss Tangent** (can be Anisotropic or Simple)
 - ▲ Magnetic Loss Tangent controls the ratio of imaginary and real permeabilities.



The dialog box titled "View / Edit Material" contains the following fields and controls:

- Material Name:** A text field containing "vacuum".
- Material Coordinate System Type:** A dropdown menu set to "Cartesian".
- Properties of the Material:** A table with 5 columns: Name, Type, Value, and Units.

Name	Type	Value	Units
Relative Permittivity	Simple	1	
Relative Permeability	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
Dielectric Loss Tangent	Simple	0	
Magnetic Loss Tangent	Simple	0	
- View/Edit Material for:** A group box containing three radio buttons: "Active Design" (selected), "This Product", and "All Products".
- Buttons:** "Validate Material", "Reset", "OK", and "Cancel".
- Calculate Properties for:** A dropdown menu at the bottom.

Eddy Current Boundary Conditions

 **Default** - The default boundary conditions for the Eddy Current solver are:

 **Natural** boundaries on the interface between objects.

- This means that the H Field is continuous across the boundary.

 **Neumann** boundaries on the outer boundaries.

- This means that the H Field is tangential to the boundary and flux cannot cross it.

 **Zero Tangential H Field** - This boundary (similar to the Magnetostatic version) is often used to model an applied uniform, external field. This would model outer boundaries of the Region that are perpendicular to the applied field. In this case, the boundary should be placed far from the structure so that the simulation is not over-defined. This is equivalent to the even symmetry boundary definition.

 **Tangential H Field** - This boundary (similar to the Magnetostatic version) is used primarily to model an applied uniform, external field. This would model outer boundaries of the Region that have a defined tangential magnetic field. This boundary should always be placed far from the structure so that the simulation is not over-defined. Faces must be planar and must be defined one at a time, due to the U-V field definition on each face.

 To apply a uniform field along any orthogonal axis of a bounding box, first, apply a Zero Tangential H Field on the top and bottom faces (with respect to the direction of the axis) of the box. Then apply a Tangential H Field to each side face - define each U vector to be parallel to the selected axis (the V vector does not matter because no field will exist in that direction), and assign the real and imaginary values of U with constant values. The field will be in the direction of the selected axis (perpendicular to the top and bottom faces), and will have a constant value away from the defined objects in the simulation.

 Note that it is very easy to create impossible field assignments with these boundary conditions. If your simulation does not converge when using these boundaries, try the boundary conditions without any objects included to see if the boundaries are assigned correctly (the simulation will have difficulties converging if boundaries are incorrect).

Eddy Current Boundary Conditions (Continued)

-  **Insulating** - This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate coils from magnetic steel (defined on surfaces between the two objects), but there are many other applications.
-  **Symmetry** - There are two magnetic symmetries - odd symmetry (flux tangential) and even symmetry (flux normal). Odd symmetry defines H to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Even symmetry defines H to be normal to the boundary (this is equivalent to the Zero Tangential H Field boundary condition). Remember that geometric symmetry may not mean magnetic symmetry in all cases. 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. Other considerations for a Symmetry boundary condition are the same as for a Magnetostatic symmetry boundary.
-  **Master/Slave** - This boundary condition is also known as a matching boundary condition because it matches the magnetic field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement for matched geometries) and the faces need to be planar. It is required to define a U-V coordinate system along each face to properly map the matched boundary as desired. Master and Slave boundaries enable you to model only one period of a periodic structure, which will reduce the size of a design. Example applications are periodic rotational machines or infinite arrays.
-  **Radiation** - This boundary condition is specific to the Eddy Current solver. See the discussion of the Radiation boundary in the Radiation Boundary example.
-  **Impedance** - This boundary can model induced currents within excluded objects without explicitly solving within the objects. This can decrease simulation time because the difficult to mesh areas near the surface of objects can be ignored and approximated with this boundary. By excluding the object (accomplished by deselecting Solve Inside in the object properties), there will be no solution inside the object. This approximation is good for good conductors (where the skin depth is less than half the width of the excluded conductor).

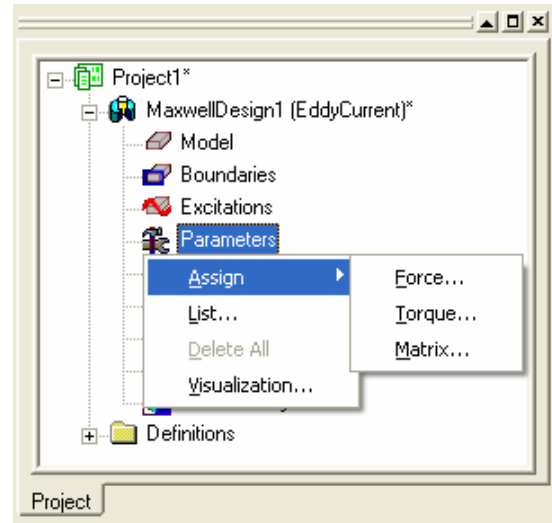
Eddy Current Excitations

-  Typical sources for eddy current problems include current and current density. In applying the sources for the magnetic field problems, keep in mind that the applied current distribution must be divergence free in the entire space of the solution as it is physical for (quasi) stationary conduction current density distributions. Thus, the conduction paths(s) for the applied current distributions must be closed when totally contained within the solution space for the problem or must begin and end at the boundaries. The total current applied to conductors that touch the boundaries doesn't require the existence of terminals at the ends where the current is applied, the respective planar surfaces of the conductors in the plane of the region (background) can be used to apply the excitations.
-  **Current Excitations** - This excitation can be assigned on any conductor to define the total peak current (amp-turns) through the cross-section of a loop or to define the current into and out of the opposing, external faces of a conducting object. This is a very general purpose excitation that comes in two flavors - Solid or Stranded. More information about Stranded Current excitations (along with a Magnetostatic example and explanation) can be found in the Magnetostatic Switched Reluctance Motor example. The only difference between the use of stranded and solid conductors between the magnetostatic solver and the eddy current solver, is that in the eddy current solver eddy effects are not considered in stranded conductors, but not in solid conductors.
-  **Current Density** - These excitations are used to define a known current density throughout an object and must be used with a Current Density Terminal. The peak Current Density (magnitude and phase) is defined on the 3D object, and the Terminal is defined on either an internal cross-sectional sheet, or on all external cross-sectional faces.

Eddy Current Analysis

Parameters

- ▲ There are three parameters that can be automatically calculated in an Eddy Current simulation - **Force**, **Torque**, and Inductance **Matrix**.
- ▲ All three quantities are computed directly from the magnetic field solution.
- ▲ Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).



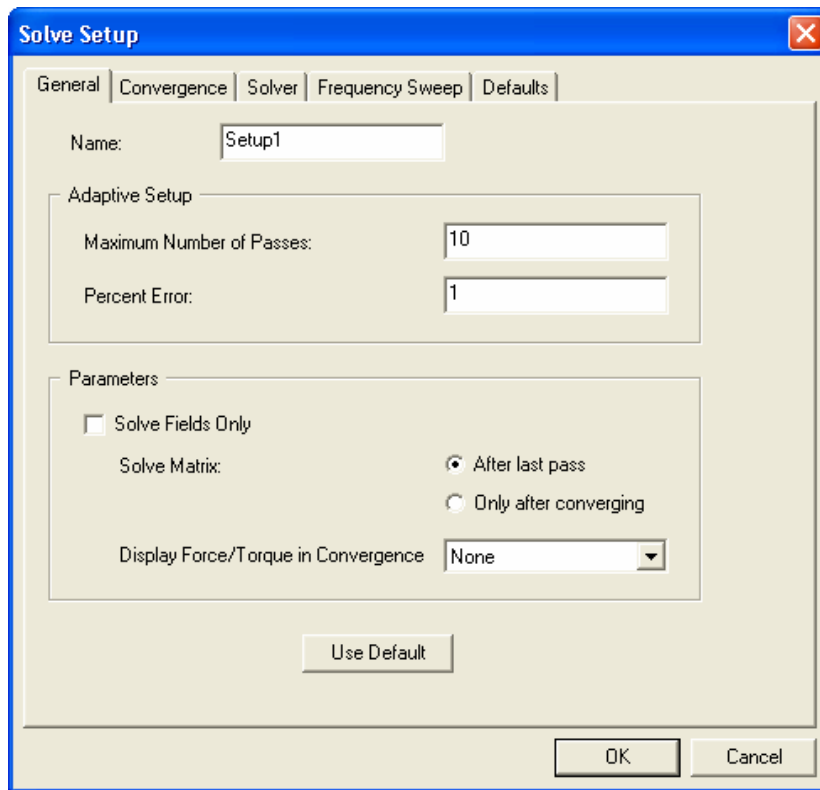
- ▲ The results of any parameters can be found by selecting **Maxwell > Results > Solution Data...** or by clicking on the  icon.
- ▲ Further results can be obtained manually through the field calculator.

Mesh Operations

- ▲ Mesh operations are described in detail in the Mesh operations section.
- ▲ Remember that the Eddy Current solver has an adaptive mesh solution, so excessive mesh operations are not always required. It can often be worse to over-define the mesh than to not define mesh operations at all (it will take longer to solve, and it will be more difficult to adapt correctly).
- ▲ However, it is very important to mesh properly to account for currents with small skin depths - this is important on solid conducting objects.

Solution Setup

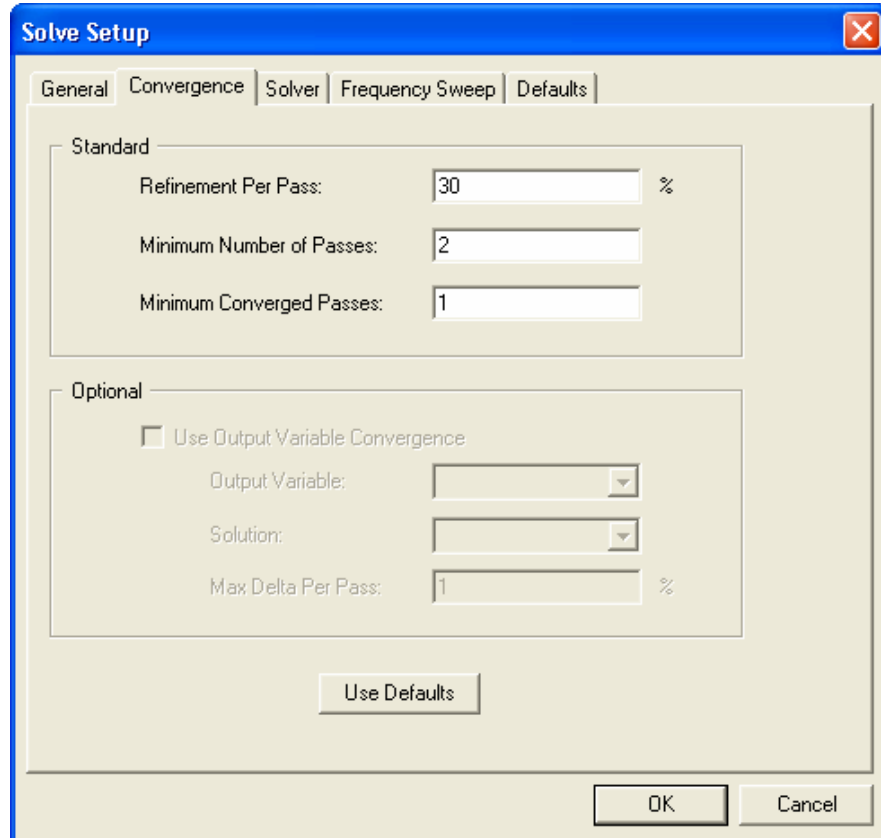
-  The solution setup defines the parameters used for solving the simulation.
-  Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
-  The following window appears with the General Setup parameters.



-  You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
-  **Maximum Number of Passes** defines a limit to the number of adaptively refined passes that the solver performs (the default value is 10).
-  **Percent Error** is the error goal for both the Error Energy and Delta Energy.
-  **Solve Fields Only** ignores any defined parameters if checked.
-  **Solve Matrix** has the options of calculating the matrix after the last solved pass or calculating the matrix only if the solution converges.
-  An option is included to display one Force or Torque parameter in the Convergence table.

Solution Setup (Continued)

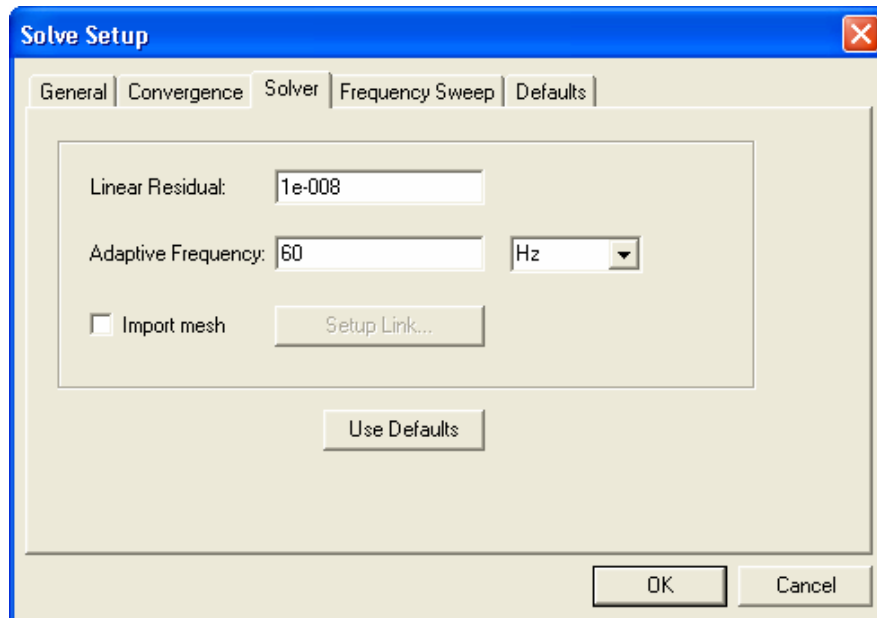
- The second tab of the Solution Setup contains information about Convergence.



- Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
- Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
- Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).
- Use Output Variable Convergence** is an option to include a defined output variable as an additional convergence criterion with a specified maximum percent change per pass (an output variable must be defined for this option to be available).

Solution Setup (Continued)

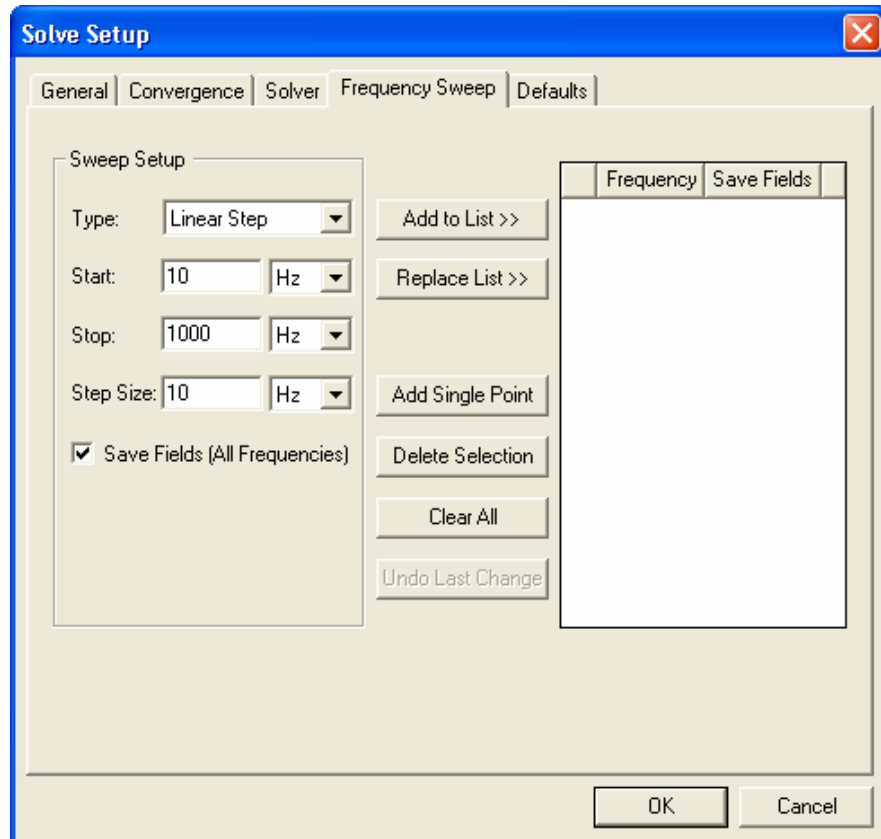
-  The third tab of the Solution Setup contains information about the Solver.



-  **Linear Residual** defines the residual limit of an iterative solver that is automatically used if necessary, but the default solver for eddy current simulations is the direct solver where this limit has no significance (the default value is 1e-8).
-  **Adaptive Frequency** defines the frequency at which the mesh is constructed and adapted, and at which solution is obtained (the default value is 60 Hz).
-  **Import Mesh** allows the initial mesh to be imported from another solution - the linked solution must have the exact same geometry as the current simulation.
-  **Setup Link** must be defined when selecting **Import Mesh**.

Solution Setup (Continued)

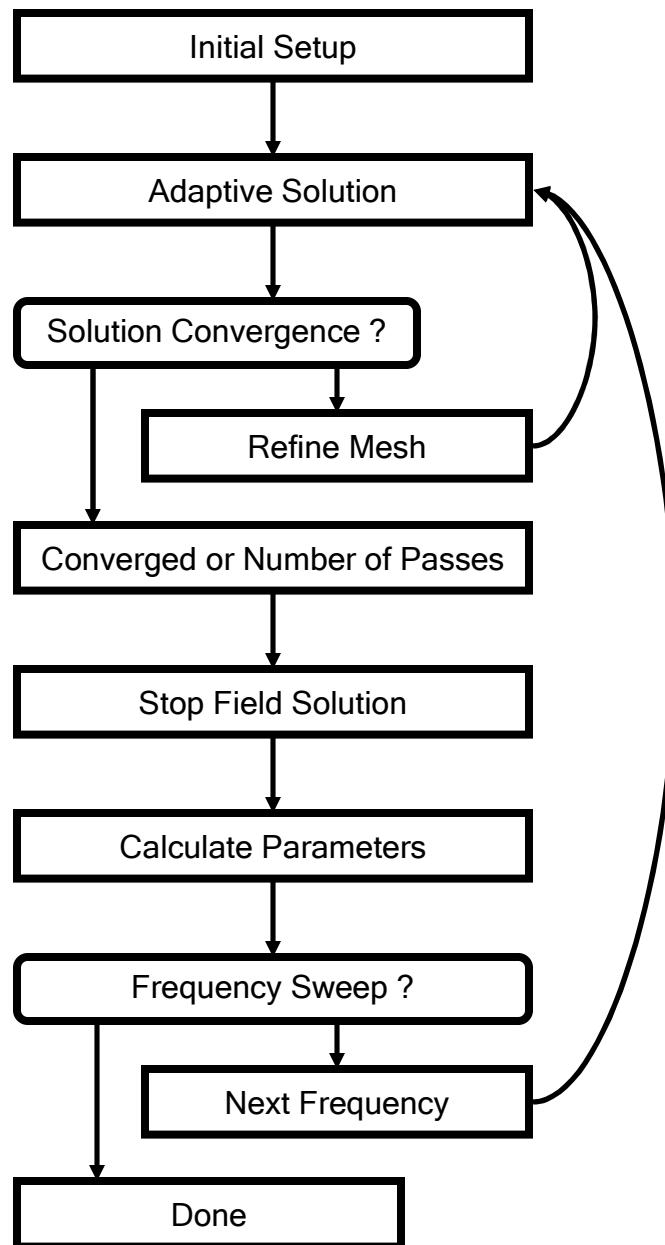
-  The fourth tab of the Solution Setup contains information about an optional Frequency Sweep.



-  Define the sweep (**Type**, **Start**, **Stop**, **Step**) in the left panel.
-  Check **Save Fields (All Frequencies)** to save the fields for all frequencies in this sweep definition.
-  Select **Add to List >>** to place this sweep definition in the Sweep List (the Sweep List is displayed in the right panel).
-  Edit any entries in the Sweep List to adjust solution frequencies or whether to save fields at specific frequencies in the list.

Eddy Current Solution Process

- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



Transient Analysis

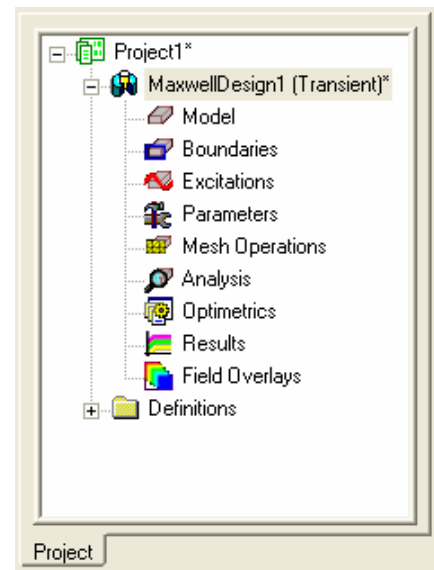
-  Transient Analysis is performed by choosing the Transient solution type.
-  Applications that use Transient Analysis can be solenoids, inductors, motors, actuators, permanent magnets and many others.

Overview

-  The Transient solver computes magnetic fields in the time domain (instantaneously at each time step).
-  The solver formulation is based on a current vector potential in solid conductors, and a scalar potential over the entire field domain.
-  The source of the static magnetic field can be:
 -  Arbitrary time-varying current in conductors.
 -  Permanent magnets.
-  Field Quantities are strongly coupled with circuit equations to allow voltage sources and/or external driving circuits.
-  The quantity solved is the magnetic field (H) and the current density (J) while magnetic flux density (B) is automatically calculated from the H-field.
-  Derived quantities such as forces, torques, flux linkage and core loss may be calculated from these basic field quantities.
-  Material permeabilities can be nonlinear and/or anisotropic.
-  Permanent magnets are considered.
-  Excitations can be sinusoidal or non-sinusoidal including:
 -  Voltages and currents applied to windings.
 -  External circuits attached to windings.
 -  Permanent magnets.

Setup

- ▲ The options in the project tree for a transient simulation control all the simulation setup parameters.
 - ▲ Notice that right-clicking on any of the options will open a menu with important options for each step of the simulation setup.
- ▲ The Model definition refers to the geometry and material, as well as motion definitions.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in a transient simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ Mesh Operations are discussed in a separate section.
- ▲ Analysis defines the solution setup.
- ▲ Optimetrics defines any automatic variational analyses.
- ▲ Results and Field overlays are discussed in a separate section.
- ▲ These options are displayed in an order that can be followed in creating a new Transient simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.
- ▲ However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.



Transient Material Definition

In a Transient simulation, the following parameters may be defined for a material:

Relative Permeability (can be Anisotropic and/or Nonlinear, or Simple)

Relative Permeability plays a large role in determining the magnetic field solution.

Bulk Conductivity (can be Anisotropic or Simple)

Bulk Conductivity is used both in determining the current distribution in current carrying conductors and in finding the eddy currents in solid conductors, which affect the magnetic solution.

Magnetic Coercivity (defined as a vector magnitude and direction)

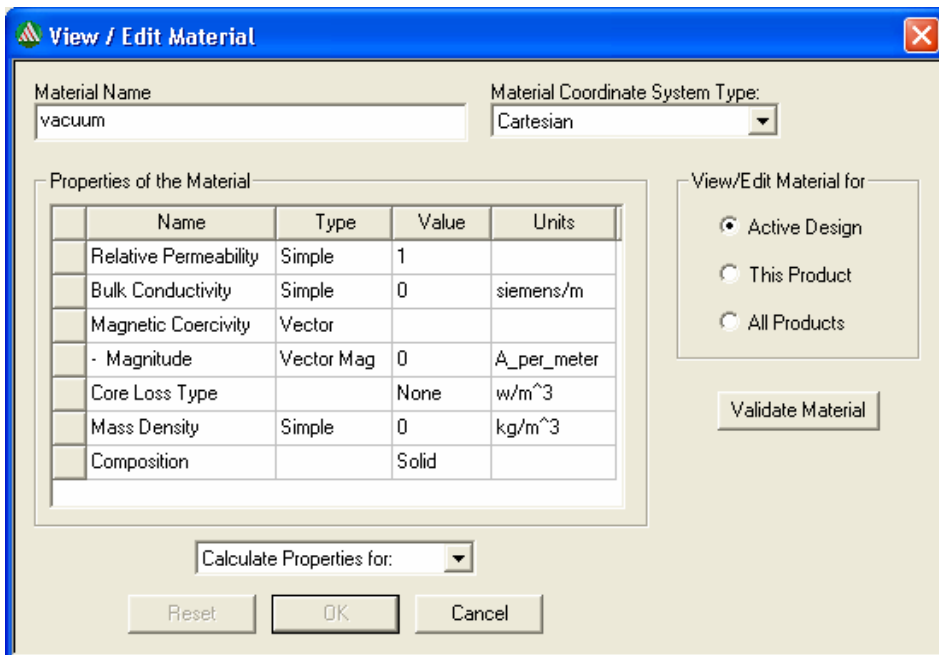
Magnetic Coercivity is used to define the permanent magnetization of magnetic materials. When a non-zero magnitude is entered, the directional entries are visible. The direction (like all directional material properties) are determined by the coordinate system type and the object orientation.

Core Loss Type (can be Electrical Steel, Power Ferrite, or None)

Mass Density

Composition (can be Solid or Lamination)

Setting Composition to Lamination creates an anisotropic magnetization effect. This is discussed in the Magnetostatic Anisotropic Material example.



The dialog box shows the material properties for a material named 'vacuum'. The Material Coordinate System Type is set to 'Cartesian'. The Properties of the Material table is as follows:

Name	Type	Value	Units
Relative Permeability	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
Core Loss Type		None	w/m^3
Mass Density	Simple	0	kg/m^3
Composition		Solid	

On the right, the 'View/Edit Material for' section has three radio buttons: 'Active Design' (selected), 'This Product', and 'All Products'. A 'Validate Material' button is located below these. At the bottom, there is a 'Calculate Properties for:' dropdown menu and 'Reset', 'OK', and 'Cancel' buttons.

Transient Boundary Conditions

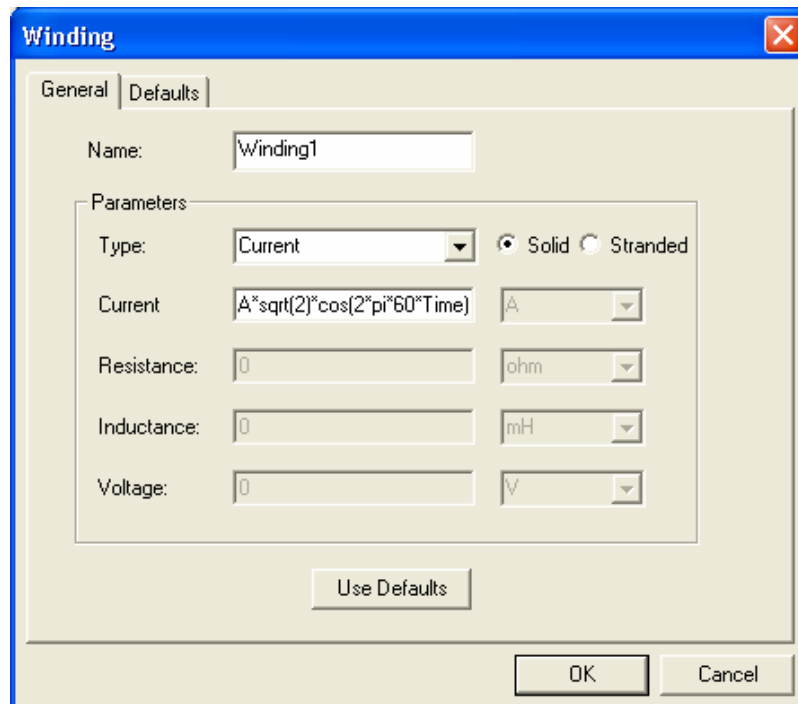
-  **Default** - The default boundary conditions for the Transient solver are:
 -  **Natural** boundaries on the interface between objects.
 - This means that the H Field is continuous across the boundary.
 -  **Neumann** boundaries on the outer boundaries.
 - This means that the H Field is tangential to the boundary and flux cannot cross it.
-  **Zero Tangential H Field** - This boundary acts equivalently on the field as the even symmetry boundary, but is used in special cases only. Use the even symmetry boundary to model symmetries with normal H fields to the symmetry plane.
-  **Insulating** - This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate coils from magnetic steel (defined on surfaces between the two objects), but there are many other applications.
-  **Symmetry** - There are two magnetic symmetries - odd symmetry (flux tangential) and even symmetry (flux normal). Odd symmetry defines H to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Even symmetry defines H to be normal to the boundary (this is equivalent Zero Tangential H Field boundary condition). Remember that geometric symmetry may not mean magnetic symmetry in all cases. 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. Other considerations for a Symmetry boundary condition are the same as for a Magnetostatic symmetry boundary.
-  **Master/Slave** - This boundary condition is also known as a matching boundary condition because it matches the magnetic field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement for matched geometries) and the faces need to be planar. It is required to define a U-V coordinate system along each face to properly map the matched boundary as desired. Master and Slave boundaries enable you to model only one period of a periodic structure, which will reduce the size of a design. Example applications are periodic rotational machines or infinite arrays.

Transient Excitations

-  Typical sources for transient field problems include coil terminals of type voltage, current or external circuit. When applying the sources for the magnetic field problems, the applied current distribution must be divergence free in the entire space of the solution as it is physical for quasi-stationary conduction current density distributions. Thus, the conduction path(s) for the applied current distributions must be closed when totally contained within the solution space for the problem, or must begin and end at the boundaries.
-  Permanent Magnets can also act as sources for a transient analysis, however, those are defined elsewhere.
-  **Coil Terminals** - Coil terminals are defined to designate the cross sectional faces of the 3D conductors. These can either be located internally to a closed loop, or on the external faces of a conduction path. The coil terminals are grouped into a Winding that controls the current in one or multiple conduction paths. The only things that are defined by the Coil Terminals are as follows:
 -  Number of conductors (solid windings require 1 conductor)
 -  Direction of the current
 -  A cross section of the conductor
-  **Windings** - Windings control the current flowing through the conductors and are therefore crucial to the magnetic solution. There are three different types of winding:
 -  **Current** - defines a specified functional current through the conducting paths.
 -  **Voltage** - defines a specified functional voltage across the coil terminals (and a series resistance and inductance).
 -  **External** - defines that an external circuit will control the current and voltage associated with the conducting path.
-  Coil terminals must be added to a Winding to complete the excitation setup.
-  Coil terminals will automatically report flux linkage and induced voltage vs. time in the 2D reporter.

Winding Setup

- Assign a coil terminal to a winding by either:
 - Right-clicking on the winding and choosing **Add Terminals...**
 - or
 - Right-clicking on the terminal and choosing **Add to Winding...**
- Group coil terminals by adding them to the same winding.
- Grouped conductors are considered in series - the defined current goes through each conductor, voltage is defined across all conductors plus additional resistance and/or inductance - total winding inductance is treated as the sum of each coil's inductance.
- In the Winding setup:
 - Define the Winding to be the desired Type and Solid vs. Stranded.
 - Then, define the necessary parameters - current or voltage can be a function of time in their respective source types. Resistance and inductance are available for voltage sources (to determine the current), and are considered in series with winding.

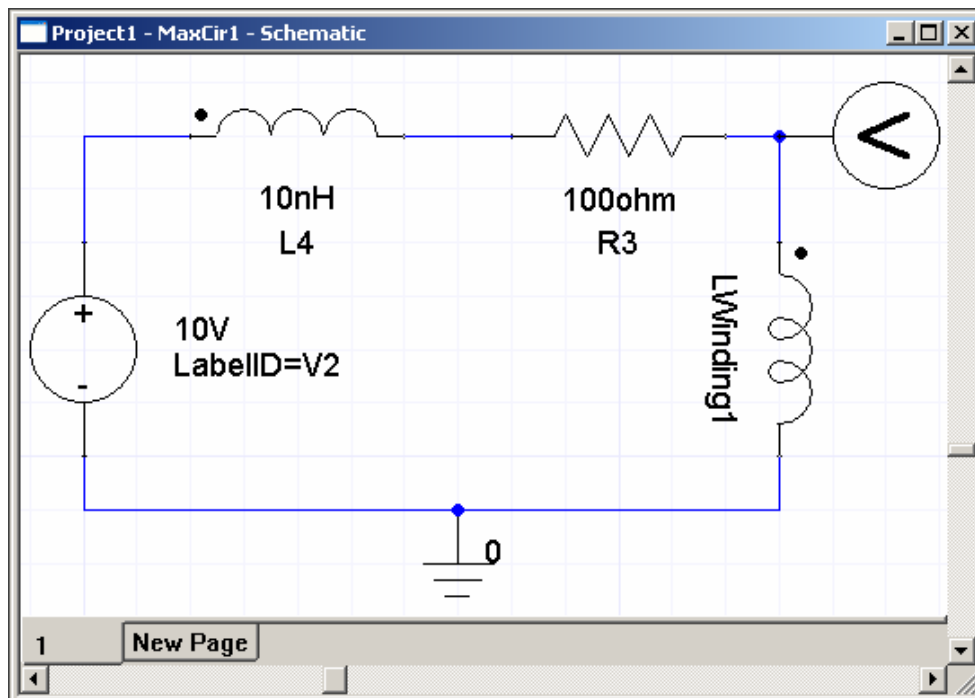


External Circuits

- ▲ Step1: Open Maxwell Circuit Editor
(*Windows Start Menu > Programs > Ansoft > Maxwell 11 > Maxwell Circuit Editor*)



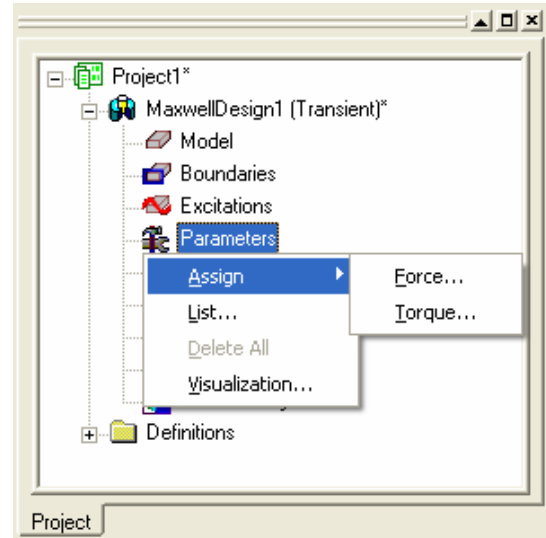
- ▲ Step 2: Create Circuit in Maxwell Circuit Editor Schematic
 - ▲ In the editor, each winding should appear as a Winding element, with a matching name for each winding - i.e. Winding1 in the transient simulation would require Winding1 in the schematic (displayed as LWinding1 on the schematic sheet).



- ▲ Step 3: Export netlist from the circuit editor (*Maxwell Circuit > Export Netlist*).
 - ▲ Save the Circuit Editor project so that you can edit your circuit later.
- ▲ Step 4: Import the netlist into Maxwell (*Maxwell > Excitations > External Circuit > Edit External Circuit...*)
 - ▲ Choose **Import Circuit...** to import the netlist.
 - ▲ You must re-import the netlist every time that you make a change to the circuit, otherwise the change will not take effect.

Parameters

- ▲ There are two parameters that can be automatically calculated in a Transient simulation - **Force** and **Torque**.
- ▲ Both quantities are computed directly from the magnetic field solution.
- ▲ Force and torque are calculated with the Virtual work method.
- ▲ With the motion setup, a force or torque associated with the translational or rotational motion is automatically calculated - this force or torque (from the motion setup) may or may not be exactly equal to a similar parameter assigned on the same set of moving objects.
- ▲ The results of the parameters are not located in the solution results for a transient simulation - however, they can be obtained by creating a 2D report to find the parameter values as a function of time.
- ▲ Further results can be obtained manually through the field calculator.

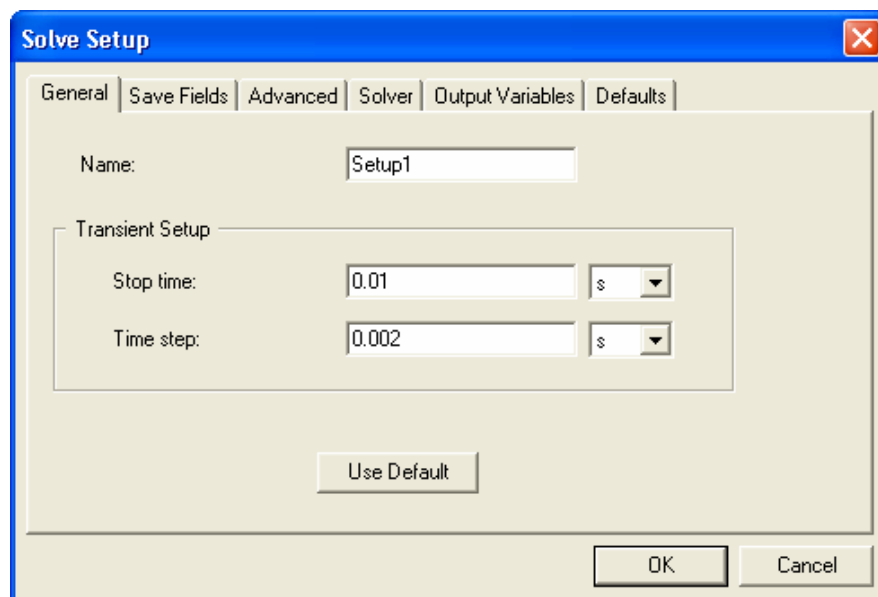


Mesh Operations

- ▲ Mesh operations are described in detail in the Mesh operations section.
- ▲ Remember that the Transient solver does not have an adaptive mesh solution, so significant, intelligent mesh operations are required. There are techniques to obtain a more defined mesh, such as linking a transient simulation to the mesh from a Magnetostatic simulation. Also, skin-depth meshing can be very important if solid conductors have significant eddy effects.

Solution Setup

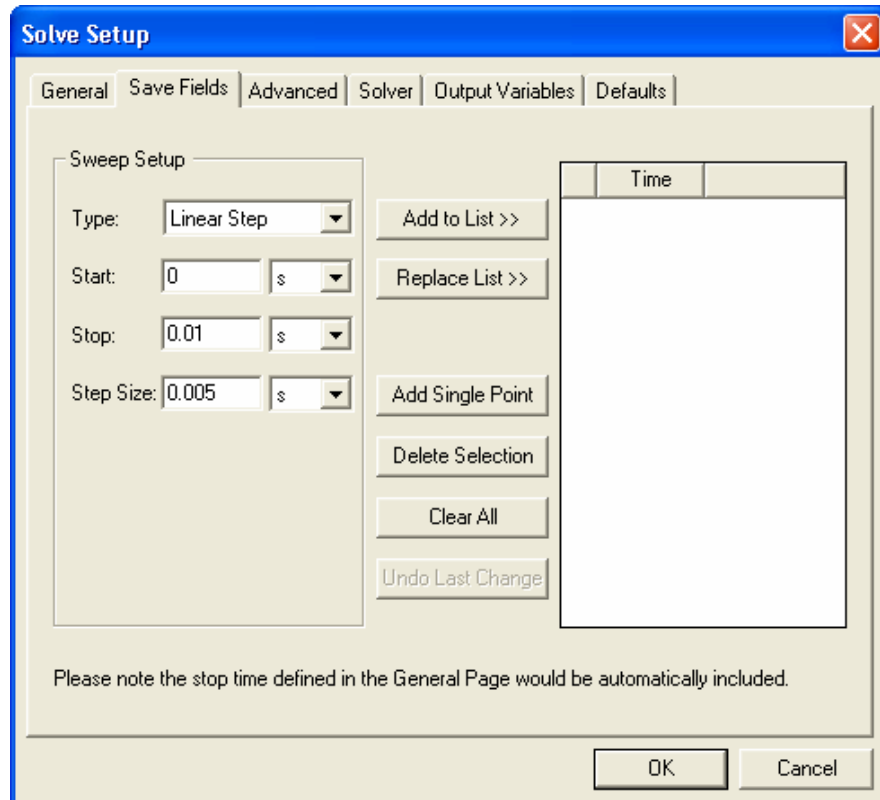
-  The solution setup defines the parameters used for solving the simulation.
-  Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
-  The following window appears with the General Setup parameters.



-  You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
-  **Stop time** indicates the transient simulation time at which the simulation will stop.
-  **Time step** indicates the discrete lengths of time used in the transient simulation.
-  Notice that the general information is related to the transient nature of this simulation - there is no information about convergence, because there is no adaptive solution to converge with.
-  Choose the time steps appropriately for the physical time-constants of the simulation (about 20 timesteps per cycle).

Solution Setup (Continued)

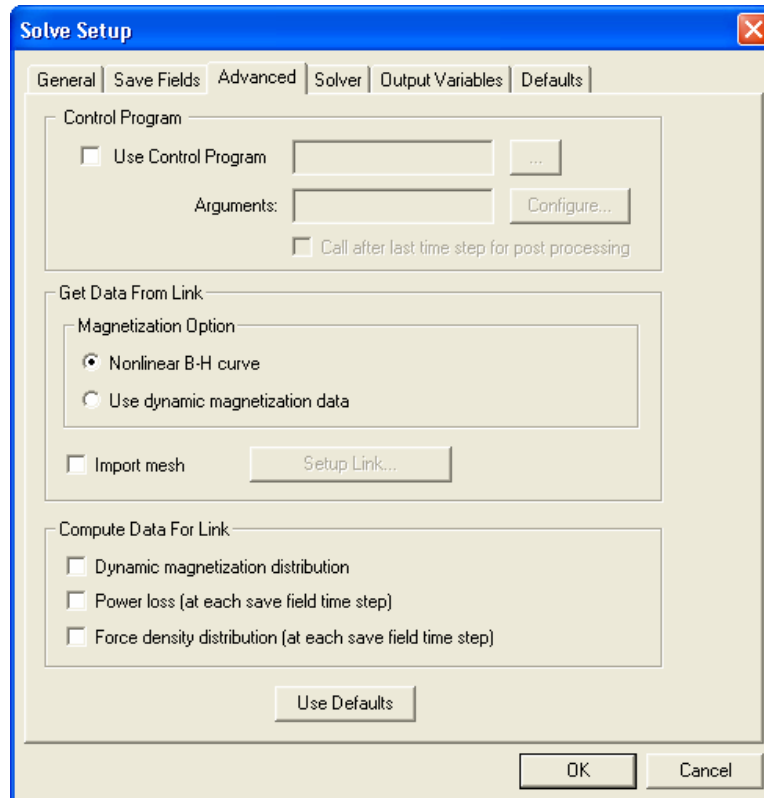
-  The second tab of the Solution Setup contains information about Saving Fields.



-  Define the sweep (**Type, Start, Stop, Step**) in the left panel.
-  Select **Add to List >>** to place this sweep definition in the Sweep List (the Sweep List is displayed in the right panel).
-  Edit any entries in the Sweep List to adjust saved fields times.
-  The times that are included in this list will have a full field solution available for post-processing. All other time-steps that are solved will not keep their field solutions after solving and moving on to the next time step.
-  The last time step in a simulation will retain its fields automatically.

Solution Setup (Continued)

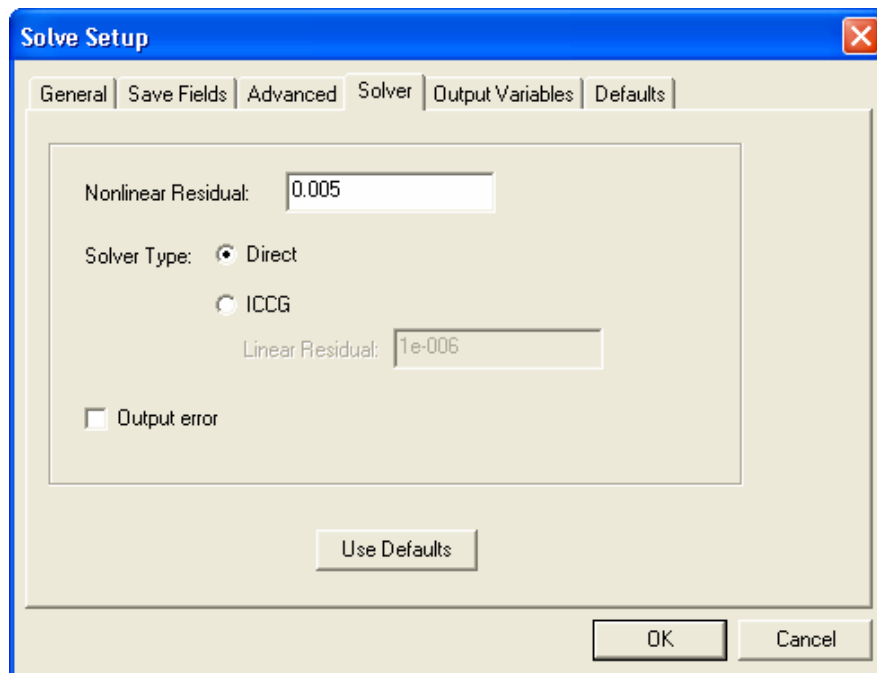
-  The third tab of the Solution Setup contains Advanced options.



-  **Control Programs** are used to dynamically adjust parameters and control the simulation - information can be found in the Maxwell Help.
-  **Permeability Option** allows the nonlinear B-H operating points either to be calculated by the solver from the **Nonlinear B-H curve** or to use frozen permeabilities **From Link** - the linked solution must have the exact same geometry as the current simulation (Nonlinear B-H curve is the default).
-  **Magnetization Option** allows the permanent magnetization to be determined from the **Nonlinear B-H curve** or to use demagnetized values **From Link** - where the linked solution selected **Compute demagnetized operating points** - the linked solution must have the exact same geometry (Nonlinear B-H curve is the default).
-  **Import Mesh** allows the initial mesh to be imported from another solution - the linked solution must have the exact same geometry as the current simulation.
-  **Setup Link** must be defined when selecting **From Link** or **Import Mesh**.

Solution Setup (Continued)

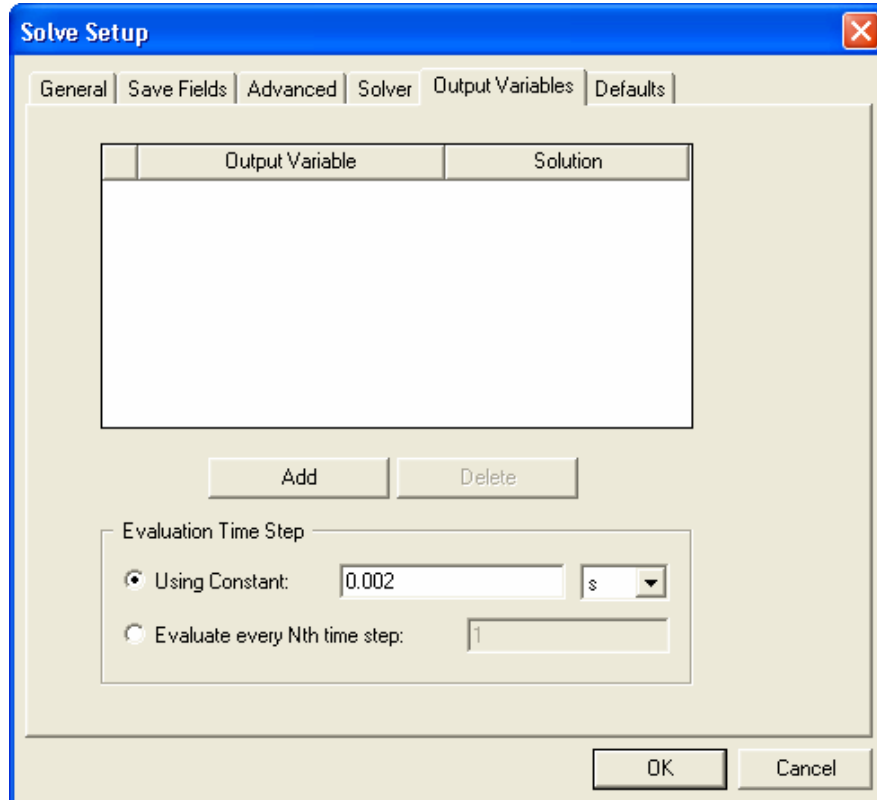
-  The fourth tab of the Solution Setup contains information about the Solver.



-  **Nonlinear Residual** defines how precisely the nonlinear solution must define the B-H nonlinear operating points (the default value is 0.005).
-  **Solver Type** has options for the **Direct** or **ICCG** solvers (Direct is the default).
-  **Output Error** will calculate the percent energy error (as described in the Solver section). This provides some measure of convergence of the total field solution at each time step. Remember however, that this does not guarantee that the field solution is converged at all points. Also, the calculation of this quantity requires a moderate increase in solution time (because the calculation is evaluated at every time step).

Solution Setup (Continued)

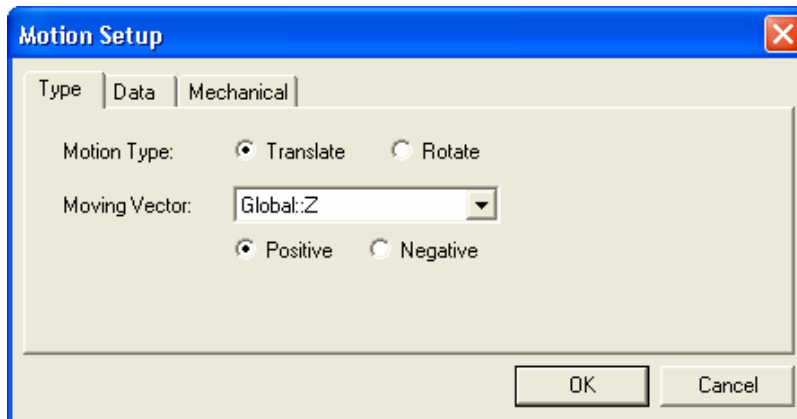
- The fifth tab of the Solution Setup contains Output Variable options.



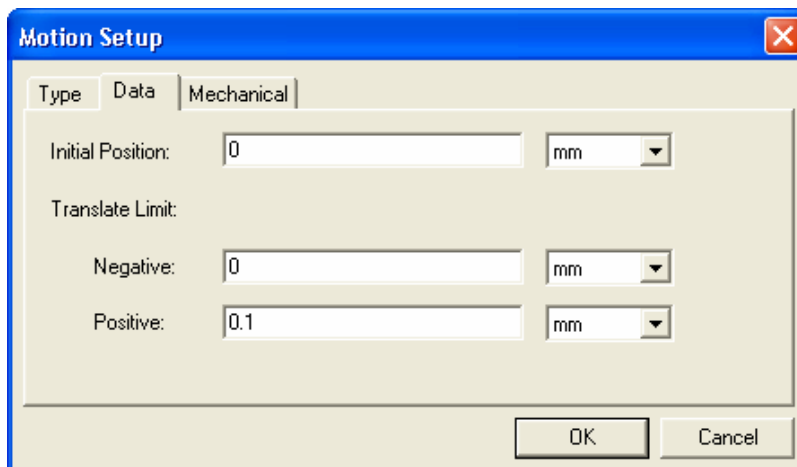
- Output Variables** are added to the list by selecting **Add** then selecting the available output variables from the pull down list.
- This output variable option requires that you first define an output variable before adding it (however, you often need to define the analysis setup before you can define an output variable - so do not be disoriented if you need to return to these settings after defining the analysis setup).
- There are two options for defining the time steps at which these output variables are calculated.
 - The first option is to evaluate the output variables at evenly spaced increments of time by using a constant time step.
 - The second option is to evaluate the output variables every Nth step, where N is some integer. The output variable will then be computed at 0s, and then at step N, step 2*N and so on.

Motion Setup

-  Choose **Assign Band** to specify the band object (*Maxwell > Model > Motion Setup > Assign Band...*).
-  The Motion Setup window appears when the band is assigned.

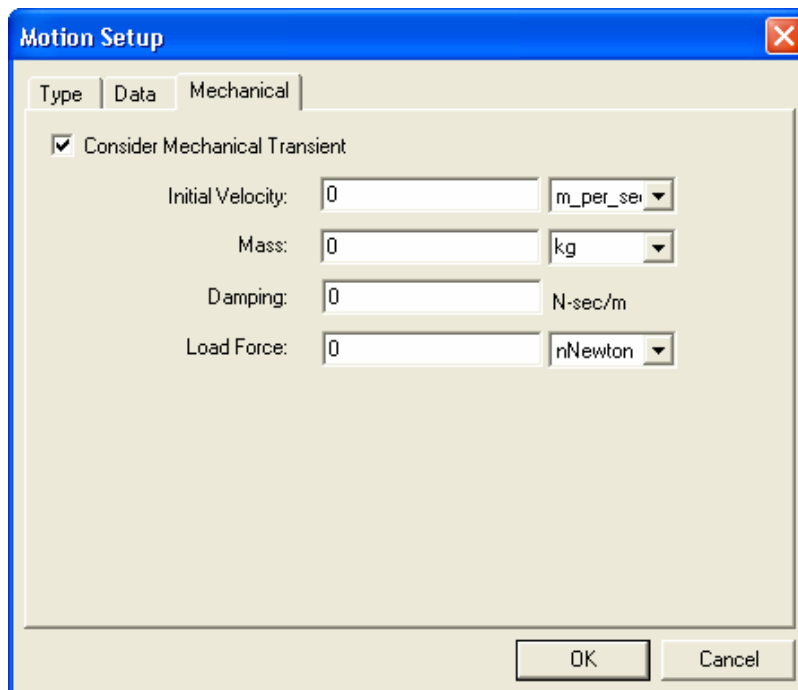


-  Specify **Rotational** or **Translational**
-  Set the **Moving Vector** (this vector can be defined along one of the axis of any coordinate system - you may need to construct a new coordinate system to properly assign the proper direction of motion or rotational axis).
-  Set **Positive** or **Negative** to define the direction with respect to the direction of the positive axis of the Moving Vector.
-  Then define the **Initial Position** and translational or rotational limits.



Motion Setup (Continued)

-  In the Mechanical tab of the motion setup, there are two fundamental options:
 -  Velocity Definition
 - or
 -  Consider Mechanical Transient
-  Velocity Definition is useful when the velocity is constant or follows a known trajectory that can be expressed as a velocity as a function of time. Use this option by de-selecting Consider Mechanical Transient.
-  Consider Mechanical Transient is useful for situations when the velocity varies dynamically or is unknown. Requires the following inputs:
 -  **Initial Velocity or Initial Angular Velocity**
 -  **Mass or Moment of Inertia**
 -  **Damping**
 -  **Load Force**



Motion Setup

Type | Data | **Mechanical**

☒ Consider Mechanical Transient

Initial Velocity: 0 m_per_sec

Mass: 0 kg

Damping: 0 N-sec/m

Load Force: 0 nNewton

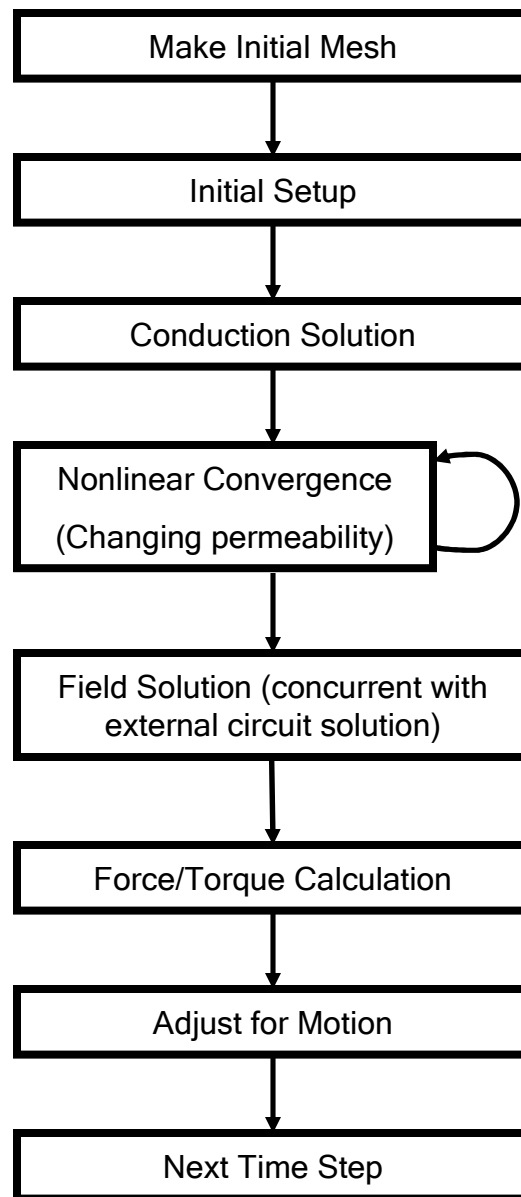
OK Cancel

Motion Setup - Suggestions

-  Read the Maxwell Help note called *Meshing Aspects for 3D Transient Applications With Motion*
-  When performing translational or non-cylindrical rotational motion, always include a vacuum container object within the band that contains all the moving parts. This container object should be spaced slightly away from all the objects that it contains and should provide clearance when the objects and container move within the band. This container object is necessary for both meshing purposes with multiple objects and to produce a better force calculation surface. A vacuum container object is often useful in cylindrical motion too, but not necessary due to meshing considerations. The objects within the vacuum container are assumed to move as one rigid body (all moving objects are assumed to move as one rigid body by definition of the transient motion solution).
-  When conceptualizing translational or non-cylindrical rotational motion, remember that the moving parts within the vacuum container move as one solid body and the remeshing occurs between the surface of the band object and the moving objects (generally a container object).
-  When conceptualizing cylindrical rotational motion, remember that the band object and all moving parts rotate without remeshing between.
-  The conceptual difference between the two is that the band is more solid in the cylindrical case (in that it does not change position with respect to the moving parts), and the band is more fluid in the translational and non-cylindrical case (in that the moving parts change position with respect to the band).
-  In the translational and non-cylindrical cases, the mesh the is created within the band is created so that the edge length is not larger than the average edge length for the elements on the surface of the band and the moving parts. If you wish to better define the mesh within the band, you should apply a length-based mesh operation on the surfaces of the band and moving parts (simply on the surface of the vacuum container if used).

Transient Solution Process

- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



THIS PAGE INTENTIONALLY LEFT BLANK

Electrostatic Analysis

-  Electrostatic Analysis is performed by choosing the Electric solution type and selecting the Electrostatic option.
-  Applications that use Electrostatic Analysis can be capacitors, high voltage lines, breakdown voltage calculations and many others.

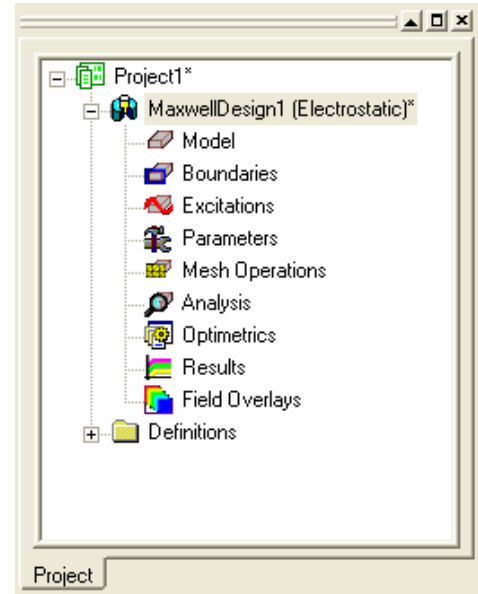
Overview

-  The electrostatic solver computes static (DC) electric fields.
-  All objects are stationary.
-  The source of the static magnetic field can be:
 -  Applied potentials.
 -  Charge distributions.
-  The quantity solved is the electric scalar potential (ϕ).
-  Electric Field (E) and Electric Flux Density (D) are automatically calculated from the scalar potential (ϕ).
-  Derived quantities such as forces, torques, energy, and capacitances may be calculated from these basic field quantities.
-  Material permittivities and conductivities can be anisotropic.
-  All fields inside conductors are assumed to be perfect and equipotential in an electrostatic equilibrium (no current flow), therefore Joule losses are zero everywhere.
-  Conductivity is irrelevant except to define conductors from insulators.
-  Can be coupled with a DC conduction simulation, where the electric potential from the DC conduction solution is used as a voltage boundary condition for the electric field solution in the insulators in an electrostatic simulation.

Setup

- ▲ The options in the project tree for an electrostatic simulation control all the simulation setup parameters.
 - ▲ Notice that right-clicking on any of the options will open a menu with important options for each step of the simulation setup.

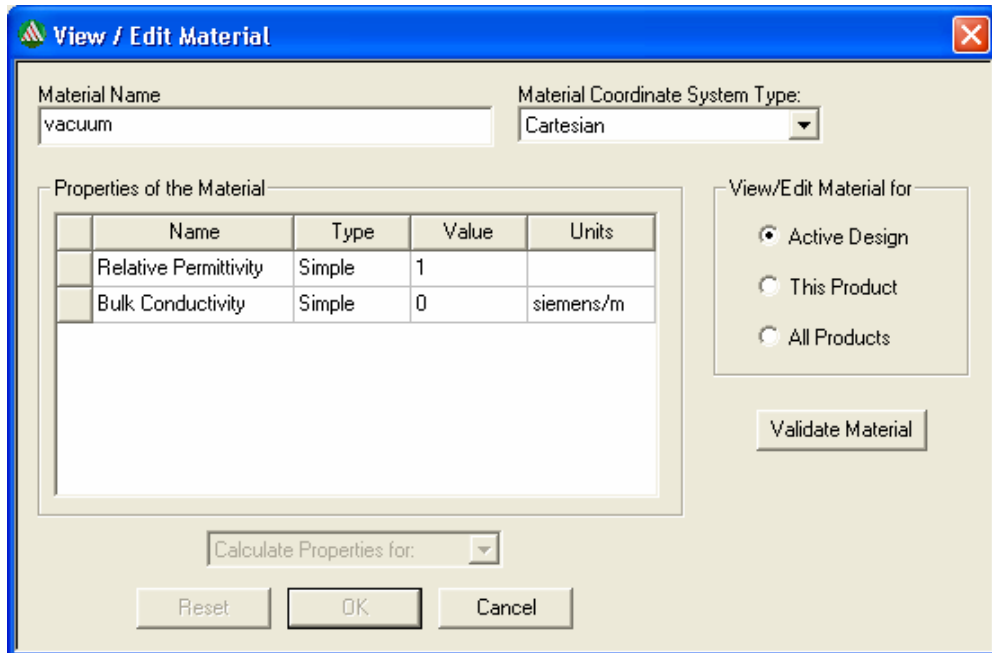
- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in an electrostatic simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ Mesh Operations are discussed in a separate section.
- ▲ Analysis defines the solution setup.
- ▲ Optimetrics defines any automatic variational analyses.
- ▲ Results and Field overlays are discussed in a separate section.



- ▲ These options are displayed in an order that can be followed in creating a new Electrostatic simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.
- ▲ However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.

Electrostatic Material Definition

- ▲ In an Electrostatic simulation, the following parameters may be defined for a material:
 - ▲ **Relative Permittivity** (can be Anisotropic or Simple)
 - ▲ Relative permittivity determines the electric field solution in the insulators.
 - ▲ **Bulk Conductivity** (can be Anisotropic or Simple)
 - ▲ Bulk Conductivity defines whether an object is a conductor (treated as a perfect conductor in the Electrostatic solver) or an insulator. This separation is determined by the insulator/conductor material threshold setting.



The dialog box titled "View / Edit Material" contains the following fields and controls:

- Material Name:** A text field containing "vacuum".
- Material Coordinate System Type:** A dropdown menu set to "Cartesian".
- Properties of the Material:** A table with the following data:

Name	Type	Value	Units
Relative Permittivity	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
- View/Edit Material for:** A group box containing three radio buttons: "Active Design" (selected), "This Product", and "All Products".
- Buttons:** "Validate Material", "Reset", "OK", and "Cancel".
- Calculate Properties for:** A dropdown menu at the bottom.

Electrostatic Boundary Conditions

-  **Default** - The default boundary conditions for the Electrostatic solver are:
 -  **Natural** boundaries on the interface between objects.
 - This means that the normal component of the D Field at the boundary changes by the amount of surface charge density on the boundary.
 -  **Neumann** boundaries on the outer boundaries.
 - This means that the E Field is tangential to the boundary and flux cannot cross it.

-  **Symmetry** - There are two Electric symmetries - even symmetry (flux tangential) and odd symmetry (flux normal). Even symmetry defines E to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Odd symmetry defines E to be normal to the boundary. Remember that geometric symmetry may not mean electric symmetry in all cases. 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. 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

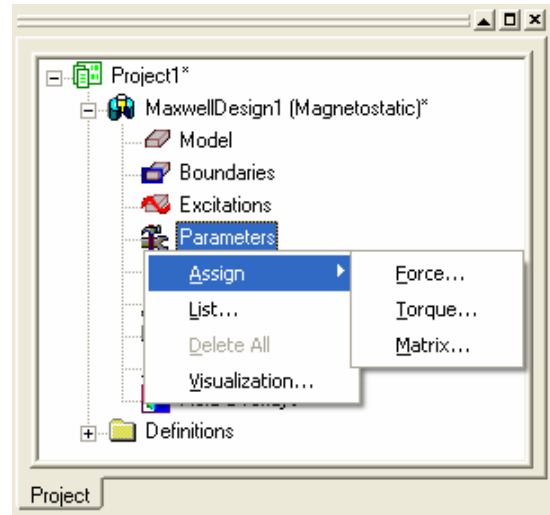
-  **Master/Slave** - This boundary condition is also known as a matching boundary condition because it matches the electric field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement for matched geometries) and the faces need to be planar. It is required to define a U-V coordinate system along each face to properly map the matched boundary as desired. Master and Slave boundaries enable you to model only one period of a periodic structure, which will reduce the size of a design. Example applications are periodic rotational machines or infinite arrays.

Electrostatic Excitations

-  Typical sources for electrostatic problems are net charges (assumed to have a uniform distribution) applied to perfect insulator model objects or on surfaces that cannot touch conductors and voltages (electric potential applied to perfect conductor model objects or on surfaces, also called a Dirichlet boundary condition). Additionally, a floating boundary condition can be applied to perfect conductors (surrounded by insulators) or to surfaces surrounded by perfect insulators.
-  **Voltage Excitations** - surface or object is at a constant, known potential - E field is normal to the boundary.
-  **Charge** - The total charge on a surface or object (either a conductor or dielectric).
-  **Floating** - used to model conductors at unknown potentials
-  **Volume Charge Density** - The charge density in an object.

Parameters

- ▲ There are three parameters that can be automatically calculated in an Electrostatic simulation - **Force**, **Torque**, and Capacitance **Matrix**.
- ▲ All three quantities are computed directly from the electric field solution.
- ▲ Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).



- ▲ The results of any parameters can be found by selecting **Maxwell > Results > Solution Data...** or by clicking on the  icon.
- ▲ Further results can be obtained manually through the field calculator.

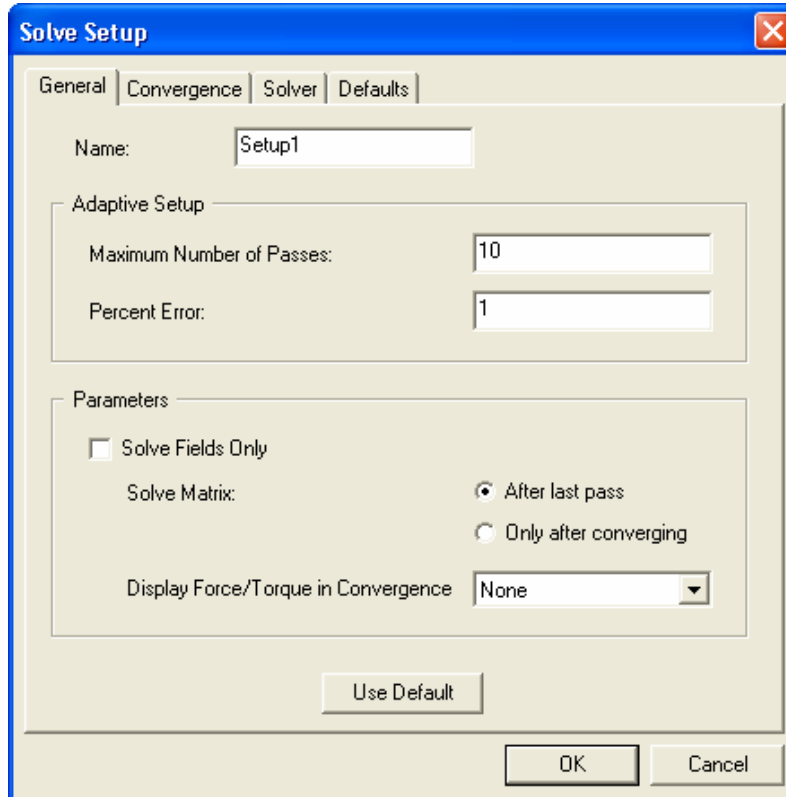
Mesh Operations

- ▲ Mesh operations are described in detail in the Mesh operations section.
- ▲ Remember that the Electrostatic solver has an adaptive mesh solution, so excessive mesh operations are not usually required. It can often be worse to over-define the mesh than to not define mesh operations at all (it will take longer to solve, and it will be more difficult to adapt correctly).

Solution Setup

-  The solution setup defines the parameters used for solving the simulation.
-  Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.

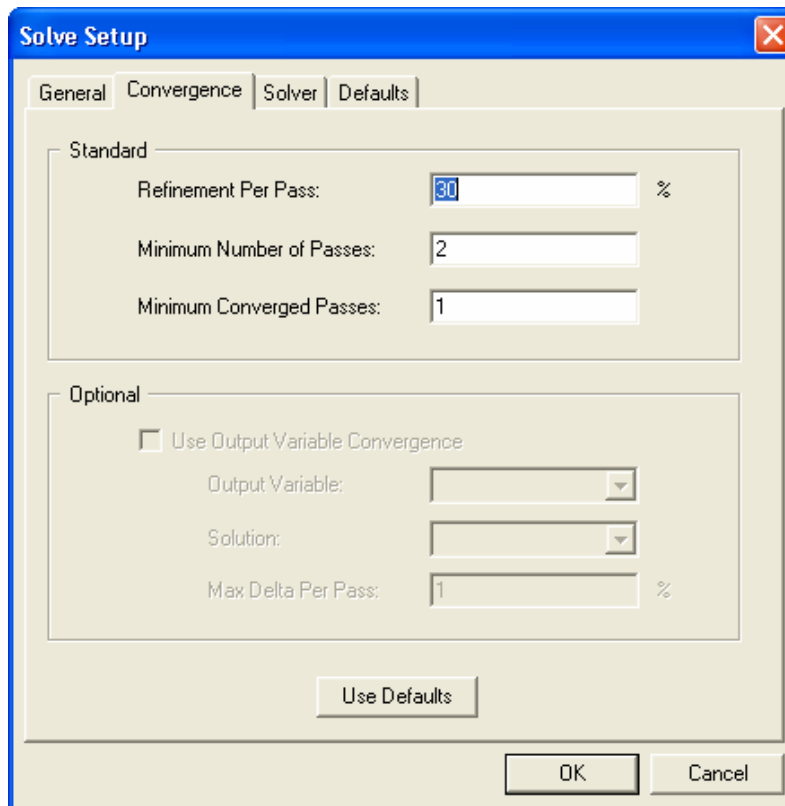
-  The following window appears with the General Setup parameters.



-  You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
-  **Maximum Number of Passes** defines a limit to the number of adaptively refined passes that the solver performs (the default value is 10).
-  **Percent Error** is the error goal for both the Error Energy and Delta Energy.
-  **Solve Fields Only** ignores any defined parameters if checked.
-  **Solve Matrix** has the options of calculating the matrix after the last solved pass or calculating the matrix only if the solution converges.
-  An option is included to display one Force or Torque parameter in the Convergence table.

Solution Setup (Continued)

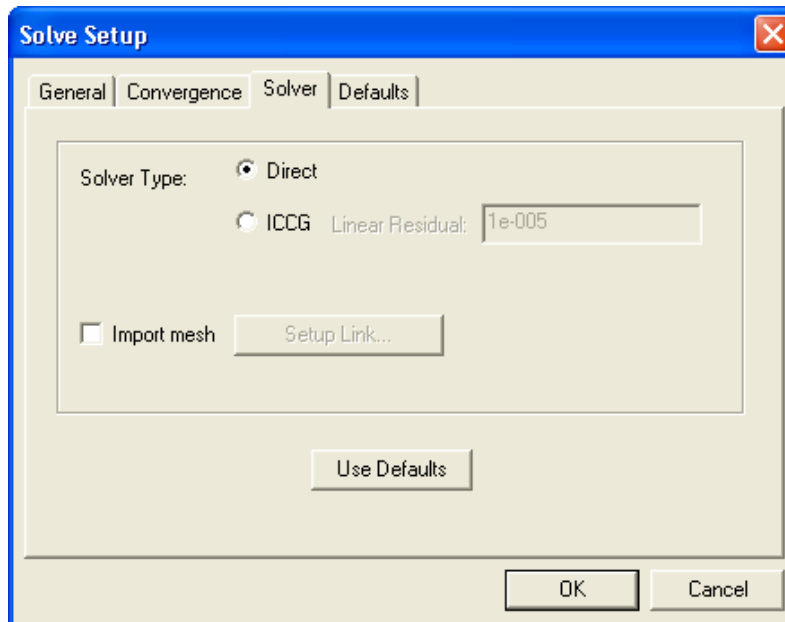
-  The second tab of the Solution Setup contains information about Convergence.



-  **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
-  **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
-  **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).
-  **Use Output Variable Convergence** is an option to include a defined output variable as an additional convergence criterion with a specified maximum percent change per pass (an output variable must be defined for this option to be available).

Solution Setup (Continued)

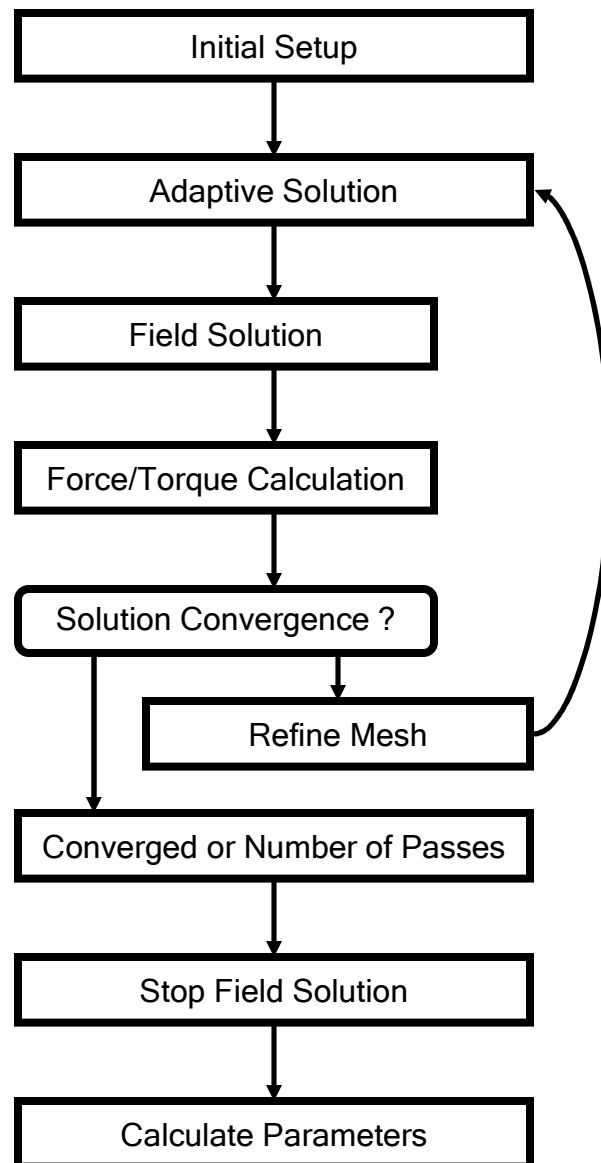
-  The third tab of the Solution Setup contains information about the Solver.



-  **Solver Type** has options for the **Direct** or **ICCG** solvers (Direct is the default).
-  **Import Mesh** allows the initial mesh to be imported from another solution - the linked solution must have the exact same geometry as the current simulation.
-  **Setup Link** must be defined when selecting **Import Mesh**.

Electrostatic Solution Process

- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



DC Conduction Analysis

-  DC Conduction Analysis is performed by choosing the Electrostatic solution type and selecting the DC conduction option.
-  Applications that use DC Conduction Analysis can be bus bars, power supplies, and many others.

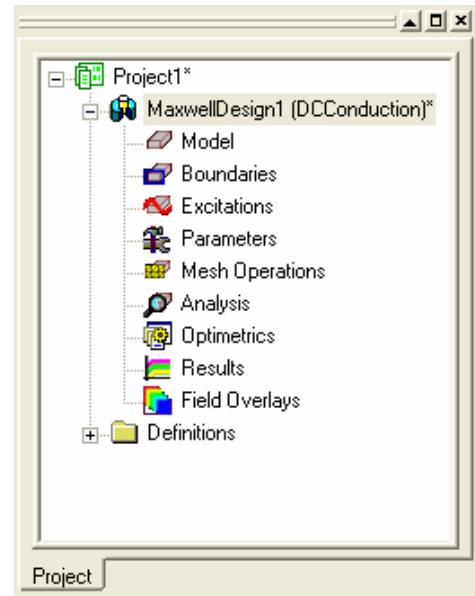
Overview

-  The DC Conduction solver computes static (DC) currents in conductors.
-  All objects are stationary.
-  The source of the static magnetic field can be:
 -  Voltages at different ends of solid conductors.
 -  Currents applied on surfaces of conductors.
-  The quantity solved is the electric scalar potential (ϕ).
-  Current density (J) and Electric Field (E) are automatically calculated from the electric scalar potential (ϕ).
-  Material conductivities can be anisotropic.
-  All fields outside of the conductors are not calculated and totally decoupled from the electric field distribution in the conductors - permittivity is irrelevant in this calculation.
-  There is non-zero Joule loss (ohmic power loss) in the conductors.
-  Can be coupled with an electrostatic simulation, where the electric potentials found in the conductors are used as a voltage boundary condition for the electric field solution in the insulators in an electrostatic simulation.

Setup

- ▲ The options in the project tree for a DC conduction simulation control all the simulation setup parameters.
 - ▲ Notice that right-clicking on any of the options will open a menu with important options for each step of the simulation setup.

- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in a DC Conduction simulation.
- ▲ There are no parameters for a DC conduction simulation.
- ▲ Mesh Operations are discussed in a separate section.
- ▲ Analysis defines the solution setup.
- ▲ Optimetrics defines any automatic variational analyses.
- ▲ Results and Field overlays are discussed in a separate section.



- ▲ These options are displayed in an order that can be followed in creating a new DC Conduction simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.
- ▲ However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.

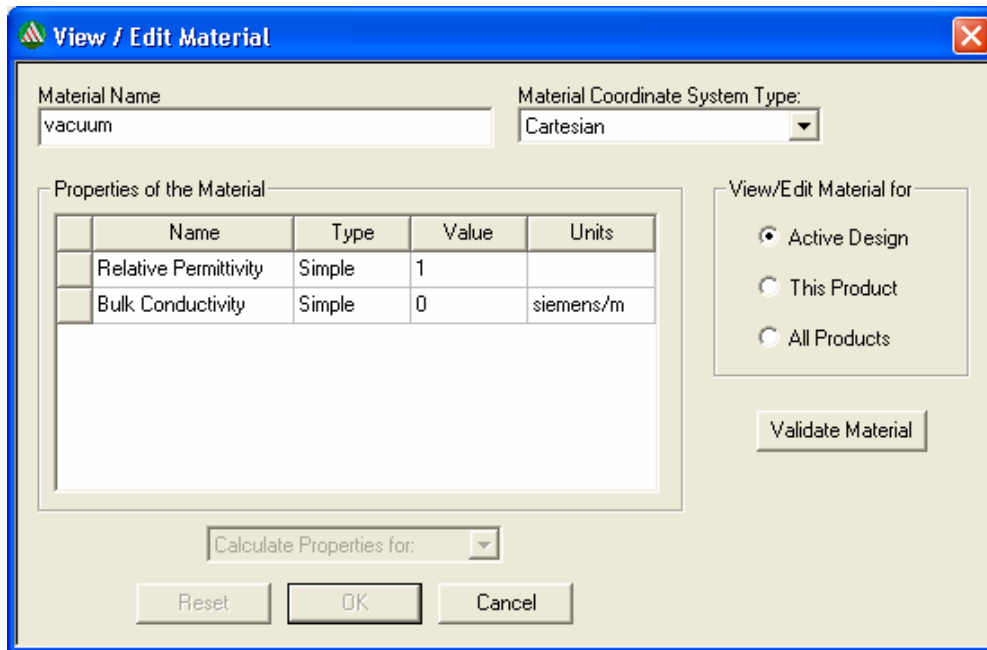
DC Conduction Material Definition

In a DC Conduction simulation, the following parameters may be defined for a material:

- Relative Permittivity** (can be Anisotropic or Simple)

 - Relative permittivity does not affect the DC conduction calculation, but will be important in insulators if this is coupled with an electrostatic simulation.
- Bulk Conductivity** (can be Anisotropic or Simple)

 - Bulk Conductivity defines whether an object is a conductor (treated as a perfect conductor in the Electrostatic solver) or an insulator. This separation is determined by the insulator/conductor material threshold setting.



The dialog box titled "View / Edit Material" contains the following fields and controls:

- Material Name:** A text field containing "vacuum".
- Material Coordinate System Type:** A dropdown menu set to "Cartesian".
- Properties of the Material:** A table with the following data:

	Name	Type	Value	Units
	Relative Permittivity	Simple	1	
	Bulk Conductivity	Simple	0	siemens/m
- View/Edit Material for:** A group box containing three radio buttons: "Active Design" (selected), "This Product", and "All Products".
- Validate Material:** A button located below the radio buttons.
- Calculate Properties for:** A dropdown menu at the bottom.
- Buttons:** "Reset", "OK", and "Cancel" at the bottom.

DC Conduction Boundary Conditions

-  **Default** - The default boundary conditions for the DC Conduction solver are:
 -  **Natural** boundaries on the interface between objects.
 - This means that the normal component of the current density at the boundary is continuous.
 -  **Neumann** boundaries on the outer boundaries.
 - This means that the E Field is tangential to the boundary and flux cannot cross it (current cannot leave conductors).
-  **Insulating** - This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate distinct conductors (defined on surfaces between the objects).
-  **Symmetry** - There are two Electric symmetries - even symmetry (flux tangential) and odd symmetry (flux normal). Even symmetry defines E to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Odd symmetry defines E to be normal to the boundary. Remember that geometric symmetry may not mean electric symmetry in all cases. 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. 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** - This boundary condition is also known as a matching boundary condition because it matches the electric field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement for matched geometries) and the faces need to be planar. It is required to define a U-V coordinate system along each face to properly map the matched boundary as desired. Master and Slave boundaries enable you to model only one period of a periodic structure, which will reduce the size of a design. Example applications are periodic rotational machines or infinite arrays.

DC Conduction Excitations

-  Typical sources for DC current flow problems are currents applied on surfaces of conductors and voltages (electric potential applied to surfaces of conductors). The direction of the applied current is either “in” or “out”, always normal to the respective surfaces. Multiple conduction paths are allowed. Each conduction path that has a current excitation must also have either a voltage excitation applied or a sink to ensure a unique solution.
-  **Voltage Excitations** - These are used in conjunction with the material conductivity to define the current through a solid conductor. Either multiple Voltage excitations can be used to define a voltage difference across two faces of a conductor (creating a current) or a Voltage can be defined along with a current excitation to define a voltage reference for the electric field solution.
-  **Current Excitations** - This excitation can be assigned on any conductor to define the total current (amp-turns) through the cross-sectional face of a conductor.
-  **Sink** - This excitation is used when only current excitations are defined in a conduction path and there is no voltage excitation. This excitation ensures that the total current flowing through the outside surface of a conduction path is exactly zero.

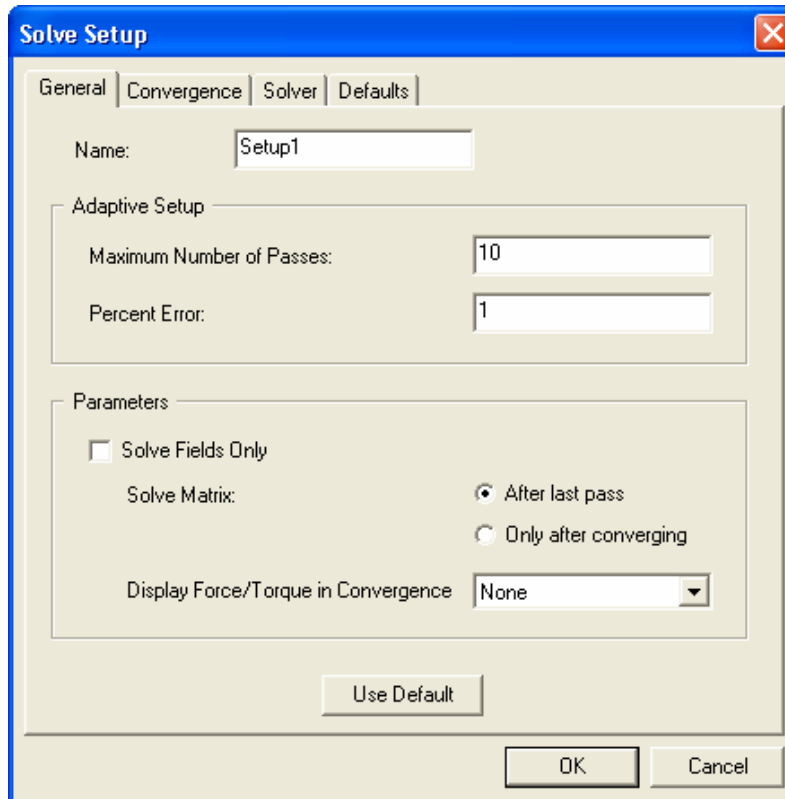
Mesh Operations

-  Mesh operations are described in detail in the Mesh operations section.
-  Remember that the Magnetostatic solver has an adaptive mesh solution, so excessive mesh operations are not usually required. It can often be worse to over-define the mesh than to not define mesh operations at all (it will take longer to solve, and it will be more difficult to adapt correctly).

Solution Setup

-  The solution setup defines the parameters used for solving the simulation.
-  Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.

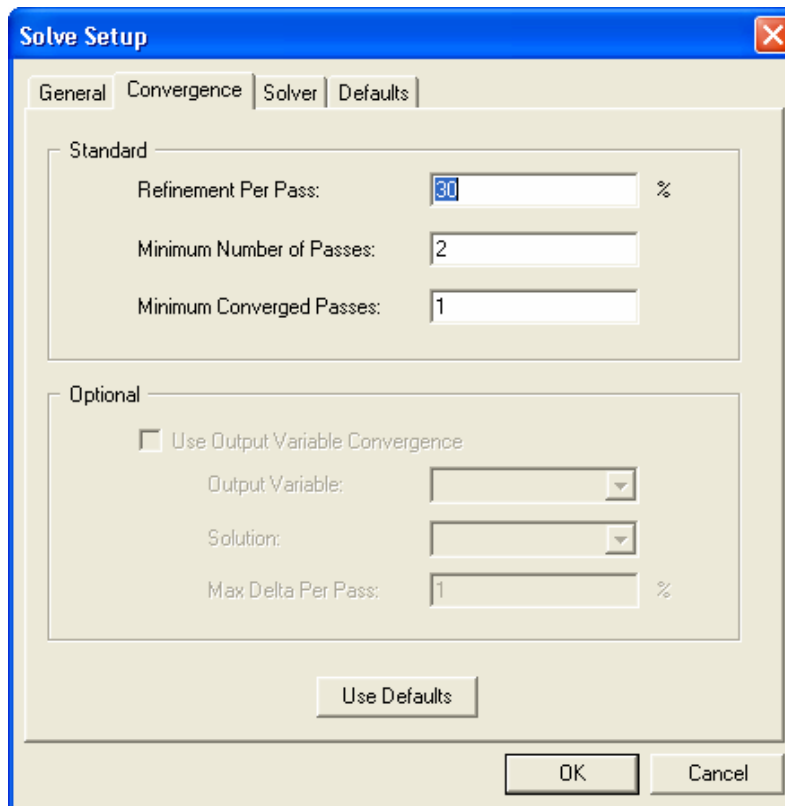
-  The following window appears with the General Setup parameters.



-  You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
-  **Maximum Number of Passes** defines a limit to the number of adaptively refined passes that the solver performs (the default value is 10).
-  **Percent Error** is the error goal for both the Error Energy and Delta Energy.
-  **Solve Fields Only** ignores any defined parameters if checked.
-  **Solve Matrix** has the options of calculating the matrix after the last solved pass or calculating the matrix only if the solution converges.
-  An option is included to display one Force or Torque parameter in the Convergence table.

Solution Setup (Continued)

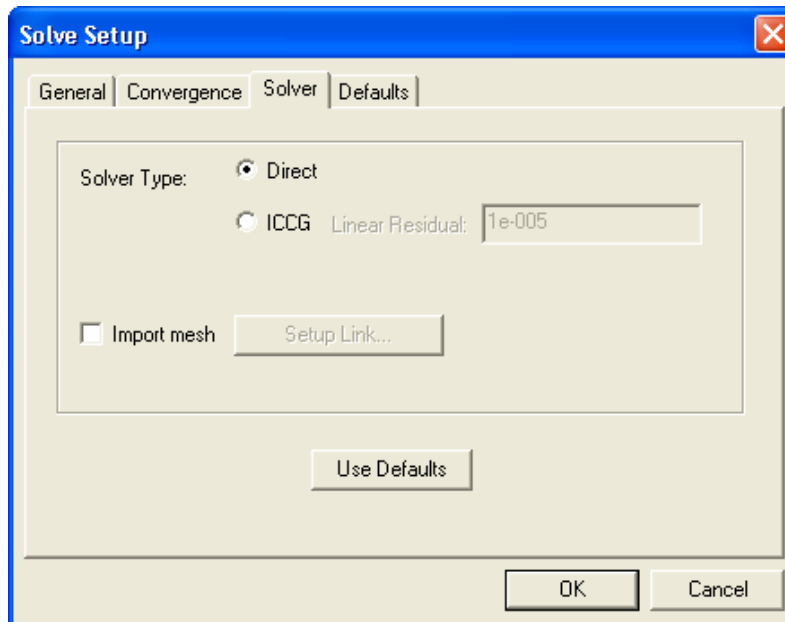
-  The second tab of the Solution Setup contains information about Convergence.



-  **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
-  **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
-  **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).
-  **Use Output Variable Convergence** is an option to include a defined output variable as an additional convergence criterion with a specified maximum percent change per pass (an output variable must be defined for this option to be available).

Solution Setup (Continued)

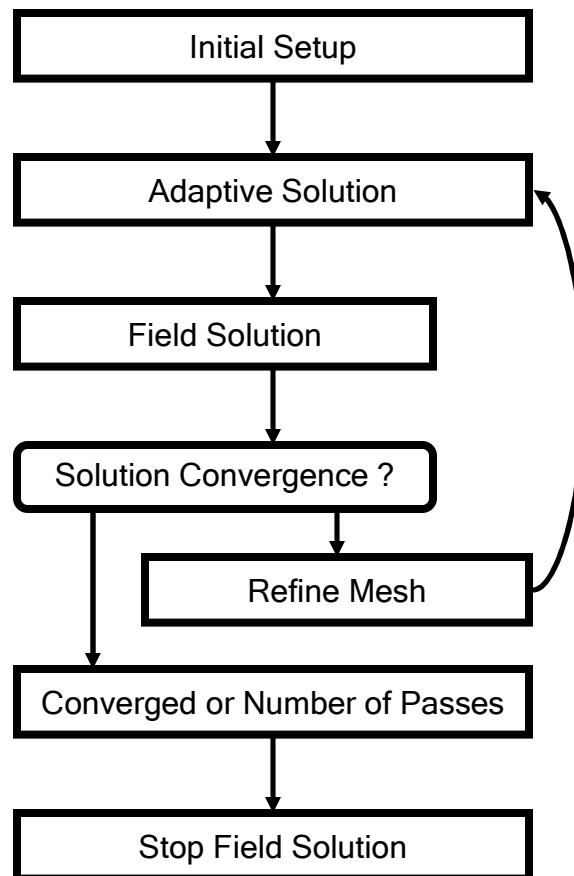
-  The third tab of the Solution Setup contains information about the Solver.



-  **Solver Type** has options for the **Direct** or **ICCG** solvers (Direct is the default).
-  **Import Mesh** allows the initial mesh to be imported from another solution - the linked solution must have the exact same geometry as the current simulation.
-  **Setup Link** must be defined when selecting **Import Mesh**.

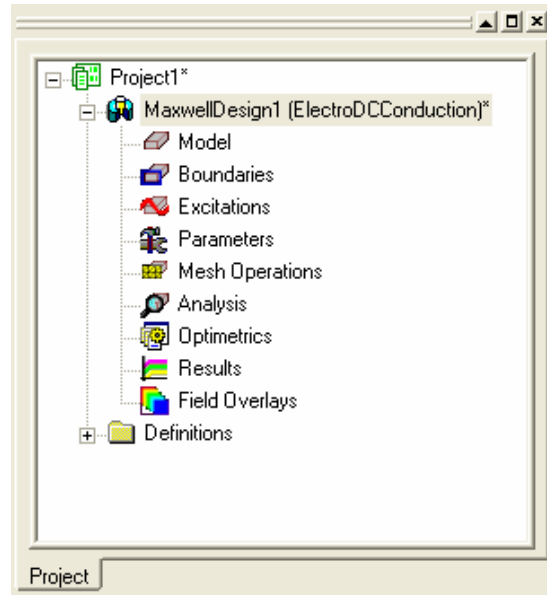
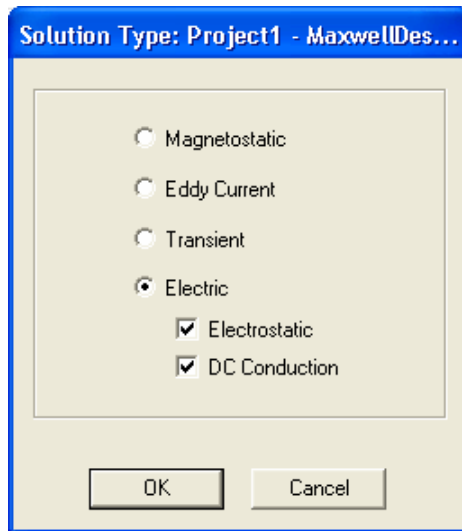
DC Conduction Solution Process

- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



Electrostatic and DC Conduction Combination

- The combination of the DC current flow and electrostatic solution is based on the division of the arrangement into conductors and insulators (determined by the object's conductivity and the insulator/conductor material threshold). The solution of such problems is performed in two steps: first the DC conduction problem in the conductors is computed, then the electrostatic solution is calculated using the electric scalar potential of the conductors as a voltage boundary condition.
- Select the Electrostatic and DC Conduction combination by choosing the Electrostatic solution type and selecting both the electrostatic and the DC conduction options.



- The setup for a ElectroDCConduction simulation is the same as if setting up a DC Conduction simulation in the conducting objects, and setting up an Electrostatic simulation in the insulators. All boundary conditions and excitations for both setups are available when both options are selected. It is up to the user to set up the simulation appropriately in each domain.

Mesh Operations

-  This chapter provides details on meshing in the Ansoft Maxwell v11 software. It discusses the default process of creating a mesh, meshing of curvature, user control of meshing, and intermediate methods of manual mesh refinement. The following topics are discussed:

-  Initial Mesh Process
-  Adaptive Mesh Process
-  Mesh Considerations and Impact on Solutions

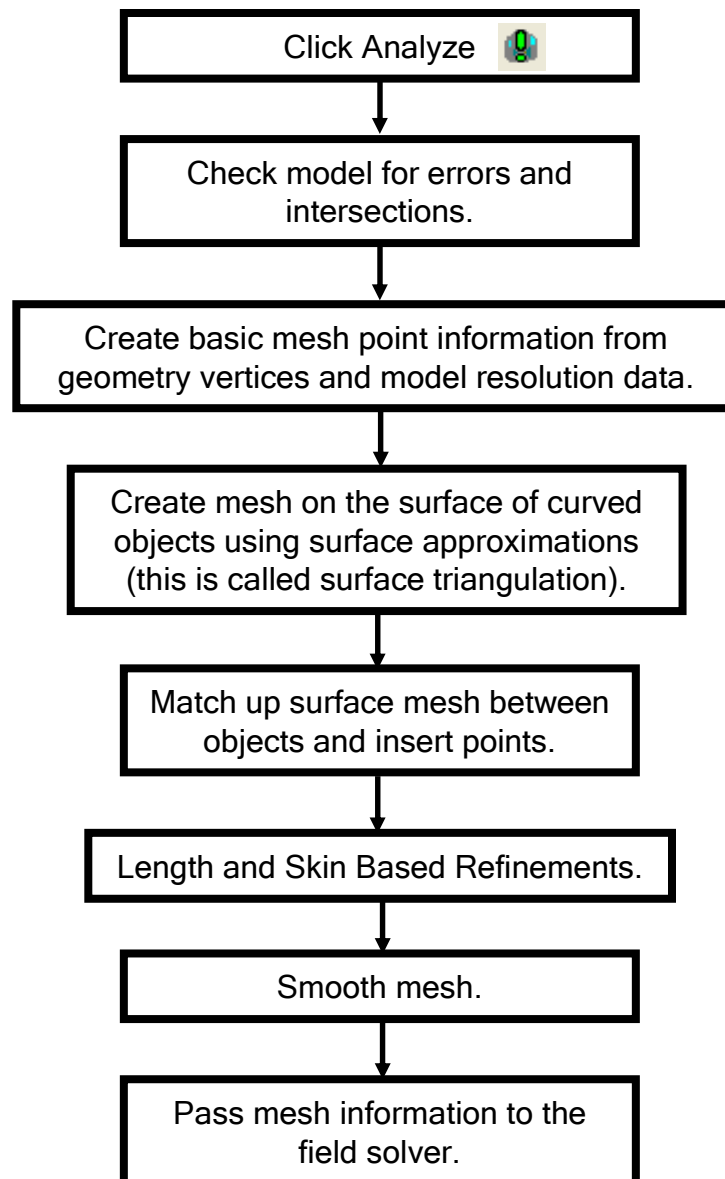
-  Applying Mesh Operations
 -  Surface Approximations
 -  Curved Geometry Mesh Adaptation
 -  Model Resolution
 -  Length Based Mesh Operations
 -  Skin Depth Based Mesh Operations

-  Mesh Reduction Techniques
-  Dummy Objects
-  Linking Mesh
 -  Transient
 -  Non-transient

-  Mesh failures and suggestions

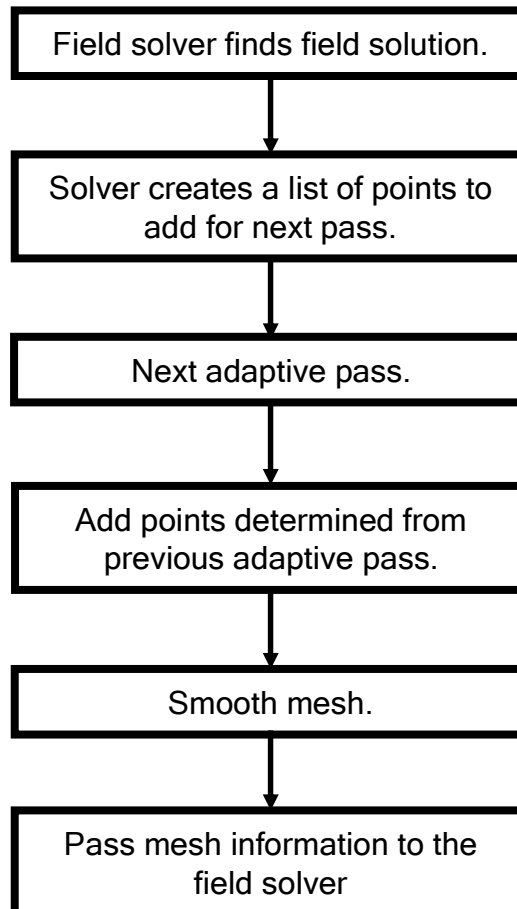
Initial Mesh Process

-  When you first analyze a problem, there must be an initial mesh in place to perform a field calculation. The initial mesh is automatically constructed without any instructions from the user (besides for the geometry definition) when the analysis is first performed. The initial mesh is only constructed if no current mesh exists - if a current mesh exists Maxwell will solve without creating a new initial mesh. The automatic mesh process goes something like the following:



Adaptive Mesh Process

-  The adaptive mesh process takes numerous factors into consideration when deciding where to refine the mesh at each pass. There are geometry factors, field solution factors, and everything is based on the percent refinement number defined in the analysis setup dialog.
-  If the percent refinement is set to 30%, and there are 1000 elements at pass 1, then 300 points will be added for pass 2. This increase may not be exactly 30% due to smoothing and other factors that will adjust the actual percent refinement.
-  The refined mesh points are calculated by the field solver and placed at points where there are strong fields, large errors, large field gradients, or areas that generally have a large impact on the field solution. This list of refined points is then passed on to the meshing procedure which places the points and creates an optimal refined mesh for the next adaptive pass.



Mesh Considerations and Impact on Solutions

-  The mesh is important for two separate, yet related reasons - first, the mesh is used to directly determine the numerical field solution, second, the mesh is used to produce all secondary results such as volume integrations or other field calculations.
-  There is a trade-off among the size of the mesh, the desired level of accuracy and the amount of available computing resources.
-  The accuracy of the solution depends on the size of each of the individual elements (tetrahedra). Generally speaking, solutions based on meshes using thousands of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each element must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.
-  However, generating a field solution involves inverting a matrix with approximately as many elements as there are tetrahedra nodes. For meshes with a large number of elements, such an inversion requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.
-  Generally, uniform mesh elements with equilateral triangular faces are best suited for second order interpolated field solutions. However, these triangles are not easy to produce with complex geometries. The important thing to remember is not to over-define the mesh in any region (this is especially true for non-transient simulations). It is often necessary to define the mesh on specific surfaces or volumes for further calculations, however there are efficient and inefficient methods to achieve good results.
-  Inefficient meshes pick up all the details of every curve and joint, even in unimportant areas for the field solution. There are methods that we will discuss that can decrease the mesh in areas of low importance and promote overall convergence of the field solution.

The Initial Mesh and Why Mesh Operations are Often Necessary

-  The initial mesh is created only by taking into account the design's geometry. For example, if only one box is included in the simulation, there will be 5 elements generated inside the box in the initial mesh. Only the corners of the geometry are used to create the initial mesh - this is constructed by an initial mesh-maker that only looks at the geometry and knows nothing about the field structure (there is no solution yet, so there can be no knowledge of the field solution).
-  It is often the case that the initial mesh is too coarse in the regions of interest to produce an efficient, accurate field solution (this is certainly true in transient simulations). Mesh operations are able to define a manual refinement to the initial mesh that can improve simulation time and provide enhanced solutions in some cases.
-  It is sometimes the case that the initial mesh is overly defined in some places, due to joints in the geometry or difficult to mesh areas. Mesh operations can better define the initial mesh in the entire solution region so that the mesh is most efficient and improvements are possible for both simulation time and solution accuracy. This can even allow the simulation of geometries that would be impossible without mesh operations.
-  Sometimes the initial mesh has difficulties that can be fixed by assigning mesh operations. This means that problems that are not solvable with the default initial mesh can sometimes be solved with the addition of a few mesh operations (or with mesh considerations in mind during geometry creation).

Applying Mesh Operations

-  If you want to refine the mesh on a face or volume you do not necessarily have to generate a solution. Do either of the following after defining mesh operations to apply mesh operations:
 -  Select **Maxwell > Analysis Setup > Apply Mesh Operations**, or right-click on the setup name in the project tree and choose **Apply Mesh Operations**.
 -  Analyze the design - the mesh operations will take effect when creating the mesh for the current adaptive pass.
-  For non-transient simulations, the following behaviors can be expected when applying mesh operations using either of the above methods:
 -  If a current mesh has been generated, Maxwell will refine it using the defined mesh operations.
 -  If a current mesh has not been generated, Maxwell will apply the mesh operations to the initial mesh.
 -  If an initial mesh has not been generated, Maxwell will generate it and apply the mesh operations to the initial mesh.
-  For transient simulations, the mesh must be the same from one time step to the next, therefore, adjusting mesh operations of a transient simulation will force the simulation to start from time zero.
-  Select **Maxwell > Analysis Setup > Revert to Initial Mesh** to clear the mesh information (as well as any solution information) and revert to the initial mesh. This can also be accessed by right-clicking on the setup name and choosing **Revert to Initial Mesh**.

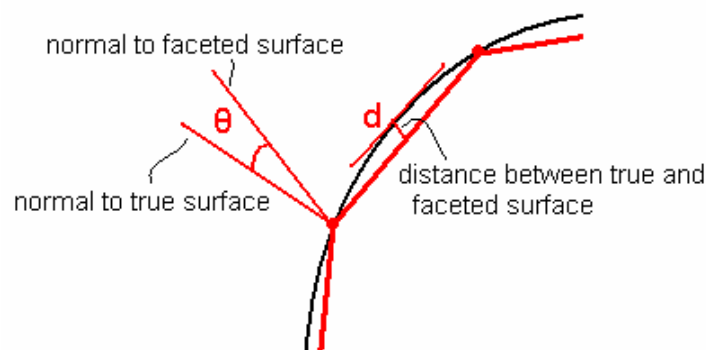
Applying Mesh Operations (Continued)

Note the following:

-  If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.
 -  Define a new mesh operation rather than modify an existing mesh operation. Maxwell will not re-apply a modified mesh operation.
 -  Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them.
 -  Model Resolutions adjust the effective geometry used for the initial mesh (and the mesh maker used in solving the problem), so, applying a model resolution will invalidate any solutions and revert to the initial mesh.
 -  Surface approximations only work on the initial mesh. If a surface approximation is applied with an existing current mesh, it will not take effect until the mesh has been cleared.
-
-  You can look at the mesh by selecting **Maxwell > Fields > Plot Mesh...** or by right-clicking on **Field Overlays** in the project tree and choosing **Plot Mesh**. A setup must be created to define the plot. An object or surface must be selected to create the mesh plot. Read the section on Data Reporting to find out more about field plotting on 3D geometries.
 -  You can view mesh statistics by right-clicking on the setup name and choosing **Mesh Statistics** (except for Transient simulations), or by going to the Solution Data or Analysis Profile and choosing the Mesh Statistics tab. This mesh statistic information represents information about element length (the length of the sides of each element), the tet volume (the volume of the tetrahedral mesh elements), and the total number of elements in each object.

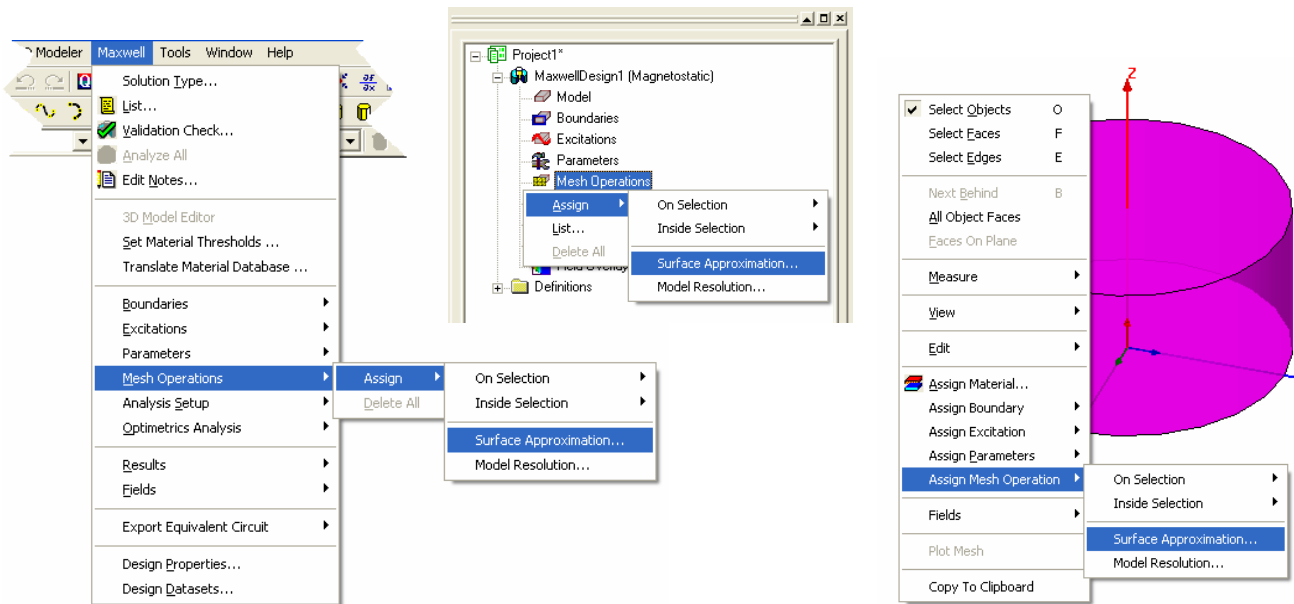
Surface Approximations

- Object surfaces in Maxwell may be planar, cylindrical or conical, toroidal, spherical or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, Maxwell first divides all true surfaces into triangles. These triangulated surfaces are called *faceted surfaces* because a series of straight line segments represents each curved or planar surface.
- For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.
- The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees (15deg is the default).
- The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner (see the aspect ratio comments after the usage suggestions for a detailed explanation).
- You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click **Maxwell > Mesh Operations > Assign > Surface Approximation**.)
- The surface approximation settings are applied to the initial mesh.
- Below is a diagram that illustrates the surface deviation, "d", and the normal deviation, " θ ".



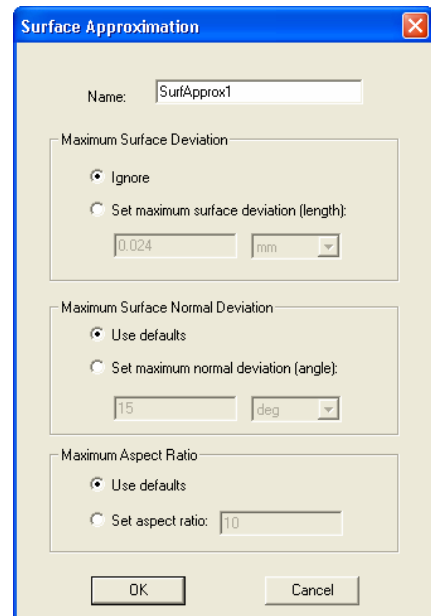
Surface Approximations (Continued)

- Since refining curved surfaces is not always enough to produce an efficient solution, users can control the fidelity to which the initial mesh faceting conforms to geometric curvature by assigning Surface Approximations to appropriate objects and/or object faces
- Mesh Operations can be assigned from the **Maxwell** menu, from the **Project Tree**, or from the geometry interface's context-sensitive menu.



- The Surface Approximation options are shown below right. Definitions follow:

- Surface Deviation** is the maximum spacing, in drawing units, that the tetrahedral surfaces may be from the true-curved geometry's surface.
- Normal Deviation** is the maximum angular difference, in degrees, that a tetrahedral face's normal can have from the surface normal for the true geometry which it is meant to represent.
- Aspect Ratio** refers to the maximum allowed aspect ratio of all faces of all tetrahedra of the selected object or face. This setting influences mesh quality rather than actual meshed volume or surface locations.



Surface Approximations - Usage Suggestions

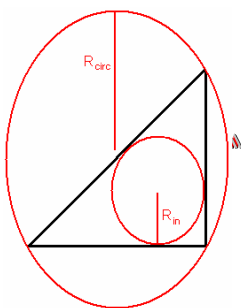
Do not overspecify.

- It is always easier to 'add' than subtract mesh, by running more adaptive passes or by adding supplemental mesh instructions
- Too stringent a setting (e.g. Normal Deviation of 1 degree) can result in poor mesh qualities due to aspect ratios, poor mesh gradients to surrounding objects, etc.
- Use Aspect Ratio settings along with Normal or Surface Deviation settings
 - For cylindrical type objects where curved and planar faces meet, the normal and surface deviation settings apply to the curved faces only. Setting an aspect ratio limit as well (e.g. 4:1) will force a few additional triangles on the planar faces and help preserve a cleaner overall mesh
- Consider using Polyhedrons or Polygons instead if using to 'reduce' mesh
 - If your design has many curved objects which you want only very coarsely meshed (e.g. individual current-carrying wires with no eddy or proximity effects, for which 15 degree default normal deviation is unnecessary), and the geometry is not imported, consider drawing the objects as hexagonal or even square solids instead.
- Surface approximations can be applied to either entire objects or faces (in groups or individually) or sheets. The surface approximations take effect during the surface meshing, so it does not matter if an object is selected whole, or if all faces of an object are selected to apply these operations - either way, the volume of the object will be meshed after the surface approximation is considered.

Surface Approximations - Aspect ratio comments

- There is a different definition of aspect ratio on curved and planar surfaces, but in either case, the aspect ratio is defined by the triangles on the selected surfaces.

- On curved surfaces, the aspect ratio is essentially defined by the height to width ratio of the triangles. The default minimum aspect ratio is around 1.2 and will tend to form nearly isosceles right triangles on the faceted faces. The default aspect ratio on curved surfaces is 10.

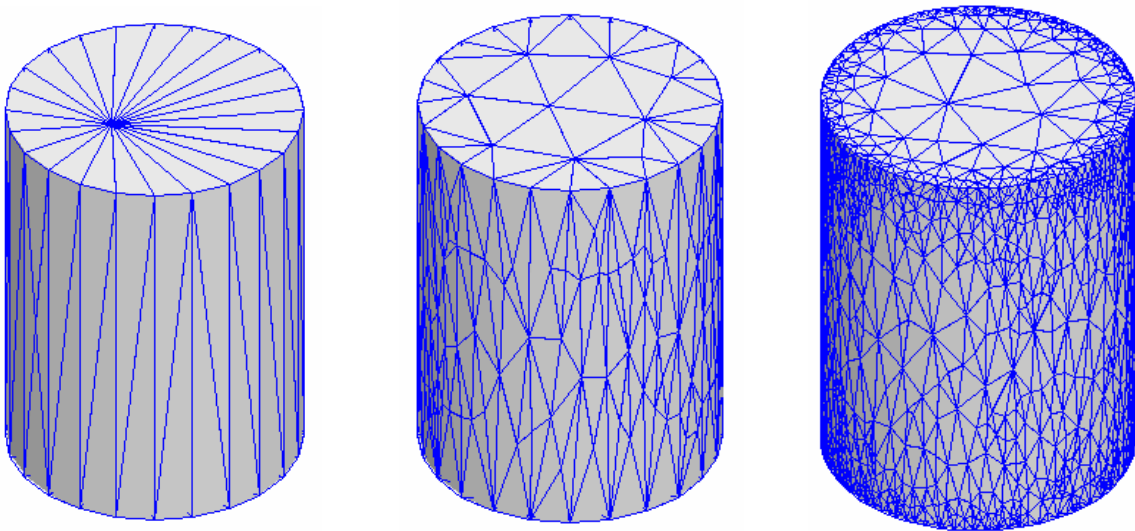


- On planar surfaces, the aspect ratio is equal to the circumscribed radius over twice the inner radius of each triangle - this produces 1 for an equilateral triangle. (See the diagram at left to see what is meant by circumscribed circle and inner circle of a triangle.) The default minimum aspect ratio is around 2. The default aspect ratio on planar surfaces is 200.

- Specifying an aspect ratio of 1 will not produce perfectly equilateral triangles.

Curved Geometry Mesh Adaptation

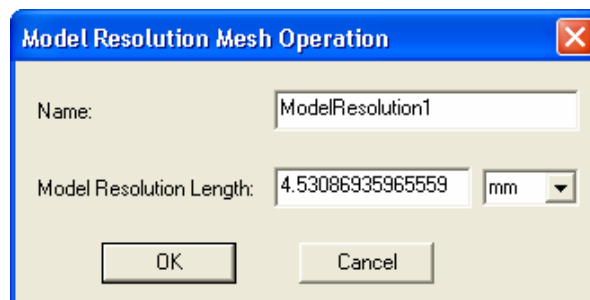
- ▲ The new graphical drawing interface encourages the use of true-curved drawing by removing the option to assign a facet count to the construction of primitives such as circles, cylinders, spheres, and ellipses.
 - ▲ Faceted primitives are available however as polyhedrons and polyhedral solids, if desired.
- ▲ Initial meshing is constrained by faceting decisions made by the first pass of the meshing algorithms. However, adaptive mesh points can be placed 'anywhere' on the true-surface of the curved object(s), as shown in the before and after images below. (Initial mesh left, partly adapted in middle, fully adapted at right.) Note that regular faceting is not maintained after adaptive mesh alteration.



- ▲ This example documents a cylindrical, linear magnet oriented in the Z-direction, with a 200% padded region. The pictures correspond to the initial pass, the 10th pass, and the 18th pass, with total mesh sizes of 401, 4969, and 14135 mesh elements for each respective pass.
- ▲ The increase in mesh size due to the refinement of the cylinder may increase model fidelity, but it may also increase the solution time.
- ▲ Note that if you wish to maintain the initial faceting, you can use faceted primitives during geometry creation.

Model Resolution

- ▲ In many models (especially imported models), there are small geometric details that are present, yet do not affect the field solution to a large extent.
- ▲ Small features in the geometry can lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower.
- ▲ For these reasons and more, Model resolution allows the initial mesh to generate without including all the little details that can slow down simulations and do not add to the quality of the solution.
- ▲ Model resolution improves model convergence and reduces solve times. If in previous versions the tetrahedron count got too large, the solver could run out of memory. It would be trying to add meshing elements to areas of the model that are not important. Now with model resolution, we are able to crank down the mesh size in order to actually get convergence before memory becomes an issue.
- ▲ Model resolution is a length based value that modifies the initial mesh. This meshing operation allows the user to specify a minimum edge length of any tetrahedron used for the mesh. By specifying a minimum edge length for a tetrahedron, the mesher will have to coarsely mesh geometric detail that may not be important for the field solution. This saves the solver a tremendous amount of solution time since the initial mesh is smaller, and the mesher does not have to add mesh elements to areas that are not important for the field solution.
- ▲ Model resolution can be assigned from the **Maxwell** menu, from the **Project Tree**, or from the geometry interface's context-sensitive menu, and is found immediately below Surface Approximation in the Mesh Operations list.
- ▲ The Model Resolution dialog is presented below. The suggested value for the resolution length is determined by the smallest edge length on the selected objects. This may not be the best value to use, but it might be a good place to start or, at least, a good reference.



Model Resolution (Continued)

-  Model resolution capabilities are an expanded capability of the ACIS engine and are incorporated into the new fault tolerant meshing features. By default, ACIS uses 1.0e-6 model units as the smallest possibly resolved length. Any two geometry points that are closer than this absolute resolution length (ResAbs) will not be resolved by ACIS (so choose the model units appropriately).
-  By default, if there are difficulties creating an initial mesh, a second initial mesh will be attempted with a model resolution automatically applied to all model objects with 100 times the absolute resolution length (ResAbs). If continued difficulties are experienced, a third initial mesh will be attempted with an automatic model resolution of 1000 times ResAbs on all model objects. After the third attempt a mesh failure is reported (you will be able to see three distinct “mesh_init” lines in the analysis profile).
-  If the user specifies the model resolution length so that the mesh grossly misinterprets the model or changes the contacts between the objects, Maxwell will detect this and report this as an error (*“MRL too large or object_name lost all surface triangles”*).

Model Resolution (Continued)

Practical Example

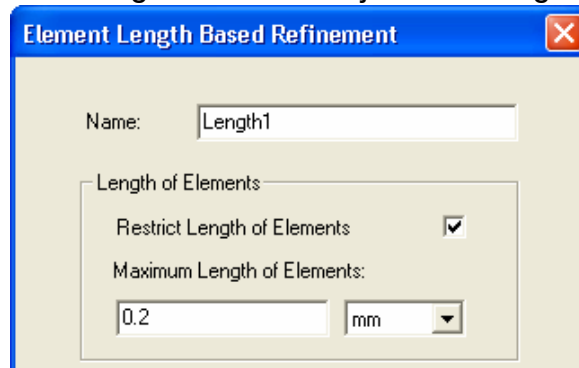
-  This is part of a complex design that benefited from model resolution. The grey outline is the reduced model. The pink lines were the original model details that were removed due to the mesh operation. The original model had an initial mesh of 184,675 tets. After model resolution, the initial mesh was 24,691 tets.

Model Resolution - Usage Suggestions

-  Apply model resolution to one specific object or a group of objects. Be aware of what the size of important details are when assigning a new model resolution to an object or group of objects. Measure various lengths (*3D Modeler > Measure*) to survey some nominal lengths for the specific objects. Then, when applying the model resolution, look at the number suggested in the model resolution assignment dialog - this number is the smallest edge found on the selected objects. If this edge is very much smaller than some of the measured and expected lengths, apply a model resolution that is larger than the suggested value, yet significantly smaller than the expected length (for example, if you have an object that is cube-like with 10in sides, yet the smallest length is returned as 1e-6in, you should apply a model resolution of about 0.1in to the object).
-  If the mesh fails with a vague error, occasionally, applying a model resolution to a complicated object will assist the initial mesh generation.
-  Model resolution is very useful when dealing with imported geometries.
-  When dealing with models that have very high aspect ratios due to small geometric detail, use a model resolution of 1/10 to 1/20 of the thinnest conductor to start with. Then adjust the value accordingly.
-  It is often good to be conservative with model resolutions as a large model resolution can cause mesh errors.
-  Model resolution is often most useful when fillets, chamfers, and other small features are included at joints and corners - these can often be ignored for electromagnetic simulations and are often small enough to use an appropriate model resolution.
-  Always create the initial mesh and look at the surface mesh on objects before solving the problem. If too aggressive of a model resolution is used, then important features can be skipped with a model resolution - it is important to know that the mesh is appropriate for the simulation.

Length Based Mesh Operations

- ▲ Length based mesh operations are perhaps the most basic conceptual mesh refinement option. The assignment of a length based mesh operation limits the edge length of all tetrahedral elements inside an object or on an object's surface.
- ▲ There are two types of length based mesh operations:
 - ▲ On Selection
 - ▲ Inside Selection
- ▲ The Length-based On-selection refinement will limit the edge length of all triangles formed on the surface of a selected object (or group of objects), or any selected faces.
- ▲ The Length-based Inside-selection refinement will limit the edge length of all tetrahedral elements inside the selected object (or group of objects).
- ▲ The assignment of these two types are similar - simply choose either:
Maxwell > Mesh Operations > Assign > On Selection > Length Based ... or
Maxwell > Mesh Operations > Assign > Inside Selection > Length Based ...
- ▲ The same dialog appears for both types of assignment - the only difference is whether it affects all edges within an object or all edges on a face (or surface).



- ▲ The default value for **Maximum Length of Elements** is 20% of the largest edge-length of the bounding boxes of each selected face.
 - ▲ For a cube, the default length-based on-selection refinement will produce about 7 triangles along an edge for a total of about 100 triangles per face.

Length Based Mesh Operations (Continued)

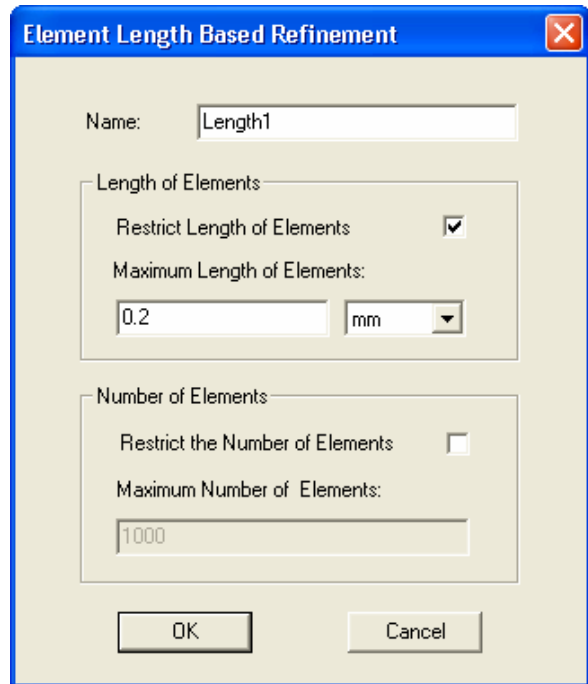
- There is one other option that is provided when assigning a length based mesh operation that will limit the number of elements that are added into the simulation with that particular mesh refinement.

- Restrict the Number of Elements** is a hard limit and no more than the specified number of elements will be added to the simulation for the particular mesh operation than are specified here (unless the box is un-checked).

- For example: if the restriction to the length of elements requires 3000 additional elements, yet the number of elements is restricted to 1000, then only 1000 elements will be added, and the elements will not meet the Length of Elements requirement (they will be larger than specified).

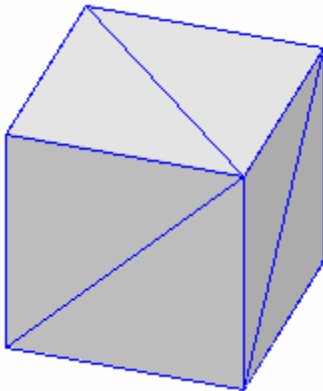
- This second restriction brings another possible method of adding elements to objects without regard to the dimension of the object - you may uncheck the Length of Elements restriction and enter a value for the Number of Elements restriction, thereby adding a defined number of elements to the simulation.

- This second method of adding elements is good and bad, because it allows control over the number of elements that are inserted into the defined objects for the particular mesh operation, however, it does not necessarily need to meet any length requirements (so the elements could still be too large).

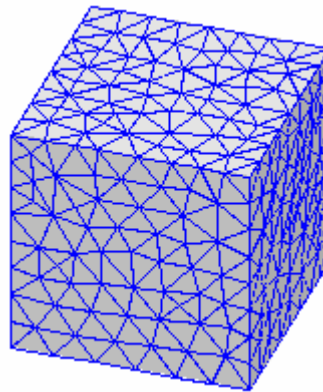


▀ Length Based Mesh Operations - Example

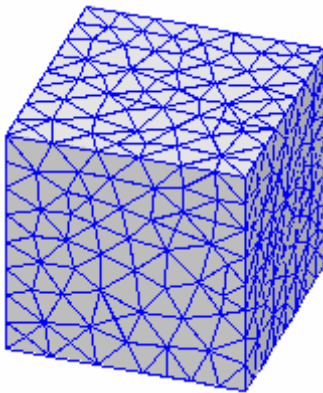
- ▀ This example uses a cube inside a Region with 100% padding. An initial mesh was created for four different cases of applying length-based mesh operations.
- ▀ First, a default initial mesh was created without defining any mesh operations.
- ▀ Second, an initial mesh was created by defining a length-based mesh operation on the surface of the box at the default maximum length setting (20% edge length).
- ▀ Third, an initial mesh was created by defining a length-based mesh operation inside the box at the default maximum length setting (20% edge length).
- ▀ Fourth, an initial mesh was created with a length-based mesh operation defined both on and inside the box at the default maximum length settings - 2 mesh operations were used.



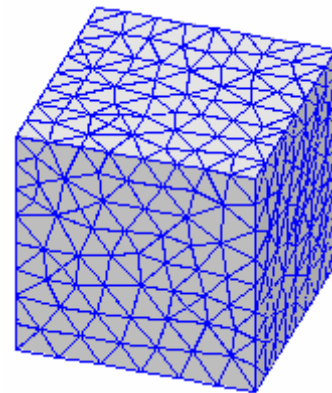
Default Initial mesh
42 elements



Initial mesh with Length-Based
On-Selection set to 20%
6467 elements



Initial mesh with Length-Based
Inside-Selection set to 20%
6698 elements



Initial mesh with Length-Based Inside
& On Selection set to 20%
9042 elements

Skin Depth Based Mesh Operations

-  Skin depth based mesh operations are similar to On-selection length-based mesh operations. These operations are defined by both the triangles on the surface and a seeding inside the selected faces.

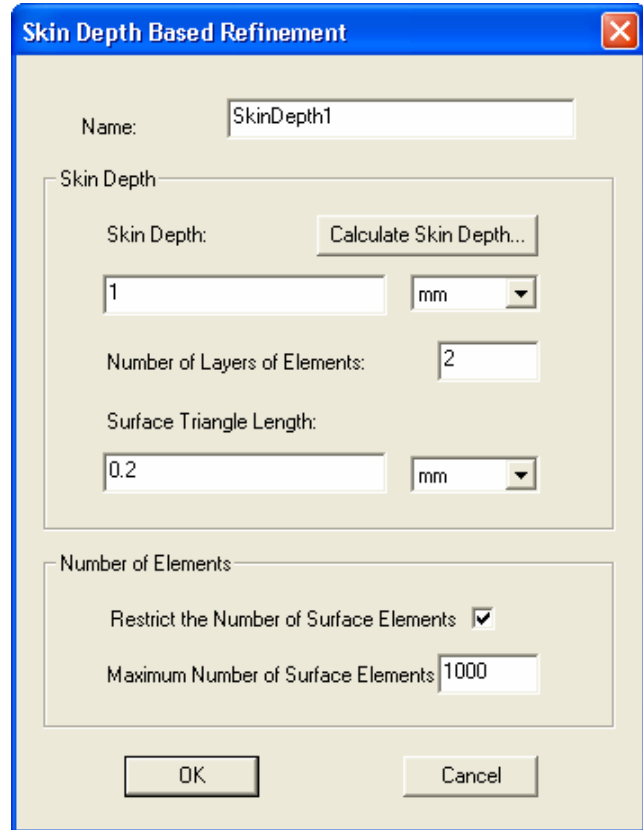
-  This refinement method creates layers of mesh within the selected surfaces of objects - this is useful for modeling induced currents near the surface of a conducting object.

-  **Skin Depth** defines the classical skin depth value (discussed on the next page).

-  **Number of Layers** defines the number of layers of points that are placed at distances according to the skin depth value.

-  The **Surface Triangle Length** is equivalent to the surface length restriction for length-based refinement.

-  The **Number of Elements** is equivalent to the surface element restriction for length-based refinement. The total number of elements added may exceed this number by a factor of the Number of Layers.

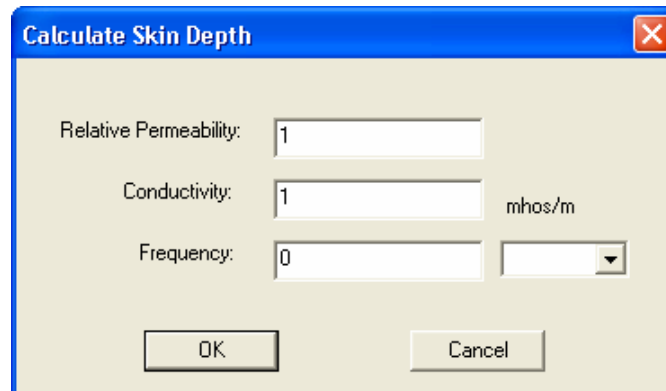


The dialog box titled "Skin Depth Based Refinement" contains the following fields and controls:

- Name:** A text field containing "SkinDepth1".
- Skin Depth:** A section containing:
 - Skin Depth:** A text field with "1" and a unit dropdown menu set to "mm". A "Calculate Skin Depth..." button is to the right.
 - Number of Layers of Elements:** A text field with "2".
 - Surface Triangle Length:** A text field with "0.2" and a unit dropdown menu set to "mm".
- Number of Elements:** A section containing:
 - Restrict the Number of Surface Elements:** A checked checkbox.
 - Maximum Number of Surface Elements:** A text field with "1000".
- Buttons:** "OK" and "Cancel" buttons at the bottom.

Skin Depth Based Mesh Operations (Continued)

-  Instead of inserting a value straight into the skin depth box you can use the automatic skin depth calculator. Simply fill in the relative permeability and conductivity of the material, and the measurement frequency to calculate and insert the skin depth in the skin depth box.



The dialog box titled "Calculate Skin Depth" contains three input fields: "Relative Permeability" with a value of 1, "Conductivity" with a value of 1 and units of mhos/m, and "Frequency" with a value of 0 and a dropdown menu. At the bottom are "OK" and "Cancel" buttons.

-  The skin depth of an object is calculated as:

$$\delta = \sqrt{\frac{2}{\omega \sigma \mu_0 \mu_r}} = \frac{1}{\sqrt{\pi f \sigma \mu_0 \mu_r}}$$

-  Currents are concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.
-  The skin depth based mesh operation is most important in eddy current and transient simulations where eddy currents and proximity effects are important to the solution of the simulation.

Mesh Reduction Techniques

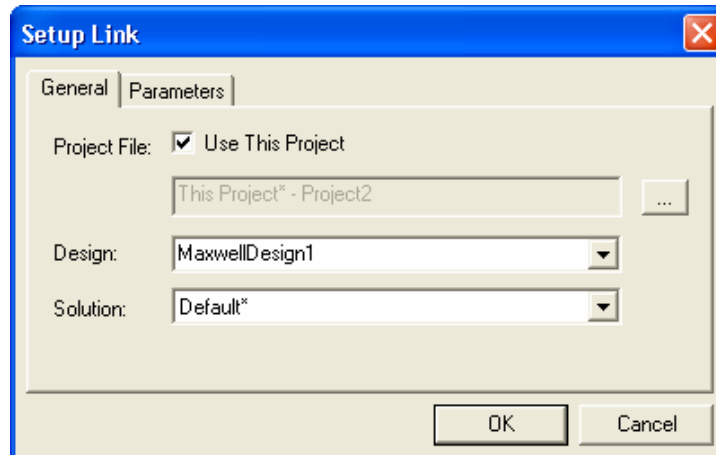
-  Because the mesh size affects the simulation time, it is often desired to decrease the total number of elements in a simulation to also decrease total simulation time. This can be done either at the expense of accuracy or not depending on the efficiency of the mesh construction. There are several ways to reduce the mesh size - here are a few examples.
- 1. Mitigate large aspect ratios with intelligent geometry construction.
 -  If you have a thin conductor that is far from other objects, this conductor should not be drawn with curved surfaces. Draw a square or regular polygon and sweep the square along the path of the conductor. This will limit the number of triangles necessary in a large aspect ratio object and will decrease the total number of mesh elements in the simulation.
- 2. Add a vacuum container object (dummy object) around all objects in simulation.
 -  This can be a good method to limit the aspect ratios of triangles in the region. The region needs to be defined a certain distance away from the components (due to fringing fields) - so this vacuum container can assist the mesh construction and reduce aspect ratios in the region near to the structure.
- 3. Use a model resolution when it will reduce the size of the mesh but not affect the solution of the simulation.
 -  Look at the Model Resolution usage suggestions for more information.
- 4. Use faceted objects instead of curved surfaces when the surface is not in an area of high fields.
 -  Faceted objects are easier to mesh and do not add more surface elements in the same manner as curved objects. This can reduce the mesh significantly in many cases. Curved surfaces may provide a more true representation of objects, especially important in areas with large fields. However, using faceted surfaces even with large fields may be appropriate and may reduce the mesh in many cases.
- 5. Add small mesh operations on conductors and inside magnetic materials.
 -  These will intelligently assist the initial mesh formation, so that less adaptive passes are necessary.

Dummy Objects

-  Placing an object of the same material inside another object does not affect the field solution for the field solver. However, this additional object defines another set of surfaces on which the initial mesh will form. This object can also have additional mesh operations defined on it. For these reasons and more, adding additional objects may assist the mesh formation process, and, moreover, increase the amount of control with which you can define the initial mesh. These objects are called dummy or virtual objects.
-  Some example places to use dummy objects are the following:
 -  In the gap of a magnetic core.
 -  As a container object for subgroups of objects (such as the inner rotor of an rotational machine).
 -  As a container object for all objects.
 -  Around a sensor.
 -  Around a surface that will be integrated on.
 -  Around a line that will be used for plotting or calculations.
-  These dummy object examples serve different purposes - the first three examples in the above list are intended to improve the general convergence of the solution, while the last three examples are intended to improve the post-processing results due to the local mesh in the vicinity of the measurement.
-  When placing dummy objects, use simple objects (such as rectangles, or regular polyhedrons) when possible. The size of the dummy objects should be comparable to the objects around it. The placement is sometimes best to be placed slightly away from model objects (such as a container object for a rotational machine - which should have its radius in the middle of the gap, or equally spaced in the gap with the band object) where the placement creates an extra layer of mesh elements that allow for better calculation of changing fields. Other times the placement is best when the dummy object touches the model objects (such as in the gap of a magnetic core) where the dummy object helps to both reduce the aspect ratio in the surroundings and to provide an object to apply mesh operations if desired.

Linking Mesh - Transient

-  Every solution type has a mesh linking capability. It is sometimes necessary to have a simulation linked (when using demagnetization options, or otherwise). However, it can be beneficial for mesh purposes to link a mesh from another simulation. This is nowhere as powerful as in a transient simulation.
-  Because a transient simulation does not adaptively refine its mesh, the mesh that it uses is constrained the initial mesh. Mesh operations can assist in manually refining the mesh, but this can be insufficient sometimes. What is provided with mesh linking, is the power of an adaptively refined mesh used in a transient simulation.
-  A transient simulation can be linked to any other simulation with the exact same geometry. This means that a magnetostatic or eddy current simulation can be adaptively solved to any accuracy, then the design can be copied and converted to a transient simulation that uses the same adaptively refined mesh.
-  In the transient simulation Analysis Setup, simply check **Import Mesh**. When this box is initially checked on, a dialog appears that sets up the mesh link.



-  Either select **This Project** or browse to the desired project file. Then choose the design name within the selected project and the desired setup name within the design. Adjust any parameters if necessary and then select **OK**. The mesh in the transient simulation will now be identical to the mesh in the selected project/design/setup.

Linking Mesh - Non-Transient

-  There are several important reasons why mesh linking capabilities are important in non-transient simulations. One example is if DC power flow calculations are performed, both an electrostatic and a magnetostatic solution are required, and the mesh must be identical to perform the necessary calculations (see the Field Calculator topic for an example). Another example is if the exact same mesh is required in a parametric sweep (especially important with distributed analysis).
-  Note that if mesh operations are included in a linking simulation, the model resolution and surface approximations are ignored - however, the length-based and skin-depth based resolutions are performed. If identical meshes are desired, delete mesh operations from the linking simulation.
-  You cannot link two analysis setups within the same design.
-  Any simulation can be linked to any other simulation with the exact same geometry (regardless of solution type).
-  In the Analysis Setup, simply check **Import Mesh**. When this box is initially checked on, a dialog appears that sets up the mesh link. In the dialog, either select **This Project** or browse to the desired project file. Then choose the design name within the selected project and the desired setup name within the design. Adjust any parameters if necessary and then select **OK**. The mesh in the linking simulation will now be identical to the mesh in the selected project/design/setup.
-  To guarantee identical meshes in a parametric sweep (with no geometry variations) do the following:
 1. Create a nominal design and solve it.
 2. Copy the design and delete mesh operations.
 3. Set up the mesh link in the copied design.
 4. Create a parametric sweep in the linking design.
-  This will guarantee that every row of the parametric sweep (assuming no geometry variations) will use an identical mesh.
-  To re-initialize the linked data, select **Maxwell > Analysis Setup > Clear Linked Data** or right-click on **Analysis** and choose **Clear Linked Data**.

Mesh Failures and Suggestions

-  There are several different points at which the mesh can fail even if the design passes all model verifications (no ACIS errors, no non-manifold objects, no partial intersections).
-  If an ACIS error or other geometry error occurs, please see the section on geometry import/healing.
-  Information about a mesh failure will appear in **3D Modeler > Model Analysis > Show Analysis Dialog > Last Simulation Mesh...**
-  Generally, the mesh will fail at the point where it tries to match the surface mesh between objects. The error for this failure will state *"Volume Meshing Failed - Stitching flag <#> failure mode <#>"*. This is often the case if surfaces are not perfectly coincident - if there is a small gap between two objects, the mesh will have difficulties filling the gap yet retaining a reasonable aspect ratio and form.
-  Another semi-frequent mesh error is *"Incompatible Faceting"*. This error is almost exclusively related to coincident true surfaces. There is some information on coincident true surfaces in the geometry import/healing section.
-  You can determine which part of the model is causing problems for the mesh by making an object non-model (double-click on the object to bring up the Properties dialog - then deselect the **Model** check-box). If it meshes, then that object is the problem.
-  Can you move the faces of the object or move the object so that it is not coincident (or is fully coincident)? Try to change the geometry a little bit without changing the solution.

Mesh Failures and Suggestions (Continued)

Look in **3D Modeler / Model Analysis > Show Analysis Dialog > Last Simulation Mesh.**

Follow up on what the dialog says

Consider eliminating tiny details through healing

Consider aligning faces through healing

Determine which object or part of object is the trouble maker

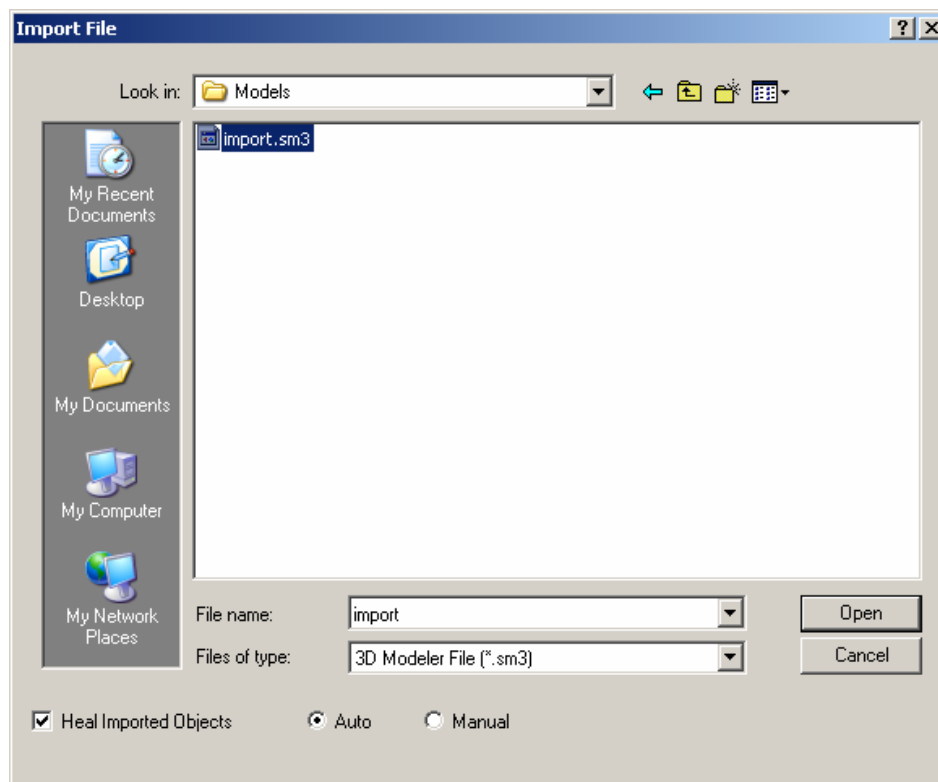
Vary Model Resolution and Surface Deviation

Try miscellaneous modifications

Redraw as much as is needed to get the geometry to mesh

Geometry Import/Healing

- 
Importing a geometry into the 3D modeler and Healing of geometries
 - 
 Geometries can be imported from many sources in numerous formats. The importing of geometry allows complicated structures from CAD type tools to be used within an electromagnetic simulation. Several points must be kept in mind when importing geometries, as important details in a CAD tool may not be important to (or may even be detrimental to) an electromagnetic simulation. However, with some general guidelines and knowledge of what to look for, most geometries can be imported from many of the standard modeling tools.
 - 
 When geometries are imported, conversions may be necessary to allow the file format to be read by the 3D Modeler. There may be very small errors in this conversion process which require healing. Also, the geometry of your model will determine the meshing characteristics of the simulation, so proper consideration for meshing should be given when creating, importing, and healing objects. Several different methods of healing imported objects and objects created in the 3D Modeler exist and will be discussed.



Ansoft Maxwell Design Environment

-  The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic

- **Edit**

-  Duplicate: **Around Axis**

- **3D Solid Modeling**

-  **Import**

-  **Purge History**

-  Model Analysis: **Analyze Objects**

-  **Heal**

-  **Align Faces**

-  Surface Operations: **Move Faces**

-  Boolean Operations: **Unite**

-  **Subtract**

-  **Intersect**

-  **Separate Bodies**

- **Mesh Operations**

-  **Surface Approximation**

-  **Model Resolution**

- **Analysis**

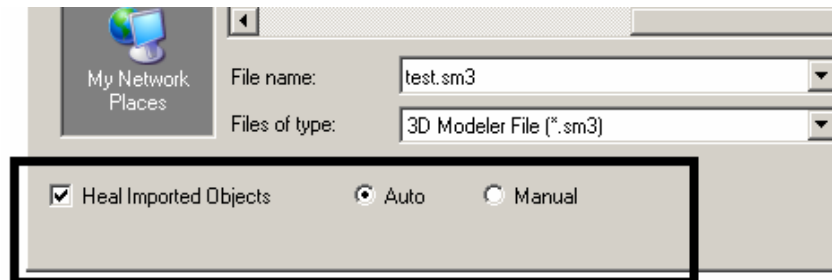
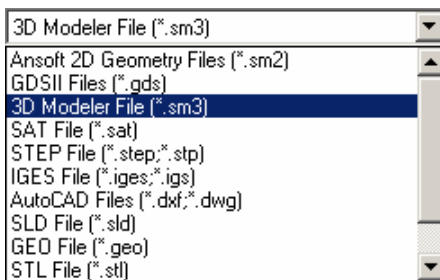
-  **Validation Check**

Geometry Import/Healing

Importing

1. To import a model, select **3D Modeler > Import ...**.
 1. An **Import File** dialog pops up, allowing you to browse to the desired file.

- On the left is a list of import files that we support. For some of these import options you will need an add-on translator feature in your license file.

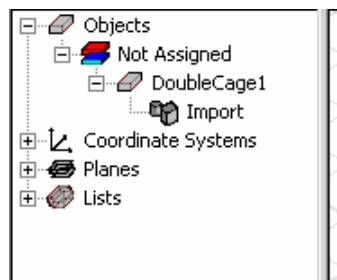


- On the right is the first available option for healing an imported model. This will be discussed in detail later.
1. Select the desired model and then **Open**.
 1. The model should appear in the window with any defined healing applied to it.

Geometry Import/Healing

Characteristics of Imported Objects

- ▲ The imported objects will appear with the same units and dimensions as exported (a 1in rod will appear as a 2.54cm rod when imported in those units).
- ▲ The imported parts will arrive separately if defined separately within the import file.
- ▲ The imported parts will retain the names assigned in the import file.
- ▲ The imported file will be added to the existing model; it will not replace it.
- ▲ The location of the imported file will be relative to the current coordinate system.
- ▲ The material assigned to the imported parts will be **Not Assigned**.
- ▲ The imported model will not have any history defined (the history tree will simply say **Import**), so it requires special operations to alter the model.
- ▲ If the object can not be classified as either solid, sheet or wire (e.g. it is some combination of the three), the object will be placed in an **Unclassified** folder in the history tree.



Suggestions for Exporting Geometries from Drawing Tools

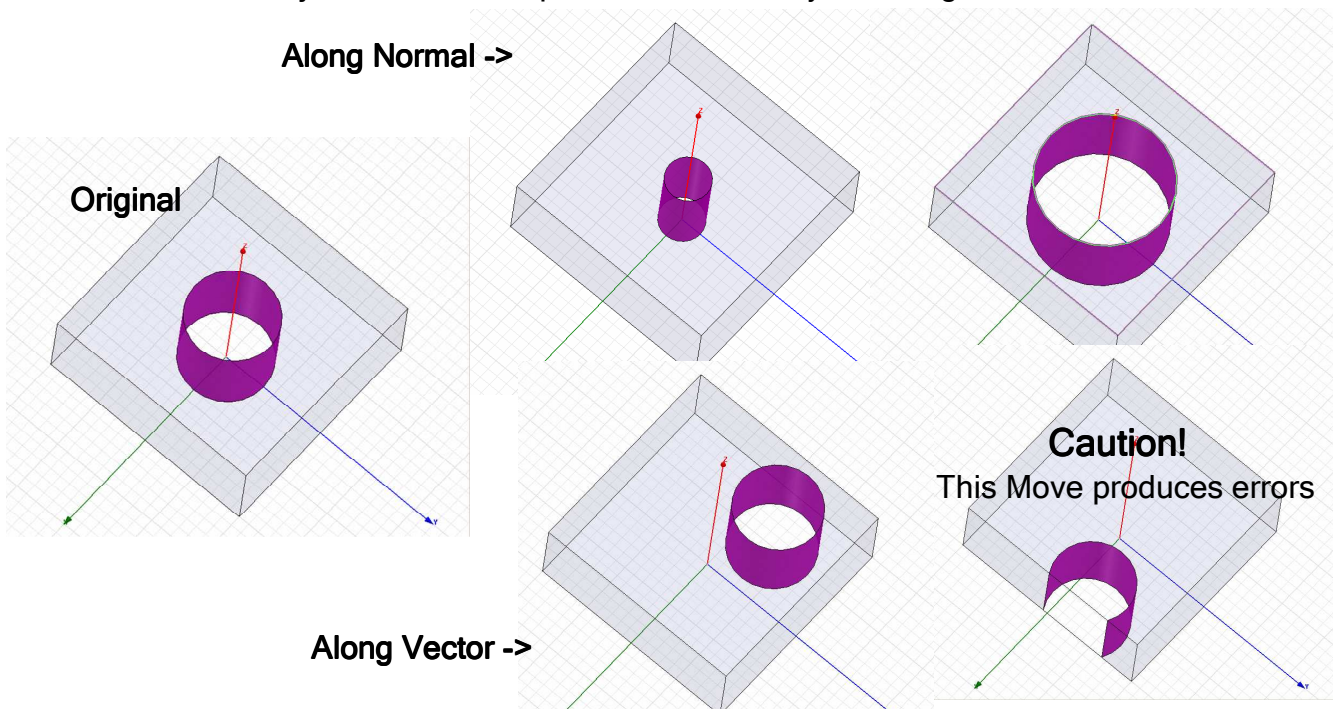
- ▲ De-feature the model to get rid of details that are not important for EM simulations (e.g. small rounds, chamfers, fillets, and mechanical connectors). Small features will result in a large and inefficient mesh.
- ▲ Use caution when exporting parts with touching (coincident) parts. Overlapping objects will need adjusting in the imported model.
- ▲ All parts should fit exactly when you create the assembly (materials should be in contact where required for electrical connections).
- ▲ Thin parts may also be successfully removed and replaced later with appropriate 2D objects (e.g. an insulating boundary in place of a thin insulating material).
- ▲ Do not use the interference fit for your exported assemblies - use the slide fit so that parts contact exactly as desired in the EM simulation.

Adjusting Imported Geometries

- Several methods can be used to adjust an imported geometry - two methods are described below. Creativity can expand the following basic strategies to adjust most imported geometries and even allow parametric manipulation of these static imports.
- The first method is to use a combination of the geometry primitives (cylinders, cubes, etc.) and Boolean expressions to add, subtract, and manipulate the geometries and therefore allow access to many dimensional variables.
- The second method is to use surface commands on the faces of imported objects as described below.

 - Select the desired face to adjust (type **f** while in the 3D Modeler to switch to face select mode - type **o** to switch back to object select).
 - Select **3D Modeler > Surface > Move Faces > Along Normal** and type in the distance the face should move along the normal. This will contract (negative distance) or expand (positive distance) the object along the normal of the face. This offset is now a parameterizable distance that is accessible in the history tree.
 - The following is an example of how to manipulate an imported object to adjust the size and position of a hole by selecting the walls of the hole.

Along Normal ->



Along Vector ->

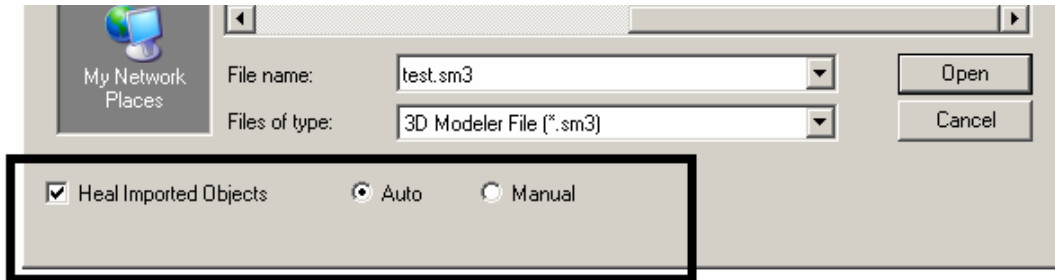
Common Model Errors

-  The following are common model errors that can be encountered during validation checks and when creating a mesh.
 1. `api_check_entity()` errors. These are errors detected by ACIS and are geometry and topology errors.
 2. non-manifold topology. This detects non-manifold edges and vertices that are present in the model (non-manifold means that a 3D object has no thickness at some point - i.e. two faces meet and produce a 2D sheet, edge, or vertex in a 3D object).
 3. Body pair intersection. This detects if pairs of bodies intersect.
 4. Small feature detection. This detects small edge length, small face area and sliver faces.
 5. Mis-aligned entities detection. This detects pairs of faces from bodies that can be aligned to remove interbody intersections. This improves the odds of mesh success.
 6. Mesh failure error display. This is available for single body, body pairs and last simulation run (all bodies in model). Errors reported by the meshing module are reported to the user.
-  Errors of type 1, 2, and 3 must be resolved before the mesh can be applied to the model.

Imported Geometry Errors

-  There are two types of errors related most specifically to Imported Parts:
 -  Geometry errors
 -  Topology errors
-  Geometry errors are errors in definition of the underlying geometry.
-  Topology errors are errors in how the underlying components like faces, edges, and vertices are connected.
-  These errors must be fixed before mesh analysis can be performed.

Healing during Geometry Import



- ▲ In the Import File dialog there is a check box “**Heal Imported Objects**”
- ▲ This will heal small errors that occur when converting the imported file from its original format to the format of the 3D Modeler (ACIS SAT v14.0).
- ▲ There are two modes for healing the imported object - “auto” and “manual”.
 - ▲ Auto healing will try to address ACIS errors and non-manifold errors, the first two classes of potential errors listed earlier.
 - ▲ Manual healing adds small-feature removal to the auto-healing. You can remove small features at this stage if you wish. However, the usual approach is to apply auto-healing at this stage and leave small-feature removal until later.
- ▲ After you import a part, you should perform a **Validation Check** as described in the next section. This allows you to target problems before the model is altered or becomes too complex.

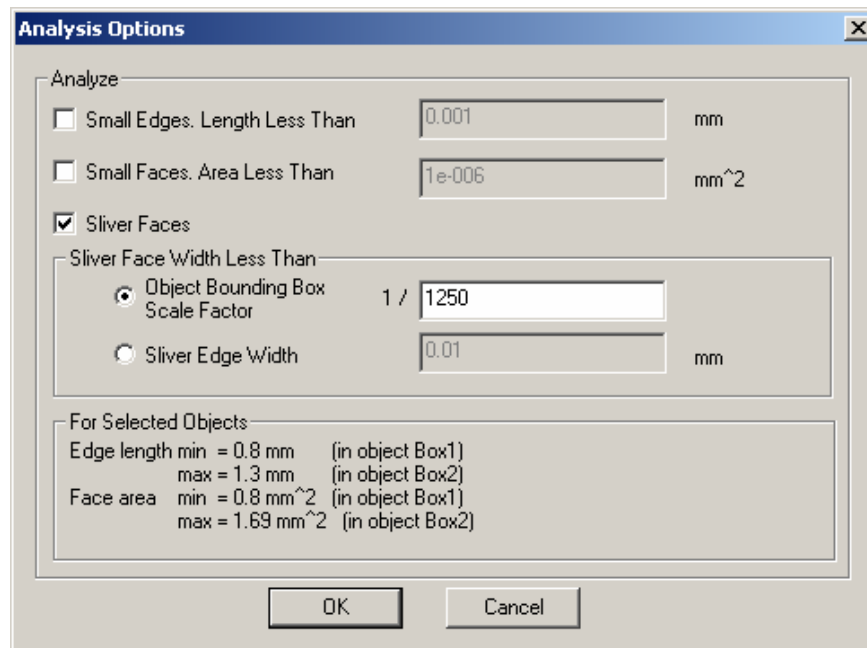
Geometry Import/Healing

Healing after Geometry Import

- ▲ Healing can only be performed on objects that have no drawing history other than “Import”.
- ▲ If necessary, object history can be deleted through **3D Modeler > Purge History**. If this causes a warning that another object will be deleted, you may need to purge the history of that other object first, or purge the histories of several objects simultaneously.
- ▲ After you import an object, you should perform a validation check **Maxwell > Validation Check**. This lets you focus on objects and object pairs that prevent the mesh from being invoked.

Fixing api_check_entity() errors

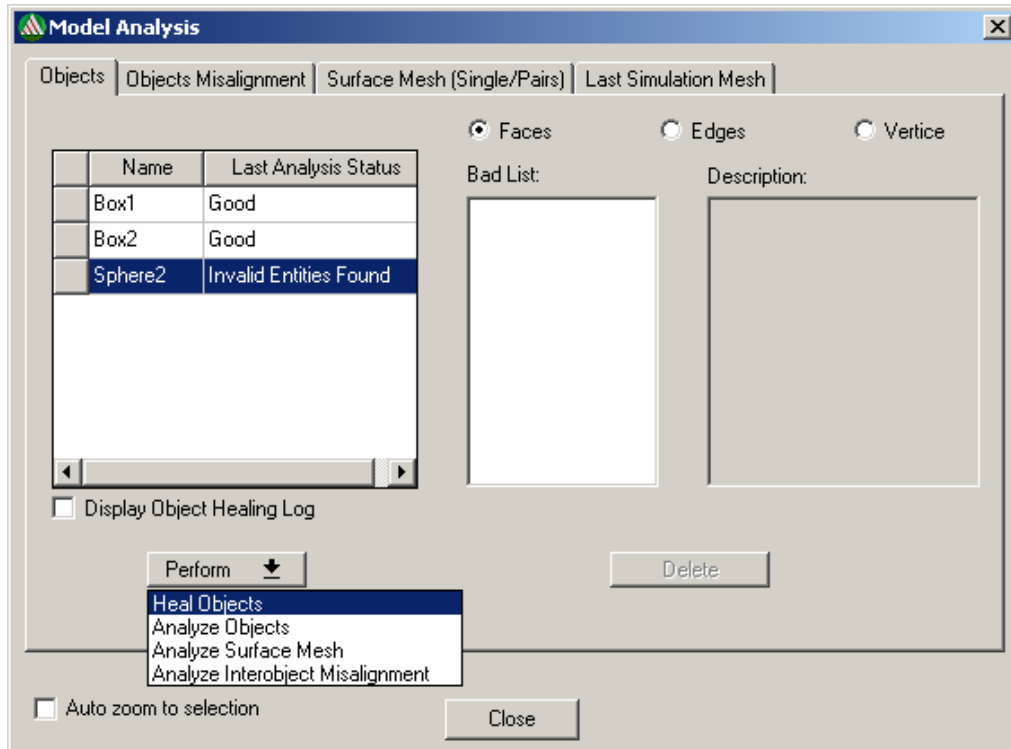
- ▲ The objects that fail api_check_entity() should be analyzed via the Analyze Objects menu item.
1. Select the objects that have ACIS errors.
 2. Select **3D Modeler > Model Analysis > Analyze Objects**.
 3. The **Analysis Options** dialog box appears with options for small feature detection.



4. Select **OK** to proceed.

Fixing api_check_entity() errors (Continued)

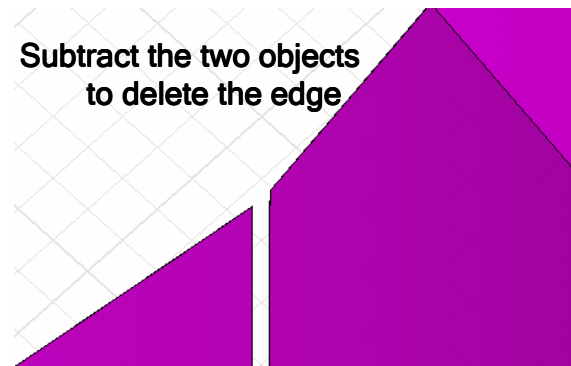
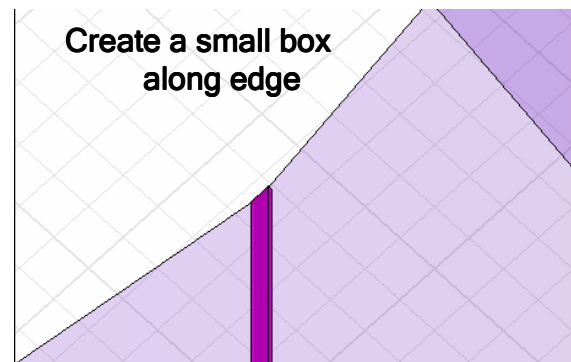
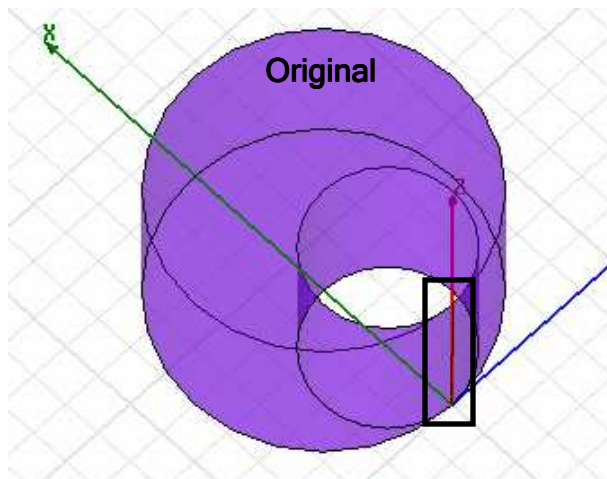
5. The **Model Analysis** dialog box appears.



- ▲ This dialog produces a list of problems affecting faces, edges, and vertices.
 - ▲ There is also the option to auto zoom to regions where problems exist.
6. Select the objects marked with “Invalid Entities Found” and click **Perform > Heal Objects**. The **Healing Options** dialog box appears. Adjust parameters as necessary. Click **OK**. The **Model Analysis** dialog box reappears.
 7. In most cases, the objects are healed, and the errors are fixed.
 8. If errors persist, select the edges and faces still containing errors and click **Delete**. This replaces each selected face/edge object by a tolerant edge/vertex, respectively. In some cases, the replacement of face/edge by edge/vertex fails.
- ▲ Objects in the Model Analysis can have the following statuses: Good, Null Body, Analysis not performed, Invalid entities found, Small-entity errors.
 - ▲ Invalid-entity errors are ACIS and non-manifold errors.
 - ▲ Invalid-entity errors must be fixed before a mesh can be generated.

Fixing non-manifold errors

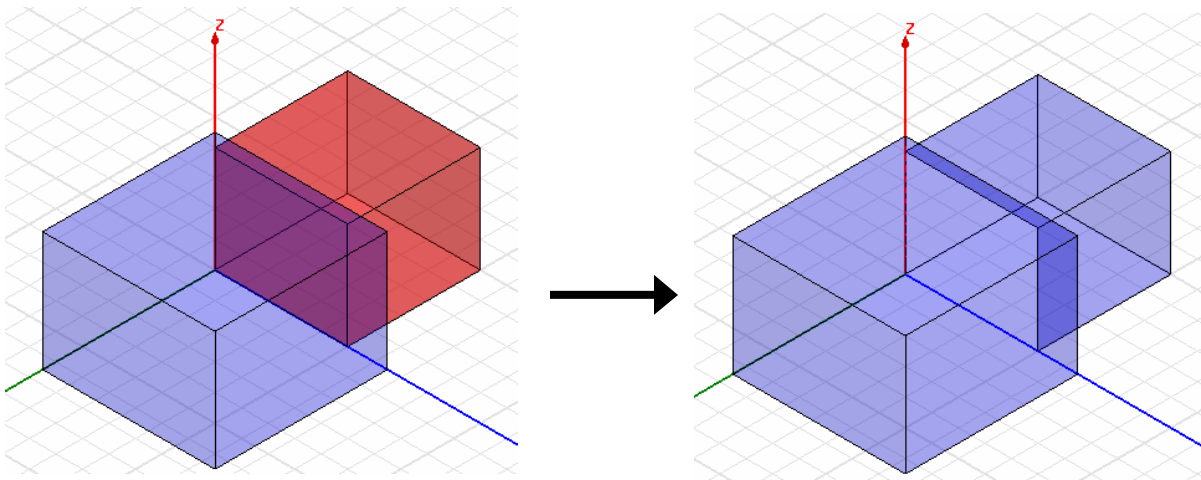
- ▲ The main strategy for fixing non-manifold errors is to slightly adjust the geometry to either add thickness to the object or delete the non-manifold edge.
- ▲ Find the non-manifold face/edge/vertex either visually or with the Analyze Objects menu item.
- ▲ Create a small box to contain the non-manifold edge.
- ▲ Either unite or subtract the non-manifold object with the small box.
- ▲ A unite operation will add thickness to the object at the required place.
- ▲ A subtract operation will delete the non-manifold edge (and a small portion of the model).
- ▲ This example shows a non-manifold edge along the z-axis



- ▲ Use engineering judgment to decide whether this edge is in an area of high fields, and is therefore important. The importance of this area can decide the size of the box and whether to add or subtract the box.

Fixing Object Pair intersection

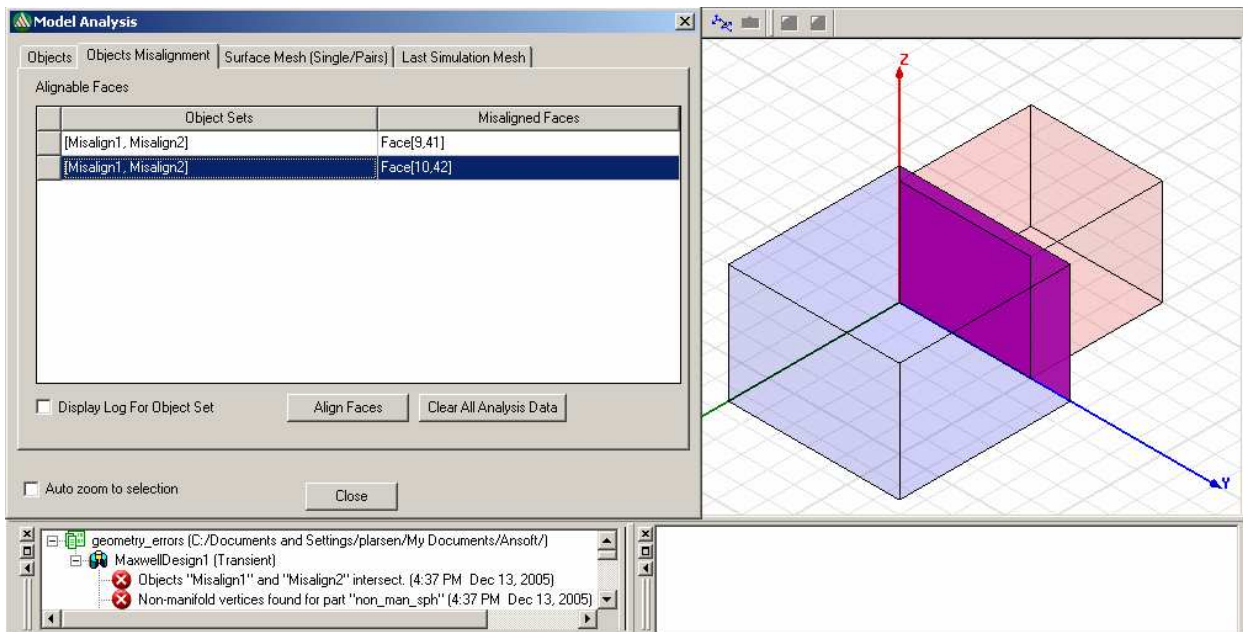
- ▲ This type of error is easily seen in a Validation Check, where the intersecting objects are designated with an error notification in the message window.
- ▲ There are several reasons why objects could be intersecting when they are imported. If care is taken with drawing and exporting the model from the desired CAD tool (i.e. no interference fit and no overlap in the model), then the problem is most likely in the translation.
- ▲ You can identify which parts of the objects are intersecting by performing an **Intersection** of the two objects (in the case where one object is supposed to be entirely inside the other you should perform a **Subtraction** to see what part of the inner object is not within the outer object).
- ▲ There are several ways to fix this issue. They are as follows:
 - ▲ The easiest circumstance for intersecting objects is if the objects are of the same material with no need for any distinction between the objects.
 - ▲ To fix intersecting objects of the same material:
 1. Select the intersecting objects.
 2. Select **3D Modeler > Boolean > Unite**.
 - ▲ This will group all the selected objects and there will be no intersection.



- ▲ In circumstances requiring separate objects the following techniques apply.

Fixing Object Pair intersection (Continued)

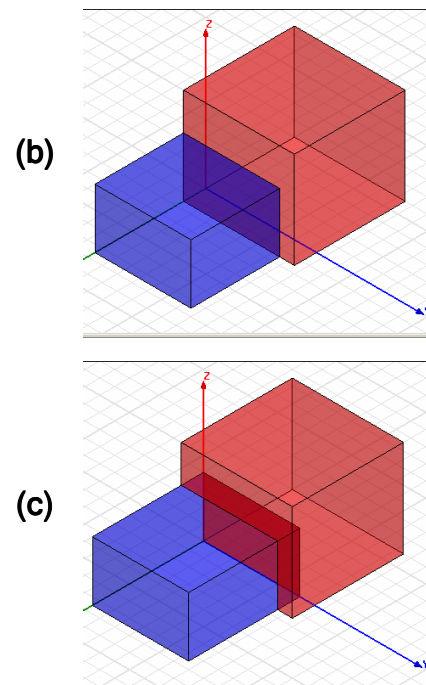
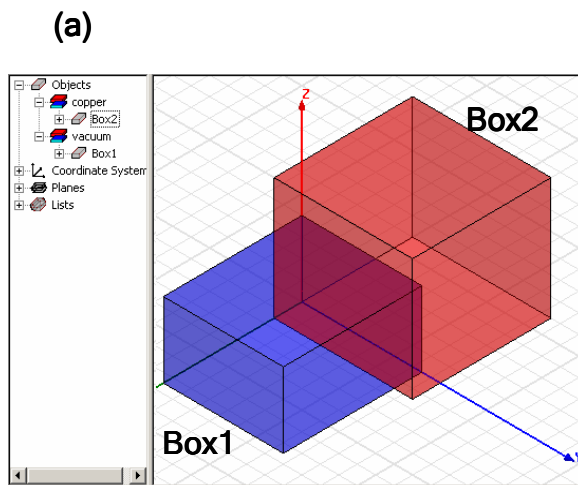
- ▲ In the circumstance that two adjacent objects are supposed to be coincident along their respective faces, yet the objects intersect, you can **Align Faces**.
- ▲ Aligning faces will take two different object faces that are very close together and make them coincident (more on mis-aligned objects will be discussed later).
- ▲ To align the faces you can use the Analyze Objects tool similar to the process in fixing `api_check_entity()` errors.
 1. Make sure the histories for intersecting objects are purged.
 2. Select the intersecting objects.
 3. Select **3D Modeler > Model Analysis > Analyze InterObject Misalignment**.
 4. Select the individual Object Sets to find which faces are mis-aligned at a possibly intersecting area.



5. Select **Align Faces** to make the faces coincident.
 6. Proceed with any other intersecting, mis-aligned faces.
 7. Select **Close**.
- ▲ This set of operations should fix the intersecting errors.
 - ▲ Run a **Validation Check** to determine that the aligned objects do not intersect.

Fixing Object Pair intersection (Continued)

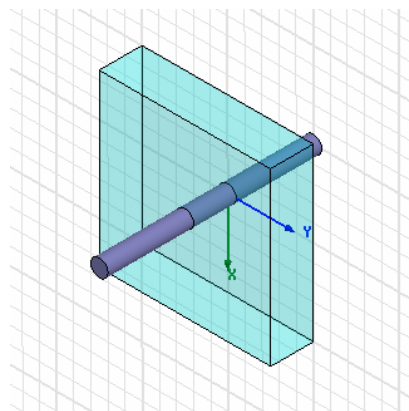
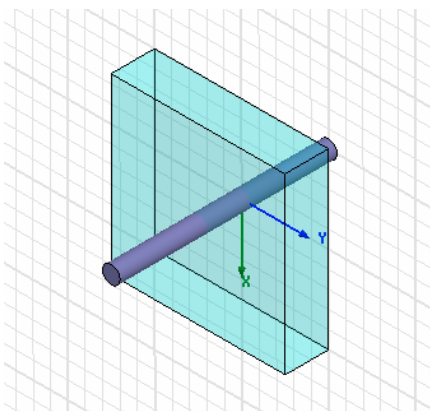
- Other circumstances for intersecting objects require manual Boolean manipulation.
- One Boolean method of fixing intersecting objects is to subtract a copy of one object from the other. In the subtract dialog, the tool object should be the object of which you want to keep the material definition in the overlapping area.
 - For example, Box1 and Box2 intersect (picture (a) below).
 - Box1 is vacuum.
 - Box2 is copper.
 - Subtraction with Box1 as the Blank Part and Box2 as the Tool Part will define the intersecting area as the copper part (picture (b) below).
 - Subtraction with Box2 as the Blank Part and Box1 as the Tool Part will define the intersecting area as the vacuum part (picture (c) below).



- Note:** Either Select **Clone** tool objects before subtracting while subtracting or Duplicate the object (Select the tool object, then *Edit > Duplicate > Around Axis* with an Angle of 0 and a Total number of 2, or use **Ctrl+C**, **Ctrl+V** if you are in the original object's coordinate system) before subtracting to retain both objects after the subtraction is performed.

Fixing Object Pair intersection (Continued)

- ▲ The final method for fixing intersecting objects is a general method that is very robust, but more advanced.
- ▲ Take for example a copper wire passing through a box that is defined as vacuum (pictured below).
- ▲ To solve this intersection easily, you would just subtract the wire from the box as described in the last method. However, this creates a situation known as Coincident True Surfaces, which is difficult for the 3D Modeler to mesh correctly.
- ▲ One way to get around this difficulty is to create three sections to the wire (one section within the box and two sections on either side).
- ▲ The process is as follows:
 1. Select the wire and the box.
 2. Select **Edit > Duplicate > Around Axis** with an Angle of 0 and a Total number of 2 (or Type **Ctrl+C**, then **Ctrl+V** if you are in the original object's coordinate system).
 3. Select the copies of both the wire and the box.
 4. Select **3D Modeler > Boolean > Intersect** to create the middle section.
 5. Select the original wire and box.
 6. Select **3D Modeler > Boolean > Subtract**.
 - ▲ Make sure the wire is in the Blank Parts and the box is in the Tool Parts, and the **Clone tool objects before subtracting** box is checked.
 7. This creates the outer sections as one part.
 8. To separate the outer sections, select the outer-section wire, then Select **3D Modeler > Boolean > Separate Bodies**.
- ▲ The wire is now either completely inside the box or completely outside the box. There are still coincident surfaces, but these are not true surfaces.
- ▲ Repeat this process for any other objects that may be intersecting.



Meshing difficulties

-  When there are no ACIS errors in the model, no non-manifold objects, and no partial object intersections, the mesh generator can be invoked to create a valid mesh for the electromagnetic analysis.
-  Even if the geometry is valid, mesh generation can still fail.
-  Possible causes of mesh failure are the presence of very short edges, very small faces, long and thin sliver faces, and slight misalignments between faces that are supposed to be coincident.

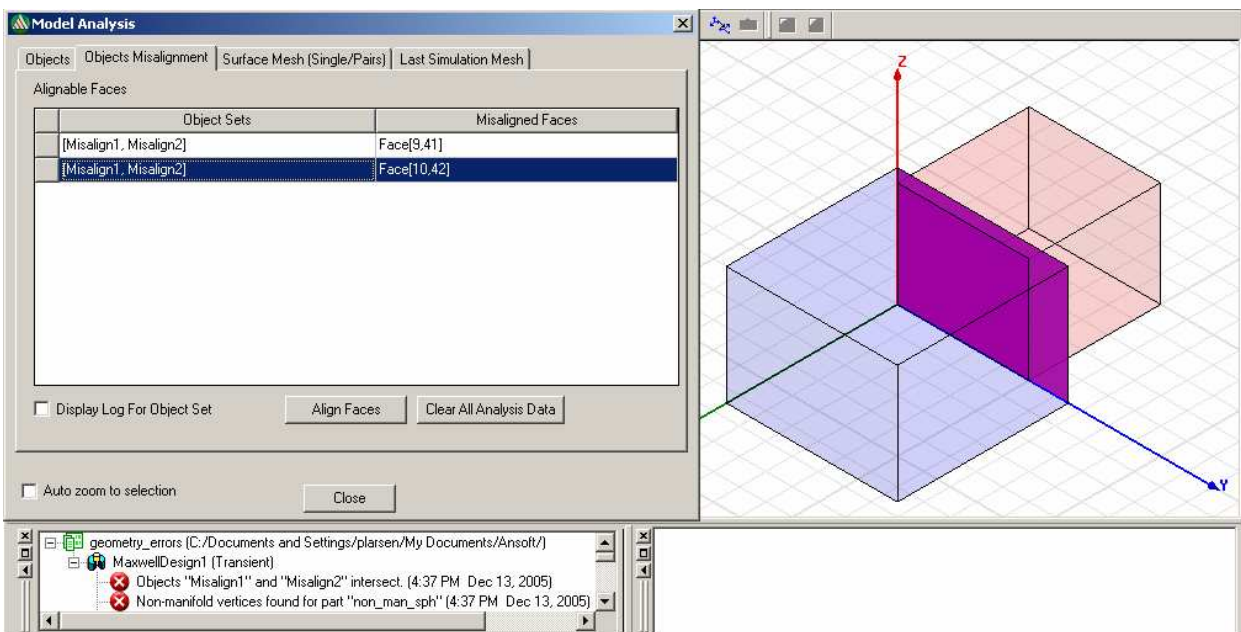
Small features

-  Small features in the geometry can lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower.
-  Small features may even cause the mesh generation to fail.
-  By small, we mean details on an object that are thousands of times smaller than the main features of the object, and that, in most cases, are unintended consequences of the drawing history in another CAD tool.
-  Therefore, it is advantageous to remove small features.
-  This step is entirely optional, and although it could have been accomplished when an object is imported (by selecting the **Manual** radio button instead of **Auto**), we present this information here because the previous steps were necessary to invoke mesh generation, while this one is optional.
-  To start the small-feature analysis:
 1. Select the objects (their histories must be purged) and invoke Object Analysis through **3D Modeler > Model Analysis > Analyze Objects**.
 2. In the **Model Analysis Window**, select **Perform > Analyze Objects**.
 3. The software will report the smallest edge length and the smallest face area, and enable you to set thresholds for the detection of short edges, small faces, and sliver faces.
 4. Click **OK**, and the analysis is performed.
-  This will report small features in the analyzed objects that can be Deleted automatically or repaired with Boolean operations in the 3D Modeler.

Geometry Import/Healing

Mis-aligned entities

- ▲ Objects that touch each other in imported geometries don't always have well-aligned faces. Often, this is a consequence of the limited level of precision in the imported file.
- ▲ Misaligned faces can cause tiny object intersections or tiny gaps between objects, which in turn can lead to an inefficient mesh or even a failure to create the mesh.
- ▲ To repair such misalignments in an automated way take the following steps:
 1. Select groups of objects.
 2. Select **3D Modeler > Model Analysis > Analyze Interobject Misalignment**.
 3. This will produce face pairs from different bodies that are slightly misaligned with respect to each other.
 4. In the window that shows this list, check the box **Auto-Zoom to Selection**.
 5. Select face pairs from the list to visualize which face pairs are misaligned.
 6. When it appears that the faces should be aligned (either they should be coincident or on the same plane), click Align Faces.



Coincident True Surfaces

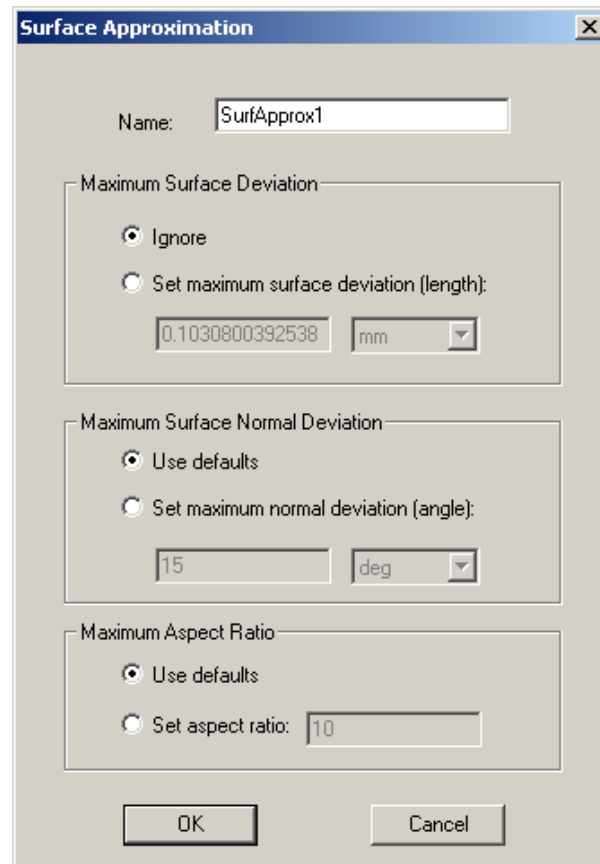
-  True Surfaces are curved surfaces in the 3D Modeler (e.g. spheres have a true surface, while boxes do not).
-  Due to the finite nature of the mesh, these true surfaces are approximated with facets and tetrahedra (as explained in the section on Mesh Operations).
-  When two true surfaces are coincident the mesh that is constructed for each object must align on the coincident face. This is difficult for the mesher.
-  A mesh failure can occur simply because of coincident true surfaces.
-  A mesh failure does not have to occur while constructing the first mesh - coincident surface mesh errors can occur on any adaptive pass.

-  There are different schools of thought to avoiding mesh problems with coincident true surfaces. Here are a few examples:
 - 1) Avoid true surfaces (e.g. use a regular polyhedron in place of a cylinder - this is usually not the preferred method).
 - 2) Avoid coincident true surfaces (see the last example in Fixing Object Pair Intersection for one method to achieve this).
 - 3) Accept the geometry you have imported (or constructed) and try to improve the chances of producing a usable mesh.

-  One method of improving the chances of a usable mesh involves increasing the number of facets on the true surfaces to make it easier for the mesh on each object to line up. After the mesher reconciles the meshes of the two coincident faces, the initial mesh size can be reduced to a more manageable size by using the model resolution. The procedure for this method is outlined in the following.

Coincident True Surfaces (Continued)

1. Select the coincident surface (type **f** in the 3D Modeler to switch to face selection - type **o** in the 3D Modeler to switch back to object selection).
 1. Select **Maxwell > Mesh Operations > Assign > Surface Approximation...**



2. Choose **Set maximum normal deviation (angle)** and enter a small number (sometimes 5 deg works well, other times a smaller angle is necessary).
 3. Select **OK**.
 4. Proceed assigning surface approximations to other coincident faces.
2. Select objects (either as a group or sequentially) that have surface approximations.
 1. Select **Maxwell > Mesh Operations > Assign > Model Resolution...**
 2. Enter a number that is half the size of the smallest necessary feature.
 3. Select **OK**.
 4. Repeat this assignment for any other objects with too fine a mesh.

Mesh Operations Examples

Mesh Operations

-  This chapter provides details on meshing in the Ansoft Maxwell v.11.0 software. The following topics are discussed:

-  Curved Geometry Mesh Adaptation
-  Faceting Default Settings
-  Volume Perturbation in FEM Solutions
-  User-Defining Surface Approximations
-  Application Recommendations
-  Model Resolution
-  Applying Mesh Operations

Examples

-  The following examples are provided to demonstrate the topics discussed in the chapter:
 -  Skin effect and adjacent effect for two bars (Eddy Current)
 -  LR Matrix of a bar of PCB (Eddy Current)
 -  3 Phase Linear PM Motor (Transient)
 -  1.5kw 4p24s IPM Synchronous Motor (Transient)

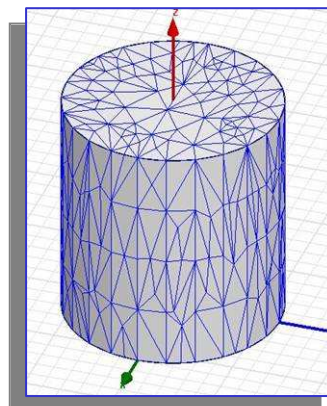
Mesh Operations Examples

Curve Mesh Adaptation in Maxwell v11

- ▲ Maxwell v11 meshes handles curved surfaces differently than prior versions of Maxwell. Proper understanding of the differences in curved mesh handling and the advantages of new capabilities in the Maxwell v11 solver are essential to obtaining accurate results.
 - ▲ The new graphical drawing interface offers the use of true-curved drawing, by removing the option to assign a facet count to the construction of primitives such as circles, cylinders, spheres, and ellipses.
 - ▲ Faceted primitives are still suggested as polyhedrons and polyhedral solids for basic needs. For examples, in rotating motion analysis with Transient solver, band have to be made as a faceted solid for Transient data mapping. It is also required for inner-region in translate motion.
 - ▲ Initial meshing is constrained by faceting decisions made by the first pass of the meshing algorithms as shown below.

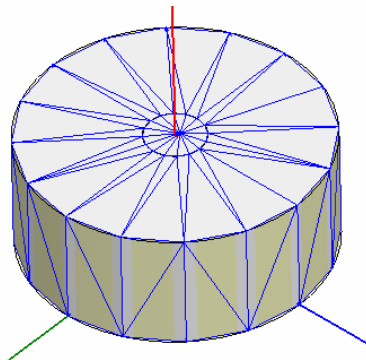


- ▲ For Maxwell v11 in order to provide more robust meshing with respect to more complex geometries, the initial faceting selections made for curved objects are respected throughout the adaptation process, so that the adapted mesh is a subdivided variation of the *same meshed volume* as the initial mesh. An adapted mesh from Maxwell v11 is shown below.



Faceter Default Settings

- ▲ In order to keep initial meshes at reasonable sizes, the initial faceting default setting is to constrain the mesh surface normals to fall within **22.5 degrees** of the true-curved surface normals.
 - ▲ This means that a cylindrical surface would be ‘faceted’ into 16 segments about its circumference, as illustrated below.
 - ▲ The normal for each flat segment cannot be off by more than 22.5 degrees from the normal for the curved true surface which that segment’s face is approximating.

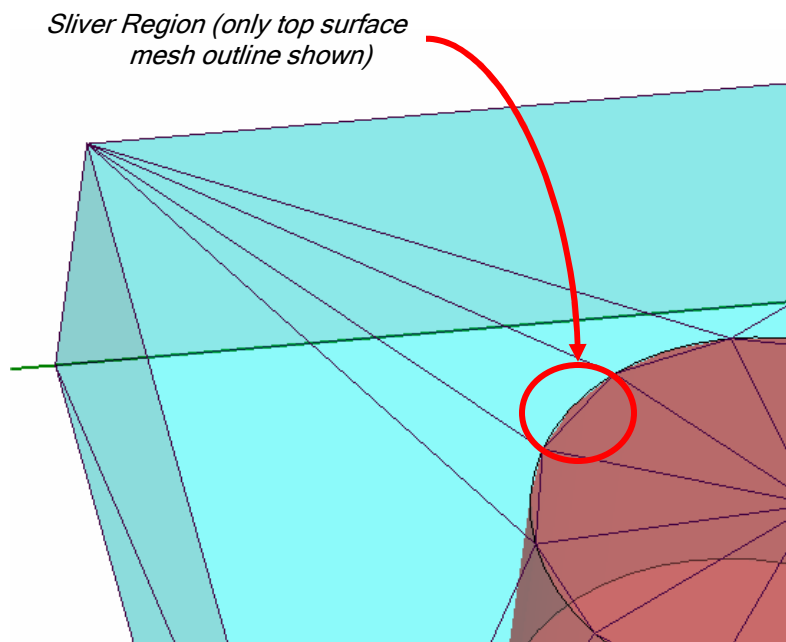


- ▲ Although aided by the **Volume Perturbation** technique now used in Maxwell v11, it is not anticipated that this default faceting will be ‘enough’ for high-accuracy analysis of motors, actuators, sensors etc.
 - ▲ The faceting default does not include a stringent aspect ratio default (note long triangles on top and bottom cylindrical faces).
- ▲ The default however is sufficient for many applications in which the curved geometry is not itself highly critical to the simulation result, yet where more faceting would result in an unnecessary large initial mesh.

Introduction of Volume Perturbation to Maxwell Solutions

- Despite lower faceting 'compliance' to the true-curved geometry as the adaptation process adds mesh points, Maxwell v11 introduces a new solution technology that should result in a more accurate solution from far smaller mesh counts than in prior versions. This technology is referred to as **Volume Perturbation** or **Volume Correction**.

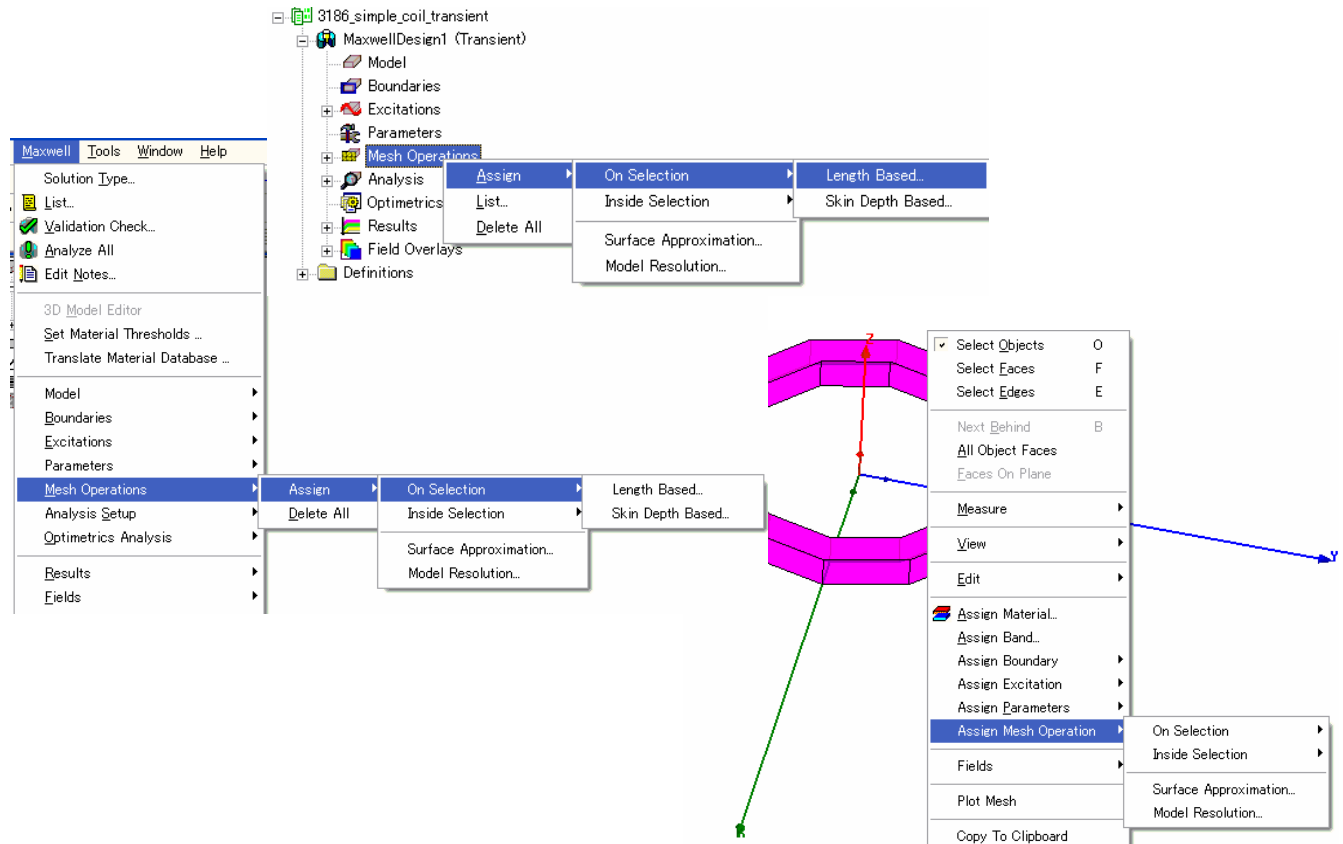
 - In a faceted mesh of a cylindrical solid, small arc-section volumes are 'lost' to the cylinder volume and become part of the volume of the cylinder's surroundings.
 - Since the true geometry definition is known before faceting, the delta-volumes represented by each of these sliver regions is computable
 - The meshing algorithm in Maxwell v11 can provide 'adjustment factors' to both the tetrahedral center-edge node locations and to the appropriate volumes of the tetrahedra on either side of the curved boundary, so that the FEM solution proceeds with the right terms even for a loosely-discretized mesh
 - The end result is higher accuracy from the same mesh, even before the benefits of adaptive mesh refinement are taken into account.
- Volume Perturbation solutions are always on. No user setting is required to active them.



Mesh Operations Examples

Mesh Operations

- Mesh Operations can be assigned from the **Maxwell** menu, from the **Design Tree**, or from the geometry interface's context-sensitive menu.



- Maxwell v11 can offer user 5 types of Mesh Operations:

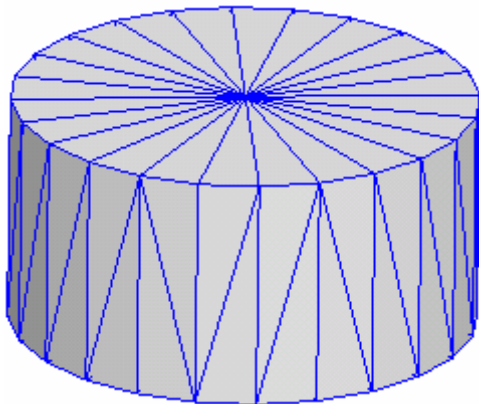
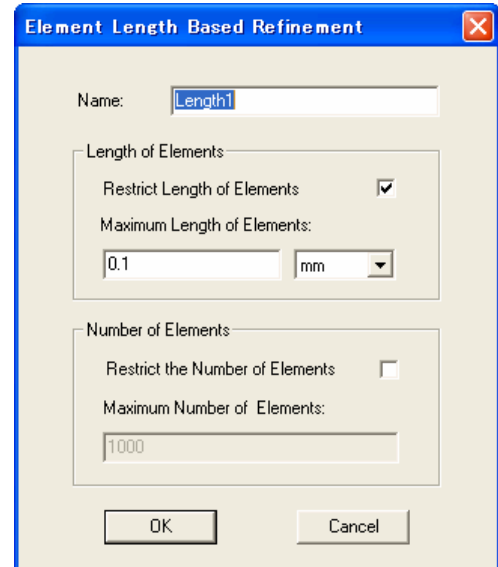
- On Selection/ Length Based;
- On Selection / Skin Depth Based
- Inside Selection / Length Based
- Surface Approximation
- Model Resolution

Mesh Operations Examples

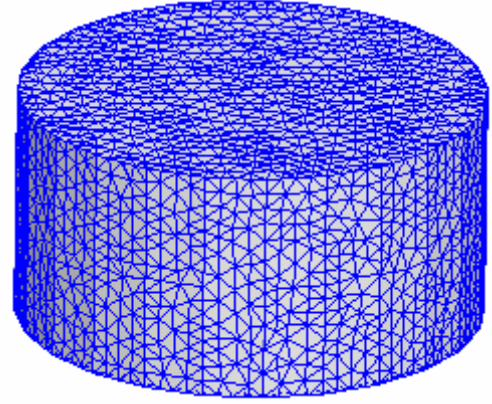
On Selection / Length Based

After user selected one or some Objects, this mesh operation can be setup as a maximum length of the edges of the mesh for it.

- The Cylinder with initial mesh below was created by Draw regular Polyhedron with $R = 1 \text{ mm}$, $H = 1 \text{ mm}$, Number of segments = 24. The left two fig. are the meshes on surface and the center cross-section. The right fig. of them are the results after using the mesh option **On Selection / Length Based .0.1mm**. User can see the surface of it was meshed with small ones on it. The Default of Maximum Length equals 20% of the maximum length of the edge of it.

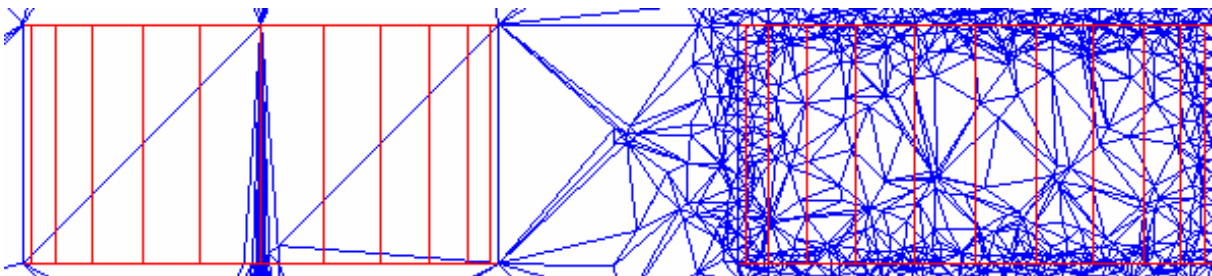


Without Mesh Operation



After Mesh Operation

Meshes on the surfaces



Without Mesh Operation

After Mesh Operation

Meshes on the center cross-section of them

Mesh Operations Examples

On Selection / Skin Depth Based

The mesh will be created with the skin depth effect in the object.

The Cylinder with initial mesh left below was created by Draw regular Polyhedron with $R = 1 \text{ mm}$, $H = 1 \text{ mm}$, No. of segments = 24. The right one was the results with the setting in **On Selection / Skin Depth Based**:

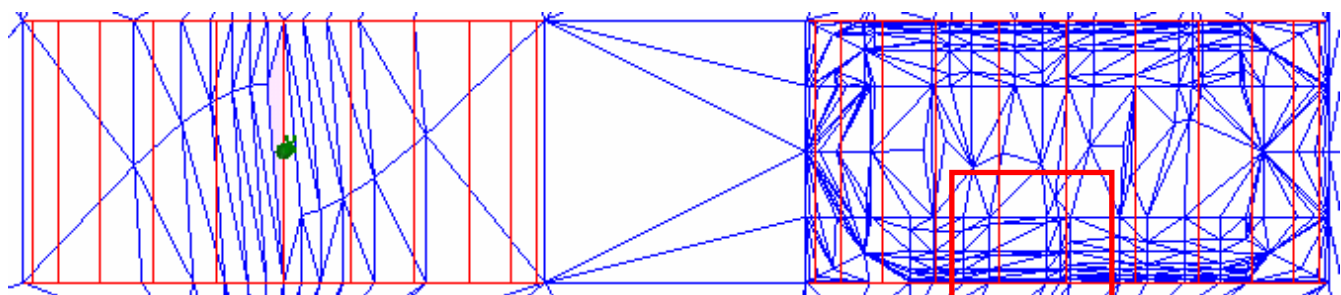
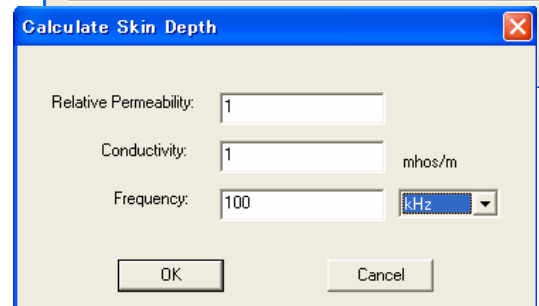
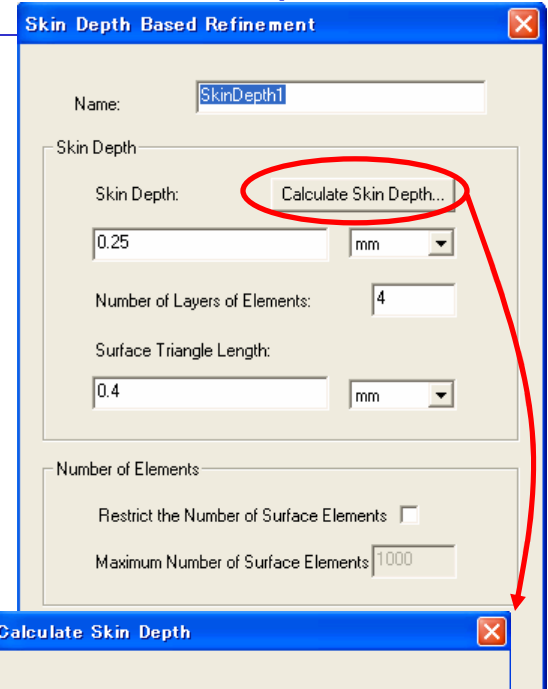
Skin Depth. 0.25mm.

Number of Layer of Element. 4;

Surface Triangle Length. 0.4mm.

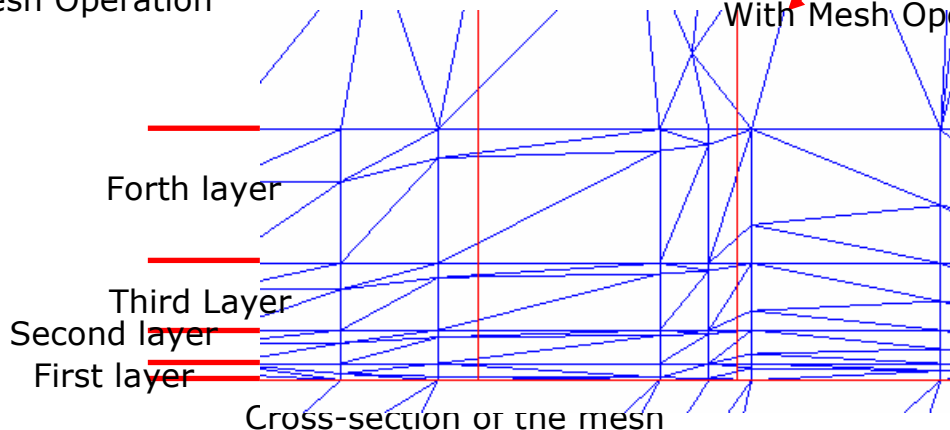
The inner mesh will be 4 layers where the skin depth effects will be considered.

if user select **Calculate Skin Depth**, the skin depth will be automatically calculated due to the frequency, permeability and conductivity set by user.



Without Mesh Operation

With Mesh Operation



Cross-section of the mesn

Mesh Operations Examples

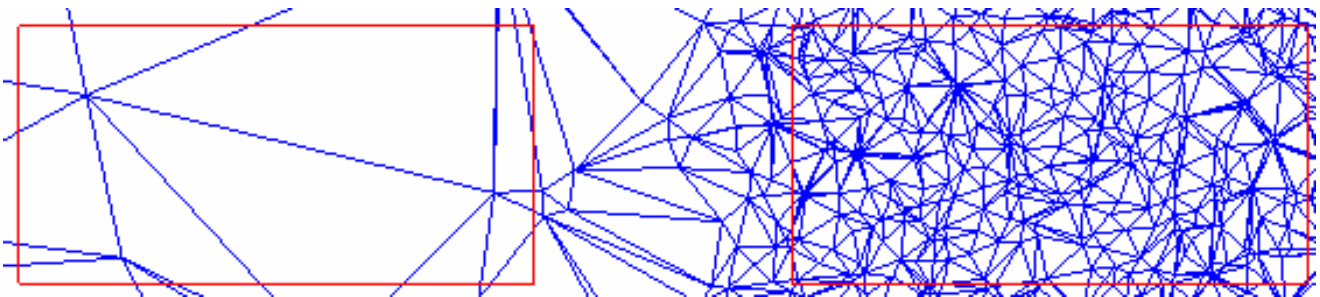
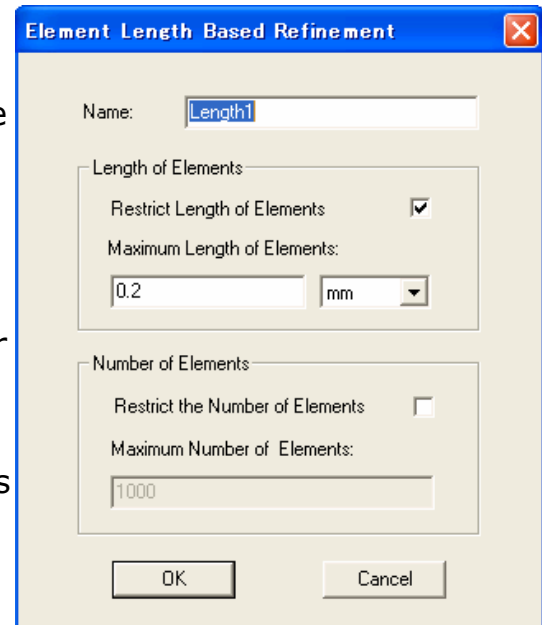
▲ Inside Selection / Length Based

Model Object of any shape can be meshed with this command for inner mesh creating. It's the most frequently used for manual mesh. It also can be used for Non Model Objects (Boxes only).

- ▲ Below is the 2x2x1 mm box with initial mesh. The right below is result with the command of **Inside Selection / Length Based** . 0.25mm

Note:

For some special points or lines where user have the measured data for comparison, the Non Model Boxes can be created to cover all these places and set small lengths in **Inside Selection / Length Based** for enough accuracy.



Without Mesh Operation

With Mesh Operation

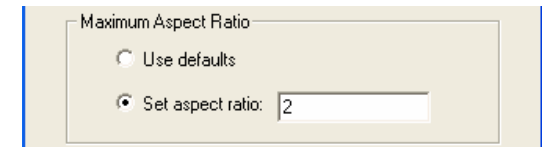
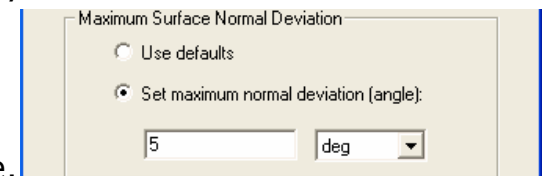
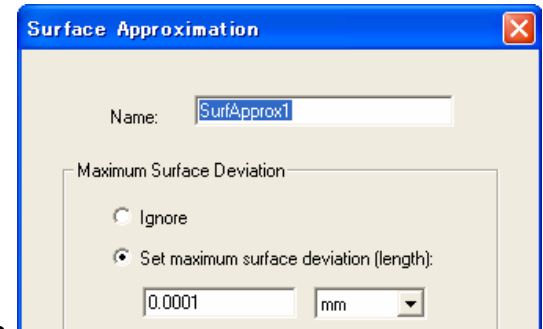
Cross-section of the mesh

Mesh Operations Examples

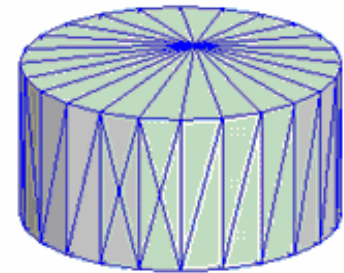
Surface Approximation

The Surface Approximation options are shown right Below.
Definitions follow:

- ▀ *Surface Deviation* is the maximum spacing, in drawing units, that the tetrahedral surfaces may be from the true-curved geometry's surface.
- ▀ *Normal Deviation* is the maximum angular difference, in degrees, that a tetrahedral face's normal can have from the surface normal for the true geometry which it is meant to represent.
- ▀ *Aspect Ratio* refers to the maximum allowed aspect ratio of all faces of all tetrahedral of the selected object or face. This setting influences mesh quality rather than actual meshed volume or surface locations.

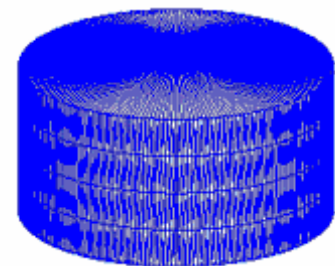


- ▀ The Cylinder with initial mesh on the right is created by Draw regular Polyhedron with R = 1 mm, H = 1 mm, No. of segments = 24.



Without Mesh Operation

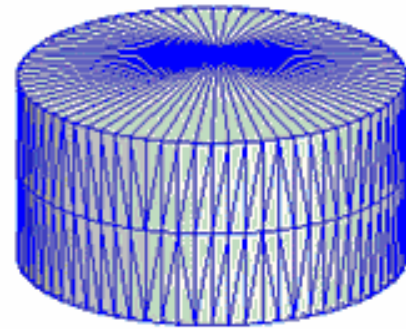
- ▀ The right is the result with **Maximum Surface Deviation . 0.000125mm.**



Maximum Surface Deviation

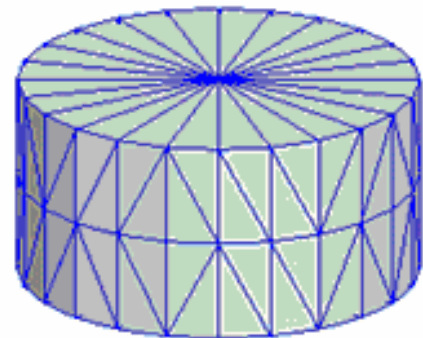
Mesh Operations Examples

Here in right is the result with **Maximum Surface Normal Deviation. 5deg.**



Maximum Surface Normal Deviation

The last in the right is the surface result with **Maximum Aspect Ratio .2**



Maximum Aspect Ratio

Mesh Operations Examples

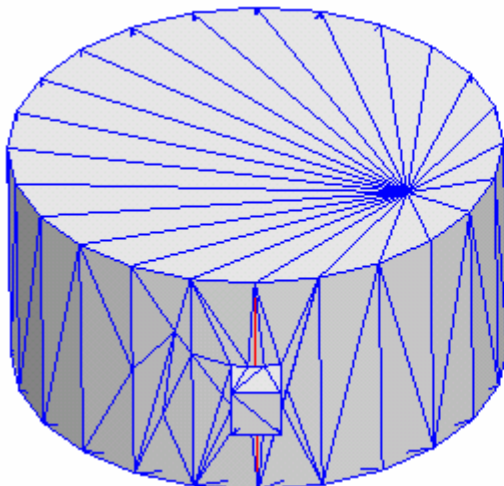
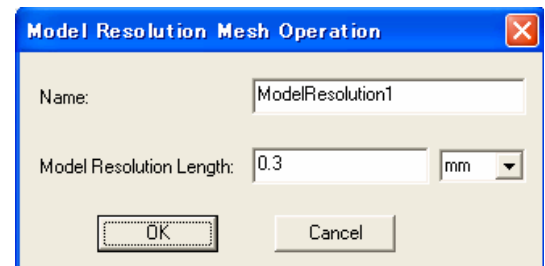
Model Resolution

- Model resolution is a length based value that modifies the initial mesh. This meshing operation allows the user to specify a minimum edge length of any tetrahedron used for the mesh. By specifying a minimum edge length for a tetrahedron, the mesher will have to coarsely mesh geometric detail that may not be electrically important. This will save the solver the amount of solution time since the initial mesh is smaller, and the mesher does not have to add mesh elements to areas that are not electrically important.
- When dealing with models that have very high aspect ratios due to small geometric detail, use a model resolution of 1/10 to 1/20 of the thinnest conductor to start with. Then adjust the value accordingly.

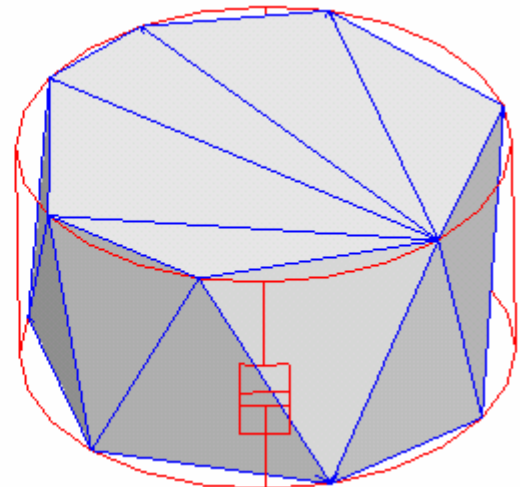
Simple example:

There is a small stub (0.2x0.2x0.2mm) on a cylinder (R=1mm, H=1mm). Below is the initial mesh without Model Resolution.

Then with **Model Resolution .0.3mm**, the mesh of it will be as below right, the stub will be neglected.



Without Mesh Operation



With Mesh Resolution

Mesh on the surface

Mesh Operations Examples

Mesh Operation Examples

EX1. Skin effect and adjacent effect for two bars (Eddy Current)

Model:

Two parallel copper bars (named coil1 & coil2) are created by Draw regular Polyhedron with $R = 0.15$ mm, $H = 3$ mm, No. of segments = 12. The distance between their axes is 0.4mm (Gap is 0.1mm)

The air box (3x3x3mm) is created for analysis region

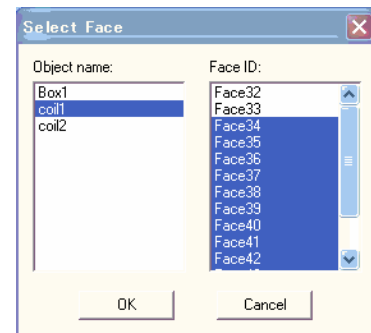
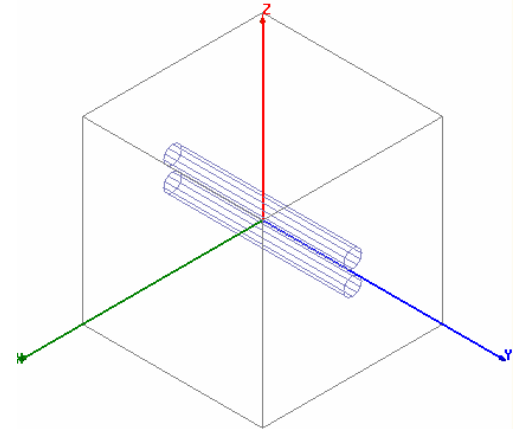
Excitations

For each end terminal of bars is set as 1A current in same Y direction (Design named .same.)

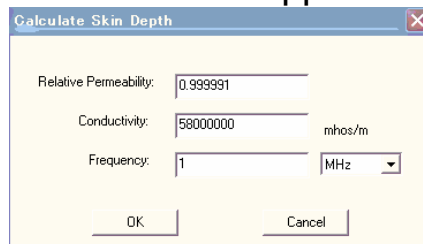
or Y direction for bar1, -Y direction for bar2 (Design named .opposite.)

Mesh Operation

Select all surface of bars except their end terminals as in right,



Then with **On Selection / Skin Depth Based**; get in **Calculate skin Depth...** and get the skin depth = 0.066mm for copper in 1MHz.



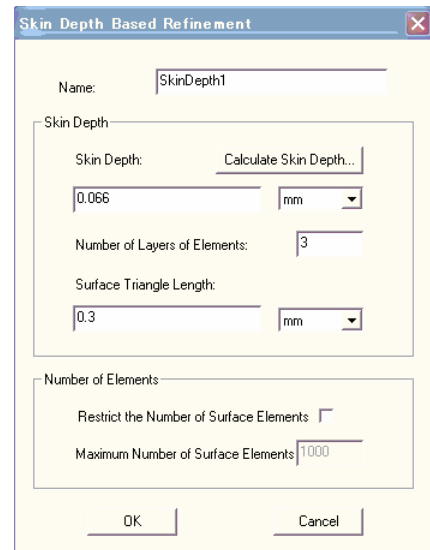
Then set **Skin Depth Based Refinement** as below:

Skin Depth. 0.066mm.

Number of Layer of Element. 3;

Surface Triangle Length. 0.3mm.

and without Number Restriction

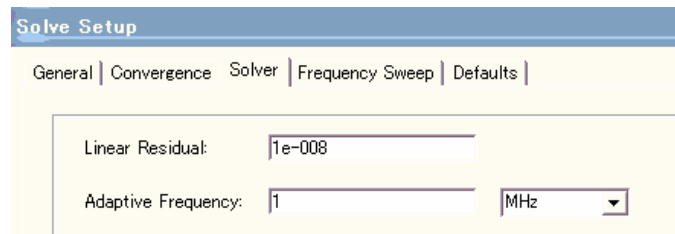
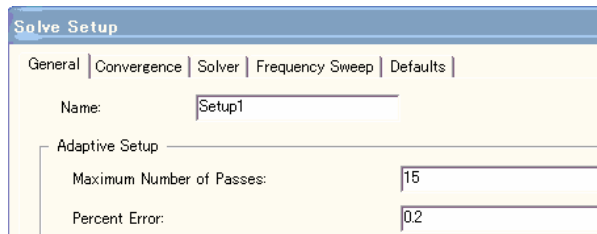


Mech Operations Examples

EX1. Skin effect and adjacent effect for two bars (Con't)

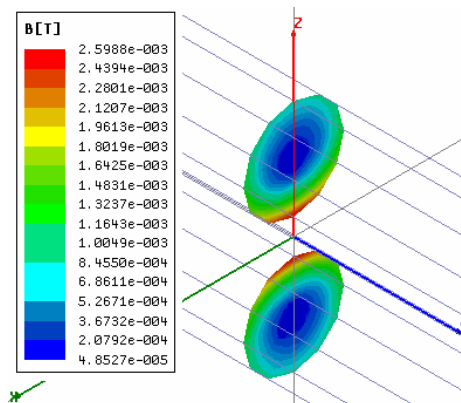
Solve Setup

Solve Setup is set as below:

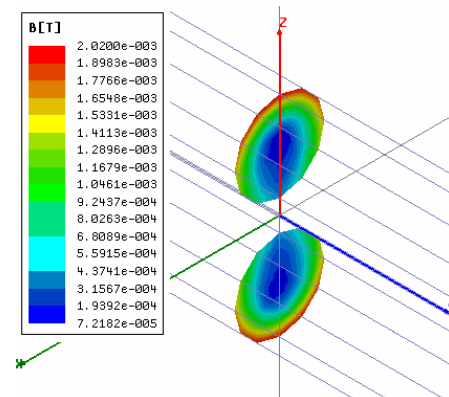


Results

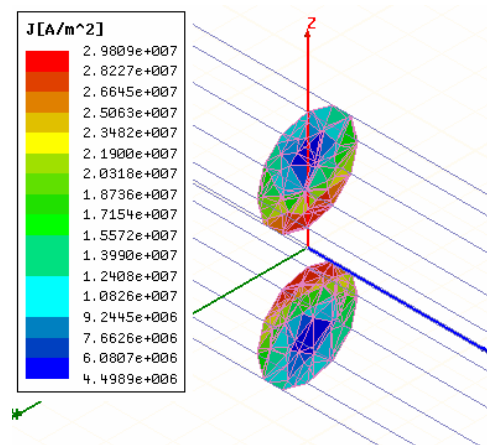
The B & J distribution on the XZ cross-section of two bars will be as below:



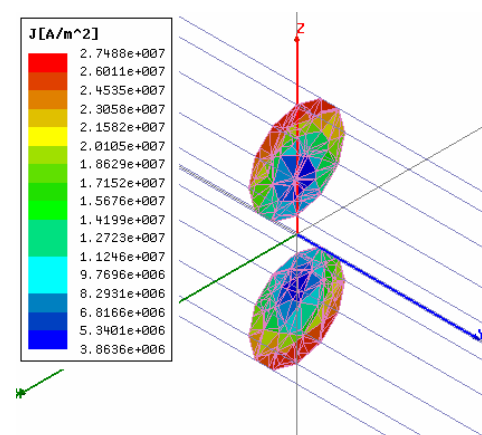
B distribution for .opposite.



B distribution for .same.



J distribution for .opposite.
With mesh plot



J distribution for .same.
With mesh plot

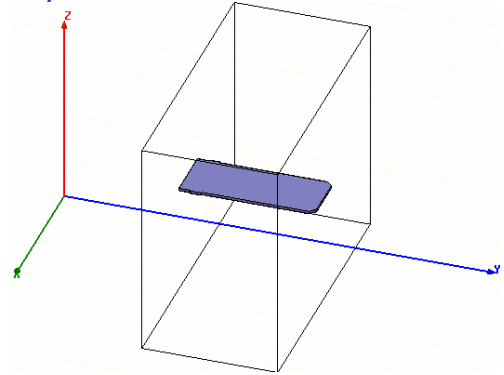
Mesh Operations Examples

EX2. LR Matrix for print bar (Eddy Current)

Model:

One copper bar (part from PCB, with Length = 24.5mm, Height = 0.5mm) on the right is imported. This model will calculate its LR from 1Mhz to 5MHz

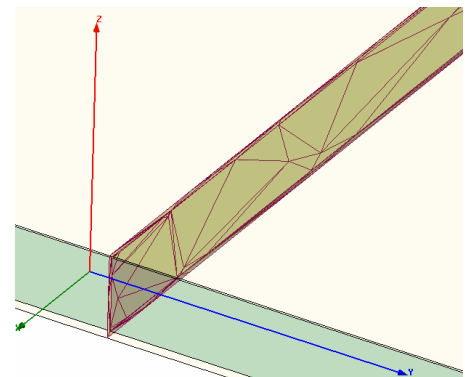
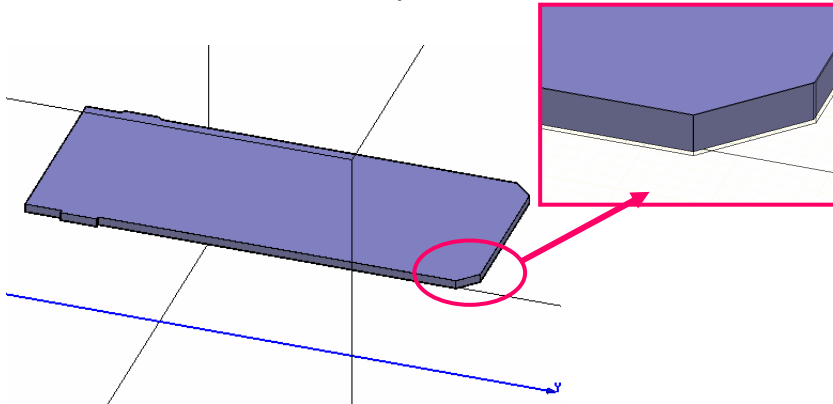
An air box (50x24.5x40 mm) is created as the analysis region



Mesh Operation

This time we will try using dummy Sheets to mesh the bar with skin depth meshes.

1. First we make the copy of the bar, and name it as .Unnamed_2.
2. Using **Calculate skin Depth** to get the skin depth = 0.029 mm as copper at 5 MHz.
3. Select the Unnamed_2 and do **/Edit/Arrange/Offset** with -0.029mm.
4. Select all surfaces of the new Unnamed_2 and do **/Edit/Surface/Create Objects** form Surfaces. (16 Sheets will be created) All the Sheets will help automesh to create skin depth meshes at the surface of the copper bar.



Mesh in cross-section of bar

Excitations

For each end terminal of bar is set as 1A current in Y direction

Matrix setup

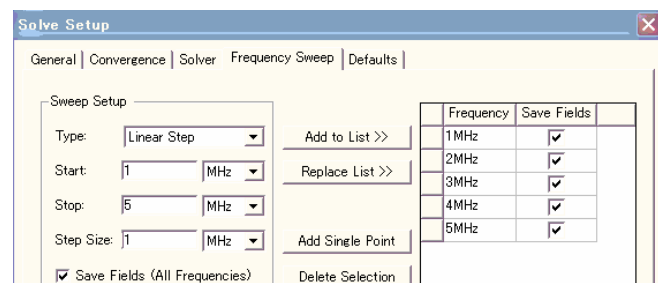
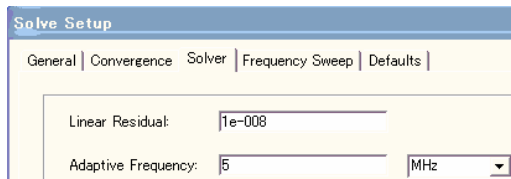
Select the copper bar and do **/Assign Parameters/Matrix** with Current1 as **Include**

Mesh Operations Examples

EX2. LR Matrix for print bar (Eddy Current) (Con't)

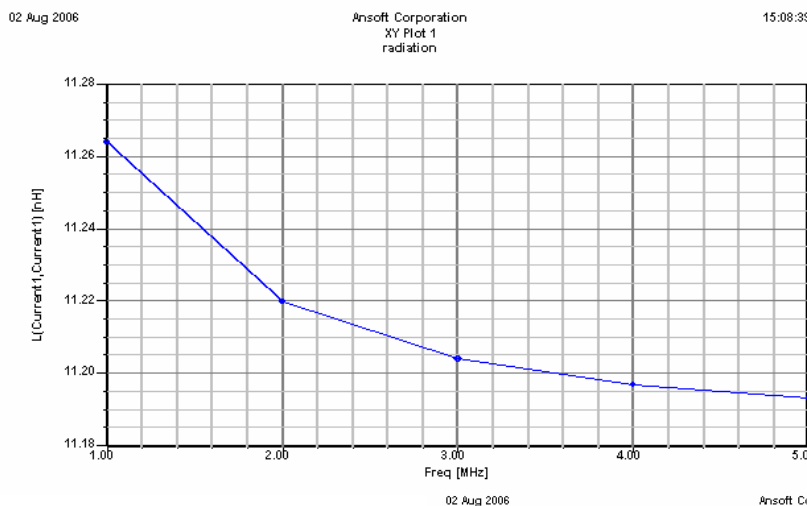
Solve Setup

Solve Setup is set as below: (Set Default In General & Convergence)

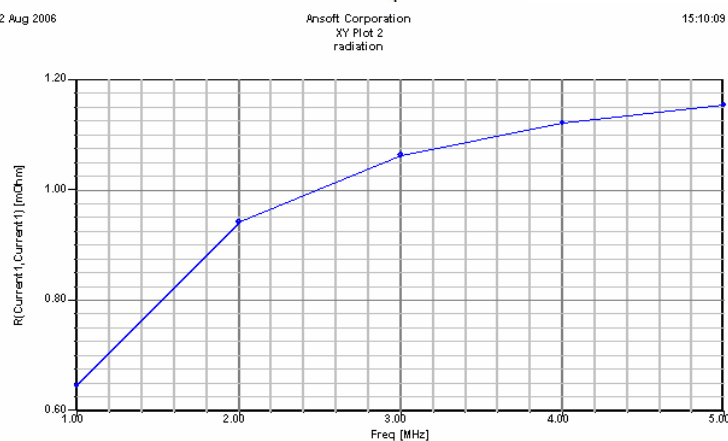


Results

The L & R vs. Frequency will be as below:



L vs. Frequency



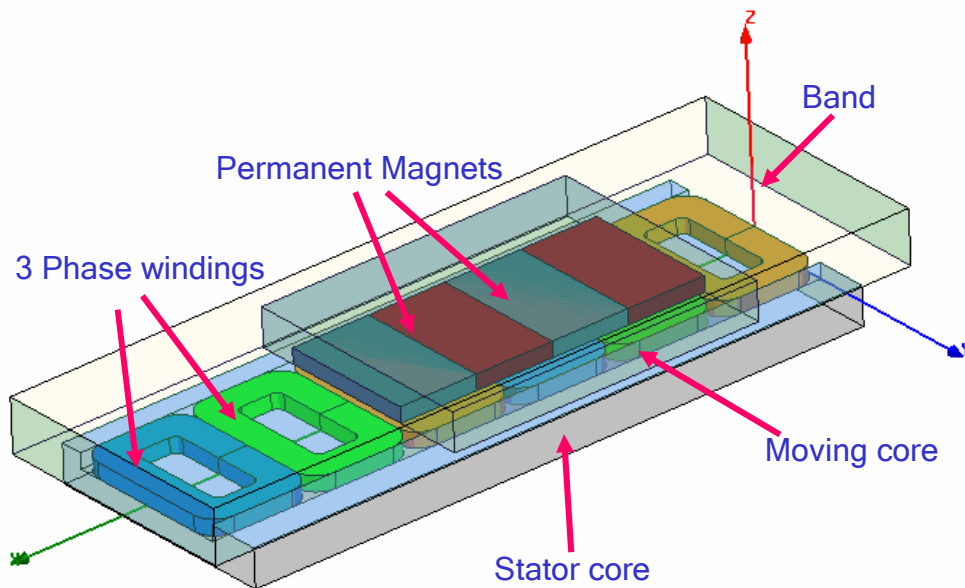
R vs. Frequency

Mesh Operations Examples

EX3. 3ph Linear PM Motor (Transient)

Model:

The basic parts of it are as below:

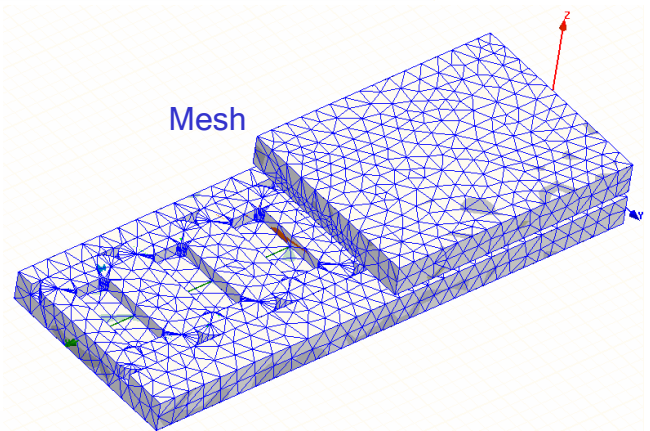


Mesh Operation

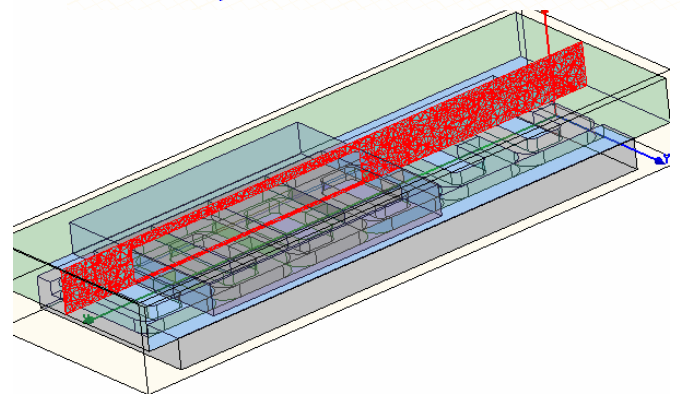
Using **Inside Selection / Length Based** to set the length of the maximum of element edge for each parts as below:

- windings: 4 mm;
- magnets: 2 mm;
- cores : 4 mm;
- Band : 3 mm

Please note for each time step the mesh of the air part in band will be recreated due to the **Inside Selection / Length Based** = 3 mm.



Band mesh in XZ cut-plane at some time step



Time=0.065s

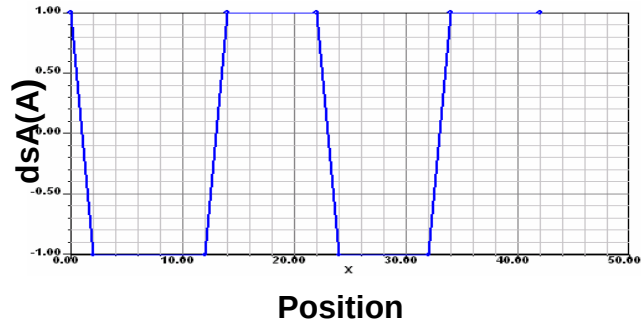
Mesh Operations Examples

EX3. Linear Motor (Transient) (con't)

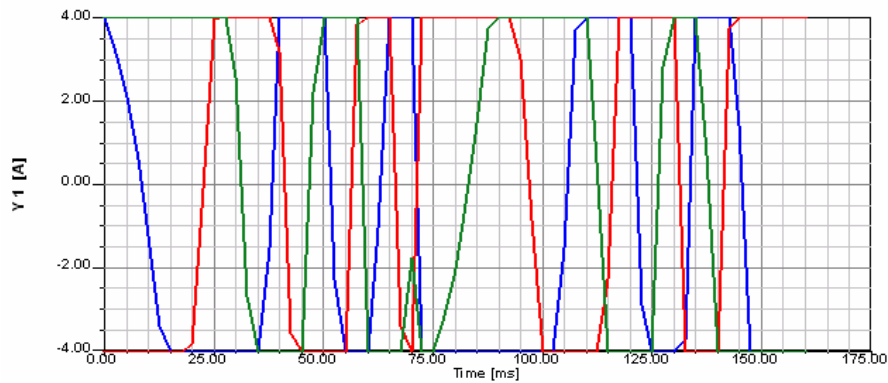
Excitation

The current excitation is set as a time function:

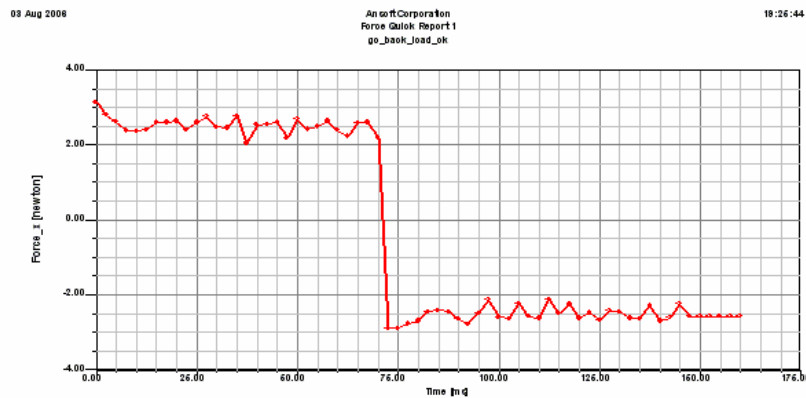
U Phase: $\text{if}((\text{Speed} > 0 \&\& \text{Position} \leq 41.5), 4 * \text{pwl\_periodic}(\text{dsA}, \text{Position}), -4 * \text{pwl\_periodic}(\text{dsA}, \text{Position}))$



Results



Transient current of 3 phase winding

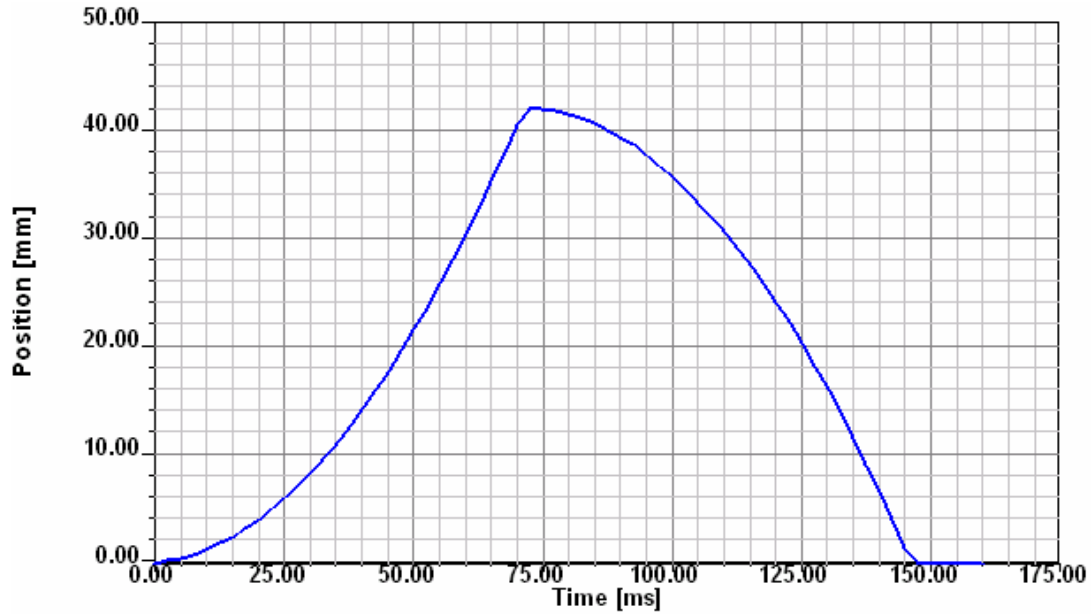


Force in X direction ($|f_x| > 2\text{N}$)

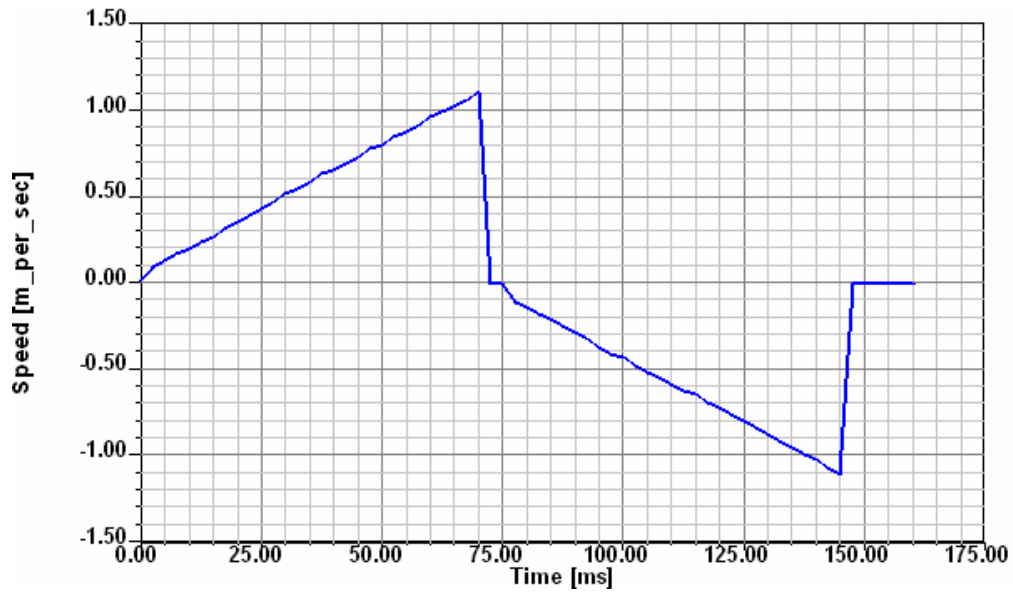
Mesh Operations Examples

EX3. Linear Motor (Transient) (con't)

Results



Position vs. Time



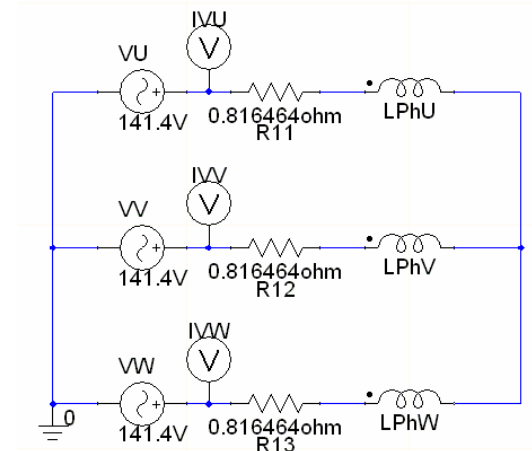
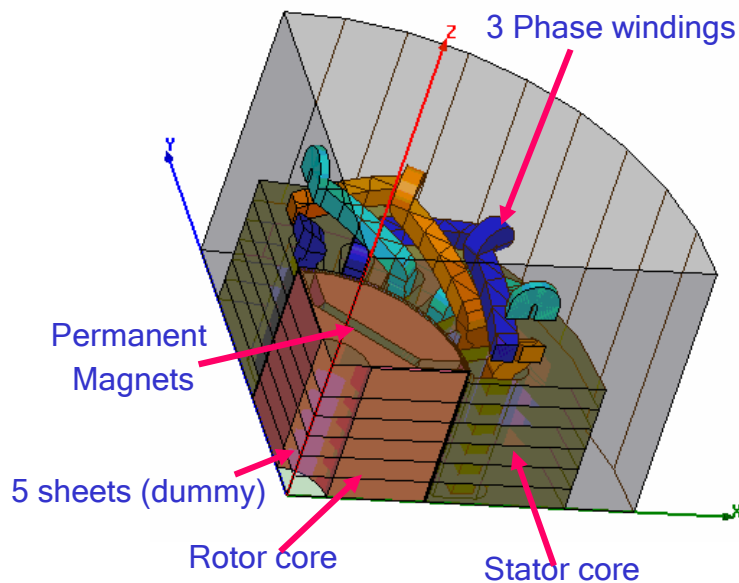
Speed vs. Time

Mesh Operations Examples

EX4. 1.5KW 4p24s IPM Synchronous Motor (Transient)

Model:

Please see the model as below:



Circuit model of 3 Phase Y connection

Mesh Operation

For making the mesh as 6 levels in motor, 5 dummy circular sheets are created. Using **Inside Selection / Length Based** to set the length of the maximum of element edge for windings: 5 mm;

For rotor and stator, we only do mesh operation for the surfaces along the gap and set **Surface Approximation** as below:

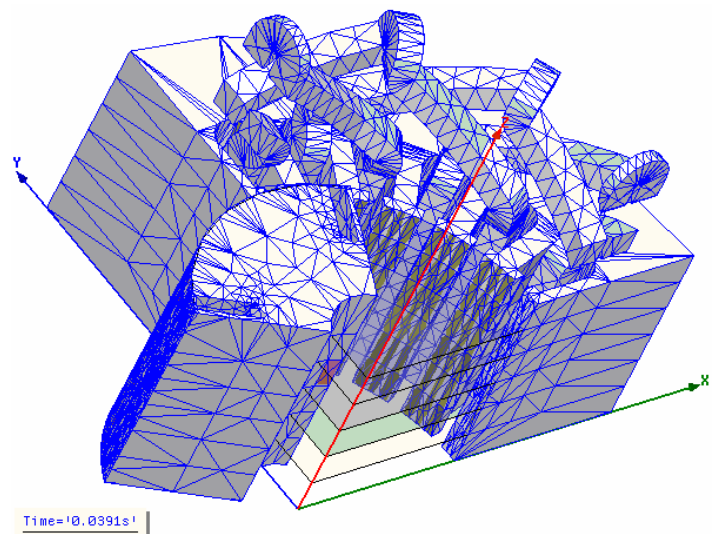
Set maximum surface deviation

(length): 2.5 mm;

Set maximum surface deviation

(angle): 3 deg

The mesh of the main parts will be shown in the right.

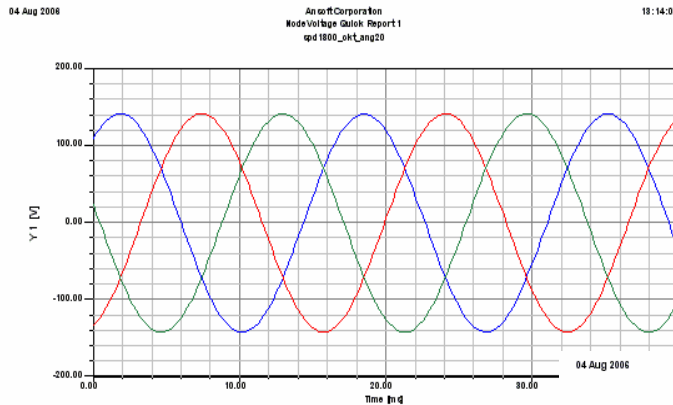


Mesh

Mesh Operations Examples

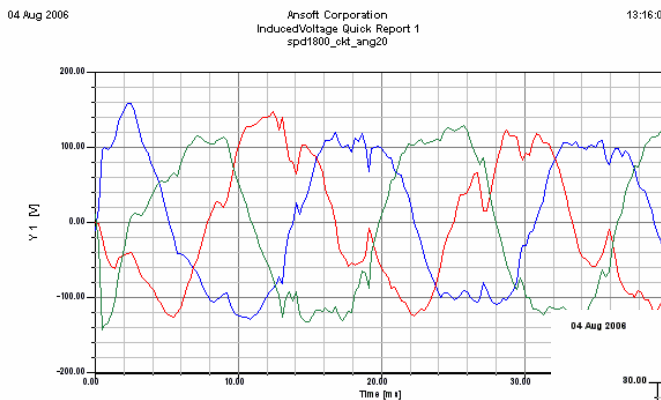
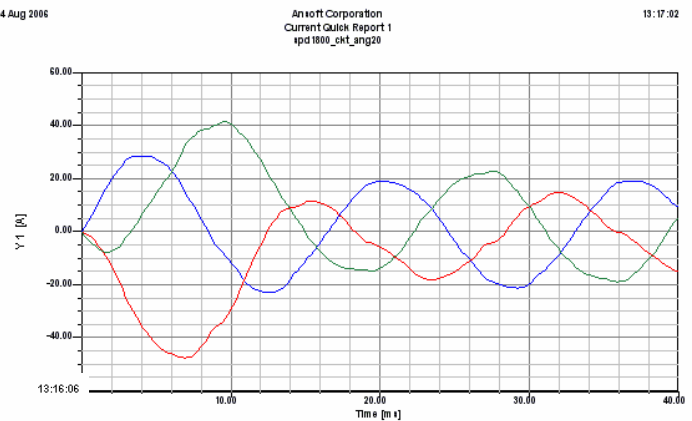
EX4. 1.5KW 4p24s IPM Synchronous Motor (Transient) (con't)

Results at Speed = 1800 rpm

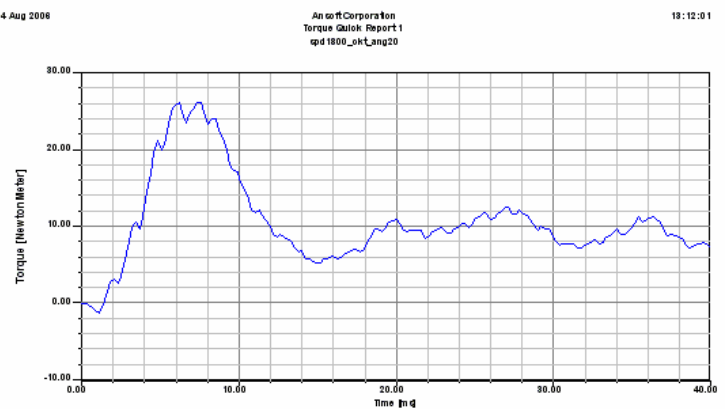


Input node voltages

Current of 3 phase



Induced Voltages



Output Torque

The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it.

Overview

- The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it. Ansoft Maxwell has very powerful and flexible data management and plotting capabilities. Once understood, it will make the whole solution process much easier, and will help craft the entire problem setup.

The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it.

Topics of Discussion

- The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it. 2D Plotting
- The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it. 3D Plotting
- The ANSOFT logo, featuring a stylized 'A' made of three red and grey triangles, with the word 'ANSOFT' in a bold, sans-serif font below it. Field Plotting

▀ Data Plotting

- ▀ Data plotting can take a variety of forms. The most often used format is 2D Cartesian plotting, but we also have the capability to plot in 3D as well. Below is a list of all the quantities that can be plotted on various graphs. For definitions of each of these quantities, see the online help or the respective solution type chapter.

- ▀ **Magnetostatic, Eddy Current, Electric**

- ▀ Force
- ▀ Torque
- ▀ Matrix

- ▀ **Transient**

- ▀ Motion
 - ▀ Position
 - ▀ Force/Torque
 - ▀ Velocity
- ▀ Winding
 - ▀ Flux Linkage
 - ▀ Induced Voltage
- ▀ Core Loss
- ▀ Power Loss
- ▀ Branch Current (external meter required)
- ▀ Node Voltage (external meter required)

- ▀ **Fields**

- ▀ Mag_H
- ▀ Mag_B
- ▀ Mag_J
- ▀ Energy Density
- ▀ Coenergy Density
- ▀ Apparent Energy Density
- ▀ Ohmic Loss Density
- ▀ Named Expressions

- ▀ **NOTE:** For all data plots of Fields data, a Line must be defined before creating the plot.

▲ Data Plotting (Continued)

▲ Types of Plots:

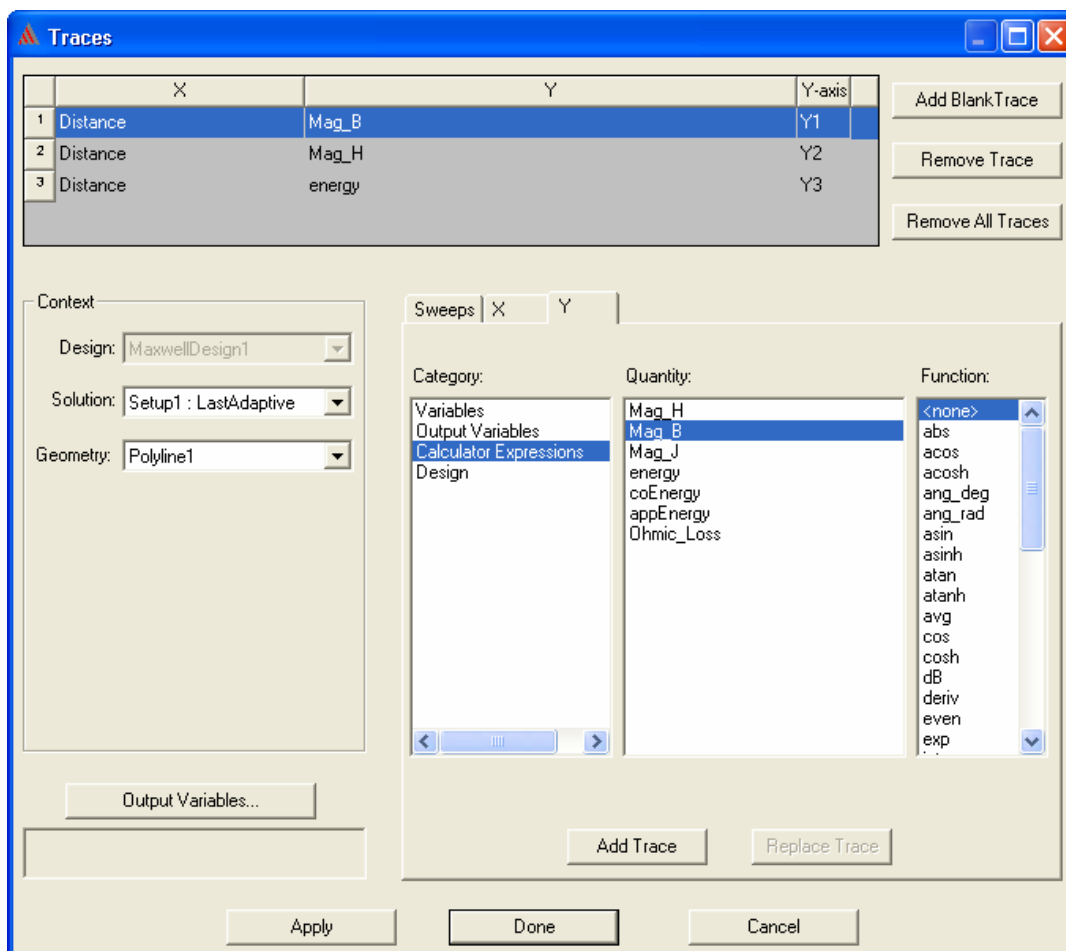
- ▲ Rectangular Plot
- ▲ Data Table
- ▲ 3D Rectangular Plot

▲ To Create a Plot:

1. Select **Maxwell > Results > Create Report**
2. Select Report Type and Display Type from the selections above
3. Click OK and the Traces Dialog will be displayed - we will go over the options in this dialog on the next page.

OR

1. Select **Maxwell > Results > Create Quick Report** (when available)
2. Choose Quick Report type and click OK.



Data Plotting (Continued)

Creating a Plot (continued)

4. Context

-  **Design** - choose from available designs within a project
-  **Solution** - choose from available setups or solution types
-  **Geometry** - if plotting Fields data, select an available Line from the list.

5. Sweep / X / Y Tabs

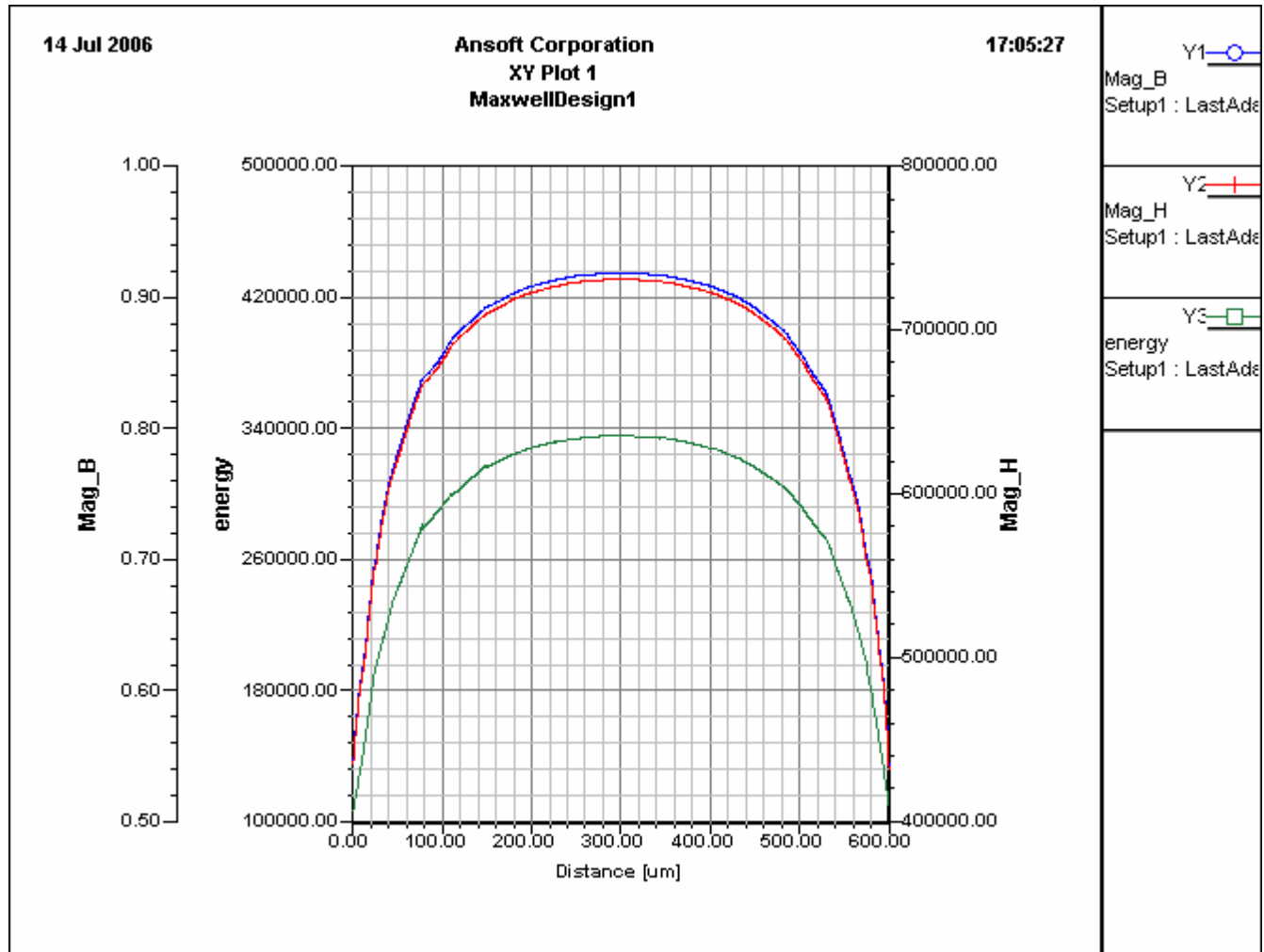
-  **Sweep** - controls the source of the independent variable in the plot.
 -  **NOTE:** By default, the Report editor selects **Use Current Design and Project variable values**. This will select the primary sweep of distance for a field data plot, or an undefined variable X for other plot types, and the current simulated values of the project variables
 -  To display a plot with multiple traces for different variable values, change this setting to **Sweep Design and Project variable values**. You can then change the primary sweep from X to a variable if desired. This is useful for looking at variations and families of curves. Simply select the value in the **Name** column next to **primary sweep**, and change the value to whatever you desire.
 -  You can also uncheck the **All Points** block to select which values of a variable are to be plotted.
 -  After adjusting the sweeps tab, select **Apply To All Selected Traces**.
-  **X** - controls any functional operator on the independent variable (this is usually set to **Use Primary Sweep** and defined in the Sweep tab).
-  **Y** - select the value to be plotted and any operator.

6. Select **Add Trace** for as many values as you would like to plot

7. Select **Done** when finished

-  An example of a multi-trace plot of the sweep tab shown on the previous page is shown next

Data Plotting (Continued)



THIS PAGE INTENTIONALLY LEFT BLANK

Examples of the Field Calculator in Maxwell3D v11

-  The Field Calculator can be used for a variety of tasks, however its primary use is to extend the post-processing capabilities within Maxwell beyond the calculation / plotting of the main field quantities. The Field Calculator makes it possible to operate with primary vector fields (such as H, B, J, etc) using vector algebra and calculus operations in a way that is both mathematically correct and meaningful from a Maxwell's equations perspective.
-  The Field Calculator can also operate with geometry quantities for three basic purposes:
 - plot field quantities (or derived quantities) onto geometric entities;
 - perform integration (line, surface, volume) of quantities over specified geometric entities;
 - export field results in a user specified box or at a user specified set of locations (points).
-  Another important feature of the (field) calculator is that it can be fully script driven. All operations that can be performed in the calculator have a corresponding “image” in one or more lines of VBscript code. Scripts are widely used for repetitive post-processing operations, for support purposes and in cases where Optimetrics is used and post-processing scripts provide some quantity required in the optimization / parameterization process.
-  This document describes the mechanics of the tools as well as the “softer” side of it as well. So, apart from describing the structure of the interface this document will show examples of how to use the calculator to perform many of the post-processing operations encountered in practical, day to day engineering activity using Maxwell. Examples are grouped according to the type of solution. Keep in mind that most of the examples can be easily transposed into similar operations performed with solutions of different physical nature.

Ansoft Maxwell Design Environment

-  The following features of the Ansoft Maxwell Design Environment are used to interact with the calculator as covered in this topic

- **Analysis**

-  Electrostatic
-  DC Conduction
-  Magnetostatic
-  Eddy Current
-  Transient

- **Results**

-  Output Variables
-  **Field Calculator**

- **Field Overlays:**

-  Named Expressions
-  Animate

Description of the interface

-  The interface is shown in Fig. I1. It is structured such that it contains a stack which holds the quantity of interest in stack registers. A number of operations are intended to allow the user to manipulate the contents of the stack or change the order of quantities being hold in stack registers. The description of the functionality of the stack manipulation buttons (and of the corresponding stack commands) is presented below:

- **Push** repeats the contents of the top stack register so that after the operation the two top lines contain identical information;
- **Pop** deletes the last entry from the stack (deletes the top of the stack);
- **RIDn** (roll down) is a “circular” move that makes the contents of the stacks slide down one line with the bottom of the stack advancing to the top;
- **RIUp** (roll up) is a “circular” move that makes the contents of the stacks slide up one line with the top of the stack dropping to the bottom;
- **Exch** (exchange) produces an exchange between the contents of the two top stack registers;
- **Clear** clears the entire contents of all stack registers;
- **Undo** reverses the result of the most recent operation.

-  The user should note that Undo operations could be nested up to the level where a basic quantity is obtained.

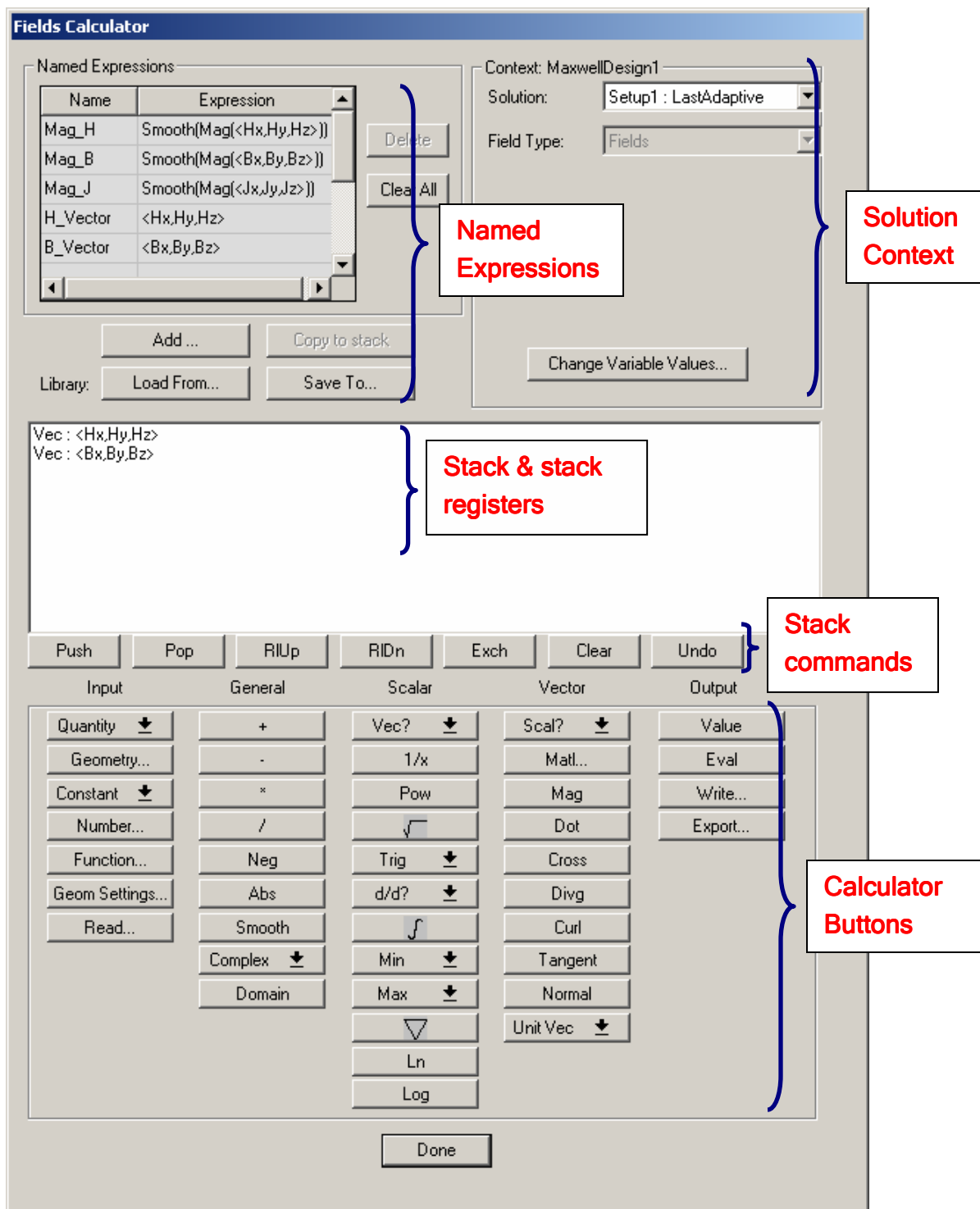


Fig. I1 Field Calculator Interface

- ▲ The calculator buttons are organized in five categories as follows:
- **Input** contains calculator buttons that allow the user to enter data in the stack; sub-categories contain solution vector fields (B, H, J, etc.), geometry(point, line, surface, volume, coordinate system), scalar, vector or complex constants (depending on application) or even entire f.e.m. solutions.
 - **General** contains general calculator operations that can be performed with “general” data (scalar, vector or complex), if the operation makes sense; for example if the top two entries on the stack are two vectors, one can perform the addition (+) but not multiplication (*); indeed, with vectors one can perform a dot product or a cross product but not a multiplication as it is possible with scalars.
 - **Scalar** contains operations that can be performed on scalars; example of scalars are scalar constants, scalar fields, mathematical operations performed on vector which result in a scalar, components of vector fields (such as the X component of a vector field), etc.
 - **Vector** contains operations that can be performed on vectors only; example of such operations are cross product (of two vectors), div, curl, etc.
 - **Output** contains operations resulting in plots (2D / 3D), graphs, data export, data evaluation, etc.
- ▲ As a rule, calculator operations are allowed if they make sense from a mathematical point of view. There are situations however where the contents of the top stack registers should be in a certain order for the operation to produce the expected result. The examples that follow will indicate the steps to be followed in order to obtain the desired result in a number of frequently encountered operations. The examples are grouped according to the type of solution (solver) used. They are typical medium/higher level post-processing task that can be encountered in current engineering practice. Throughout this manual it is assumed that the user has the basic skills of using the Field Calculator for basic operations as explained in the on-line technical documentation and/or during Ansoft basic training.
- ▲ Note: The f.e.m. solution is always performed in the global (fixed) coordinate system. The plots of vector quantities are therefore related to the global coordinate system and will not change if a local coordinate system is defined with a different orientation from the global coordinate system.

Electrostatic Examples

Example ES1: Calculate the charge density distribution and total electric charge on the surface of an object

Description: Assume an electrostatic (3D) application with separate metallic objects having applied voltages or floating voltages. The task is to calculate the total electric charge on any of the objects.

Calculate/plot the charge density distribution on the object; the sequence of calculator operations is described below:

Quantity -> D (load D vector into the calculator);

Geometry... -> Surface... (*select the surface of interest*) -> **OK**

Unit Vec -> Normal (creates the normal unit vector corresponding to the surface of interest)

Dot (creates the dot product between D and the unit normal vector to the surface of interest, equal to the surface charge density)

Add ... (*input "charge_density" as the name*) -> **OK** (creates a named expression and adds it to the list)

Done (leaves calculator)

(*select the surface of interest from the model*)

Maxwell -> Fields -> Fields -> Named Expressions (a Selecting calculated expression window appears)

(*select "charge_density" from the list*) -> **OK** (A Create Field Plot window appears)

Done

Calculate the total electric charge on the surface of an object

Quantity -> D (load D vector into the calculator);

Geometry... -> Surface... (*select the surface of interest*) -> **OK**

Normal

$\int$

Eval

Example ES2: Calculate the Maxwell stress distribution on the surface of an object

Description: Assume an electrostatic application (for ex. a parallel plate capacitor structure). The surface of interest and adjacent region should have a fine finite element mesh since the Maxwell stress method for calculation the force is quite sensitive to mesh.

The Maxwell electric stress vector has the following expression for objects without electrostrictive effects:

$$T_{nE} = (\vec{D} \cdot \vec{n})\vec{E} - \frac{1}{2}\vec{n}\epsilon E^2$$

where the unit vector $\vec{n}$ is the normal vector to the surface of interest. The sequence of calculator commands necessary to implement the above formula is given below.

Quantity -> D

Geometry... -> Surface... (*select the surface of interest*) -> OK

Unit Vec -> Normal (creates the normal unit vector corresponding to the surface of interest)

Dot

Quantity -> E

***** (multiply)

Geometry... -> Surface... (*select the surface of interest*) -> OK

Unit Vec -> Normal (creates the normal unit vector corresponding to the surface of interest)

Number... -> Scalar (0.5) OK

Constant -> Epsi0

Quantity -> E

Push

Dot

- (minus)

Continued on next page.

Example ES2: Continued

- ▲ **Add ...** (input “stress” as the name) -> **OK** (creates a named expression and adds it to the list)
- ▲ **Done** (leaves calculator)
- ▲ (select the surface of interest from the model)
- ▲ **Maxwell -> Fields -> Fields -> Named Expressions** (a Selecting calculated expression window appears)
- ▲ (select “stress” from the list) -> **OK** (A Create Field Plot window appears)
- ▲ **Done**

- ▲ If an integration of the Maxwell stress is to be performed over the surface of interest, then use the Named Expression in the following calculator sequence:

- ▲ (select “stress” from the Named Expressions list)
- ▲ **Copy to stack** (inserts the named expression to the stack)
- ▲ **Geometry... -> Surface...** (select surface of interest) -> **OK**
- ▲ **Normal**
- ▲ $\int$
- ▲ **Eval**

- ▲ *Note:* The surface in all the above calculator commands should lie in free space or should coincide with the surface of an object surrounded by free space (vacuum, air). It should also be noted that the above calculations hold true in general for any instance where a volume distribution of force density is equivalent to a surface distribution of stress (tension):

$$\vec{F} = \int_{v\Sigma} \vec{f} dv = \oint_{\Sigma} \vec{T}_n dS$$

where T_n is the local tension force acting along the normal direction to the surface and F is the total force acting on object(s) inside Σ .

- ▲ The above results for the electrostatic case hold for magnetostatic applications if the electric field quantities are replaced with corresponding magnetic quantities.

Current flow Examples

Example CF1: Calculate the resistance of a conduction path between two terminals

Description: Assume a given conductor geometry that extends between two terminals with applied DC currents.

In DC applications (static current flow) one frequent question is related to the calculation of the resistance when one has the field solution to the conduction (current flow) problem. The formula for the analytical calculation of the DC resistance is:

$$R_{DC} = \int_C \frac{ds}{\sigma(s) \cdot A(s)}$$

where the integral is calculated along curve C (between the terminals) coinciding with the “axis” of the conductor. Note that both conductivity and cross section area are in general function of point (location along C). The above formula is not easily implementable in the general case in the field calculator so that alternative methods to calculate the resistance must be found.

One possible way is to calculate the resistance using the power loss in the respective conductor due to a known conduction current passing through the conductor.

$$R_{DC} = \frac{P}{I_{DC}^2} \quad \text{where power loss is given by } P = \int_V \vec{E} \cdot \vec{J} dV = \int_V \frac{\vec{J}}{\sigma} \cdot \vec{J} dV$$

The sequence of calculator commands to compute the power loss P is given below:

Quantity -> J

Push

Number... -> Scalar (1e7) OK (conductivity assumed to be 1e7 S/m)

/ (divide)

Dot

Geometry... -> Volume... (select the volume of interest) -> OK

∫

Eval

The resistance can now be easily calculated from power and the square of the current.

Continued on next page.

- There is another way to calculate the resistance which makes use of the well known Ohm's law.

$$R_{DC} = \frac{U}{I}$$

- Assuming that the conductor is bounded by two terminals, T1 and T2 (current through T1 and T2 must be the same), the resistance of the conductor (between T1 and T2) is given the ratio of the voltage differential U between T1 and T2 and the respective current, I . So it is necessary to define two points on the respective terminals and then calculate the voltage at the two locations (voltage is called Phi in the field calculator). The rest is simple as described above.

Example CF2: Export the field solution to a uniform grid

- Description:* Assume a conduction problem solved. It is desired to export the field solution at locations belonging to a uniform grid to an ASCII file.
- The field calculator allows the field solutions to be exported regardless of the nature of the solution or the type of solver used to obtain the solution. It is possible to export any quantity that can be evaluated in the field calculator. Depending on the nature of the data being exported (scalar, vector, complex), the structure of each line in the output file is going to be different. However, regardless of what data is being exported, each line in the data section of the output file contains the coordinates of the point (x, y, z) followed by the data being exported (1 value for a scalar quantity, 2 values for a complex quantity, 3 values for a vector in 3D, 6 values for a complex vector in 3D)
- Continued on next page.

- ▲ To export the current density vector to a grid the field calculator steps are:
- ▲ **Quantity J**
- ▲ **Export... -> On Grid** (then fill in the data as appropriate, see Fig. CF2)
- ▲ **OK**

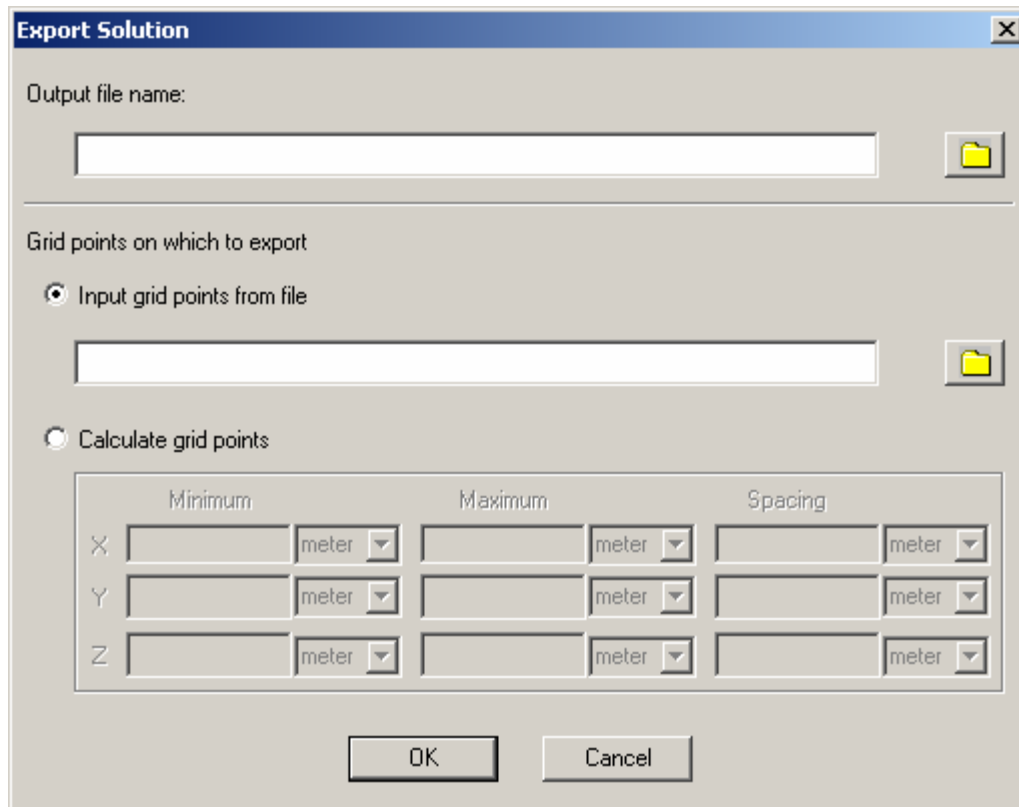


Fig. CF2 Define the size of the export region (box) and spacing within

- ▲ Select Calculate grid points and define minimum, maximum & spacing in all 3 directions X, Y, Z as the size of the rectangular export region (box) and the spacing between locations. By default the location of the ASCII file containing the export data is in the project directory. Clicking on the browse symbol one can also choose another location for the exported file.
- ▲ *Note:* One can export the quantity calculated with the field calculator at user specified locations by selecting Input grid points from file. In that case the ASCII file containing on each line the x, y and z coordinates of the locations must exist prior to initiating the export-to-file command.

Example CF3: Calculate the conduction current in a branch of a complex conduction path

- Description:* There are situations where the current splits along the conduction path. If the nature of the problem is such that symmetry considerations cannot be applied, it may be necessary to evaluate total current in 2 or more parallel branches after the split point. To be able to perform the calculation described above, it is necessary to have each parallel branch (where the current is to be calculated) modeled as a separate solid.
- Before the calculation process is started, make sure that the (local) coordinate system is placed somewhere along the branch where the current is calculated, preferably in a median location along that branch. In more general terms, that location is where the integration is performed and it is advisable to choose it far from areas where the current splits or changes direction, if possible.
- Here is the process to be followed to perform the calculation using the field calculator.

Quantity -> J

Geometry... -> Volume... (choose the volume of the branch of interest) **OK**

Domain (this is to limit the subsequent calculations to the branch of interest only)

Geometry... -> Surface... yz (choose axis plane that cuts perpendicular to the branch) **OK**

Normal

∫

Eval

- The result of the evaluation is positive or negative depending on the general orientation of the J vector versus the normal of the integration surface (S). In mathematical terms the operation performed above can be expressed as:

$$I = \int_S \vec{J} \cdot \vec{n} dS$$

- Note:* The integration surface (yz, in the example above) extends through the whole region, however because of the “domain” command used previously, the calculation is restricted only to the specified solid (that is the S surface is the intersection between the specified solid and the integration plane).

▲ Magnetostatic examples

▲ **Example MS1: Calculate (check) the current in a conductor using Ampere's theorem**

- ▲ *Description:* Assume a magnetostatic problem where the magnetic field is produced by a given distribution of currents in conductors. To calculate the current in the conductor using Ampere's theorem, a closed polyline (of arbitrary shape) should be drawn around the respective conductor. In a mathematical form the Ampere's theorem is given by:

$$I_{S\Gamma} = \oint_{\Gamma} \vec{H} \cdot \vec{ds}$$

where Γ is the closed contour (polyline) and $S\Gamma$ is an open surface bounded by Γ but otherwise of arbitrary shape. $I_{S\Gamma}$ is the total current intercepting the surface $S\Gamma$.

- ▲ To calculate the (closed) line integral of H, the sequence of field calculator commands is:

▲ **Quantity -> H**

▲ **Geometry... -> Line** (*choose the closed polygonal line around the conductor*) **OK**

▲ **Tangent**

▲ $\int$

▲ **Eval**

- ▲ The value should be reasonably close to the value of the corresponding current. The match between the two can be used as a measure of the global accuracy of the calculation in the general region where the closed line was placed.

Example MS2: Calculate the magnetic flux through a surface

Description: Assume the case of a magnetostatic application. To calculate the magnetic flux through an already existing surface the sequence of calculator commands is:

Quantity -> B

Geometry... -> Surface... (*specify the integration surface*) OK

Normal

$\int$

Eval

The result is positive or negative depending on the orientation of the B vector with respect to the normal to the surface of integration.

The above operation corresponds to the following mathematical formula for the magnetic flux:

$$\Phi_s = \int_s \vec{B} \cdot \vec{n} dA$$

Example MS3: Calculate components of the Lorentz force

Description: Assume a distribution of magnetic field surrounding conductors with applied DC currents. The calculation of the components of the Lorentz force has the following steps in the field calculator.

Quantity -> J

Quantity -> B

Cross

Scalar -> ScalarX

Geom -> Volume ... (*specify the volume of interest*) OK

$\int$

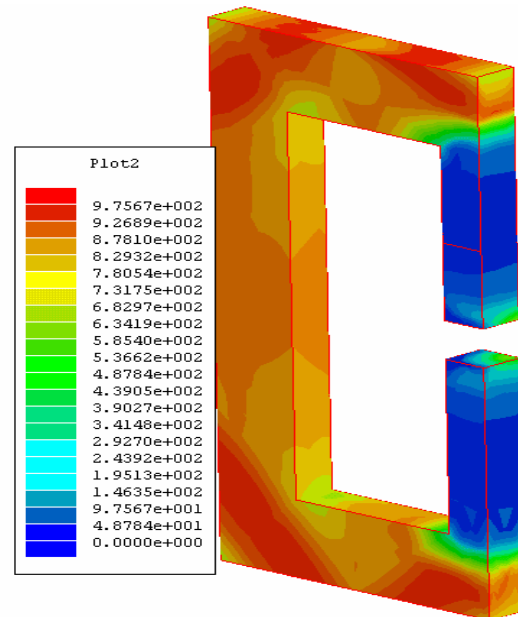
Eval

The above example shows the process for calculating the X component of the Lorentz force. Similar steps should be performed for all components of interest.

Example MS4: Calculate the distribution of relative permeability in nonlinear material

Description: Assume a non-linear magnetostatic problem. To plot the relative permeability distribution inside a non-linear material the following steps should be taken:

- Quantity -> B
- Scal? -> ScalarX
- Quantity -> H
- Scal? -> ScalarX
- Constant -> μ_0
- * (multiply)
- / (divide)
- Smooth
- Add ... (input "rel_perm" as the name)
- OK (creates a named expression and adds it to the list)
- Done (leaves calculator)



- (select the surface of interest from the model)
- Maxwell -> Fields -> Fields -> Named Expressions (a Selecting calculated expression window appears)
- (select "rel_perm" from the list) -> OK (a Create Field Plot window appears)
- Done

Note: The above sequence of commands makes use of one single field component (X component). Please note that any spatial component can be used for the purpose of calculating relative permeability in isotropic, non-linear soft magnetic materials. The result would still be the same if we used the Y component or the Z component. However, this does not apply for anisotropic materials. The "smoothing" also used in the sequence is also recommended particularly in cases where the mesh density is not very high.

Frequency domain (AC) Examples

Example AC1: Calculate the radiation resistance of a circular loop

Description: Assume a circular loop of radius 0.02 m with an applied current excitation at 1.5 GHz;

The radiation resistance is given by the following formula:

$$R_r = \frac{P_{av}}{I_{rms}^2} \quad P_{av} = \frac{1}{2} \oint_S \text{Re}(\vec{E} \times \vec{H}^*) dS = \frac{1}{2} \oint_S \text{Re} \left[\frac{1}{j\omega\epsilon_0} (\nabla \times \vec{H}) \times \vec{H}^* \right] dS$$

where S is the outer surface of the region (preferably spherical), placed conveniently far away from the source of radiation.

Assuming that a half symmetry model is used, no ½ is needed in the above formula. The sequence of calculator commands necessary for the calculation of the average power is as follows:

- Quantity -> H
- Curl
- Number... -> Complex (0 , -12) OK
- *
- Quantity -> H
- Complex -> Conj
- Cross
- Complex -> Real
- Geometry... -> Surface... (select the surface of interest) -> OK
- Normal
- ∫
- Eval

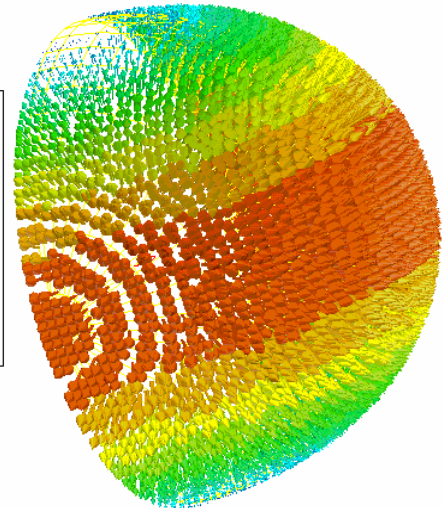
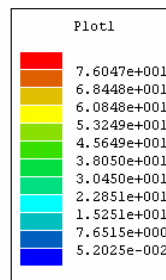
Note: The integration surface above must be an open surface (radiation surface) if a symmetry model is used. Surfaces of existing objects cannot be used since they are always closed. Therefore the necessary integration surface must be created in the example above using 3DModeler > Surface > Create Object From Face command.

Example AC2: Calculate/Plot the Poynting vector

Description: Same as in Example AC1.

To obtain the Poynting vector the following sequence of calculator commands is necessary:

- Quantity -> H
- Curl
- Number... -> Complex (0 , -12) OK
- *
- Quantity -> H
- Complex -> Conj
- Cross



To plot the real part of the Poynting vector the following commands should be added to the above sequence:

- Complex -> Real
- Number... -> Scalar (0.5) OK
- *
- Add ... (input "Poynting" as the name) -> OK (creates a named expression and adds it to the list)
- Done (leaves calculator)
- (select the surface of interest from the model)
- Maxwell -> Fields -> Fields -> Named Expressions (a Selecting calculated expression window appears)
- (select "Poynting" from the list) -> OK (a Create Field Plot window appears)
- Done

Example AC3: Calculate total induced current in a solid

Description: Consider (as example) the device in Fig. AC3.

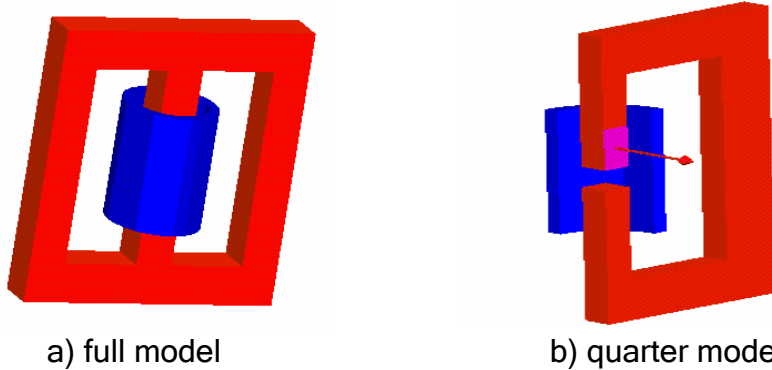


Fig. AC3 Geometry of inductor model

Assume that the induced current through the surface marked with an arrow in the quarter model is to be calculated. Please note that there is an expected net current flow through the marked surface, due to the symmetry of the problem. As a general recommendation, the surface that is going to be used in the process of integrating the current density should exist prior to solving the problem. In some cases this also means that the geometry needs to be created in such a way so that the particular post-processing task is made possible. Once the object containing the integration surface exists, use the *3DModeler > Surface > Create Object From Face* command to create the integration surface necessary for the calculation. Make sure that the object with expected induced currents has non-zero conductivity and that the eddy-effect calculation was turned on.

Assuming now that all of the above was taken care of, the sequence of calculator commands necessary to obtain separately the real part and the imaginary part of the induced current is described on the next page:

▲ For the real part of the induced current:

▲ **Quantity -> J**

▲ **Complex -> Real**

▲ **Geometry... -> Surface -> *(select the previously defined integration surface)* OK**

▲ **Normal**

▲ $\int$

▲ **Eval**

▲ For the imaginary part of the induced current:

▲ **Quantity -> J**

▲ **Complex -> Imag**

▲ **Geometry... -> Surface -> *(select the previously defined integration surface)* OK**

▲ **Normal**

▲ $\int$

▲ **Eval**

▲ If instead of getting the real and imaginary part of the current , one desires to do an “at phase” calculation, the sequence of commands is:

▲ **Quantity -> J**

▲ **Number... -> Scalar (45) OK** (assuming a calculation at 45 degrees phase angle)

▲ **Complex -> AtPhase**

▲ **Geometry... -> Surface -> *(select the previously defined integration surface)* OK**

▲ **Normal**

▲ $\int$

▲ **Eval**

▲ **Example AC4: Calculate (ohmic) voltage drop along a conductive path**

▲ Description: Assume the existence of a conductive path (a previously defined open line totally contained inside a conductor). To calculate the real and imaginary components of the ohmic voltage drop inside the conductor the following steps should be followed:

▲ For the real part of the voltage:

▲ Quantity -> J

▲ Matl... -> Conductivity -> Divide OK

▲ Complex -> Real

▲ Geometry... -> Line -> *(select the applicable line)* OK

▲ Tangent

▲ $\int$

▲ Eval

▲ For the imaginary part of the voltage:

▲ Quantity -> J

▲ Matl... -> Conductivity -> Divide OK

▲ Complex -> Imag

▲ Geometry... -> Line -> *(select the applicable line)* OK

▲ Tangent

▲ $\int$

▲ Eval

▲ To calculate the phase of the voltage manipulate the contents of the stack so that the top register contains the real part of the voltage and the second register of the stack contains the imaginary part. To calculate phase enter the following command:

▲ Trig -> Atan2

Example AC5: Calculate the AC resistance of a conductor

-  *Description:* Consider the existence of an AC application containing conductors with significant skin effect. Assume also that the mesh density is appropriate for the task, i.e. mesh has a layered structure with 1-2 layers per skin depth for 3-4 skin depths if the conductor allows it. Here is the sequence to follow in order to calculate the total power dissipated in the conductor of interest.

 **Quantity -> OhmicLoss**

 **Geometry... -> Volume -> (*specify volume of interest*) OK**

 **Eval**

 $\int$

-  *Note:* To obtain the AC resistance the power obtained above must be divided by the squared rms value of the current applied to the conductor. Note that in the Boundary/Source Manager peak values are entered for sources, not rms values.

Time Domain Examples

Example TD1: Plotting Transient Data

-  *Description:* Assume a time domain (transient application) requiring the display of induced current as a function of time. Do this before solving if fields are not saved at every step, or any time if fields are available.

 **Quantity -> J**

 **Geometry...** -> **Surface** -> *(enter the surface of interest)* **OK**

 **Normal**

 $\int$

 **Add ...** *(input "Induced_Current" as the name)* -> **OK** (creates a named expression and adds it to the list)

 **Done** (leaves calculator)

 **Maxwell -> Results -> Output Variables...** (an Output Variables Dialogue appears)

 *(select Fields from the Report Type pull down menu)*

 *(select Calculator Expressions as the Category and "Induced_Current" as the Quantity)*

 **Insert Quantity Into Expression**

 *(name your expression by writing "Ind_Cur" in the Name textbox)*

 **Add** (adds an output variable and its expression to the list)

 **Done**

-  This Output Variable can be accessed two ways. First, it can be accessed directly in the reports - make sure that the Reports Type is set to Fields (not Transient). Second, it can be included in the Solve setup under the Output Variables tab (which also makes it available in the reports with the Reports Type set to Transient).

▲ **Example TD2: Find the maximum/minimum field value/location**

▲ *Description:* Consider a solved transient application. To find extreme field values in a given volume and/or the respective locations follow these steps.

▲ To get the value of the maximum magnetic flux density in a given volume:

▲ **Quantity -> B**

▲ **Mag**

▲ **Geometry... -> Volume -> (enter volume of interest) OK**

▲ **Max -> Value...**

▲ **Eval**

▲ To get the location of the maximum:

▲ **Quantity -> B**

▲ **Mag**

▲ **Geometry... -> Volume -> (enter volume of interest) OK**

▲ **Max -> Position...**

▲ **Eval**

▲ The process is very similar when searching for the minimum. Just replace the Max with Min in the above sequences.

▲ **Example TD3: Combine (by summation) the solutions from two time steps**

▲ *Description:* Assume a linear model transient application. It is possible to add the solution from different time steps if you follow these steps:

▲ First, set the solution context to a certain time step, say t1 by selecting **View > Set Solution Context...** and choosing an appropriate time from the list.

▲ Then in the calculator:

▲ **Quantity -> B**

▲ **Write** (*enter the name of the file*) **OK**

▲ Exit the calculator and choose a different time step, say t2.

▲ Then, in the calculator:

▲ **Quantity -> B**

▲ **Read** (*specify the name of the .reg file to be read in*) **OK**

▲ **+** (add)

▲ **Mag**

▲ **Geometry... -> Surface ->** (*enter surface of interest*) **OK**

▲ *Note 1:* For this operation to succeed it is necessary that the respective meshes are identical. This condition is of course satisfied in transient applications without motion since they do not have adaptive meshing. It should be noted that this capability can be used in other solutions sequences -say static- if the meshes in the two models are identical.

▲ The whole operation is numeric entirely, therefore the nature of the quantities being “combined” is not checked from a physical significance point of view. It is possible to add for example an H vector solution to a B vector solution. This doesn’t have of course any physical significance, so the user is responsible for the physical significance of the operation.

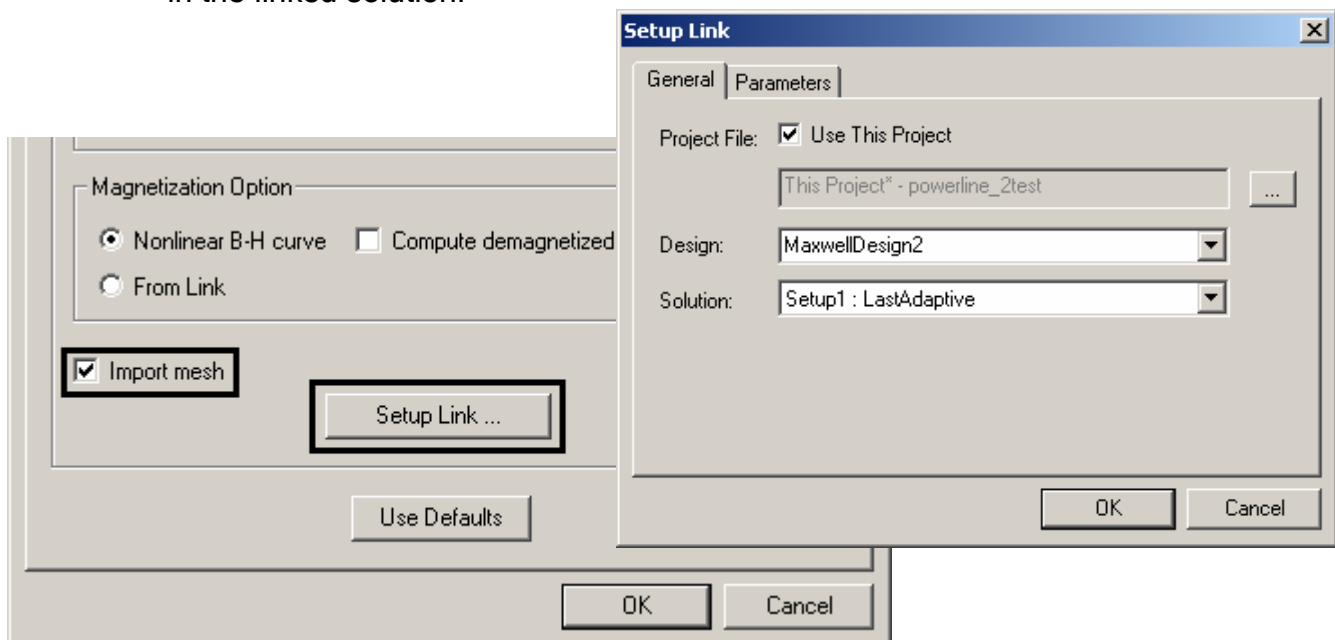
▲ For the particular case of time domain applications it is possible to study the “displacement” of the (vector) solution from one time step to another, study the spatial orthogonality of two solution, etc. It is a very powerful capability that can be used in many interesting ways.

Example TD3: Continued

Note 2: As another example of using this capability please consider another typical application: power flow in a given device.

As example one can consider the case of a cylindrical conductor above the ground plane with 1 Amp current, the voltage with respect to the ground being 1000 V. As well known, one can solve separately the magnetostatic problem (in which case the voltage is of no consequence, and only magnetic fields are calculated) and the electrostatic problem (in which case only the electric fields are calculated). With Maxwell it is possible to “combine” the two results in the post-processing phase if the assumption that the electric and magnetic fields are totally separated and do not influence each other. One possible reason that such an operation is meaningful from a physical point of view might be the need for an analysis of power flow.

Assume that a magnetostatic and electrostatic problem are created with identical geometries. Link the mesh from one simulation to the mesh of the other, so that they will be identical - this can be accomplished in the Setup tab of the Analysis Setup properties. Select the Import mesh box, and clicking on the Setup Link ... button. Then specify the target Design and Solution in the Setup Link dialog. To assure that the mesh is identical in both the linked solution and the target solution, make sure to set the Maximum Number of Passes to one (1) in the linked solution.



Example TD3: Continued

- Solve both these models and access the electrostatic results. Export the electric field solution as follows:

Quantity -> E

Write... *(enter the name of the .reg file to contain the solution)* OK

- Access now the solution of the magnetostatic problem and perform the following operations with the calculator after placing the coordinate system in the median plane of the conductor (yz plane if the conductor is oriented along x axis):

Read... *(specify the name of the file containing the electrostatic E field)* OK

Quantity -> H

Cross

Geometry... -> Volume -> background OK

Domain

Geometry... -> Surface -> yz

Normal

$\int$
Eval

- A result around 1000 W should be obtained, corresponding to 1000 W of power being transferred along the wire but NOT THROUGH THE WIRE! Indeed the power is transmitted through the air around the wire (the Poynting vector has higher values closer to the wire and decays in a radial direction). The wire here only has the role of GUIDING the power transfer! The wire absorbs from the electromagnetic field only the power corresponding to the conduction losses in the wire.

- When the integration of the Poynting vector was performed above, a domain operation was also performed limiting the result to the background only. This shows clearly that the distribution of the Poynting vector in the background is responsible for the power transfer. Displaying the Poynting vector in different transversal planes to the wire shows also the direction of the power transfer.

- This type of analysis can be very useful in studying the power transfer in complex devices.

▲ **Example TD4: Create an animation from saved field solutions**

▲ *Description:* Assume a solved time domain application. To create an animation file of a certain field quantity extracted from the saved field solution (say magnitude of conduction current density J) proceed as described below.

▲ *(select surface of interest)*

▲ **Maxwell -> Fields -> Fields -> J -> Mag_J**

▲ **Done**

▲ **Maxwell -> Fields -> Animate ...** (a Setup Animation dialogue appears)

▲ *(select desired time steps from those available in the list)*

▲ **Done**

Miscellaneous Examples

Example M1: Calculation of volumes and areas

 To calculate the volume of an object here is the sequence of calculator commands:

 **Number... -> Scalar (1) OK** (enters the scalar value of 1)

 **Geometry... -> Volume...** (*enter the volume of interest*) **OK**

 $\int$

 **Eval**

 The result is expressed in m³.

 To calculate the area of a surface here is the sequence of calculator commands:

 **- Number... -> Scalar (1) OK** (enters the scalar value of 1)

 **- Geometry... -> Surface...** (*enter the surface of interest*) **OK**

 $\int$

 **Eval**

 The result is expressed in m².

Chapter 5.0 - Magnetostatic Examples

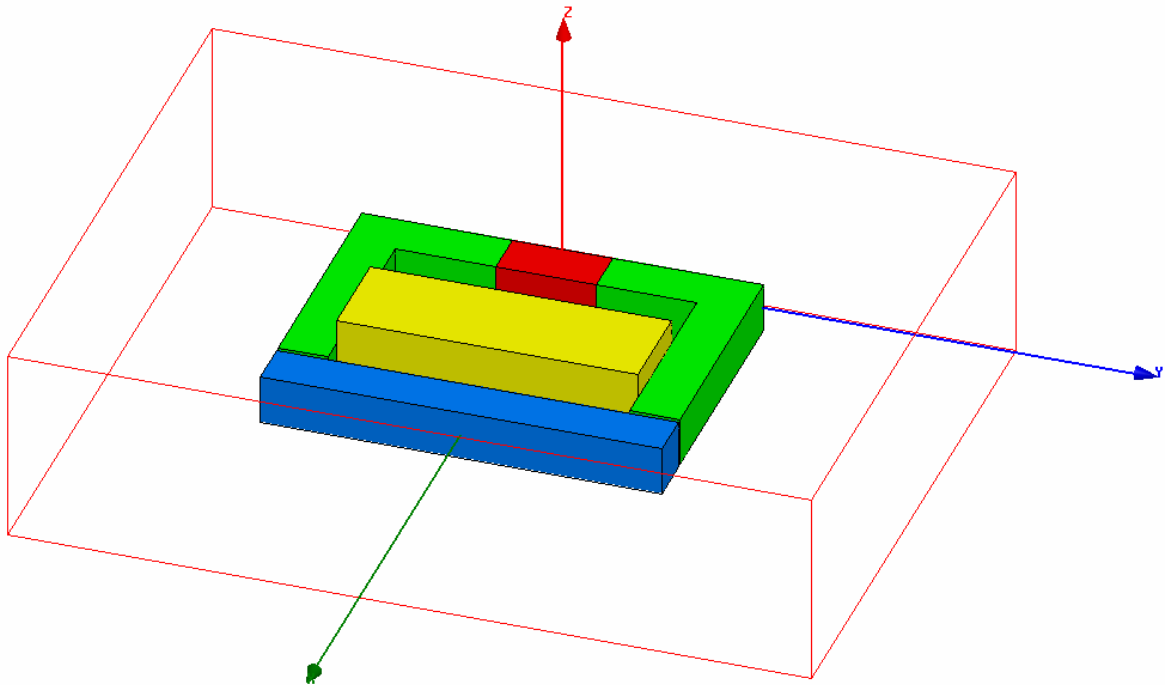
-  5.1 - Magnetic Force
-  5.2 - Inductance Calculation
-  5.3 -Stranded Conductors
-  5.4 - Equivalent Circuit Extraction (ECE) Linear Movement
-  5.5 - Anisotropic Materials
-  5.6 - Symmetry Boundaries
-  5.7 - Permanent Magnet Magnetization
-  5.8 - Master/Slave boundaries

THIS PAGE INTENTIONALLY LEFT BLANK

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](#).



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**

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**

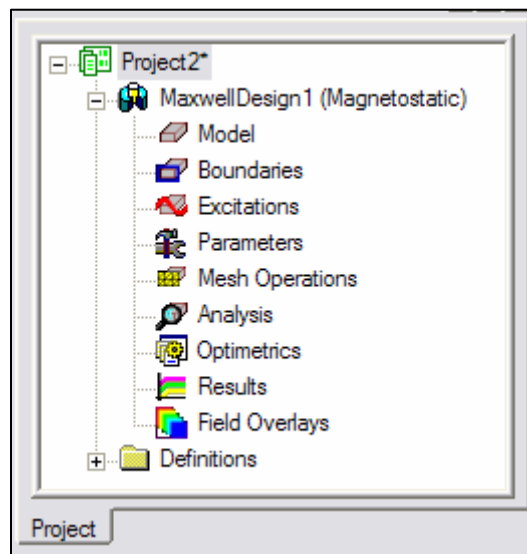
3. Click the **OK** button

Example (Magnetostatic) - Magnetic Force

Opening a New Project

To open a new project:

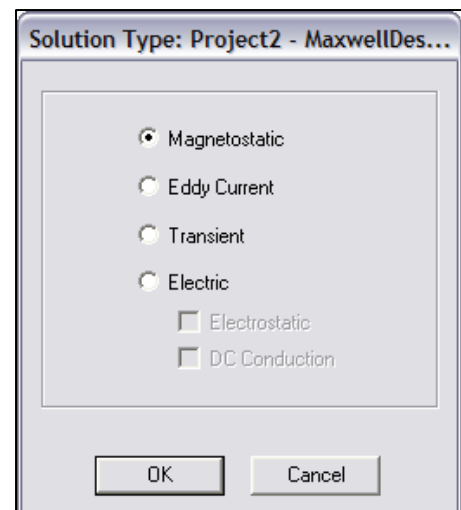
- ▲ In an Ansoft Maxwell window, click the  On the Standard toolbar, or select the menu item **File > New**.
- ▲ From the **Project** menu, select **Insert Maxwell Design**.



Set Solution Type

To set the solution type:

- ▲ Select the menu item **Maxwell > Solution Type**
- ▲ Solution Type Window:
 - ▲ Choose **Magnetostatic**
 - ▲ Click the **OK** button



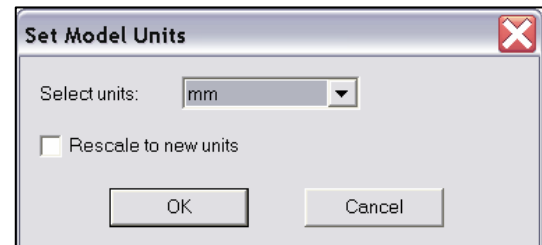
Example (Magnetostatic) - Magnetic Force

Creating the 3D Model

Set Model Units

To set the units:

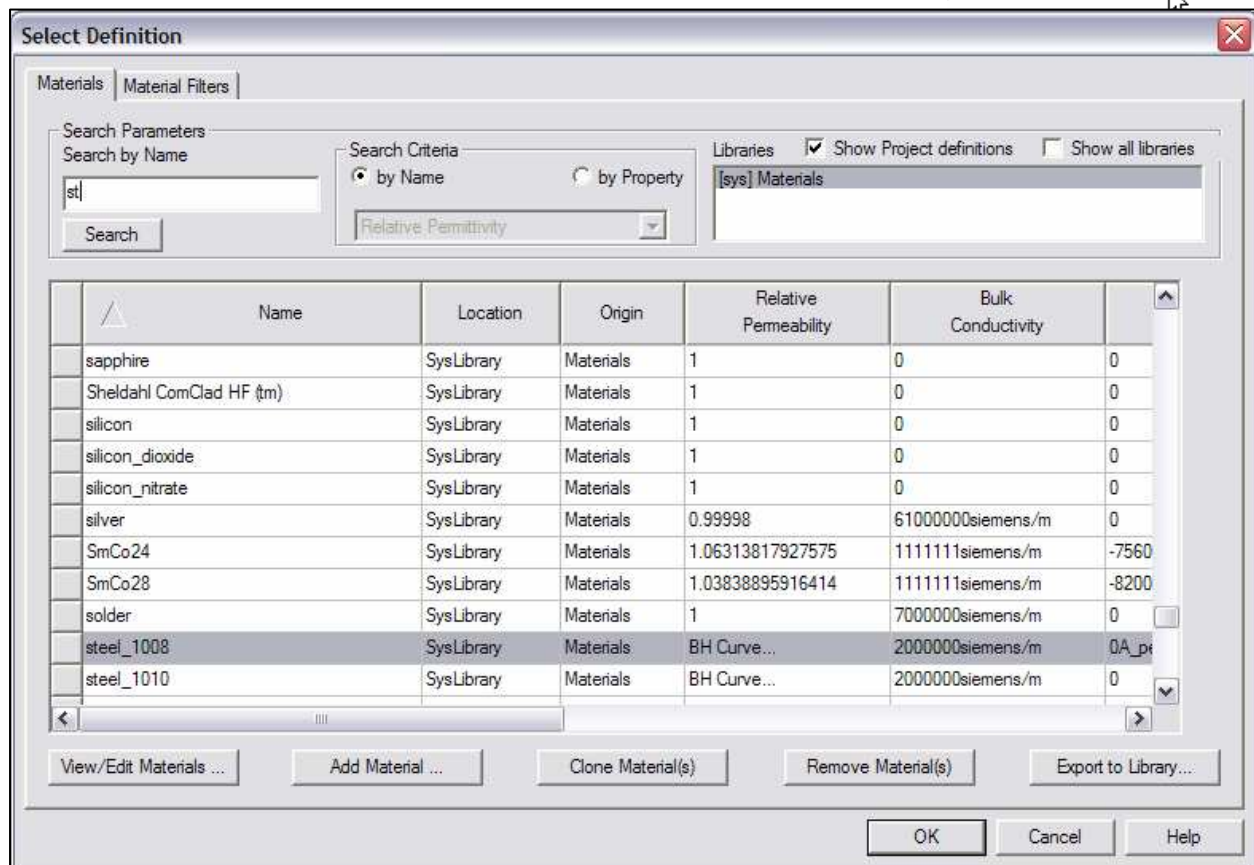
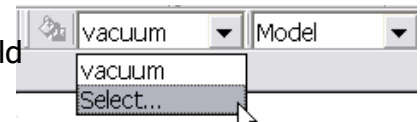
1. Select the menu item **3D Modeler > Units**
2. Set Model Units:
 1. Select Units: **mm**
 2. Click the **OK** button



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



Example (Magnetostatic) - Magnetic Force

Create Core

To create a box:

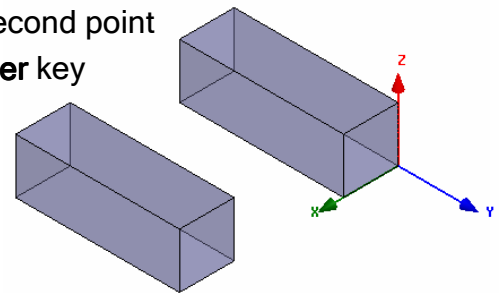
1. Select the menu item **Draw > Box**
2. Using the coordinate entry fields, enter the box position
 - ▲ X: **0.0**, Y: **0.0**, Z: **-5.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
 - ▲ dX: **10.0**, dY: **-30.0**, dZ: **10.0**, Press the **Enter** 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: **0**, Press the **Enter** key
4. Duplicate Along Line Window
 1. Total Number: **2**
 2. Click the **OK** button



To create the core:

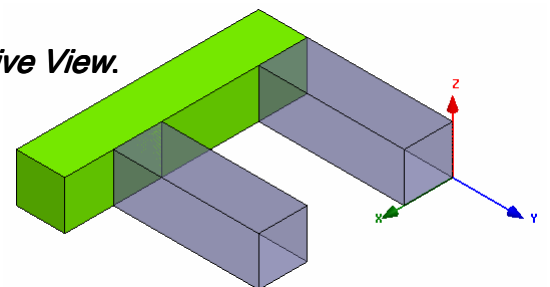
1. Select the menu item **Draw > Box**
2. Using the coordinate entry fields, enter the box position
 - ▲ X: **0.0**, Y: **-30.0**, Z: **-5.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
 - ▲ dX: **50.0**, dY: **-10.0**, dZ: **10.0**, Press the **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.**



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*.

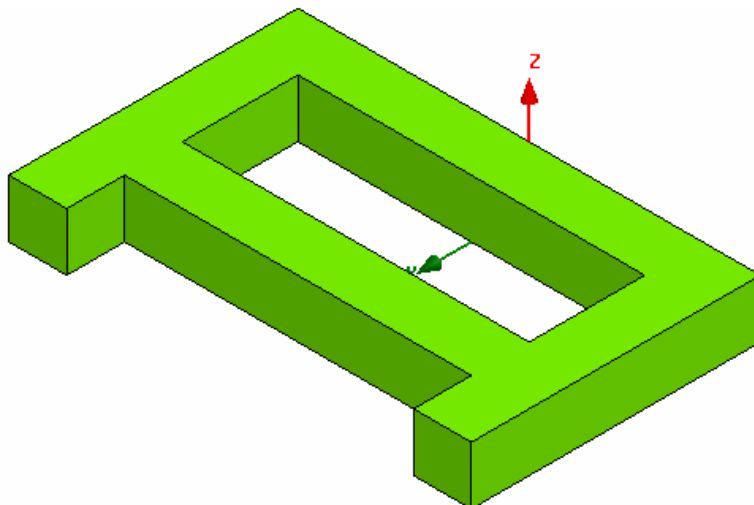
Group the Core

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*.



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:

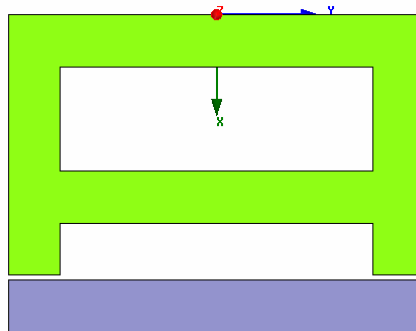
1. Select the **Command** tab from the **Properties** window
2. For **Position**, type: **50mm+mx**, **-40.0**, **-5.0**, Click the **Tab** key to accept
 - ▴ Add Variable **mx**: **1mm**, Click the **OK** button

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

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

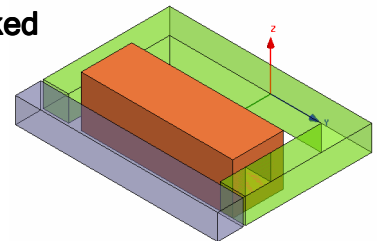
1. Select the menu item **Edit > Select > By Name**
2. Select Object Dialog,
 1. Select the objects named: **Coil, Core**
 2. Click the **OK** button

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 an insulating boundary to prevent current leakage out of the coil:

1. Select the coil **Edit > Select > By Name > Coil**
2. Click the **OK** button
3. Select the menu item **Maxwell > Boundaries > Assign > Insulating**
4. Insulating Boundary
 1. Name: **Insulating1**
 2. Click the **OK** button

Example (Magnetostatic) - Magnetic Force

Create Excitation

Section Object to create a 2D sheet object for assigning current

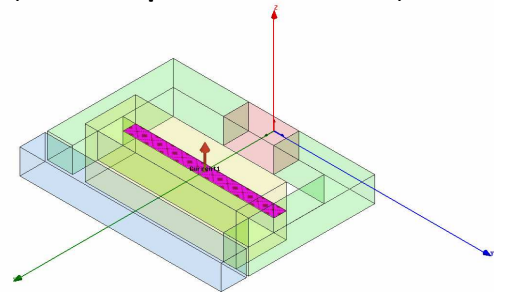
1. Select the coil **Edit > Select > By Name > Coil**
2. Click the **OK** button
3. Select the menu item **3D Modeler > Surface > Section**
 1. Section Plane: **XY**
 2. Click the **OK** button

Separate Bodies

1. Select the menu item **3D Modeler > Boolean > Separate Bodies**
2. Delete the extra sheet located closest to the bar.
3. Rename the remaining sheet to terminal and select it.

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 > Virtual**
- ▀ Click the **OK** button

Note: Virtual forces are useful on any object while Lorentz forces are only applicable on current carrying objects which have permeability = 1.

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:

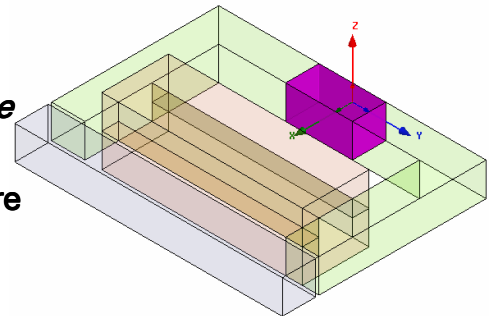
1. Select the menu item **Draw > Box**
2. Using the coordinate entry fields, enter the box position
 - 1. X: **0.0**, Y: **-10.0**, Z: **-5.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
 - 1. dX: **10.0**, dY: **20.0**, dZ: **10.0**, Press the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

Example (Magnetostatic) - Magnetic Force

Orient Magnet

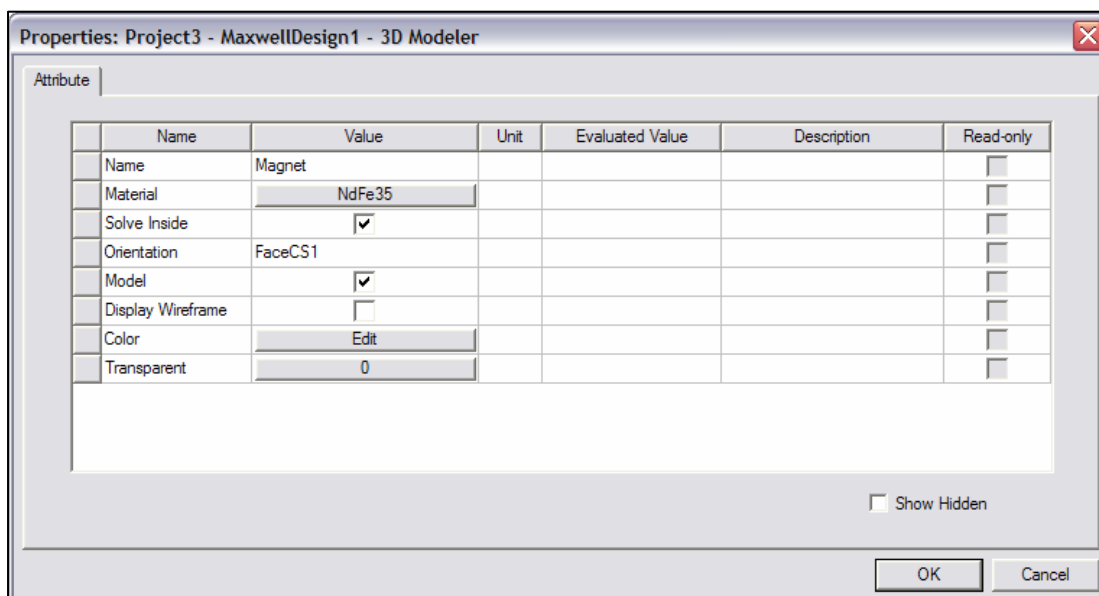
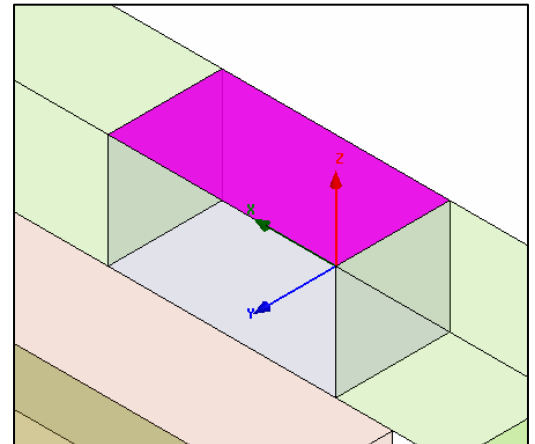
-  **Note:** By default all of the magnetic material in the material library are oriented in the x-direction. Using a local coordinate system (CS) we can correct the orientation of the geometry to align with the material definition.

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



Example (Magnetostatic) - Magnetic Force

Set Default Material

To set the default material:

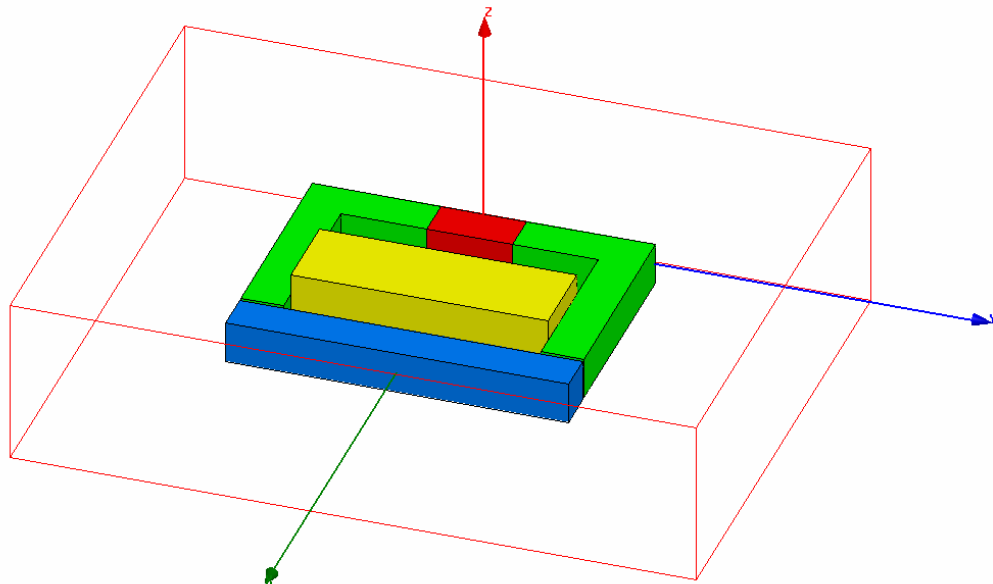
1. Using the 3D Modeler Materials toolbar, choose **vacuum**

Define a Region

To define a Region that is 50% larger than the object in the model:

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

Note: For all Maxwell 3D projects a solution space must be defined. Unless a partial model (like a motor wedge) is used, a region is usually created as described above.



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 to accept all default settings.

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:

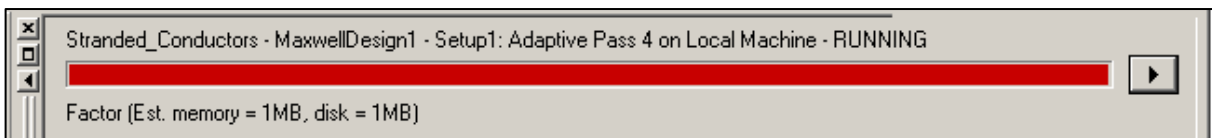
1. Select the menu item **Maxwell > Validation Check**
2. Click the **Close** button

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**



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.

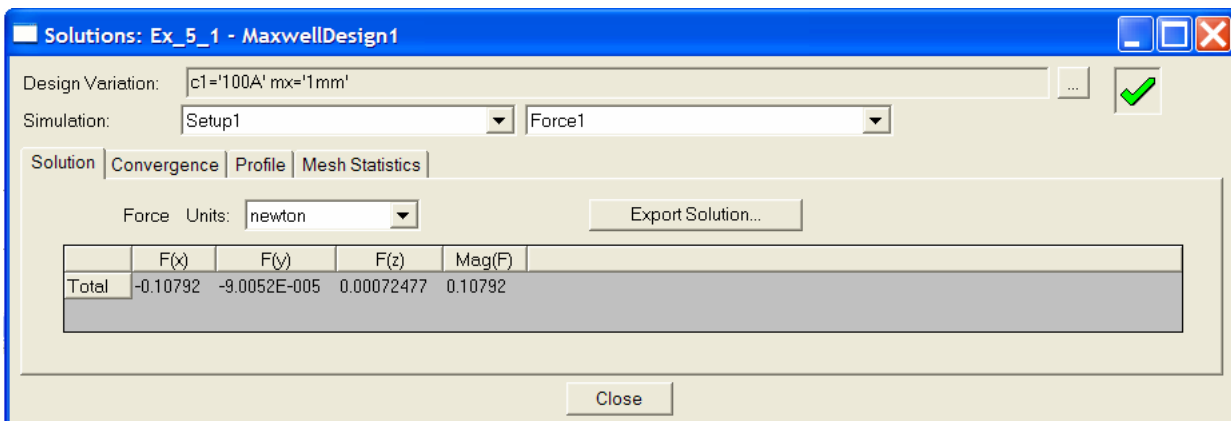
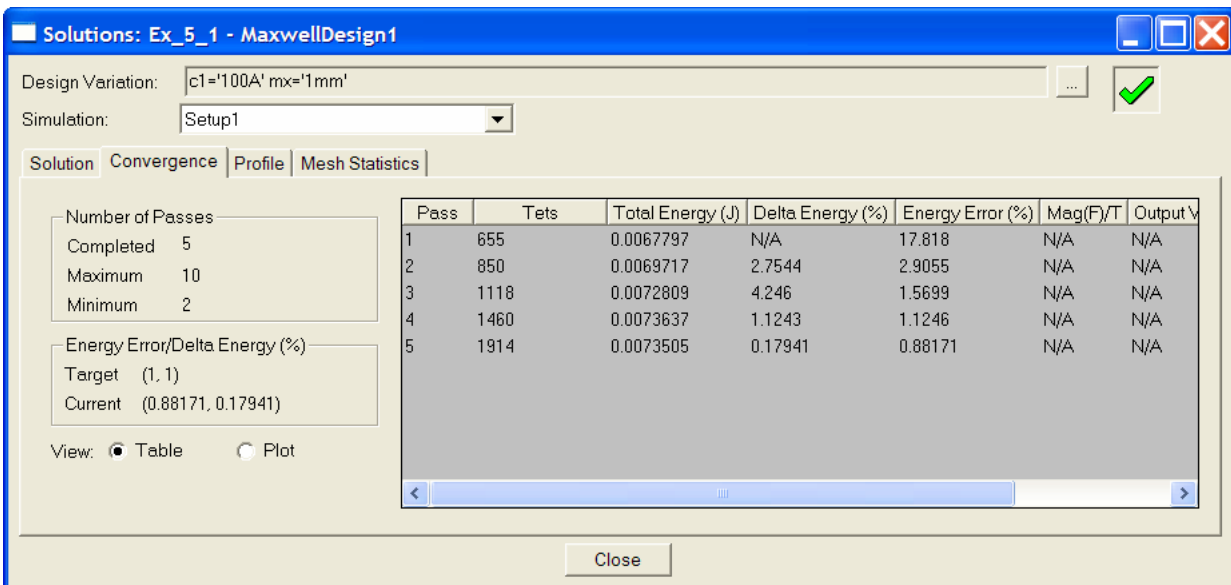
To view the Convergence:

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

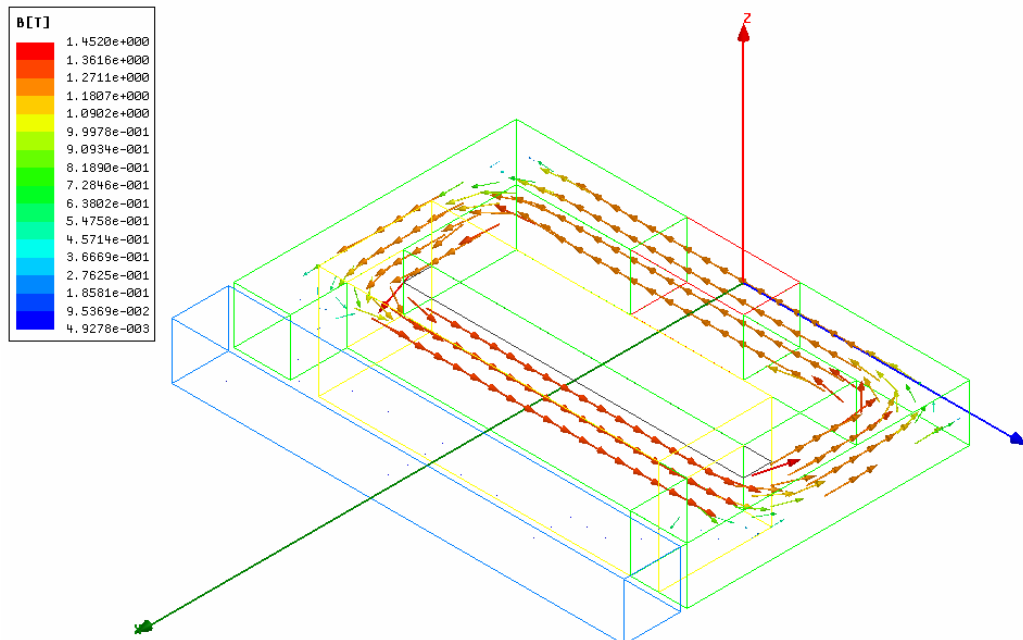


Example (Magnetostatic) - Magnetic Force

Plot Flux Density on Core Cross section

- ▲ To ensure that the current direction and magnetization are correct, create a plot the flux density vector on the core cross section, a new coordinate system is needed. Here are the steps to create this plot:
- ▲ Create an object list including the core, bar and magnet:
 - ▲ Select the menu item **Edit > Select > By Name**
 - ▲ Select Object Dialog,
 - ▲ Select the objects named: **Bar, Core, Magnet**
 - ▲ Click the OK button
 - ▲ Select **3D Modeler > List > Create > Object List > Objectlist1**
- ▲ In the history tree, expand Planes and select the geometry on which to create the plot: **Global:XY**
- ▲ Create the plot:
 - ▲ Right mouse click: **Field Overlays > Fields > B > B_vector**.
 - ▲ Choose Quantity: **B_vector** and In Volume: **Objectlist1**.
 - ▲ Click the **Done** button

Note: This field plot is for the nominal design case with: $m_x = 1\text{mm}$ and $c1 = 100\text{A}$.



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. The Setup Sweep Analysis window appears:
 1. On the **Sweep Definitions** tab:
 1. Click the **Add** button
 2. Add/Edit Sweep Dialog
 1. Select Variable: **c1** (pull down menu to to change from “mx” if necessary)
 2. Select **Linear Step**
 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

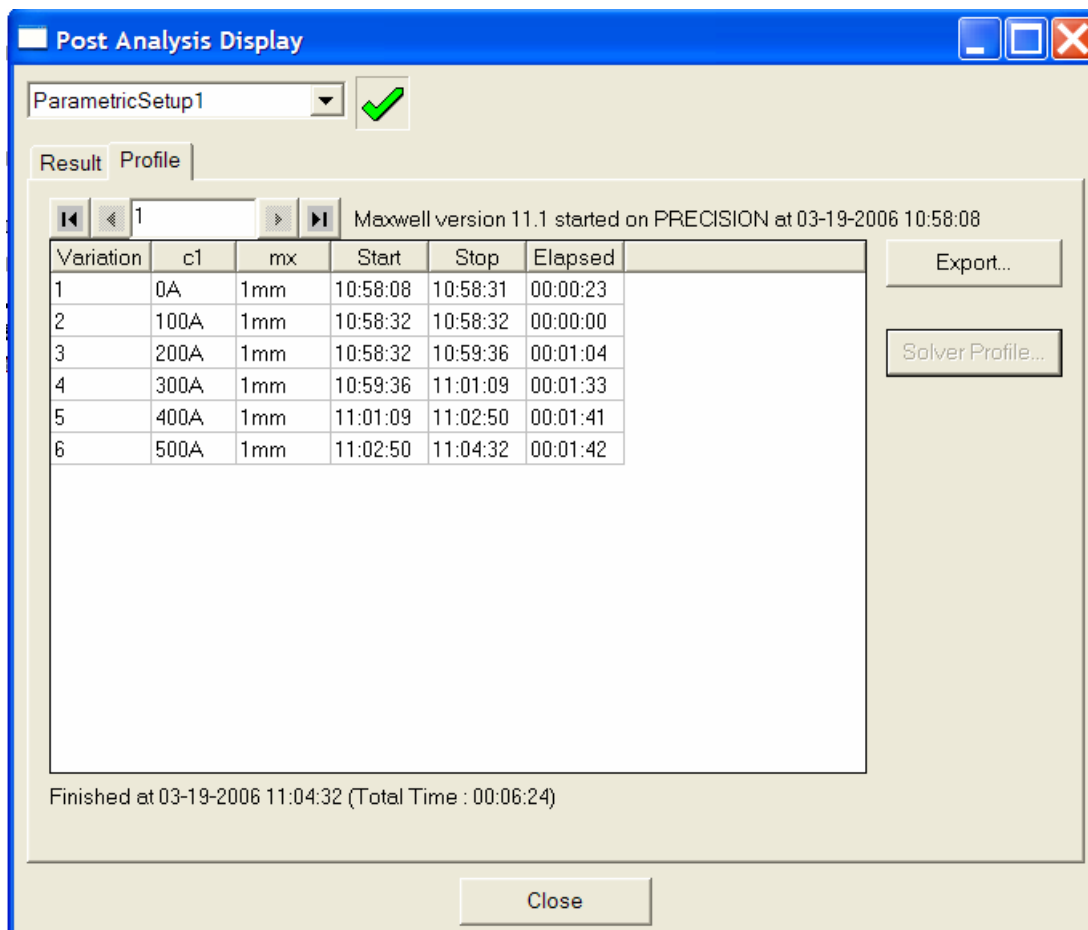
-  **To start the solution process:**
 1. Expand the Project Tree to display the items listed under **Optimetrics**
 2. Right-click the mouse on **ParametricSetup1** and choose **Analyze**

Example (Magnetostatic) - Magnetic Force

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

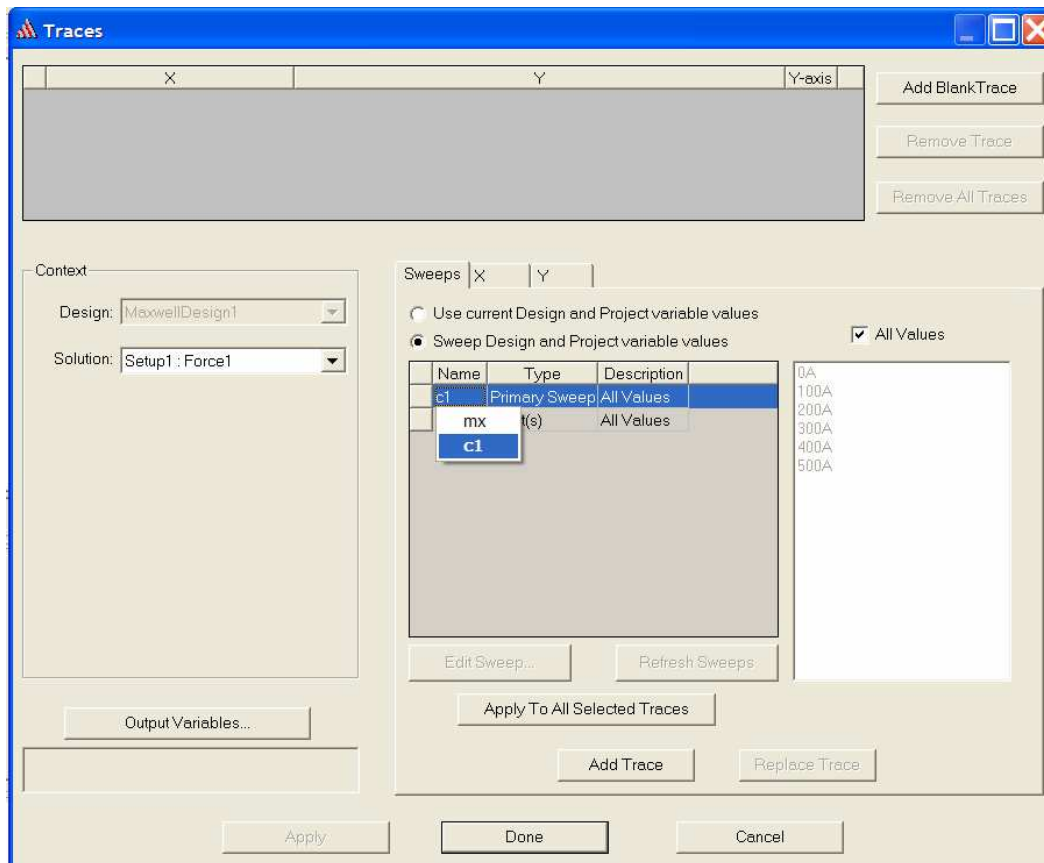


Example (Magnetostatic) - Magnetic Force

Create Plot of Force at each Current

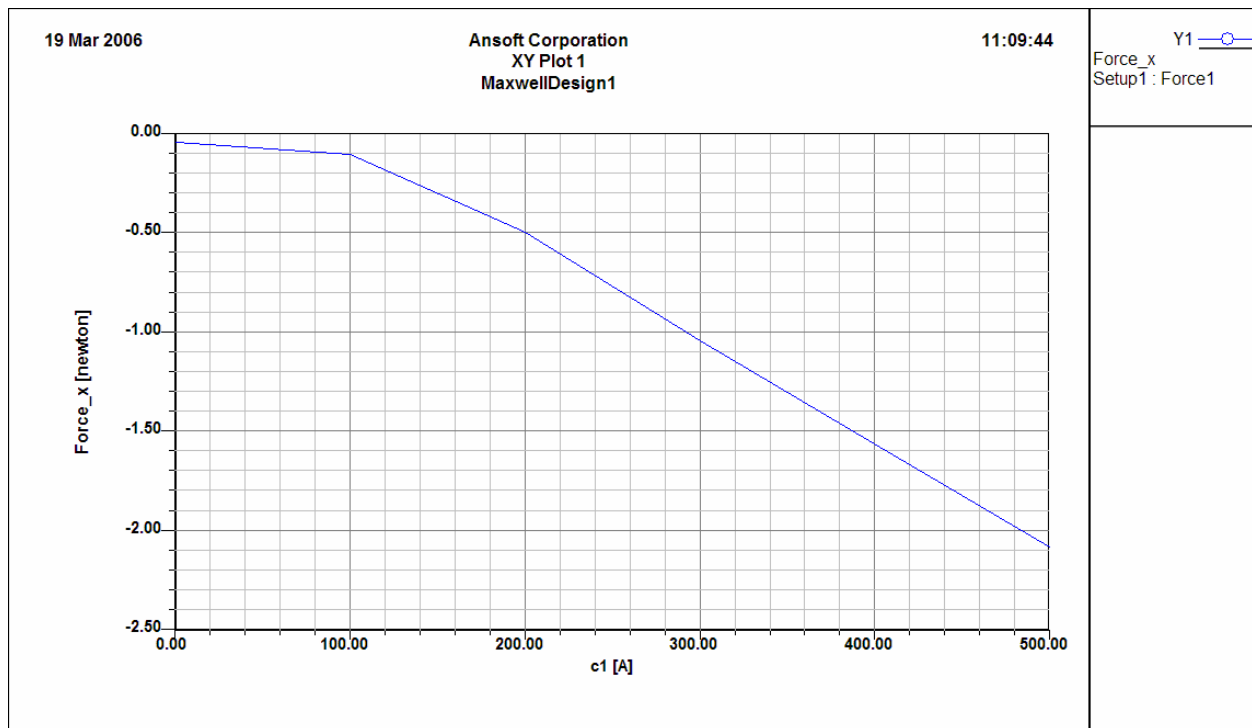
To create a report:

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



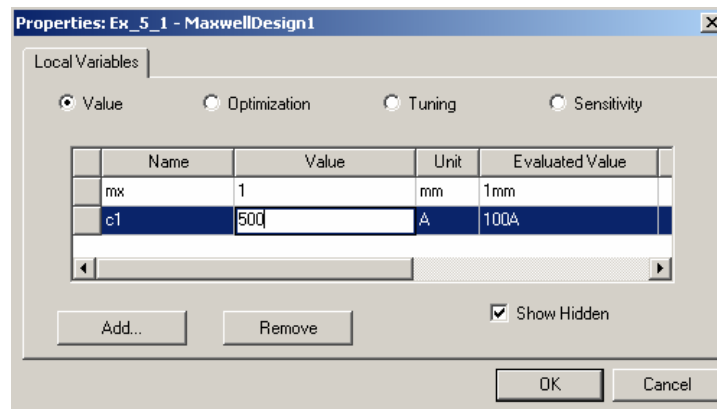
Example (Magnetostatic) - Magnetic Force

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

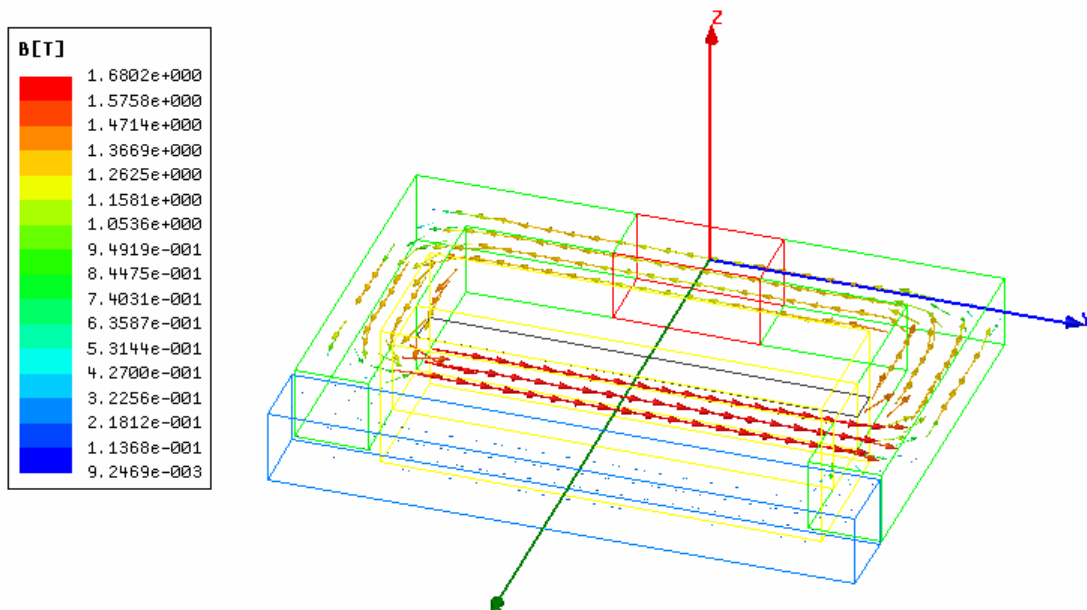


Example (Magnetostatic) - Magnetic Force

- ▲ **Plot Flux Density on Core Cross section at a different current**
 - ▲ Finally, create a plot of B_vector in the core corresponding to the last row in the parametric table with $c1 = 500A$.
 - ▲ Switch to the solution for the last row in the parametric table:
 - ▲ Select **Maxwell > Design Properties > Local Variables**. Change the value for $c1 = 500$ and click on **OK**.



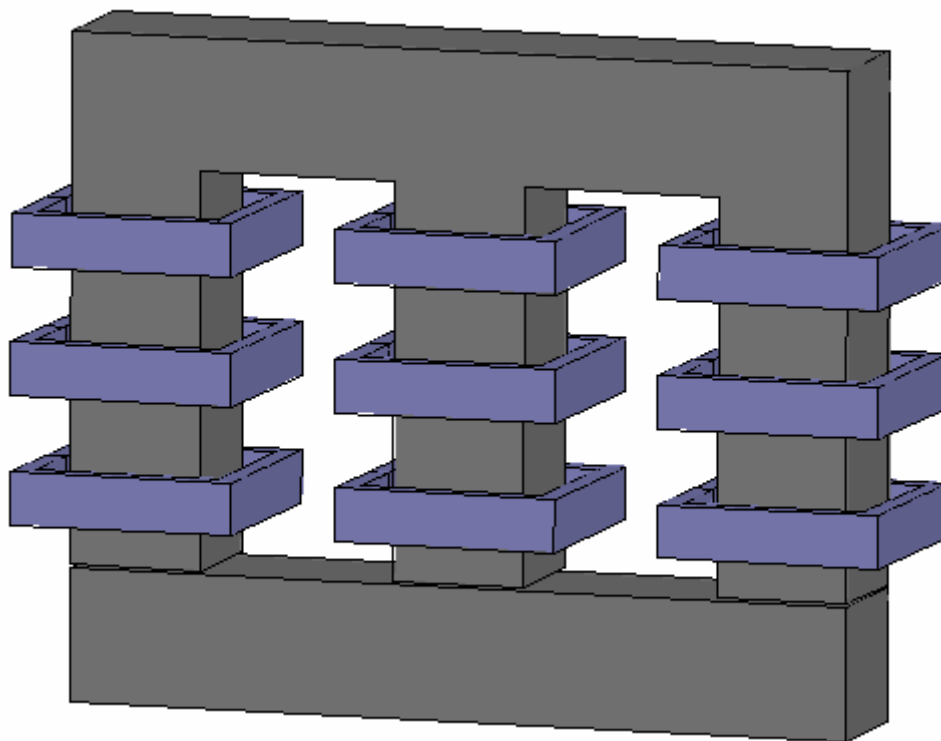
- ▲ The previous B_vector plot automatically updates to show the fields for **$c1 = 100A$** . The flux density is higher as expected



THIS PAGE INTENTIONALLY LEFT BLANK

Example (Magnetostatic) - Inductance Calculation

- 
The implementation and application of a 3-phase Inductor using the Magnetostatic Solver to calculate the nonlinear inductance
 - 
 The inductance calculation is an important part of a magnetostatic simulation. The inductive properties will, of course, change with the nonlinearities of the problem. The properties and concepts of the inductance calculation are described in this example.
 - 
 This example uses a 3-phase, 3-coil-per-phase inductor to demonstrate how the inductance calculation produces meaningful results. The setup is scrutinized and the results are examined. Then a half-symmetry model is examined to familiarize the user with a half-symmetry setup.



Example (Magnetostatic) - Inductance Calculation

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, Rectangle, Region**
-  Surface Operations: **Section**
-  Boolean Operations: **Subtract, Separate Bodies, Split**
-  3D Modeler: **Delete Last Operation**
-  Sweep: **Along Path**
-  Duplicate: **Along Line**

- **Boundaries/Excitations**

-  Current: **Stranded**

- **Analysis**

-  **Magnetostatic**

- **Results**

-  **Inductance Matrix**

Example (Magnetostatic) - Inductance Calculation

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**

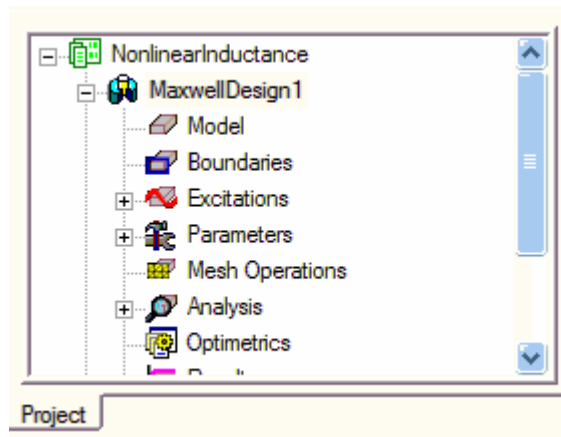
3. Click the **OK** button

Example (Magnetostatic) - Inductance Calculation

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

1. 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

1. The example that will be used to demonstrate the inductance calculation is that of a 3-phase transformer with multiple windings in series for each phase.

Set Model Units

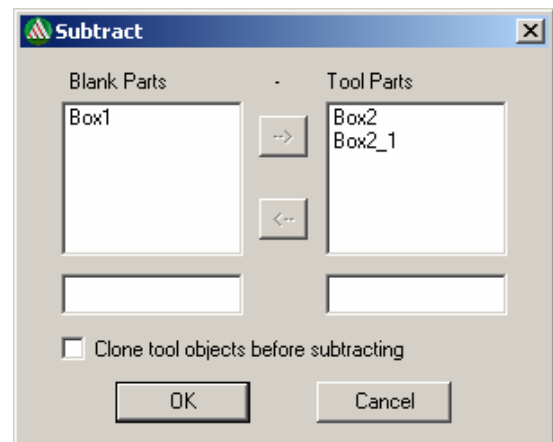
1. Select the menu item **3D Modeler > Units > Select Units: in (inches)**

Example (Magnetostatic) - Inductance Calculation



Create the Core

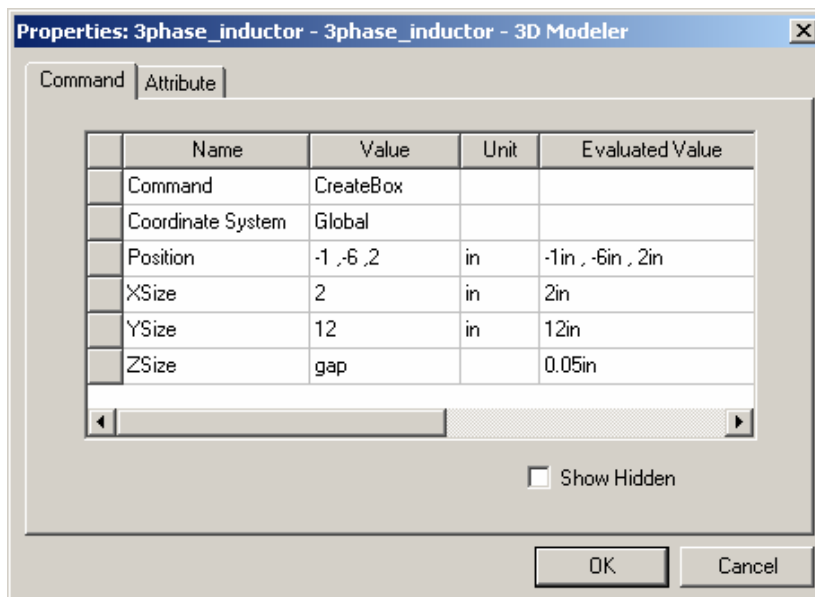
1. Select **Draw > Box** to start the drawing procedure:
 1. Enter -1, -6, 0 as the starting point of the box
 2. Enter 2, 12, 10 to define the dimensions of the box
 3. A Properties dialog appears:
 1. Select **OK**
 4. Hold down **Ctrl+D** or click the  button to view the entire geometry.
2. Select **Draw > Box** to start drawing the window.
 1. Enter -1, 1, 2 as the starting point of the box
 2. Enter 2, 3, 6 to define the dimensions of the box
 3. A Properties dialog appears:
 1. Select **OK**
3. Select **Box2** from the object tree (selected by default after creating it in step 2).
4. Select **Edit > Duplicate > Around Axis** to create a second window.
 1. Axis: **Z**
 2. Angle: **180 deg**
 3. Total number: **2**
 4. Select **OK**
5. Select **OK**
6. Select **Box1**, and then select **Box2** and **Box2_1** from the object tree.
Use **Ctrl** or **Shift** to select multiple items throughout this example.
7. Select **3D Modeler > Boolean > Subtract**
 1. Make sure that **Box1** is in the Blank Parts box and that **Box2** and **Box2_1** are in the Tool Parts box (this is true if **Box1** is selected first).
 2. Make sure that **Clone tool objects before subtracting** is deselected.
 3. Select **OK**



Example (Magnetostatic) - Inductance Calculation

Create the Core (continued)

8. Select **Draw > Box** to start drawing the gap:
 1. Enter -1, -6, 2 as the starting point of the box
 2. Enter 2, 12, 0.05 to define the dimensions of the box
 3. A Properties dialog appears:
 1. In the value box of Zsize, enter **gap** and type Enter to accept this value.
 1. In the **Add Variable** box that appears, enter 0.05in for the value.
 2. Select **OK**
 2. Select **OK**



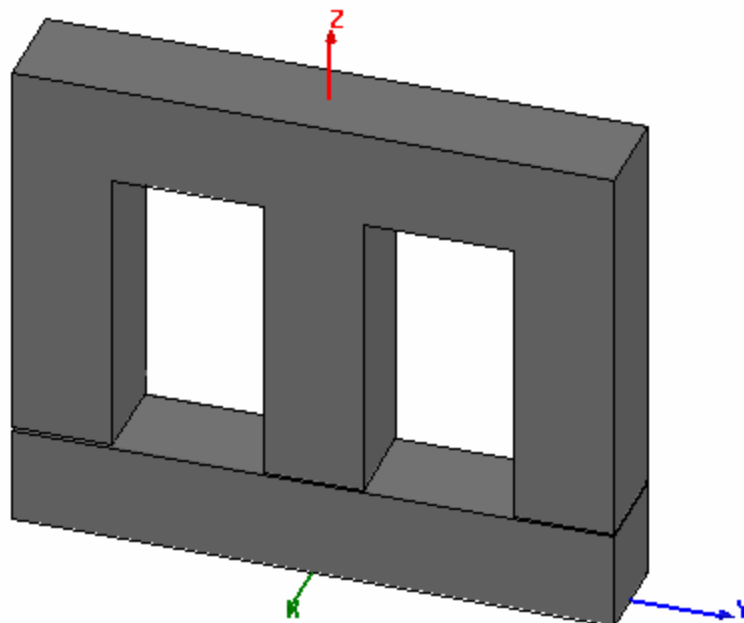
9. Select **Box1** and then **Box3** from the Object Tree.
(Use **Ctrl** or **Shift** to select multiple items.)
10. Select **3D Modeler > Boolean > Subtract**.
 1. Make sure that **Box1** is in the Blank Parts box and that **Box2** and **Box2_1** are in the Tool Parts box (this is true if **Box1** is selected first).
 2. Make sure that **Clone tool objects before subtracting** is deselected.
 3. Select **OK**

Example (Magnetostatic) - Inductance Calculation



Create the Core (continued)

11. Select **Box1**.
12. Select **3D Modeler > Boolean > Separate Bodies** (the E and I parts of the core are now separate).
13. Double-click on **Box1** in the object tree (a Properties window pops up).
 1. Change the name from **Box1** to **Core_E**.
 2. Select **OK**.
14. Double-click on **Box1_1** in the object tree (a Properties window pops up).
 1. Change the name from **Box1_1** to **Core_I**.
 2. Select **OK**.
15. Select both **Core_E** and **Core_I** from the object tree.
 1. Find the Attribute window (on the left side, under the project tree by default).
 2. Click on the Material Value **vacuum** (a Materials list appears).
 1. Find **steel_1008** and double-click on this material to accept it (or single-click and select **OK**).
16. The cores are now completely defined.

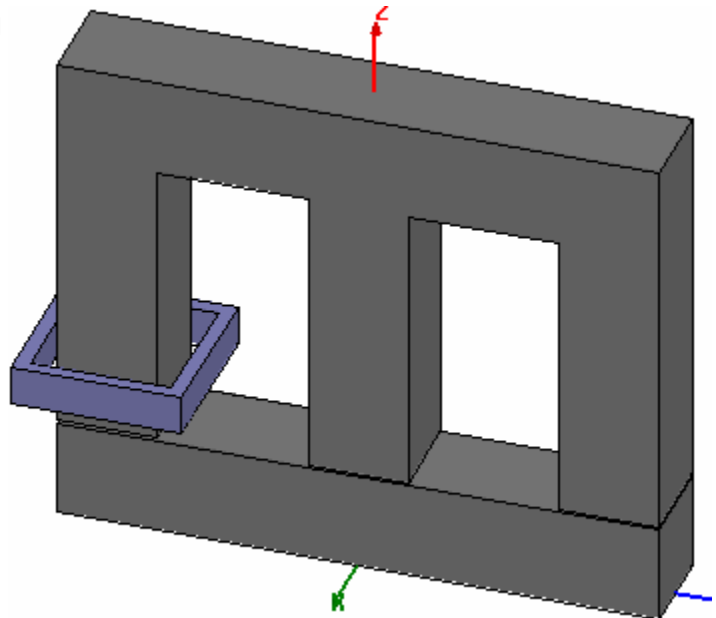


Example (Magnetostatic) - Inductance Calculation



Create the Coils

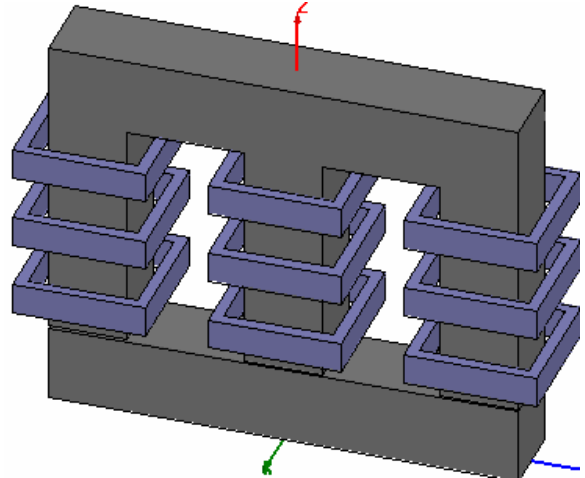
1. Select **3D Modeler > Grid Plane > YZ**
2. Select **Draw > Rectangle**.
 1. Enter 0, -3.6, 3.5 as the starting point of the rectangle
 2. Enter 0, 0.3, -0.8 to define the dimensions of the rectangle
 3. A Properties dialog appears:
 1. Select **OK**
3. Select **3D Modeler > Grid Plane > XY**
4. Select **Draw Rectangle**.
 1. Enter 1.5, -6.5, 3 as the starting point of the rectangle
 2. Enter -3, 3, 0 to define the dimensions of the rectangle
 3. A Properties dialog appears:
 1. Select **OK**
5. Select **Rectangle2** from the object tree.
6. Select **3D Modeler > Delete Last Operation** to produce a line.
7. Select **Rectangle1** and **Rectangle2** from the object tree.
8. Select **Draw > Sweep > Along Path**
 1. A **Sweep along path** window pops up:
 1. Angle of twist: 0deg
 2. Draft Angle: 0deg
 3. Draft type: Round
 4. Select **OK**
 2. Select **OK**



Example (Magnetostatic) - Inductance Calculation

Create the Coils (continued)

9. Select **Rectangle1** from the object tree.
10. Select **Edit > Duplicate > Along Line**
 1. Enter 0, 0, 0.
 2. Enter 0, 0, 1.925.
 1. Total Number: 3
 2. Select **OK**.
 3. Select **OK**.
11. Select **Rectangle1**, **Rectangle1_1**, and **Rectangle1_2** from the object tree. (Use **Ctrl** or **Shift** to select multiple items.)
12. Select **Edit > Duplicate > Along Line**
 1. Enter 0, 0, 0.
 2. Enter 0, 5, 0.
 1. Total Number: 3
 2. Select **OK**.
 3. Select **OK**.
13. Select **Rectangle1**, **Rectangle1_1**, and **Rectangle1_2** from the object tree.
14. In the Attribute window, type **Coil_left** in the Name Value.
15. Select **Rectangle1_3**, **Rectangle1_1_1**, and **Rectangle1_2_1** from the object tree.
16. In the Attribute window, type **Coil_mid** in the Name Value.
17. Select **Rectangle1_4**, **Rectangle1_1_2**, and **Rectangle1_2_2** from the object tree.
18. In the Attribute window, type **Coil_right** in the Name Value.
19. Select all **Coil** objects from the object tree.
20. In the Attribute window, click on the Material Value **vacuum** (a Materials list appears).
 1. Find **copper** and double-click on this material to accept it (or single-click and select **OK**).

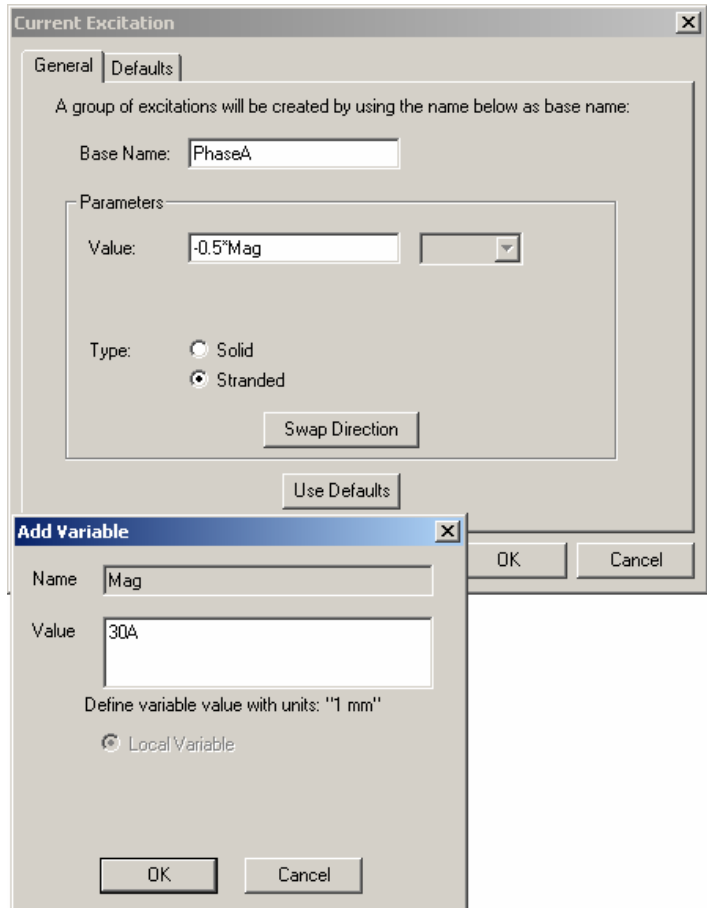


Example (Magnetostatic) - Inductance Calculation



Create the Terminals and Define the Sources:

1. Select all the **Coil** objects from the object tree.
2. Select **3D Modeler > Surface > Section...**
 1. Select **YZ**
 2. Select **OK**
3. Select all the **Section** sheets (selected by default after the last operation).
4. Select **3D Modeler > Boolean > Separate Bodies**
5. Select the extra Sections (1_1 through 9_1) and delete them (by using Delete on the keyboard or **Edit > Delete** from the command menu).
6. Select the sheets **Section1**, **Section2**, **Section3** from the object tree.
7. Select **Maxwell > Excitations > Assign > Current...**
8. The Current Excitation dialog appears.
 1. Base Name: **PhaseA**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
9. Select **OK**
10. An Add Variable dialog appears
 1. For the Variable **Mag**:
Value: **30A**
11. Select **OK**
12. Select the sheets **Section4**, **Section5**, **Section6** from the object tree.
13. Select **Maxwell > Excitations > Assign > Current...**
 1. Base Name: **PhaseB**
 2. Value: **Mag**
 3. Type: **Stranded**
14. Select **OK**



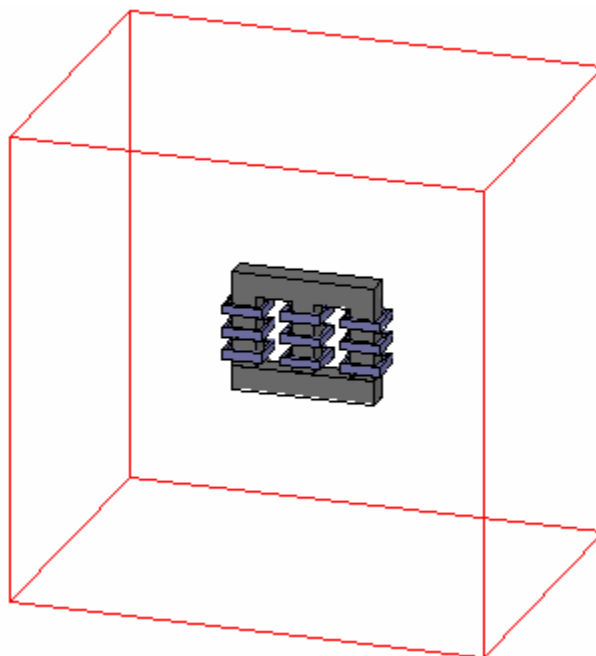
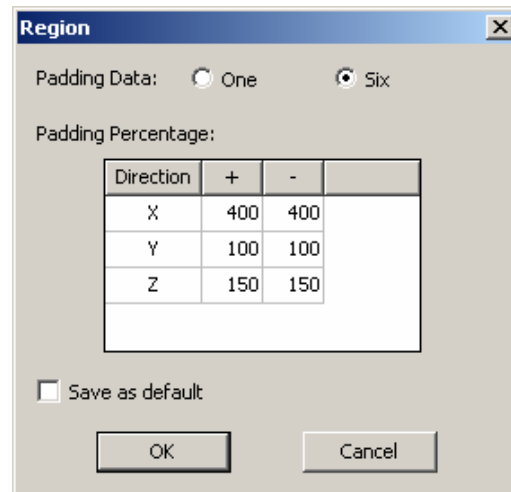
Example (Magnetostatic) - Inductance Calculation

Create the Terminals and Define the Sources (continued):

15. Select the sheets **Section7**, **Section8**, **Section9** from the object tree.
16. Select **Maxwell > Excitations > Assign > Current...**
17. The Current Excitation dialog appears.
 1. Base Name: **PhaseC**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
18. Select **OK**

Create the Region Object

1. Select **Draw > Region**
 1. A Region dialog box opens up.
 2. Fill in the information as follows:
 Padding Data: **Six**
 X: **400, 400**
 Y: **100, 100**
 Z: **150, 150**
 3. Select **OK**
2. Select **OK**



Example (Magnetostatic) - Inductance Calculation

Inductance Calculation Defining Equations

- Inductance in a magnetostatic simulation can be defined as

$$\lambda_1 = L_{11}I_1 + L_{12}I_2 + \dots$$

$$\lambda_2 = L_{21}I_1 + L_{22}I_2 + \dots$$

...

where λ is the flux linkage for an individual coil, and I is the current.

- This inductance is the apparent inductance - it defines the ratio of flux linkage to current in one coil.

- The inductance of a coil can also be defined by the energy storage as

$$L = 2 \times W / I^2$$

$$= \int \vec{B} \cdot \vec{H} d\Omega / I^2$$

where the energy is determined by the magnetic flux density and field intensity in the solution space.

- This energy storage calculation is what determines the inductance values in the matrix. The on-diagonal elements (L_{11} , L_{22} , etc.) are determined by exciting the coils individually and finding the energy from the B and H fields for each individual case. The off-diagonal elements (L_{12} , L_{21} , L_{13} , etc.) are determined from a combination of the B and H fields from the individually excited cases. The following is the exact defining equation for the inductance calculation

$$L_{ij} = \int \vec{B}_i \cdot \vec{H}_j d\Omega$$

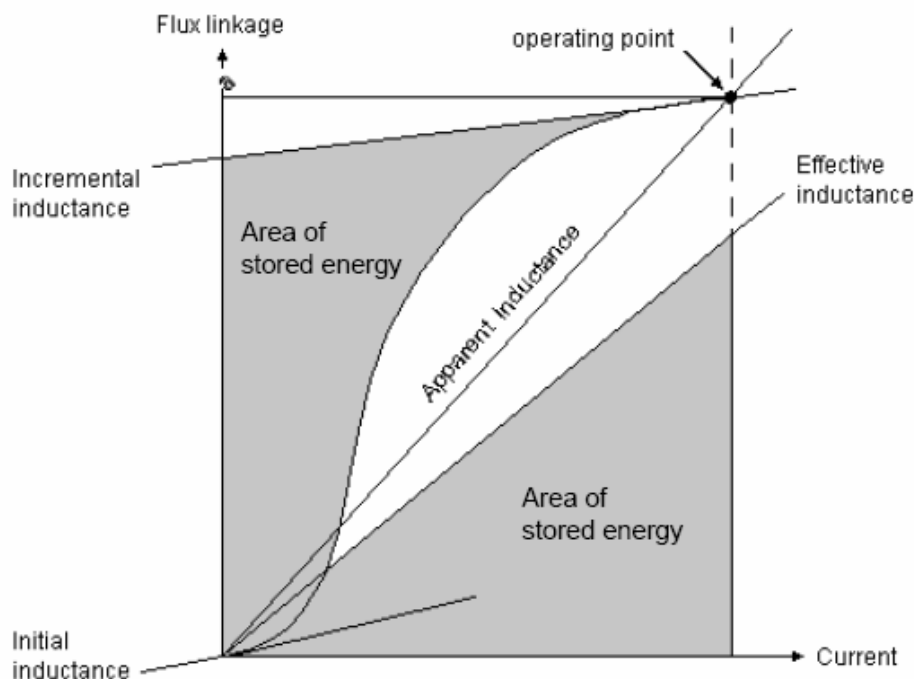
where it is assumed that one ampere is flowing through the respective coils.

- For nonlinear materials, the value of the inductance obtained from these calculations is only valid for the specific source currents. If any of the source currents change, the characteristics of the field solution will change as well as the inductance of each coil (less change when in a linear region, greatest change when moving from the linear region to a saturated region).
- If only linear materials are used, the value of the inductance will be valid for any value of source currents.

Example (Magnetostatic) - Inductance Calculation

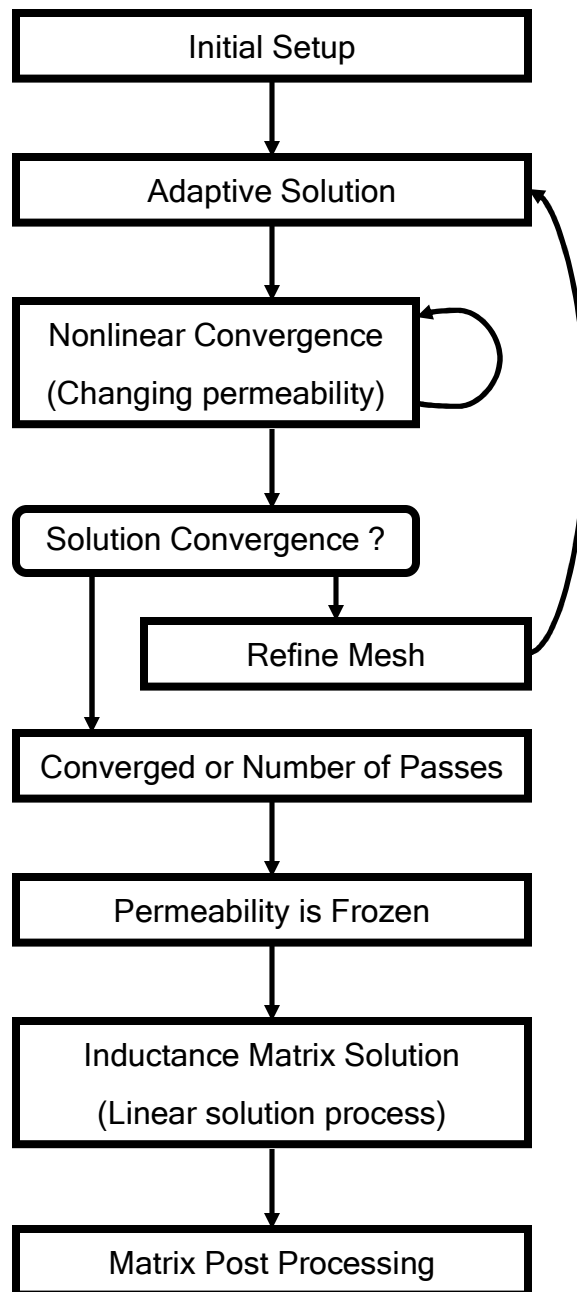
Inductance Calculation Methodology

- The inductance calculation is performed after the field solution and depends on the values of the field solution at every point in the solution space because it requires an energy integral in the entire solution space.
- After the field solution is completed, the relative permeability values are “frozen”. Then, one ampere is sequentially excited in each coil, and a field solution is performed with the frozen permeabilities. With these field solutions, the energy integral is performed and the inductance is obtained as described previously.
- The inductance obtained in this manner is the inductance per turn² (referred to as the nominal inductance in the software) and is the apparent inductance (determined by the flux linkage/current operating point in the graphic below).



- Incremental inductance can be obtained by performing energy perturbation techniques (changing the source currents and finding the slope).

Inductance Calculation Process



Example (Magnetostatic) - Inductance Calculation

Inductance Calculation Post Processing

-  The nominal inductances that are computed by the solver are returned in units of Henries/turns². In order to return each inductance in units of Henries, the number of turns for each coil must be specified at some point. Also, the inductance matrix can be grouped to represent windings in series, and a factor can be included if the winding is made up of parallel branches. These are all considered post-processing, and they can help to return meaningful quantities.

-  The post processed inductances are calculated as follows:

$$L_{ij} = N_i N_j L_{ij\_nom}$$

$$L_{group} = \sum_{i\_group} \sum_{j\_group} L_{ij}$$

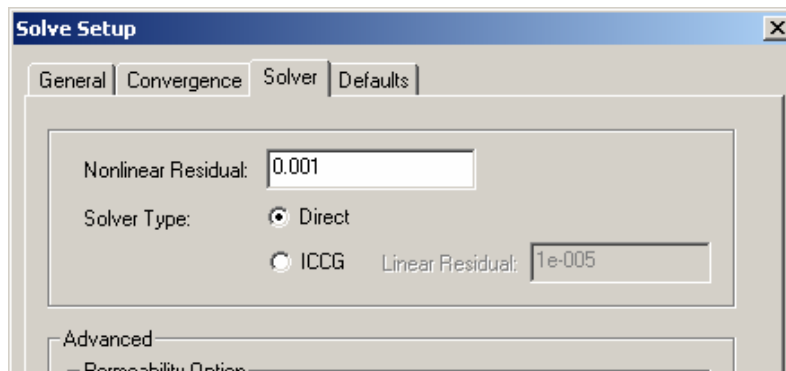
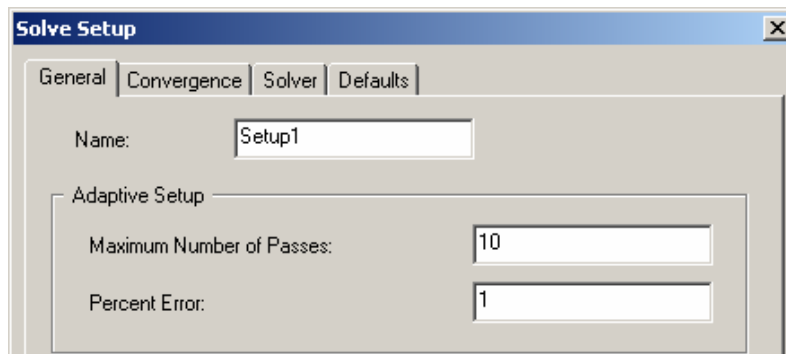
$$L_{branched} = L_{group} / b$$

-  The first step is to find the coil inductances from the nominal inductances by multiplying by the number of turns in each coil (or number of turns squared for self inductances).
-  The next step is to group the series inductances by adding up all the inductances in the block (i.e. if conductors 1 through N are grouped, then add up the N by N block of inductances to obtain the grouped self inductance - the same process is followed for grouped mutual inductances).
-  The final step is to take into account parallel branches by dividing by the number of branches squared. The inductance decreases with parallel branches because of the reduced number of turns per branch.

Example (Magnetostatic) - Inductance Calculation

Create a Solution Setup

1. Select **Maxwell > Analysis Setup > Add Solution Setup ...**
 1. In the General tab:
 1. Set Maximum Number of Passes to 10
 2. Set Percent Error to 1
 2. In the Solver tab:
 1. Set Nonlinear Residual to 0.001
 3. Select **OK**

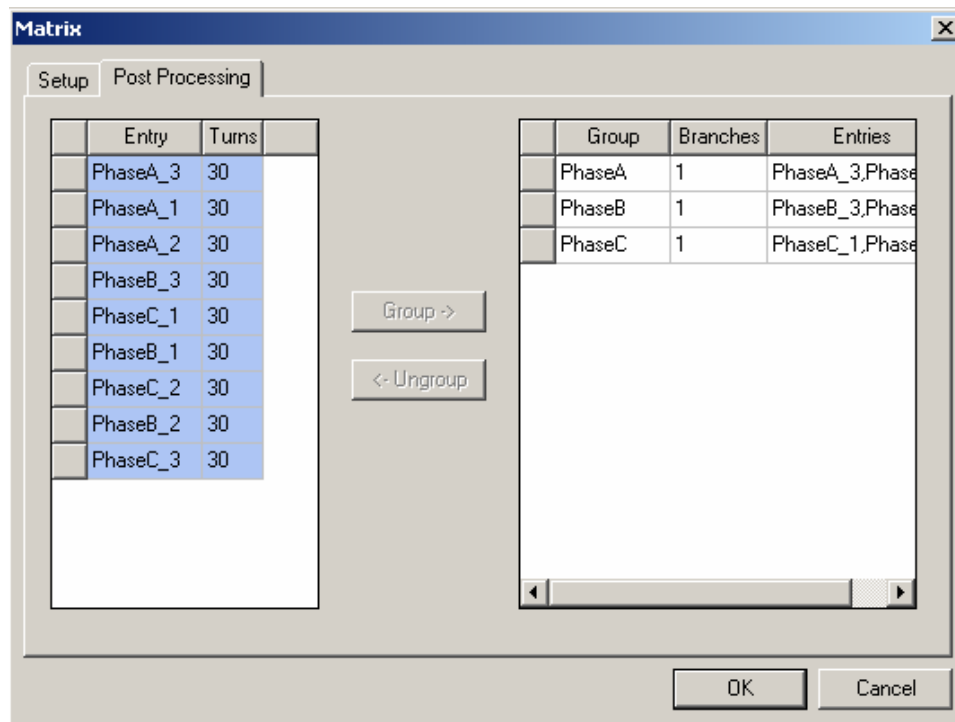


-  The Nonlinear Residual is important in simulations with nonlinear materials that operate outside of the linear region. Increasing the number will make the simulation run faster, decreasing the number will force the nonlinear solver to more precisely find the nonlinear B-H operating points within the steel (or other nonlinear magnetic material).
-  It is often the case with saturated nonlinear materials that the Nonlinear Residual will need to be reduced to obtain accurate results.

Example (Magnetostatic) - Inductance Calculation

Create Matrix Parameters

1. Select **Maxwell > Parameters > Assign > Matrix...**
 1. In the Setup tab, check the boxes to Include all the Sources (or click on the word Include to automatically check all the boxes).
 2. In the Post Processing tab, set the turns for each source to **30**
 3. Select **PhaseA_1, PhaseA_2, and PhaseA_3** and select **Group ->** (Use **Ctrl** or **Shift** to select multiple items.)
 4. Select **PhaseB_1, PhaseB_2, and PhaseB_3** and select **Group ->**
 5. Select **PhaseC_1, PhaseC_2, and PhaseC_3** and select **Group ->**
 6. Rename Group1 to **PhaseA**
 7. Rename Group2 to **PhaseB**
 8. Rename Group3 to **PhaseC**
 9. Leave Branches set to 1 for each Group



- This setup says that each source has 30 turns and that the sources on each leg are connected in series (as far as the post-processing is concerned).

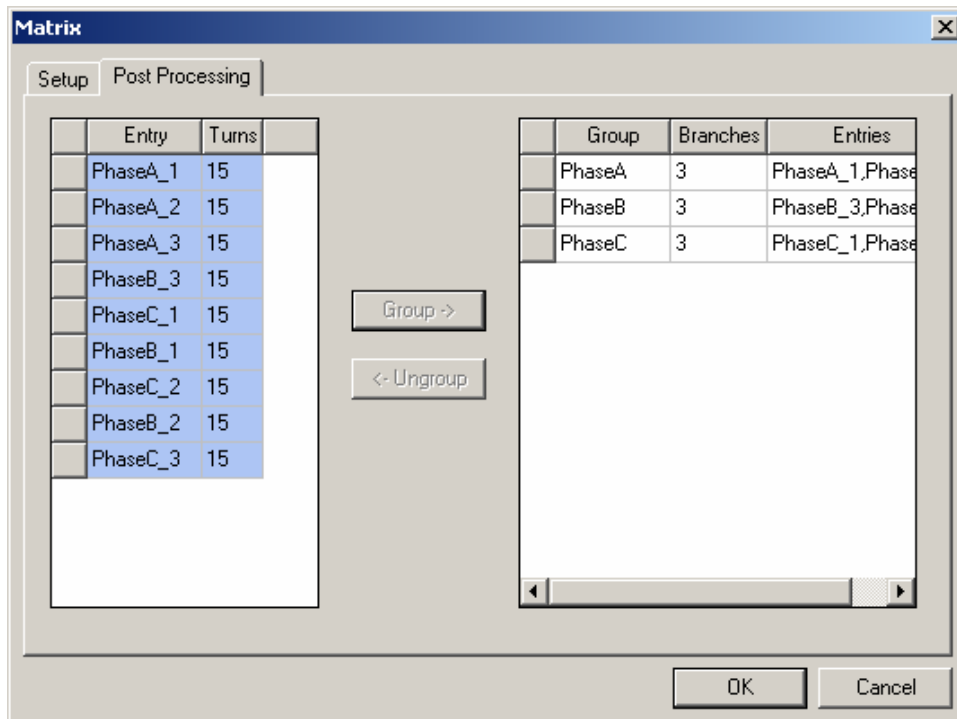
Example (Magnetostatic) - Inductance Calculation

Create Matrix Parameters (continued)

We will create a second Matrix Parameter to show what the settings signify.

2. Select **Maxwell > Parameters > Assign > Matrix...**

1. In the Setup tab, check the boxes to Include all the Sources (or click on the word Include to automatically check all the boxes).
2. In the Post Processing tab, set the turns for each source to **15**
3. Select **PhaseA_1, PhaseA_2, and PhaseA_3** and select **Group ->**
4. Select **PhaseB_1, PhaseB_2, and PhaseB_3** and select **Group ->**
5. Select **PhaseC_1, PhaseC_2, and PhaseC_3** and select **Group ->**
6. Rename Group1 to **PhaseA**
7. Rename Group2 to **PhaseB**
8. Rename Group3 to **PhaseC**
9. Set Branches to 3 for each Group



This setup says that each source has 15 turns in 3 branches of 5 turns each (with twice the current in each turn as for the 30 turn case) and that the sources on each leg are connected in series (as far as the post-processing is concerned).

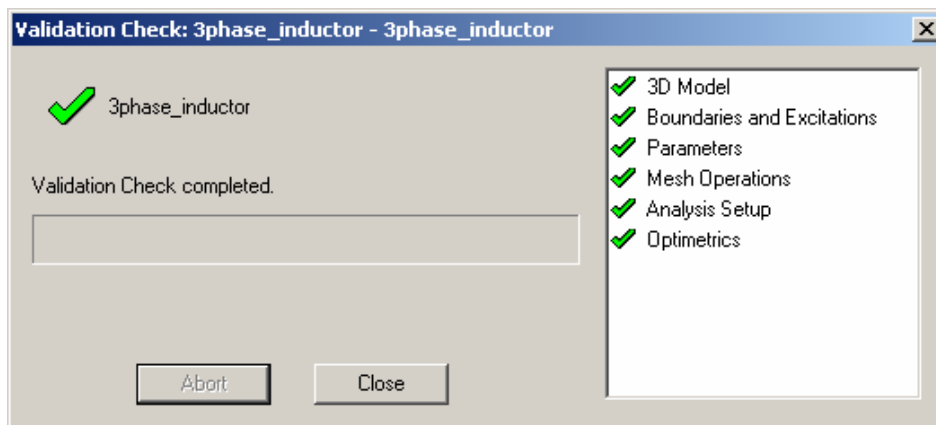
Example (Magnetostatic) - Inductance Calculation

Save the Project

1. Select the menu item **File > Save As**
2. From the Save As window, type in **3phase_inductor**
3. Click on the **Save** button

Check the Validity of the Model

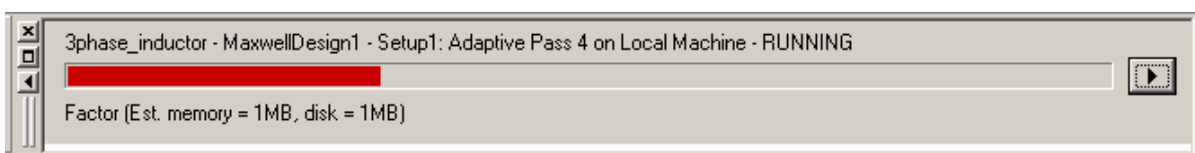
1. Select the menu item **Maxwell > Validation Check**, or click on the  icon



1. The problem won't solve if there are errors (errors are denoted by a red X), but it will solve if all the validations return a green check or a warning (warnings are denoted by a yellow triangle, and should be noted, but can often be safely ignored).

Analyze

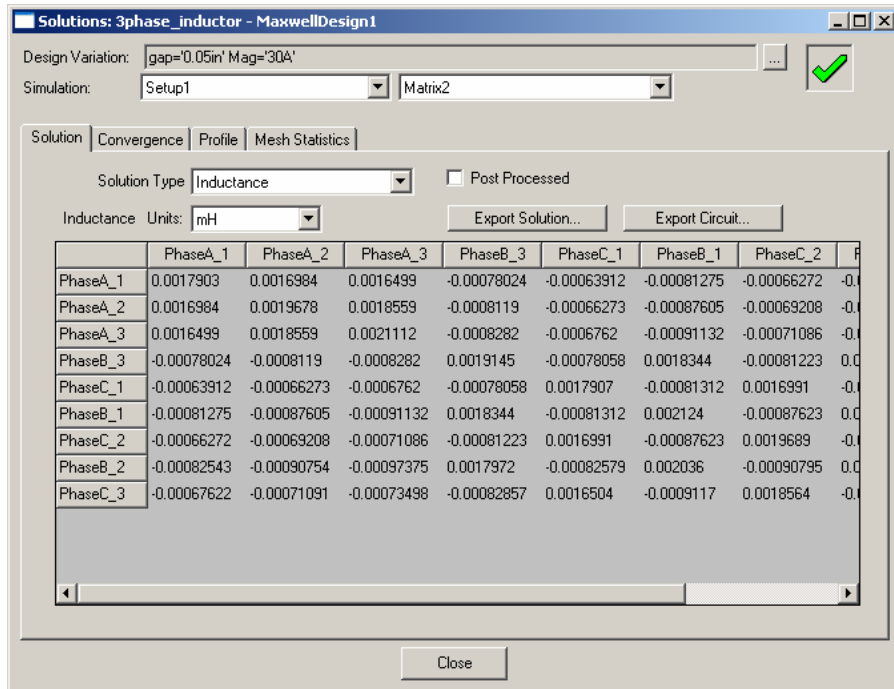
1. Select the menu item **Maxwell > Analyze**, or click on the  icon
2. This should take about 10 minutes to solve on a standard computer (such as a fast laptop or standard desktop), and about 5 minutes on a fast computer.



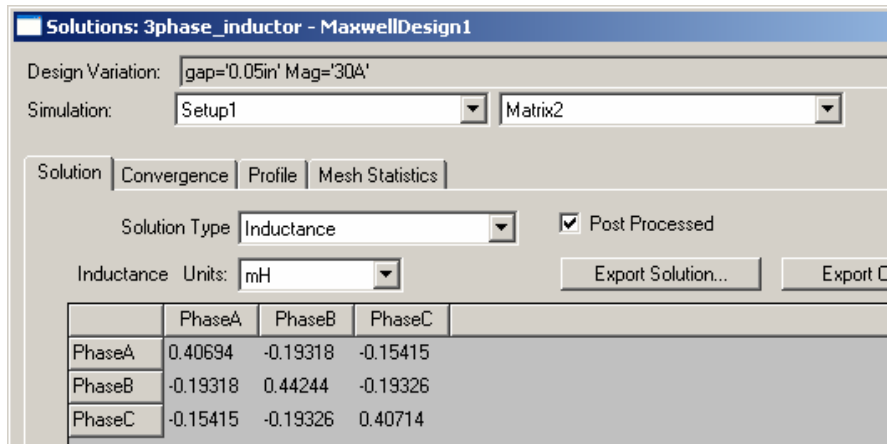
Example (Magnetostatic) - Inductance Calculation

Inductance Matrix Solutions

- Open up the matrix solution data by selecting **Maxwell > Results > Solution Data**
- The first display that appears (by default) is the non-post processed data for Matrix2



- Check the box marked **Post Processed** to see the results.

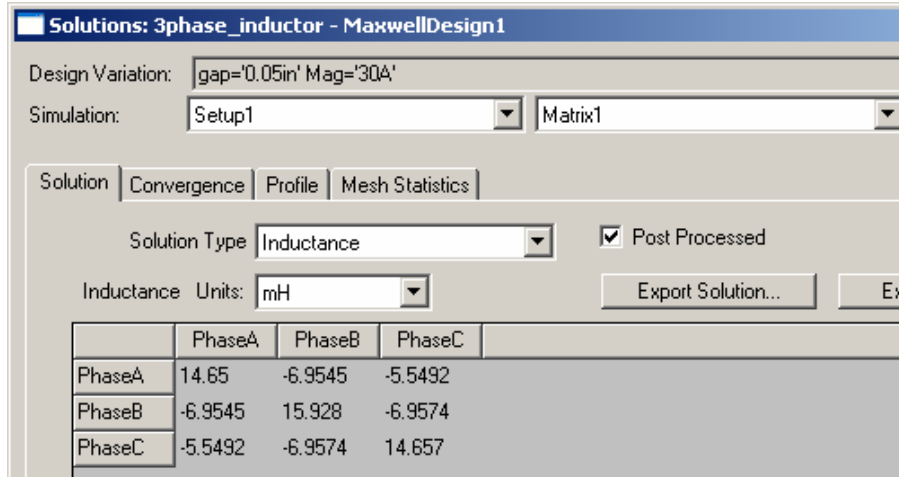


- Notice that the results are symmetric in both axes as we would expect.
- Notice that the mutual inductances are negative. It is important to note that the direction of the current used for the mutual inductances is determined by the direction of the arrow for each excitation.

Example (Magnetostatic) - Inductance Calculation

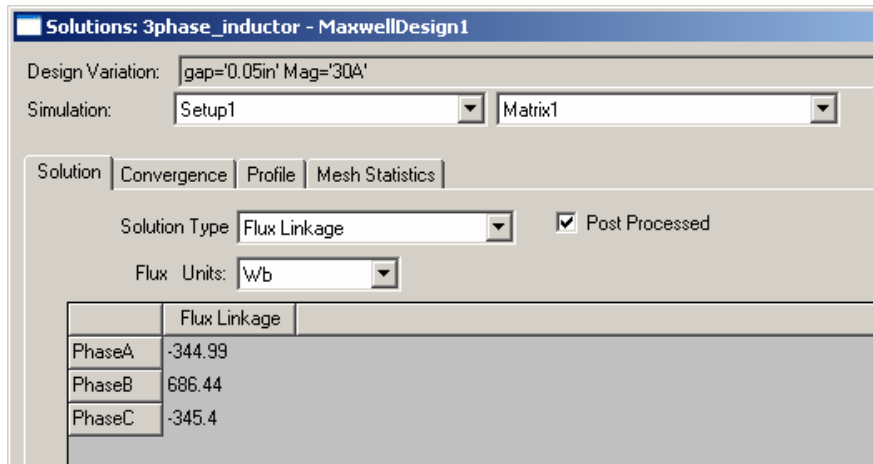
Inductance Matrix Solutions (continued)

- Change to **Matrix1** solutions by clicking on the pull-down menu with Matrix2.



	PhaseA	PhaseB	PhaseC
PhaseA	14.65	-6.9545	-5.5492
PhaseB	-6.9545	15.928	-6.9574
PhaseC	-5.5492	-6.9574	14.657

- Notice that the results change to reflect twice the turns and one-third the branches ($2^2 * 3^2 = 36$ times the previous inductance value).
- Remember that by increasing the turns, the current in each turn is decreased to keep the total ampere-turns constant - this is very important for nonlinear materials. The solutions apply only for this value of current.
- If you were to compare the non-post processed results for both matrices, you would see that these values are the same.
- Change the Solution Type from Inductance to Flux Linkage to see either the Post Processed or nominal flux linkage values.



	Flux Linkage
PhaseA	-344.99
PhaseB	686.44
PhaseC	-345.4

Example (Magnetostatic) - Inductance Calculation

Results and Interpretations

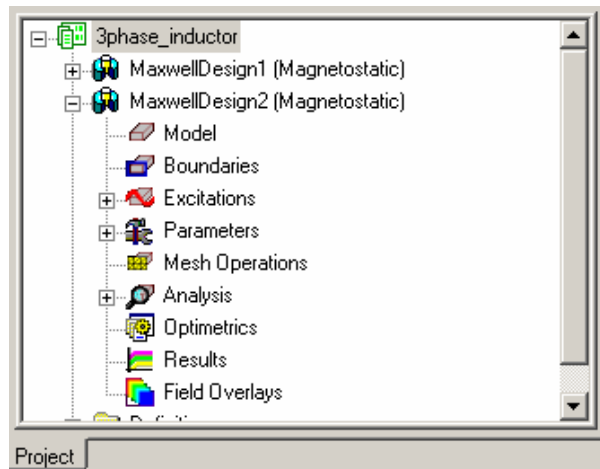
-  These results are being obtained from a magnetostatic simulation with arbitrary constant current inputs defined for each leg. The notation used was in terms of 3 phases, which might imply that this is an AC inductor of some sort. The modeling of AC currents in a magnetostatic simulation can sometimes be valid for instantaneous values (of force and inductance), but the requirements are that eddy currents do not effect the solution (either a low frequency source or low loss materials). The other two options available in the Maxwell software are eddy current simulations (which allows AC sources, but does not allow nonlinear materials), and transient simulations (which allows the study of nonlinear and AC phenomena).
-  This type of magnetostatic simulation would also be valuable for an Equivalent Circuit Extraction (again, eddy currents should be negligible) - this would require each phase to be excited as a variable, and run as a full parametric simulation including all phases across all possible current values (at least 5^3 variations to see any nonlinear behavior). Notice that if you do an equivalent circuit extraction of the inductance, the flux values automatically take into account the grouping and other post-processing values specified in the matrix parameter.
-  Other magnetostatic results can be obtained from this simulation, such as force on the Core_1 object and magnetic field in the steel (is it saturating?).
-  Remember that this simulation uses nonlinear materials, so all of the solutions are only valid for this specific set of excitations. If you were to change the input currents, then you will have to solve again to see how the inductance and other results change.
-  **Another important note is that although we included 2 matrix parameters, this is not necessary to see differently post-processed results.** You can edit the post-processing characteristics of the matrix after having solved the problem, and the results will adjust to reflect the changes without needing to resolve. The only time that you would need multiple matrices is if one matrix or the other did not include all the coils. **By using two matrices, the inductance calculation was performed twice, lengthening the simulation.** The point of this example was to show that multiple matrix parameters can be defined - not to suggest using multiple matrix parameters.

Example (Magnetostatic) - Inductance Calculation

Create a Symmetry Model

1. Right-click on **MaxwellDesign1** in the project tree (this is the design name).
2. Choose **Copy** from the menu.
3. Right-click on **3phase_inductor** in the project tree (this is the project name).
4. Choose **Paste** from the menu.
5. You now have a new design in your project that has the same setup as the first design.

Remember to double-click on the design name to switch from one design to the other!

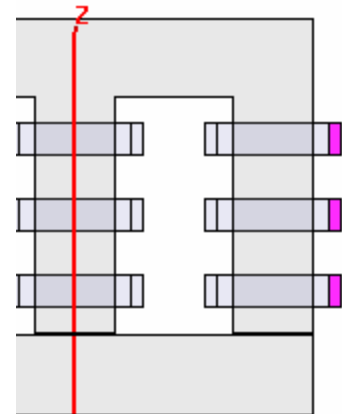
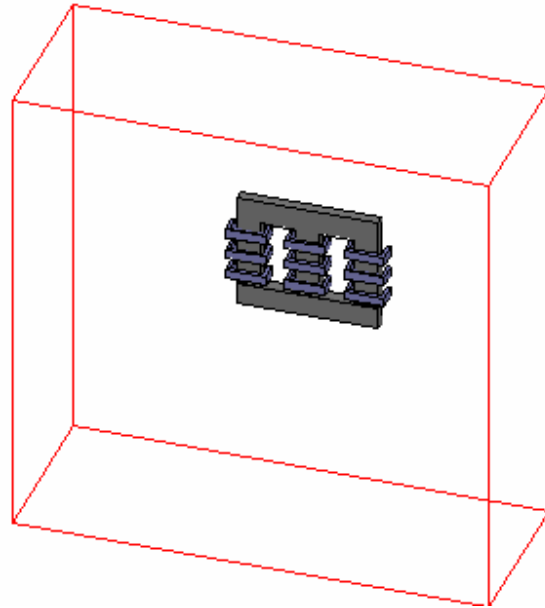


6. Select all the visible objects (except the Region) by typing **Ctrl + A** when within the 3D modeler window (or select them all in the object tree).
7. Select **3D Modeler > Boolean > Split...** to split the model at the YZ plane.
 1. In the Split window that appears, choose the following:
 1. Split Plane : **YZ**
 2. Keep Fragments: **Positive side**
 3. Split Objects: **Split entire selection**
 2. Select OK
8. Expect to see some errors and warnings that the sheet objects were deleted. There are options in the Split command to keep the sheet objects when splitting the solids, but we will not need the sheets for a symmetry simulation.

Example (Magnetostatic) - Inductance Calculation

Create a Symmetry Model (continued)

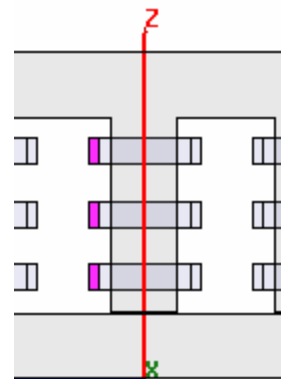
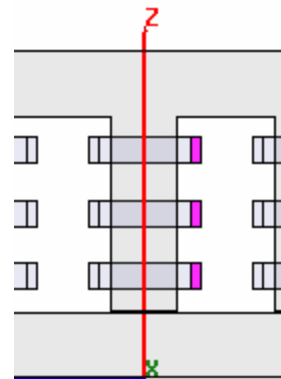
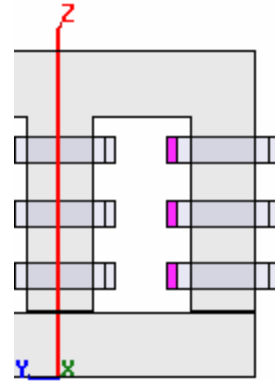
9. Select **Draw > Region** to edit the region object.
 1. Change the padding percents as follows:
 - +X: 800
 - X: 0
 2. Select **OK** to accept.
10. The entire geometry is now on the positive side of the YZ plane.
11. Adjust the visibility by going to **View > Active View Visibility...**
12. In the 3D Modeler tab, find the **Region** object and deselect Visibility.
13. Choose **OK** to accept.
14. Rotate the geometry to look at the split plane of the core and coils.
15. Type **f** in the 3D modeler to switch to face selection mode (type **o** to switch back to object selection) - face selection can also be found in **Edit > Select > Faces**.
16. Select the right-most coil faces on the YZ plane.
(Use **Ctrl** to select multiple items graphically.)
17. Select **Maxwell > Excitations > Assign > Current...**
18. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseA_in**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
19. Select **OK** to accept the excitation.



Example (Magnetostatic) - Inductance Calculation

Create a Symmetry Model (continued)

20. Select the next right-most coil faces on the YZ plane.
(Use **Ctrl** to select multiple items graphically.)
21. Select **Maxwell > Excitations > Assign > Current...**
22. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseA_out**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
 4. Click **Swap Direction** so that the arrow is pointing in the negative X direction.
23. Select **OK** to accept the excitation.
24. Continue with the next leg by selecting the next 3 coil faces on the YZ plane.
25. Select **Maxwell > Excitations > Assign > Current...**
26. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseB_in**
 2. Value: **Mag**
 3. Type: **Stranded**
27. Select **OK** to accept the excitation.
28. Now select the 3 return faces of the middle coil.
29. Select **Maxwell > Excitations > Assign > Current...**
30. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseB_out**
 2. Value: **Mag**
 3. Type: **Stranded**
 4. Click **Swap Direction** so that the arrow is pointing in the negative X direction.
31. Select **OK** to accept the excitation.

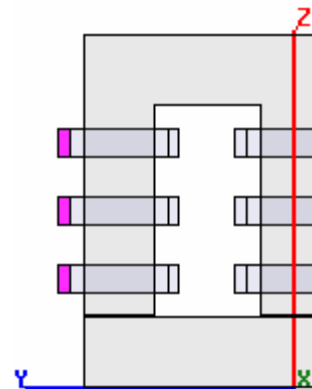
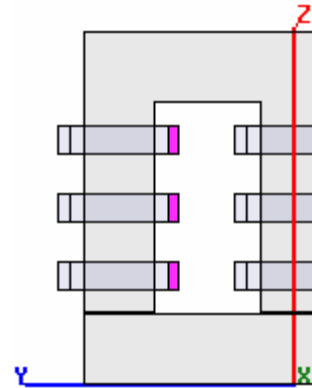


Example (Magnetostatic) - Inductance Calculation

Create a Symmetry Model (continued)

32. Continue to the last leg by selecting the next 3 coil faces on the YZ plane.
33. Select **Maxwell > Excitations > Assign > Current...**
34. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseC_in**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
35. Select **OK** to accept the excitation.

36. Finish by selecting the 3 return faces of the last coil.
37. Select **Maxwell > Excitations > Assign > Current...**
38. In the Current Excitation dialog, enter the following:
 1. Base Name: **PhaseC_out**
 2. Value: **-0.5*Mag**
 3. Type: **Stranded**
 4. Click **Swap Direction** so that the arrow is pointing in the negative X direction.
39. Select **OK** to accept the excitation.



Creating Matrix Parameters

-  Now that the excitations are redefined, the matrix has to be redefined. The matrices were deleted when the old excitations were deleted (which were deleted when the sheet objects were deleted during the split operation).

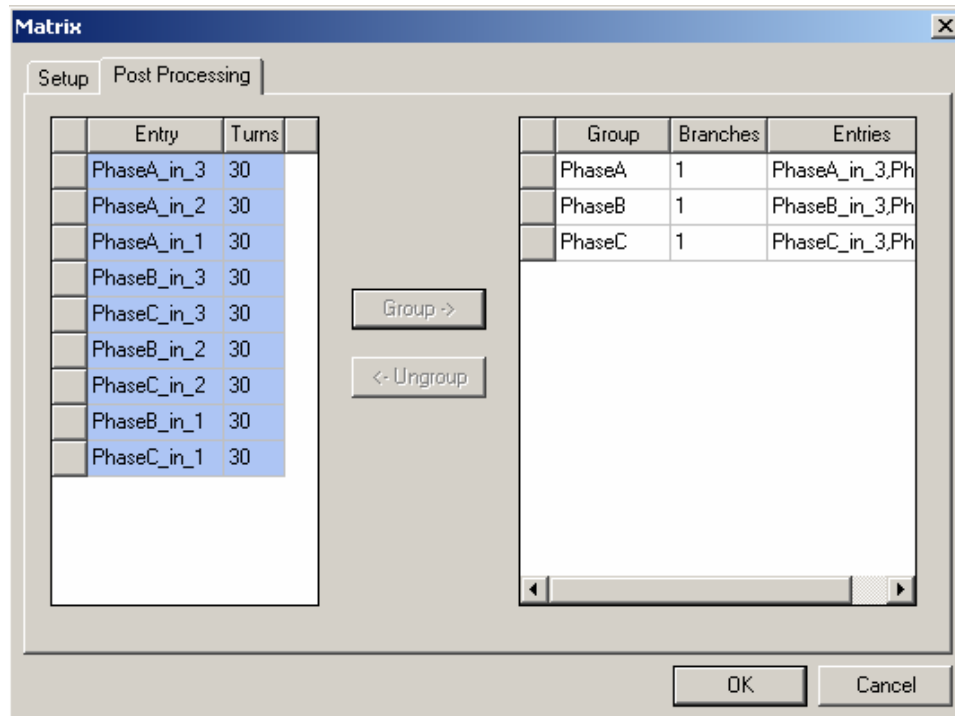
-  When matrix parameters are defined, a check will be performed for proper conduction paths and for terminals correctly defined across a cross-section of the conduction paths. Several errors can appear when defining a matrix, and these often have to do with the definition of the conducting objects or the excitation.

-  The excitations that appear in the matrix setup, are those excitations that are pointing into the conductor. So, for a symmetry model that requires two excitations per conductor, only one excitation per conductor will be listed in the matrix setup - this excitation will be the one pointing into the modeled conductor.

Example (Magnetostatic) - Inductance Calculation

Create Matrix Parameters

1. Select **Maxwell > Parameters > Assign > Matrix...**
 1. In the Setup tab, check the boxes to Include all the Sources (or click on the word Include to automatically check all the boxes).
 2. In the Post Processing tab, set the turns for each source to **30**
 3. Select **PhaseA_in_1**, **PhaseA_in_2**, and **PhaseA_in_3** and select **Group ->** (Use **Ctrl** or **Shift** to select multiple items.)
 4. Select **PhaseB_in_1**, **PhaseB_in_2**, and **PhaseB_in_3** and select **Group ->**
 5. Select **PhaseC_in_1**, **PhaseC_in_2**, and **PhaseC_in_3** and select **Group ->**
 6. Rename Group1 to **PhaseA**
 7. Rename Group2 to **PhaseB**
 8. Rename Group3 to **PhaseC**
 9. Leave Branches set to 1 for each Group

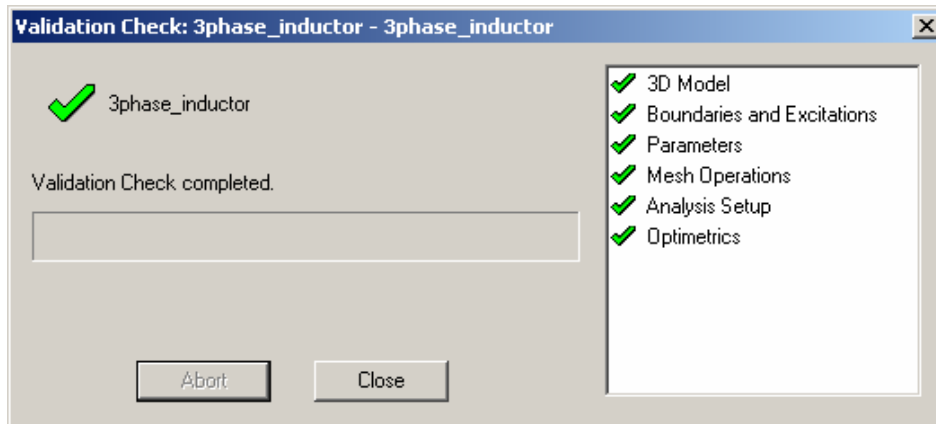


- This setup says that each source has 30 turns and that the sources on each leg are connected in series (as far as the post-processing is concerned).

Example (Magnetostatic) - Inductance Calculation

Check the Validity of the Model

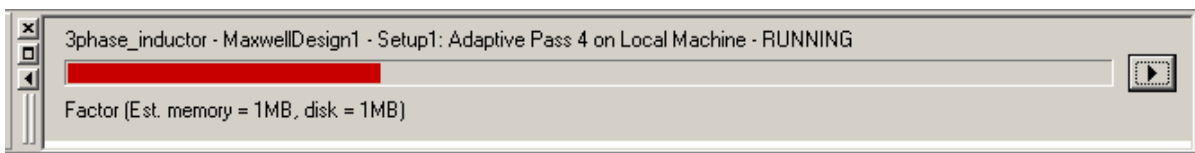
- Select the menu item **Maxwell > Validation Check**, or click on the  icon



- The problem won't solve if there are errors (errors are denoted by a red X), but it will solve if all the validations return a green check or a warning (warnings are denoted by a yellow triangle, and should be noted, but can often be safely ignored).

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the  icon
- This should take about 5 minutes to solve on a standard computer (such as a fast laptop or standard desktop), and about 3 minutes on a fast computer (half the time!).

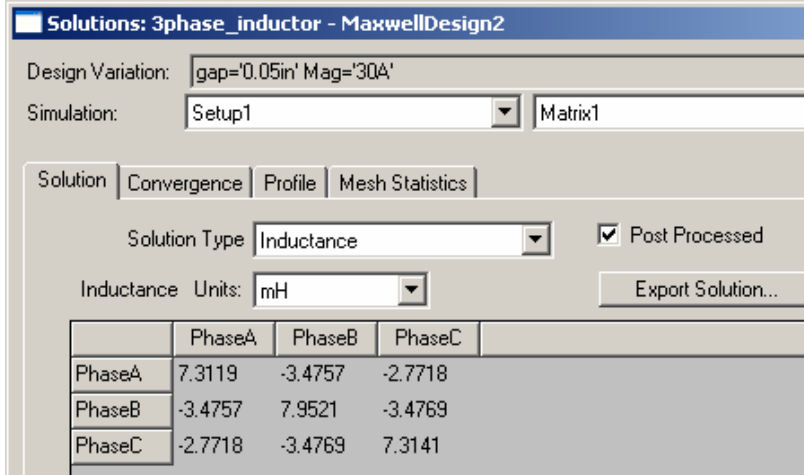


- This half model runs twice as fast - taking advantage of symmetries can have an even greater effect on larger models!

Example (Magnetostatic) - Inductance Calculation

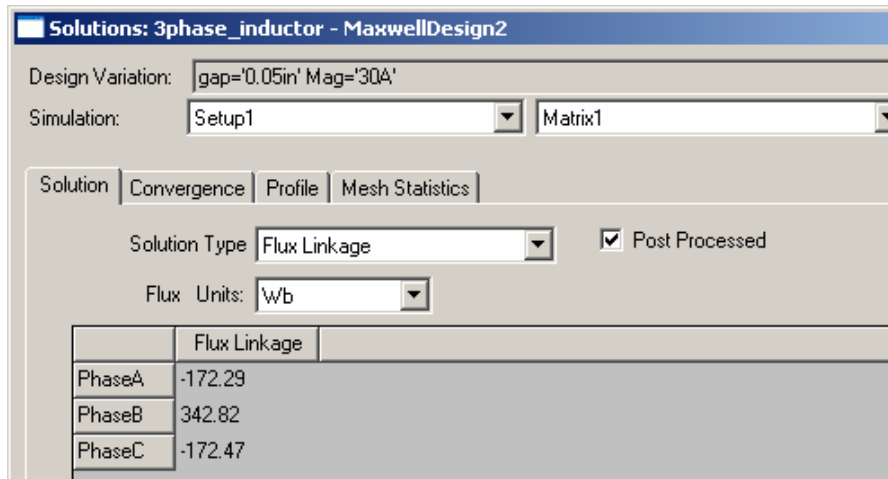
Inductance Matrix Solutions

1. Open up the matrix solution data by selecting **Maxwell > Results > Solution Data**
2. Check the box marked **Post Processed** to see the results.



	PhaseA	PhaseB	PhaseC
PhaseA	7.3119	-3.4757	-2.7718
PhaseB	-3.4757	7.9521	-3.4769
PhaseC	-2.7718	-3.4769	7.3141

- Notice that the results are half the value that we obtained for the full model. This is very important to remember when using these values. The halved results are consistent for other parameters and properties as well (i.e. torque, total energy, etc.)
 - If these values are used to create an Equivalent Circuit Extraction, remember to use a scaling factor of 2 to account for the symmetry.
3. Change the Solution Type from Inductance to Flux Linkage.
 - Notice that these numbers are halved as well.



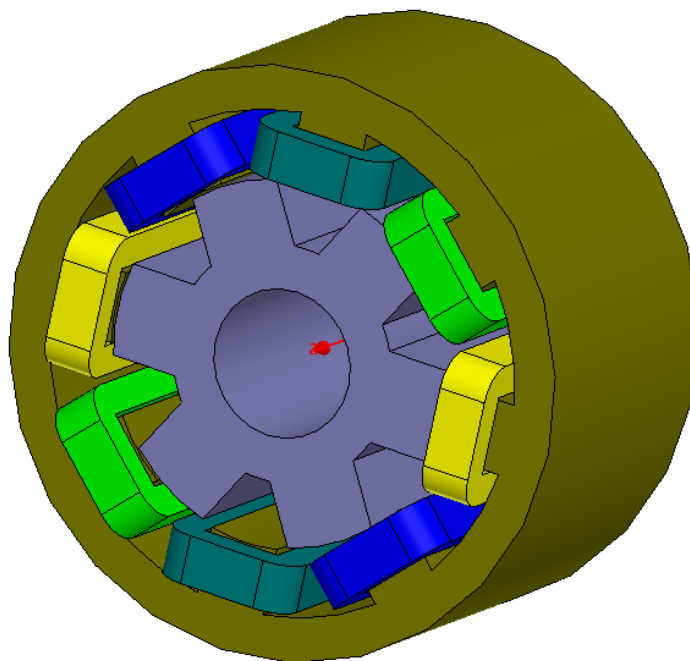
	Flux Linkage
PhaseA	-172.29
PhaseB	342.82
PhaseC	-172.47

THIS PAGE INTENTIONALLY LEFT BLANK

Example (Magnetostatic) - Stranded Conductors

Stranded Conductors

- ▲ This example is intended to show you how to create and analyze a magnetostatic problem with stranded conductors for a Switched Reluctance Motor geometry using Magnetostatic solver in the [Ansoft Maxwell 3D Design Environment](#).
- ▲ Within the Maxwell 3D Design Environment, complete 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. Secondly, defining a Stranded Conductor means that the current density will be uniform throughout the cross section of the conductor which is physically correct for a coil with many turns.
- ▲ 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.
- ▲ Note: 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.

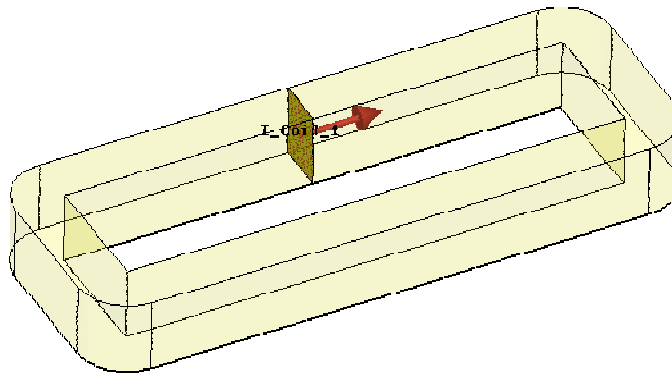


Example (Magnetostatic) - Stranded Conductors

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 as well as through the entire conductor.
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. 1 amp thru 3570 turns
2. 5 amps thru 750 turns
3. 25 amps thru 150 turns
4. 3570 amps thru 1 turn

or any such combination that produces a value of 3750 ampere-turns. The ratio of amperes to turns does not impact the field solution or the value of inductance reported in the inductance matrix. However, this value of inductance assumes a 1 turn coil, and must be multiplied by N^2 to determine the actual inductance of the coil. Please refer to the overview for details of the inductance calculation.

Example (Magnetostatic) - Stranded Conductors

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 m²

$\hat{n}$ is the unit normal direction

$\vec{J}$ is the Current Density vector in A/m²

-  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:

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

Example (Magnetostatic) - Stranded Conductors

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**

Example (Magnetostatic) - Stranded Conductors

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**

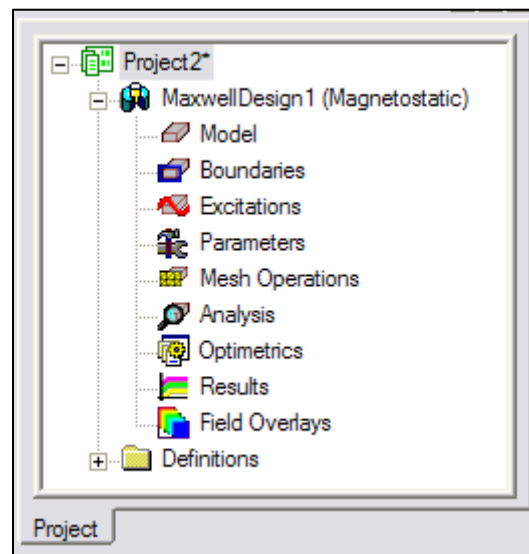
3. Click the **OK** button

Example (Magnetostatic) - Stranded Conductors

Opening a New Project

To open a new project:

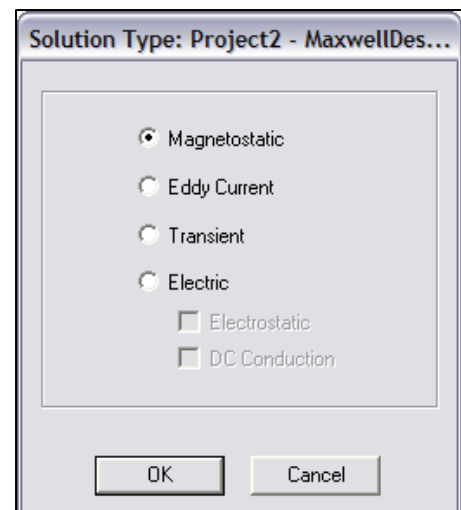
- ▲ In an Ansoft Maxwell window, click the  On the Standard toolbar, or select the menu item **File > New**.
- ▲ From the **Project** menu, select **Insert Maxwell Design**.



Set Solution Type

To set the solution type:

- ▲ Select the menu item **Maxwell > Solution Type**
- ▲ Solution Type Window:
 - ▲ Choose **Magnetostatic**
 - ▲ Click the **OK** button



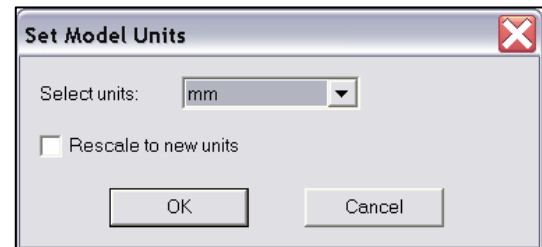
Example (Magnetostatic) - Stranded Conductors

Creating the 3D Model

Set Model Units

To set the units:

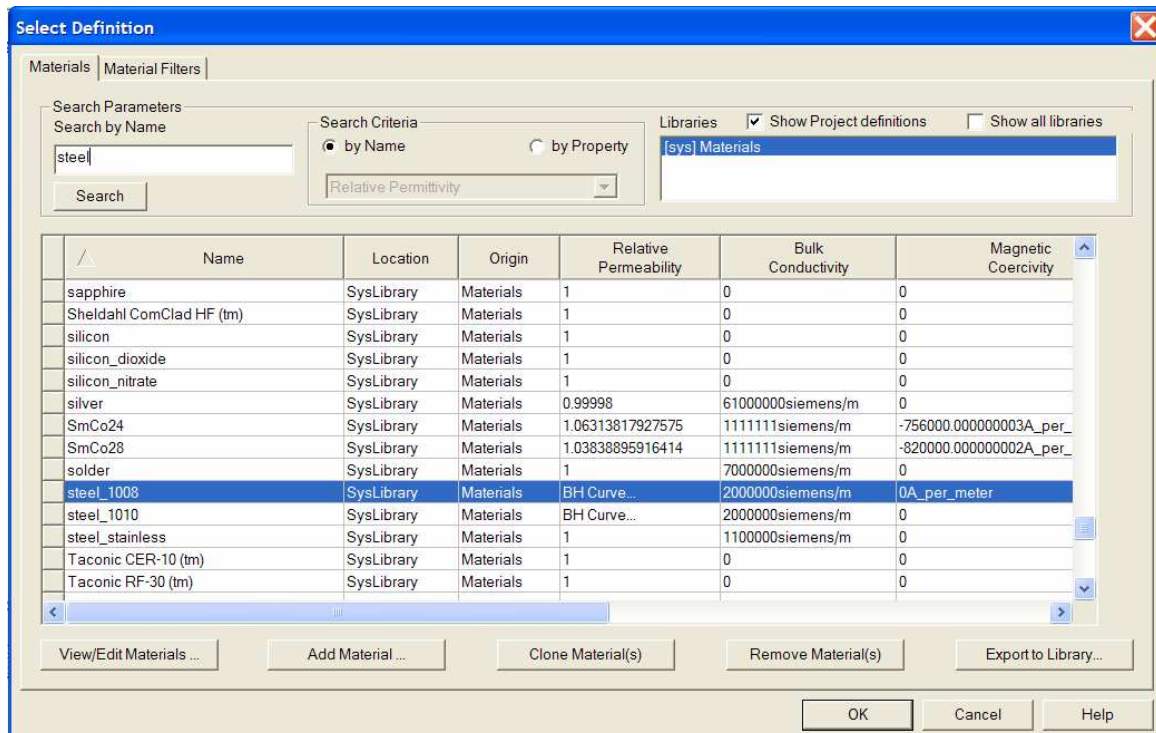
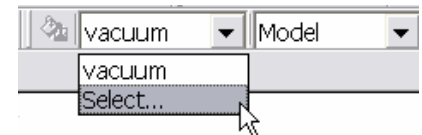
1. Select the menu item **3D Modeler > Units**
2. Set Model Units:
 1. Select Units: **mm**
 2. Click the **OK** button



Set Default Material

To set the default material:

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



Example (Magnetostatic) - Stranded Conductors

Create Rotor

To create the rotor:

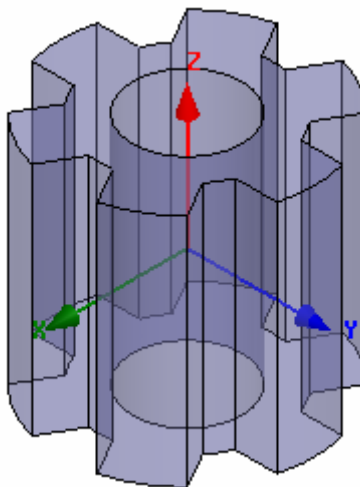
1. Select the menu item **Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore**
2. From the **Create User Defined Primitive** dialog
 1. For the value of **DiaGap**, type: **70**, Click the **Tab** key to accept
 2. For the value of **DiaYoke**, type: **30**, Click the **Tab** key to accept
 3. 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**.



Example (Magnetostatic) - Stranded Conductors

Create Stator and Coils:

To create the Stator and Coils

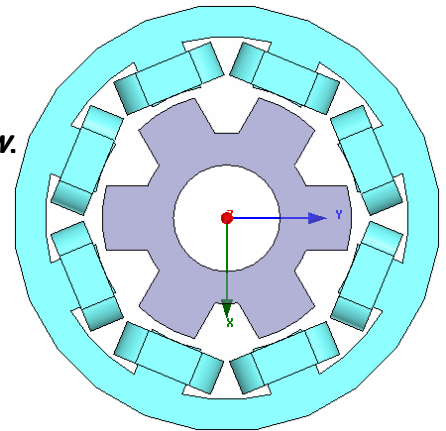
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
 1. For the value of **DiaGap**, type: **75**, Click the **Tab** key to accept
 2. For the value of **DiaYoke**, type: **120**, Click the **Tab** key to accept
 3. 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*

Example (Magnetostatic) - Stranded Conductors

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 *3D Modeler > Surface > Section*
 1. Section Plane: **XY**
 2. Click the **OK** button

Separate Bodies

1. Select the menu item *3D Modeler > 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

Example (Magnetostatic) - Stranded Conductors

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

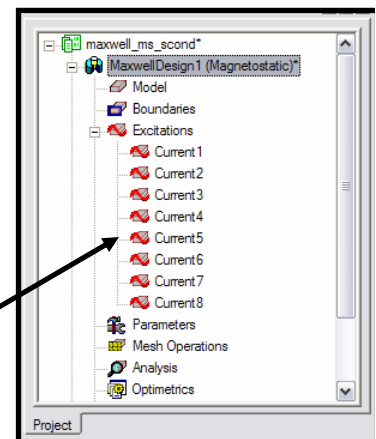
To select objects

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

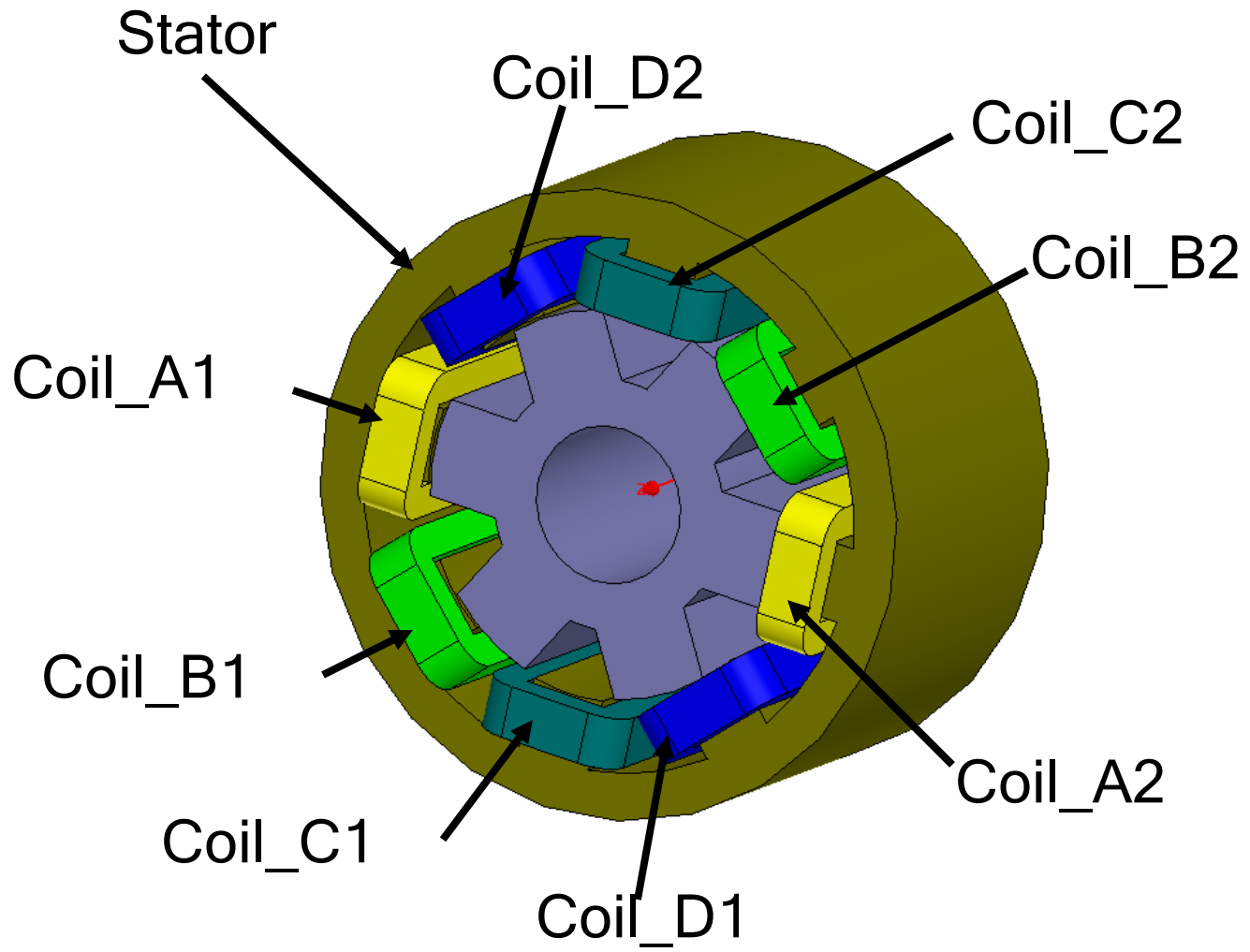
To duplicate the objects:

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



Example (Magnetostatic) - Stranded Conductors



Example (Magnetostatic) - Stranded Conductors

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:

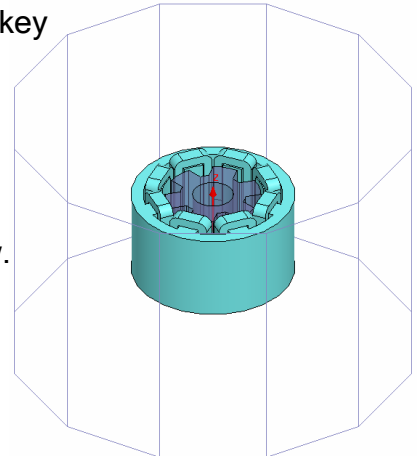
1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value of Name** type: **Region**
3. **Display Wireframe:** ☒ **Checked**
4. Click the **OK** button

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.**



Example (Magnetostatic) - Stranded Conductors

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

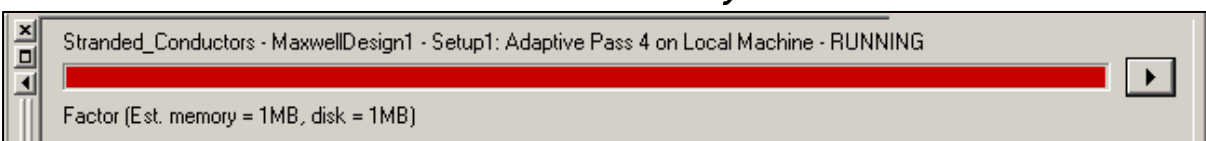
To validate the model:

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**



Example (Magnetostatic) - Stranded Conductors

Solution Data

To view the Solution Data:

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

To view the Profile:

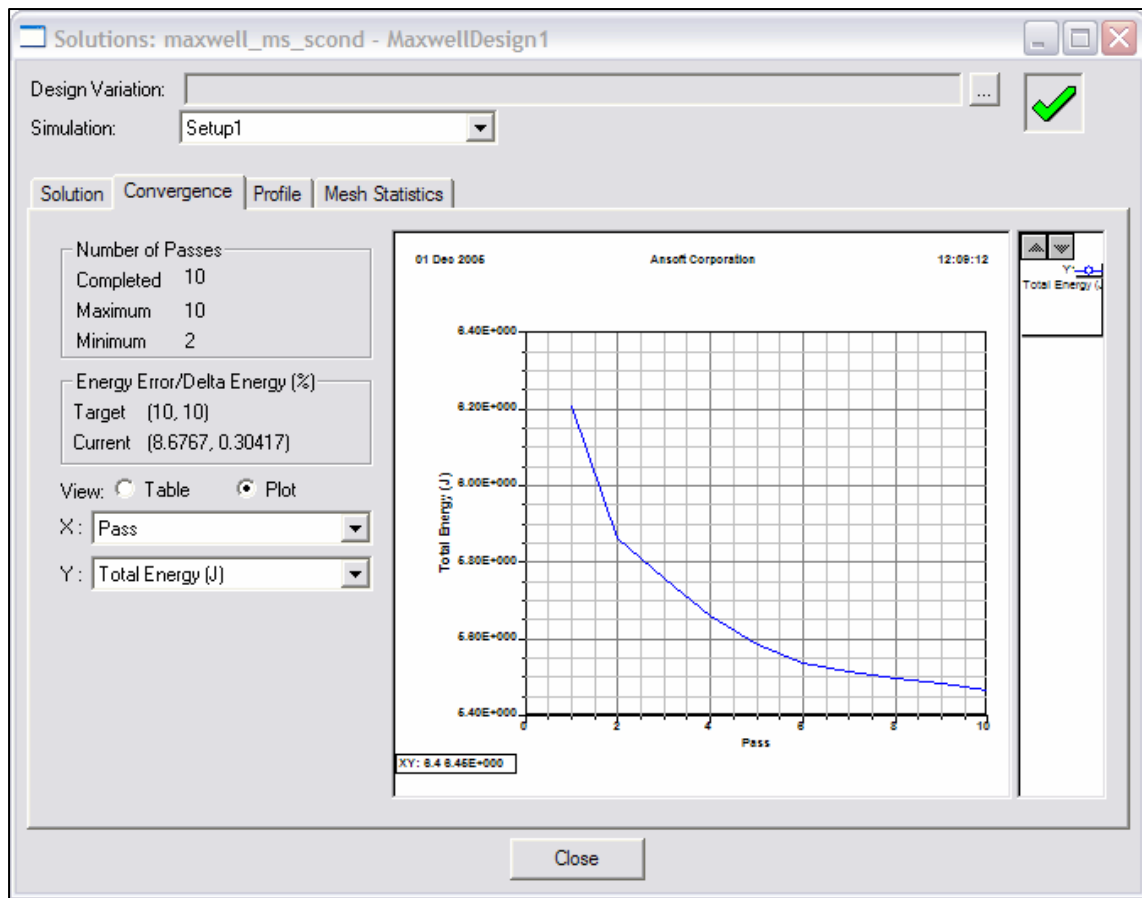
1. Click the **Profile** Tab.

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



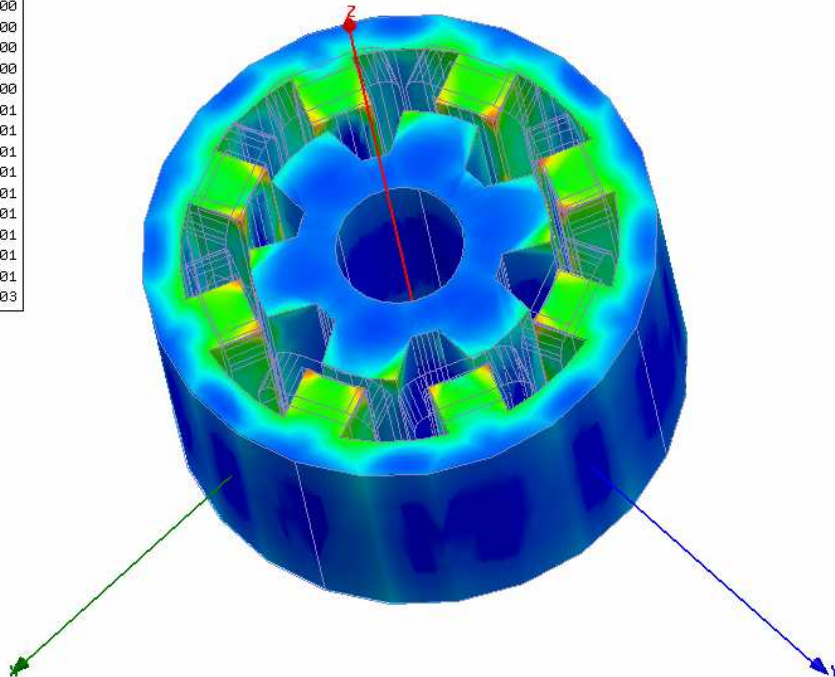
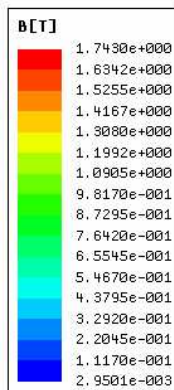
Example (Magnetostatic) - Stranded Conductors

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. In the Create Field Plot window select:
 1. Solution: **Setup1 : LastAdaptive**
 2. Quantity: **Mag_B**
 3. In Volume: **All**
 4. Plot on Surface Only: ☒ **Checked**
 5. Click the **Done** button

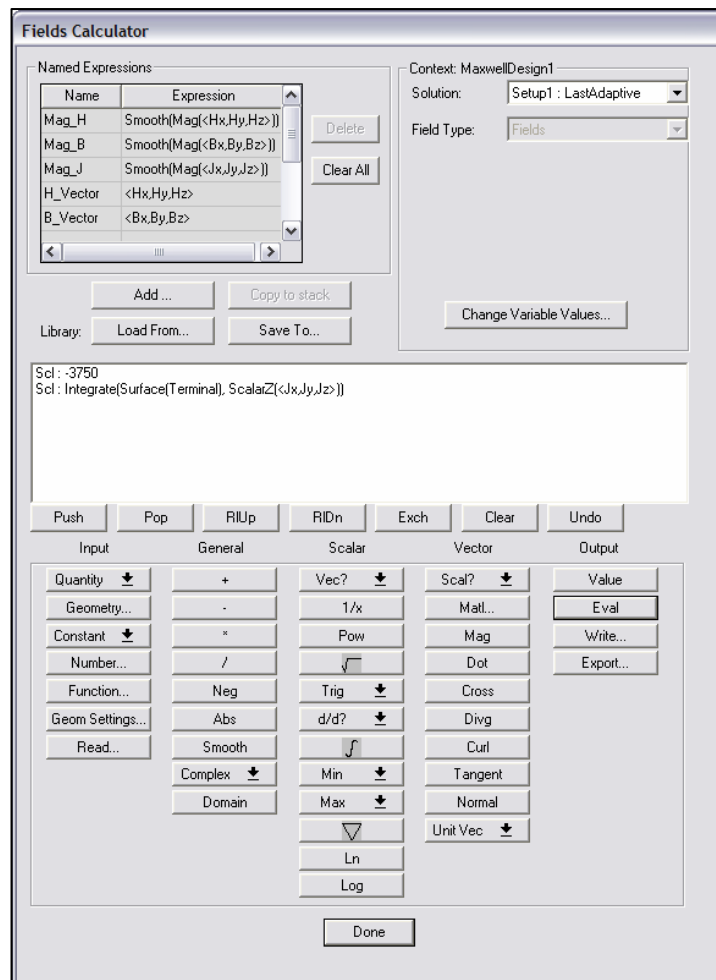


Example (Magnetostatic) - Stranded Conductors

Calculate Current

Use the field calculator to verify the total amp-turns flowing in the winding:

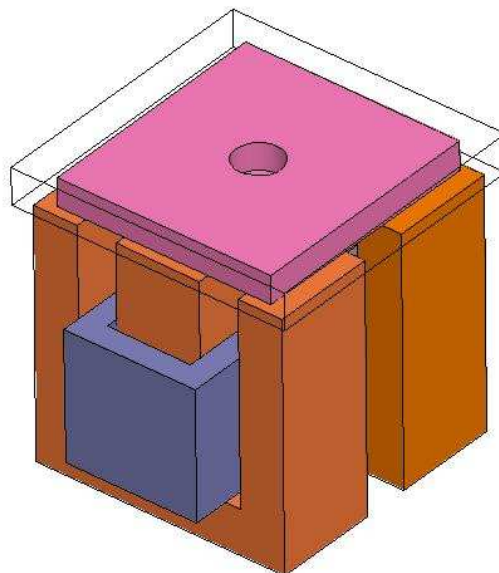
1. Select the menu item **Maxwell > Fields > Calculator**
2. Select **Quantity: J** which is a vector
3. Click the button **Geometry** to pull up the Geometry Window
 1. Select the radio button **Surface**
 2. From the list select: **terminal**
 3. Click the **OK** button
4. Click on the **Normal** button to determine the current normal to the terminal
5. Click the $\int$ (Integrate) button
6. Click the **Eval** button to evaluate the results of the integration and return the amp-turns flowing in the winding = 3750.
7. Click the **Done** button to close the calculator



THIS PAGE INTENTIONALLY LEFT BLANK

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.



▲ 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**

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

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

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

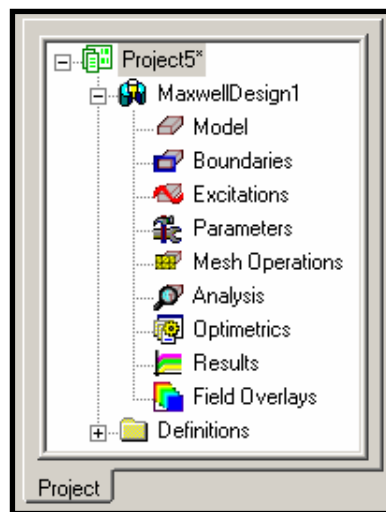
-  Edit property of new primitives: ☒ **Checked**

3. Click the **OK** button

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

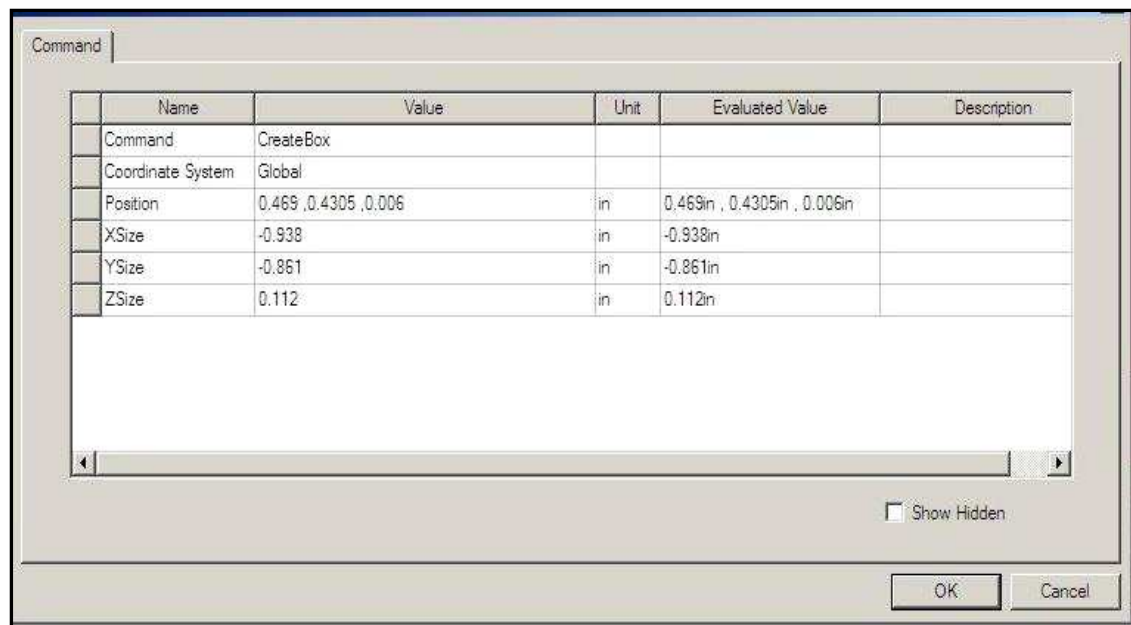
Set Model Units

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

Topic - ECE: Linear movement

▲ Create the Armature:

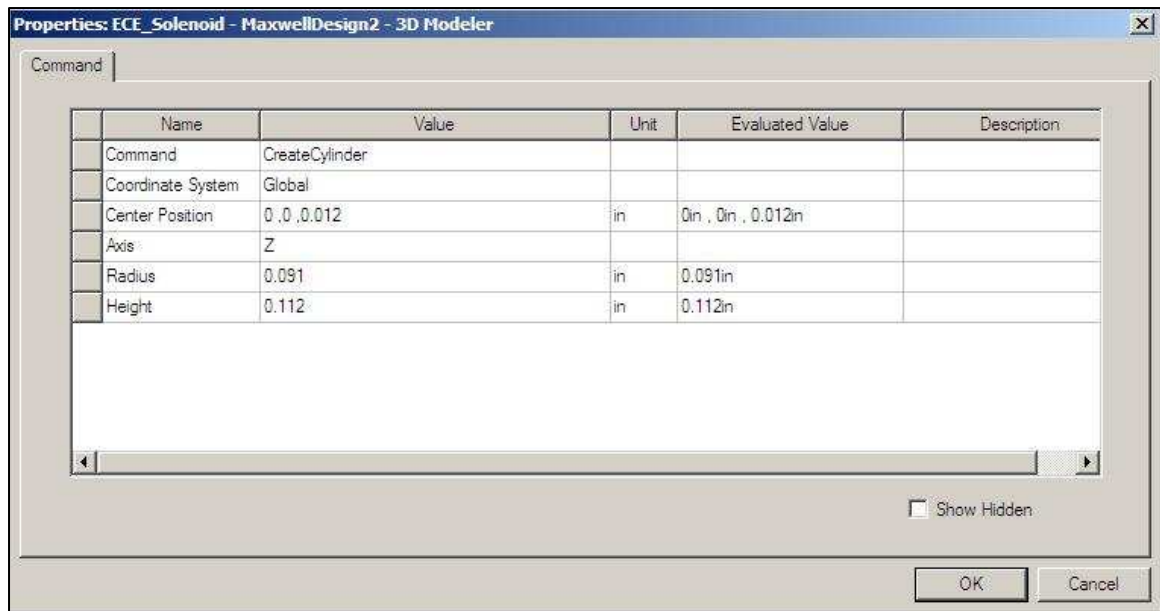
- ▲ The armature is made of a box with a cylinder hole.
- ▲ Create the armature . Select the Menu item **Draw > Box** or click on the  icon.
- 1. Using the coordinate entry field, enter the box position
 - ▲ **X: 0.469, Y: 0.4305, Z: 0.006**, Press the **Enter** key
- 2. Using the coordinate entry field, 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**.

Topic - ECE: Linear movement

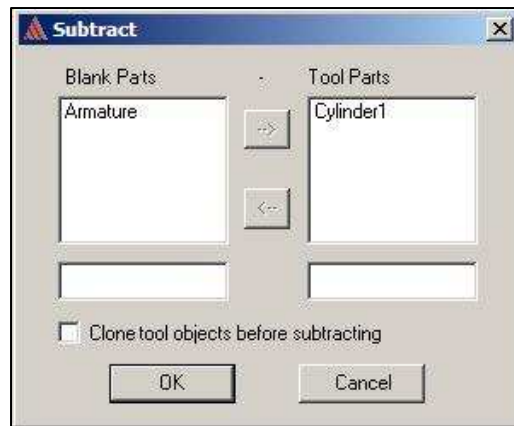
- ▲ Create the hole **Draw > Cylinder** or click on the  icon.
 1. Using the coordinate entry field, enter the center of base
 - ▲ **X: 0.0, Y: 0.0, Z: 0.012**, Press the **Enter** key
 2. Using the coordinate entry field, enter the coordinates of a external point of the cylinder base
 - ▲ **X: 0.091, 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.



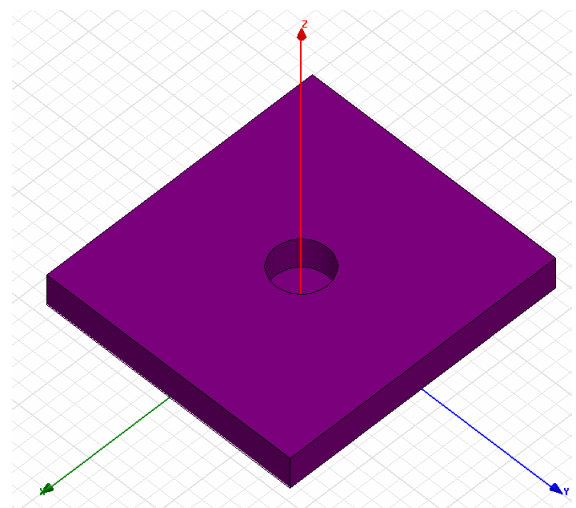
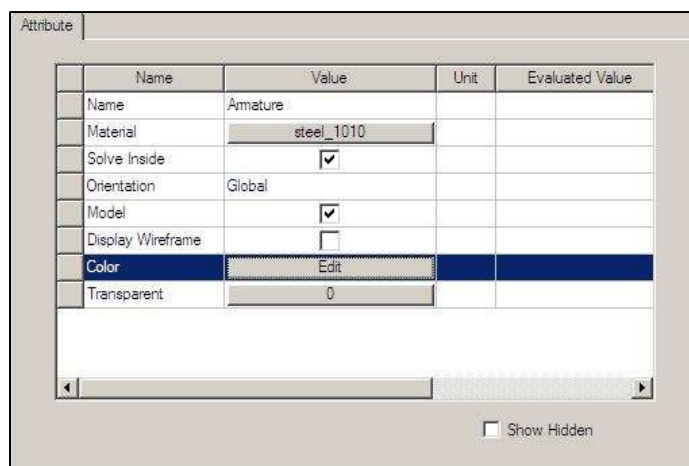
- ▲ Select the objects **Armature** and **Cylinder1** from the objects tree (maintain the Ctrl key to select both objects).

Topic - ECE: Linear movement

- ▲ 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.

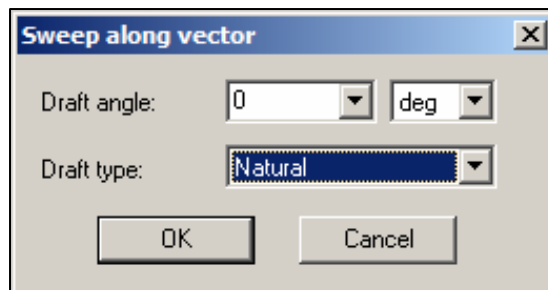


Create the Stators

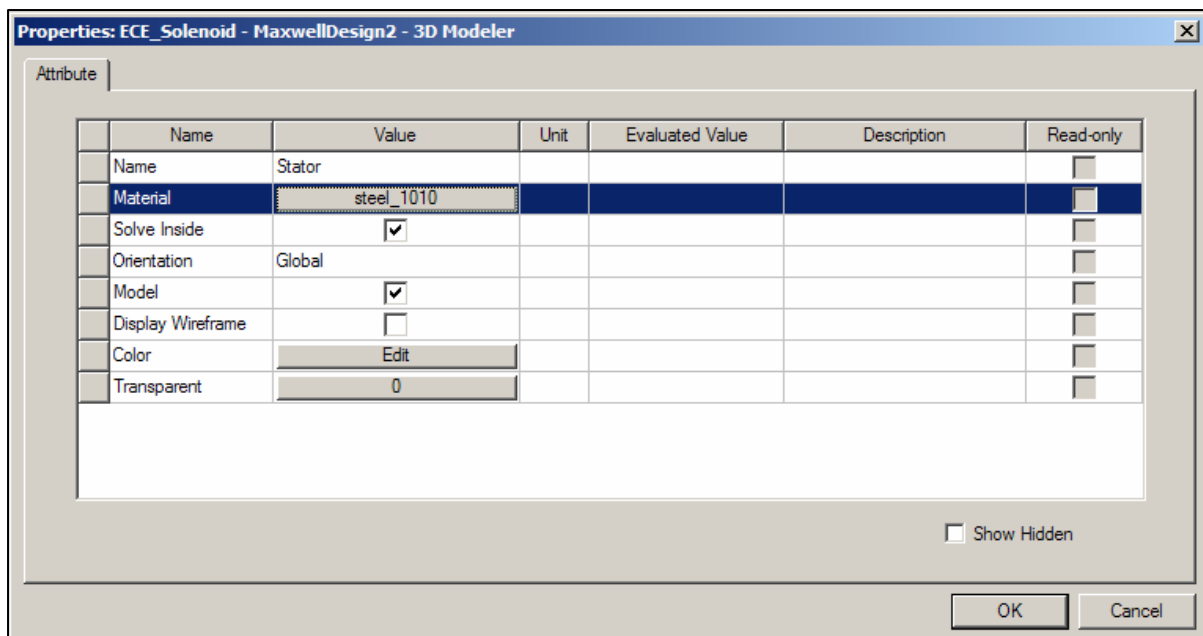
-  The two stators are created using a polyline.
-  Create the polyline **Draw > Line** or click on the  icon.
 1. Using the coordinate entry field, enter the first point of the line
 -  **X: 0.5025, Y: 0.0, Z: 0.0**, Press the **Enter** key
 2. Using the coordinate entry field, enter the second point 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: -0.324, 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: -0.803**, Press the **Enter** key
 13. 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
 14. 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**.

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
(Note that the following coordinates are ABSOLUTE not RELATIVE.)
 - ▲ **X: 0.091, Y: 0.0, Z:0.0**, Press the **Enter** key.

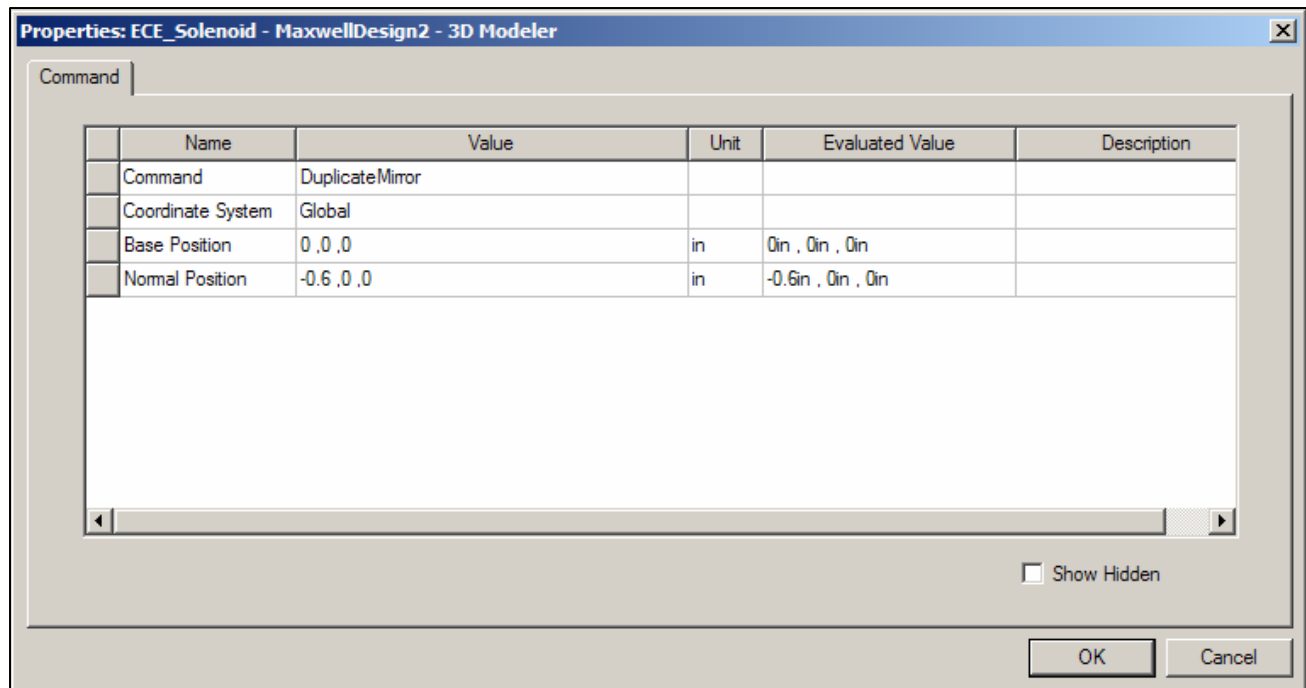


- ▲ Leave the Draft angle to 0 deg and change Draft type to Natural.

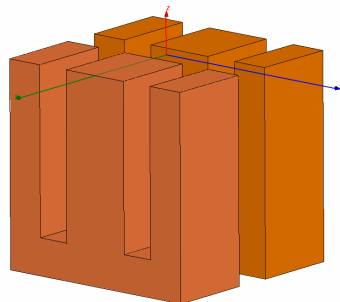


Topic - ECE: Linear movement

- ▲ 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
 - (Note that the following coordinates are ABSOLUTE not RELATIVE.)
 - ▲ **X: -0.6, Y: 0.0, Z:0.0**, Press the **Enter** key

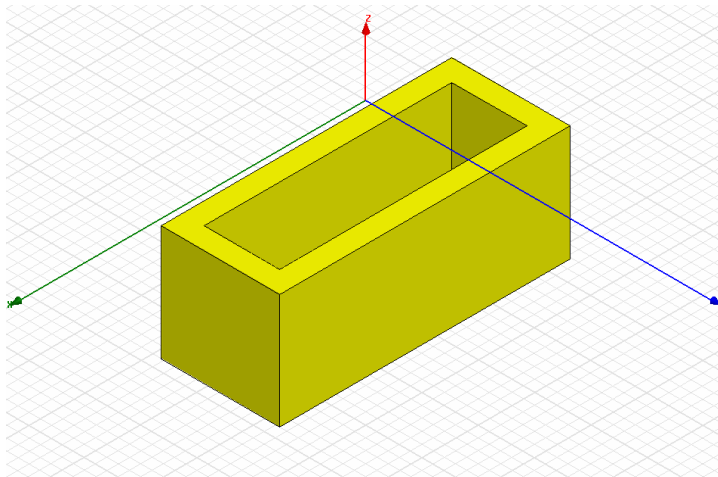


- ▲ After the operation, a window appears and recall the coordinates.
- ▲ A new object **Stator_1** has been created with the same material property and the same color as **Stator**.



Create the Coil:

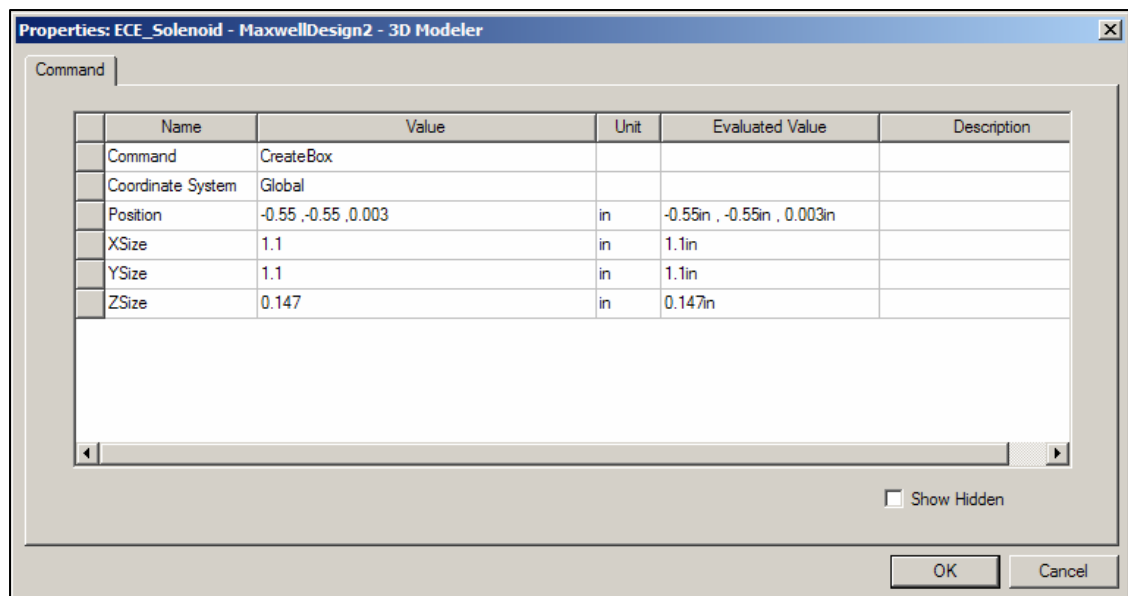
- ▲ The coil is created by drawing two boxes and subtracting the smaller box from the larger one..
- ▲ Create the first box using **Draw > Box** or click on the  icon.
 1. Using the coordinate entry field in the lower right corner of the modeling window, enter the first corner of the box
 - ▲ **X: - 0.646, Y: - 0.264, Z: -0.803**, Press the **Enter** key
 2. Next, enter the change in position to define the opposite corner of the box
 - ▲ **dX: 1.292, dY: 0.528, dZ: 0.511**, Press the **Enter** key.
 3. Change the name of the box from **Box1** to **Coil** and leave the other default parameters.
- Note:** If you happen to have an incorrect coordinate, you can modify any entry from the model's history tree.
- ▲ Create the second box using **Draw > Box** or click on the  icon.
 1. Using the coordinate entry field in the lower right corner of the modeling window, enter the first corner of the box
 - ▲ **X: - 0.55, Y: - 0.168, Z: -0.803**, Press the **Enter** key
 2. Next, enter the change in position to define the opposite corner of the box
 - ▲ **dX: 1.1, dY: 0.336, dZ: 0.511**, Press the **Enter** key.
 3. Change the name of the box from **Box1** to **Basecoil** and leave the other default parameters.
- ▲ Subtract the boxes to create the coil using **3D Modeler > Boolean > Subtract** or click on the  icon.
 1. Make sure to have **Coil** listed in Blank Parts and **Box1** listed in Tool Parts. Use the arrows to exchange the objects if necessary.
 2. Assign **Copper** as material. Change the color in blue.



Topic - ECE: Linear movement

Create the Band

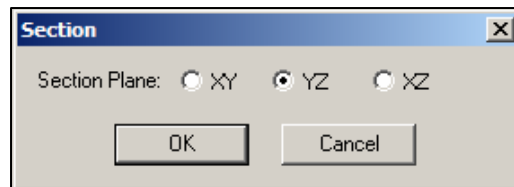
- In our study, the **Armature** will be moved up and down along the Z axis. It is always a good idea to help the mesh creation by creating an air box object around a object that will be moved (even if the project will be performed with the Magnetostatic solver).
- Create the air object **Draw > Box**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

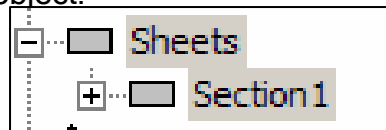
Create the Coil Terminal

- ▲ 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**.

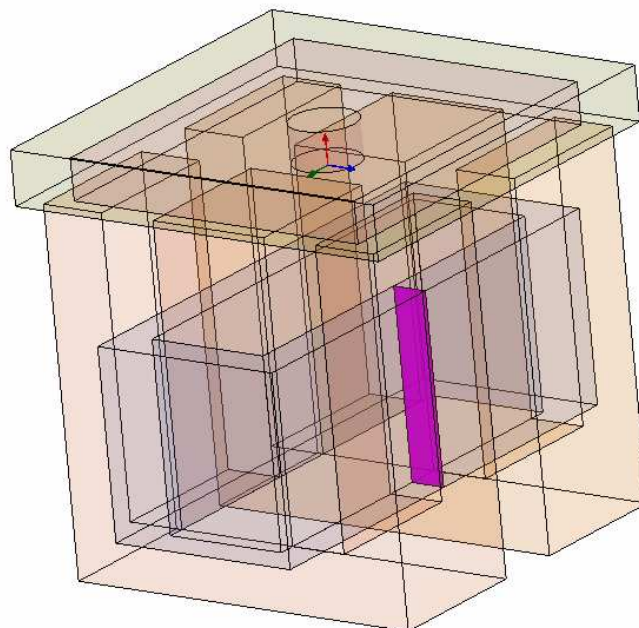


Separate the Grouped Terminals

- ▲ 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**.

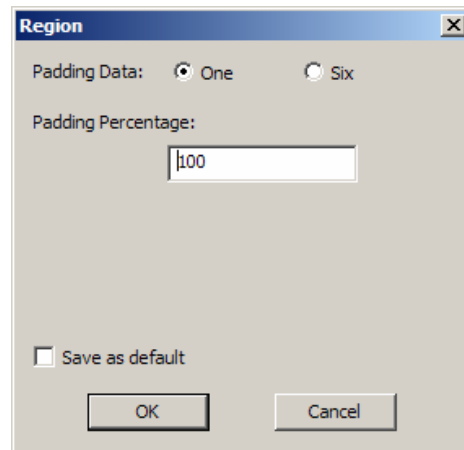


Topic - ECE: Linear movement

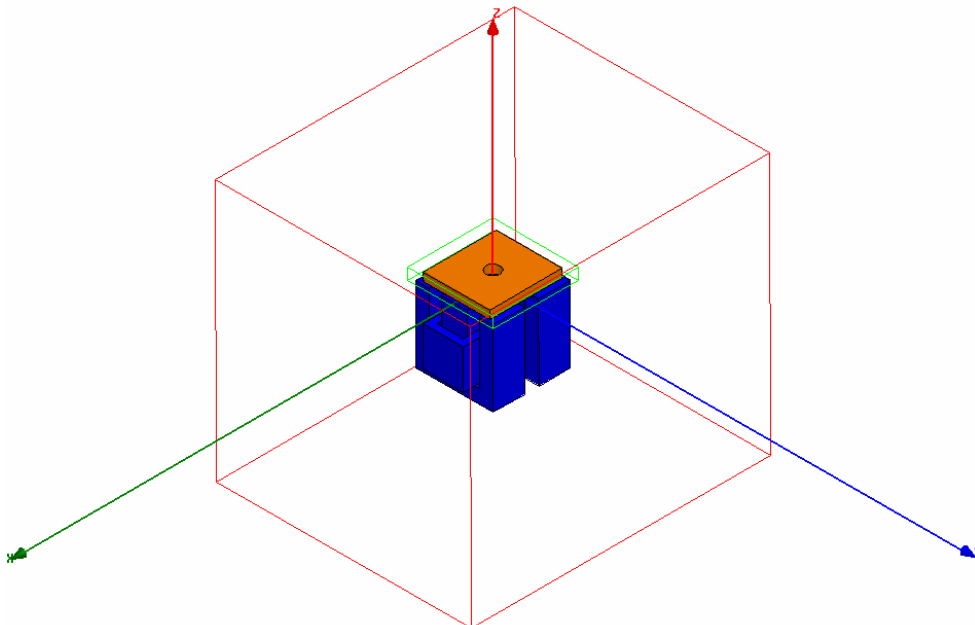
Create the Problem Region

- 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

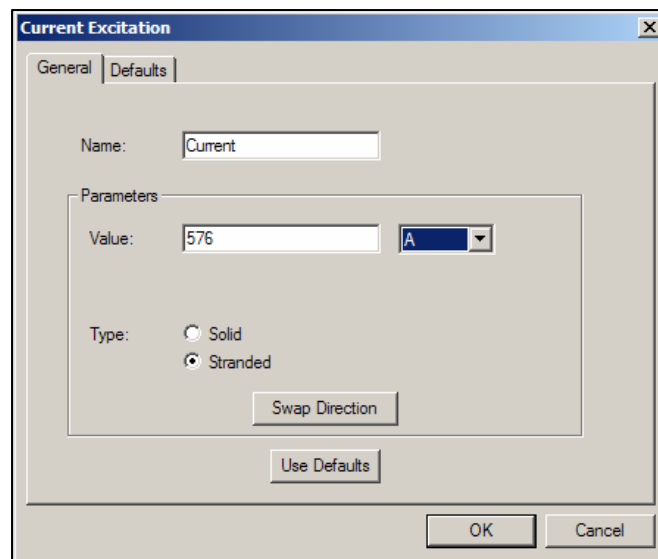


- The Padding determines how far will the limits of the region be from the objects. In this case, select 100 as Padding Percentage to have plenty of room around the solenoid.



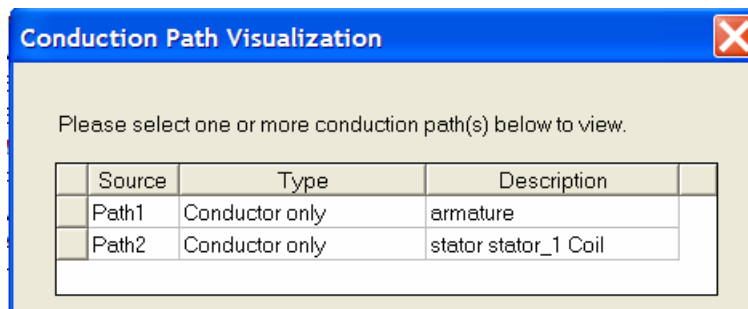
Assign Current Source to the Terminal

-  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 Boundary Conditions

-  We assume that no current leaks out of the coil and into the stators. However, the stators and the coil are touching, so an insulating boundary must be applied to the either the coil or to both stators
-  Select the menu item **Maxwell > Excitations > Conduction Paths > Show Conduction Paths** to verify the the coil is touching the stators.

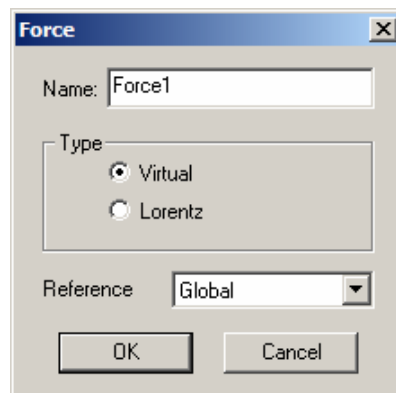


Assign Boundary Conditions (Continued)

- 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 > Boundaries > Assign > Insulating**. Leave the default name Insulating1 and select OK.

Assign Executive Parameters

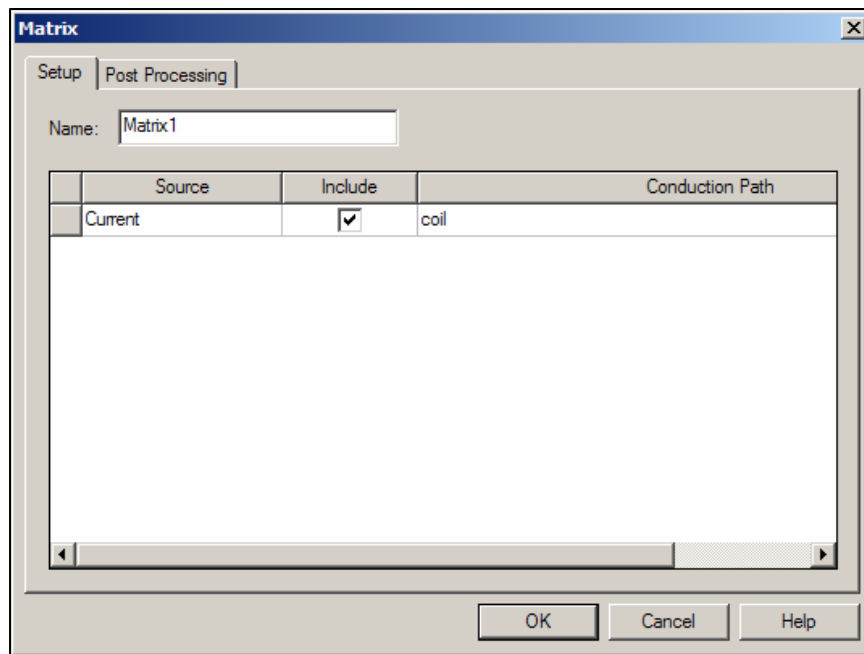
- The force applied on the armature as well as the self inductance of the coil are needed. These outputs will be used during the Equivalent Circuit Extraction process.
- Select the **Armature** from the objects tree or by using the mouse.
- Select the menu item **Maxwell > Parameters > Assign > Force**



- 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 since it works on all objects. The Lorentz force calculation works only on current carrying conductors with permeability = 1. Also, 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).

Assign Executive Parameters (Continued)

- ▲ Select the menu item Edit > Select > By Name to choose the coil.
- ▲ Select the menu item **Maxwell > Parameters > Assign > Matrix**
- ▲ Include the term current in the matrix computation (see below).

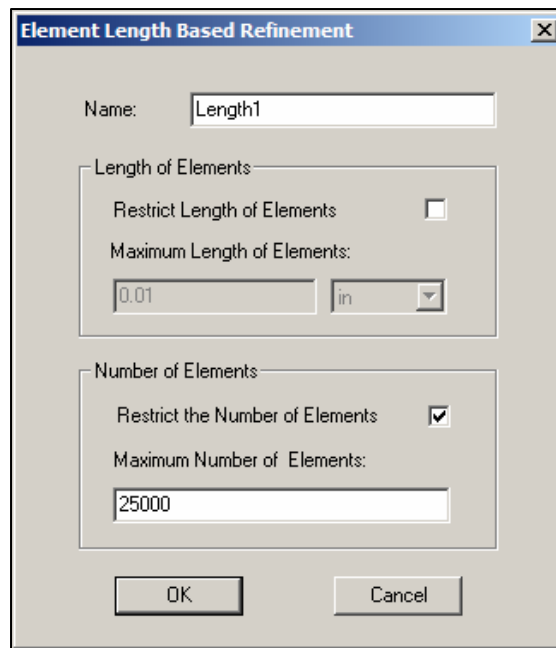


Assign Mesh Operations

- ▲ 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**

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

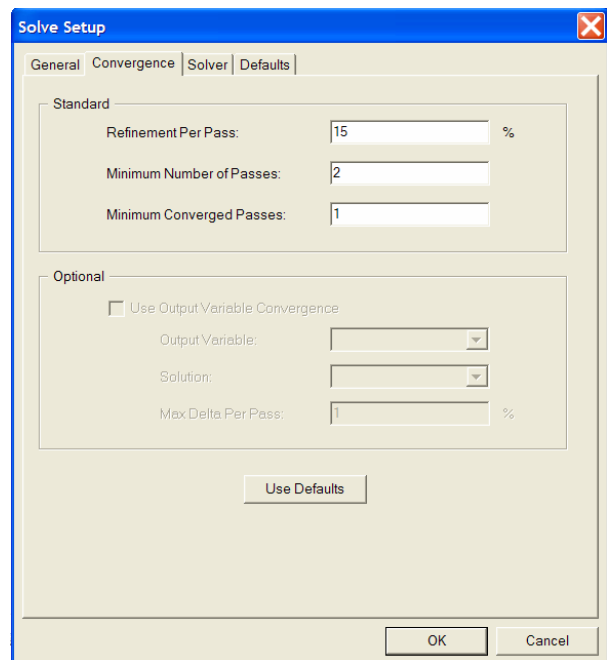
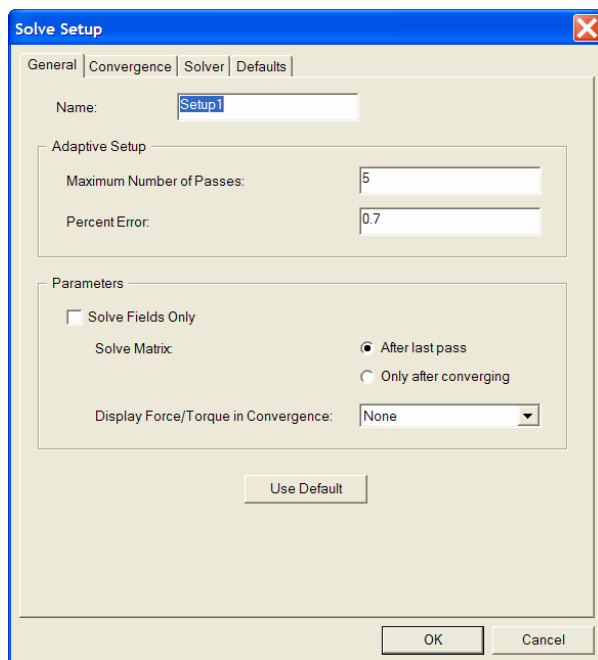
Create an Analysis Set up

- 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 1.0 to 0.7. 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).

Topic - ECE: Linear movement

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.

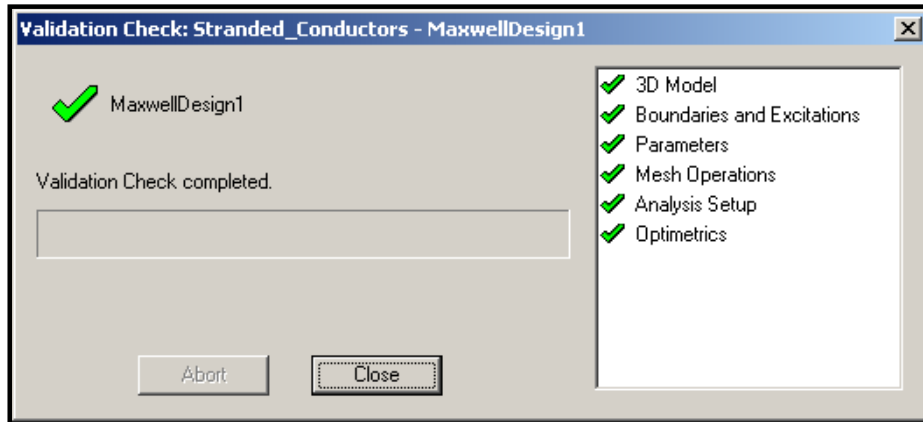


Save the Project

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

Check the Validity of the Model

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



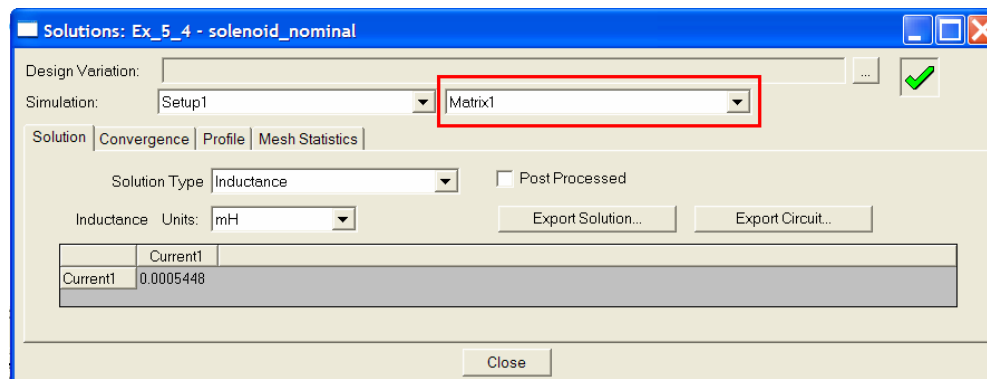
- The problem will not solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the  icon.
- The progress bar shows the different stages of the ongoing computation. This takes approximately 20 minutes on a 3GHz PC. 

Solution Data of the nominal problem

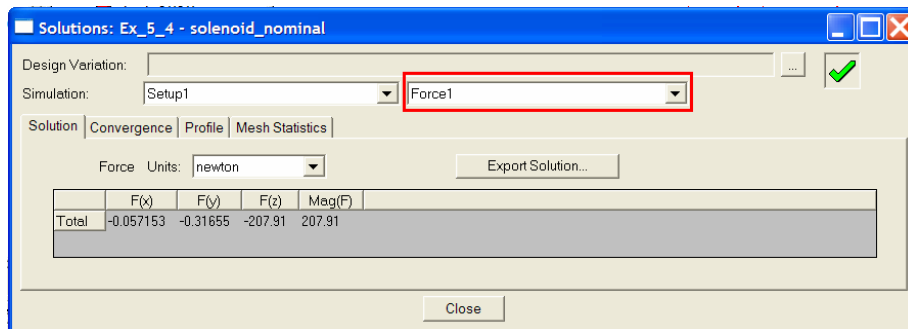
- To view the Solution Data, select the menu item **Maxwell > Results > Solution Data**. You have access from this window to the inductance and to the force computed as well as other information:
 - Select the **Solution** tab: the inductance is available by choosing Matrix1 in the pull-down menu. In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



Topic - ECE: Linear movement

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.

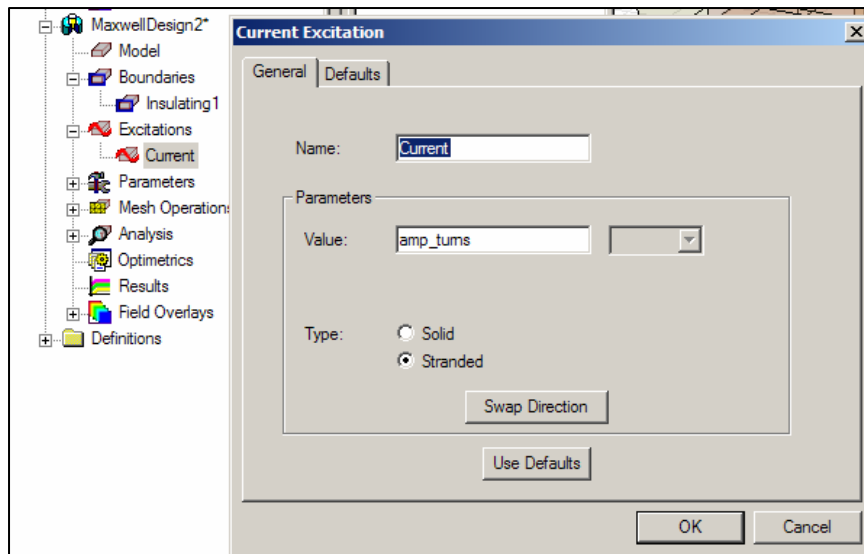


-  Select the **Convergence** tab: the log of the different adaptive passes is displayed with the successive sizes of the mesh.
 -  Select the **Profile** tab: you can find information about the CPU time, the size of the generated data.
 -  Select the **Mesh Statistics**: all the details about the final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

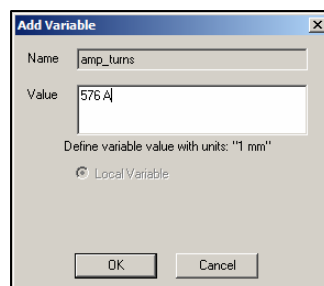
❖ Create Parametric Analysis

❖ 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*.



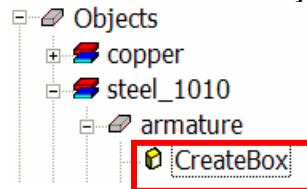
- ❖ A window pops up asking for the value of amp_turns. Leave it to 576 Amps.



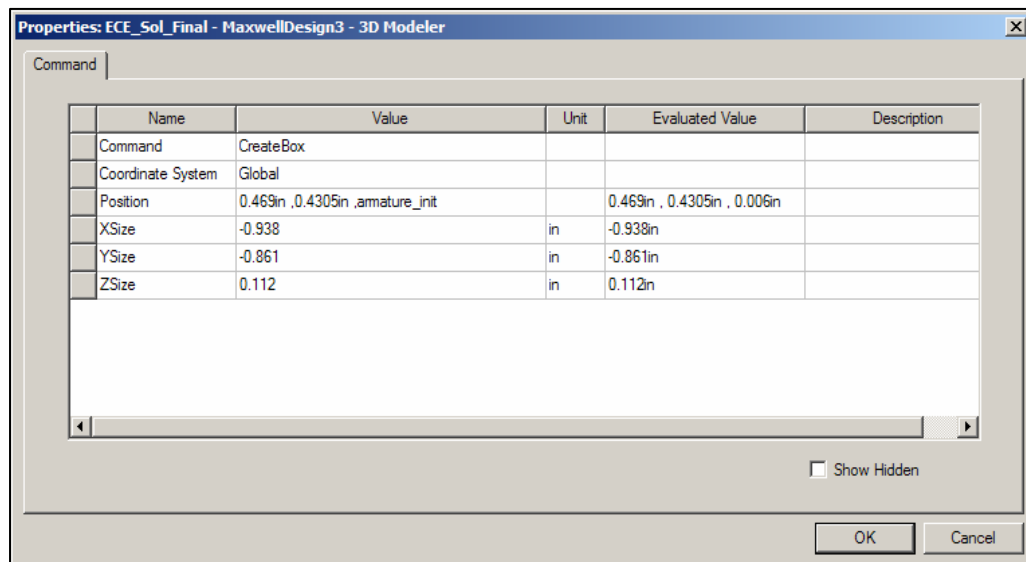
Topic - ECE: Linear movement

Assign Problem Parameters (Continued)

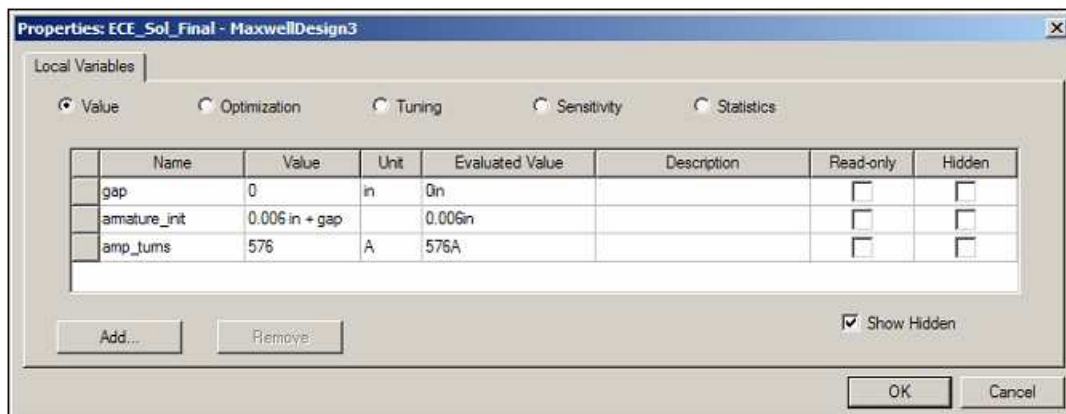
- To enter the air gap parameter, expand from the objects tree the **Armature** entry. Double click on CreateBox object.



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

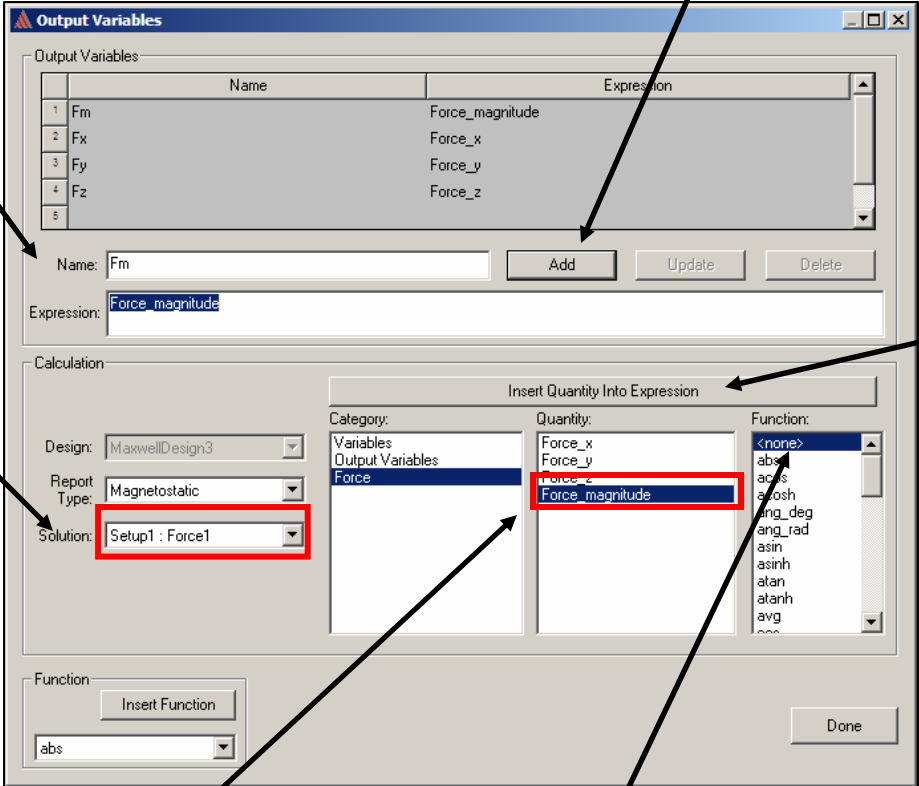


- Set the value of **armature_init** as **.006in + gap**. Set **gap** as **0.0in**.
- At any time you can access at the list of the problem parameters. Select the menu item **Maxwell > Design Properties**. The window should look like:



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:



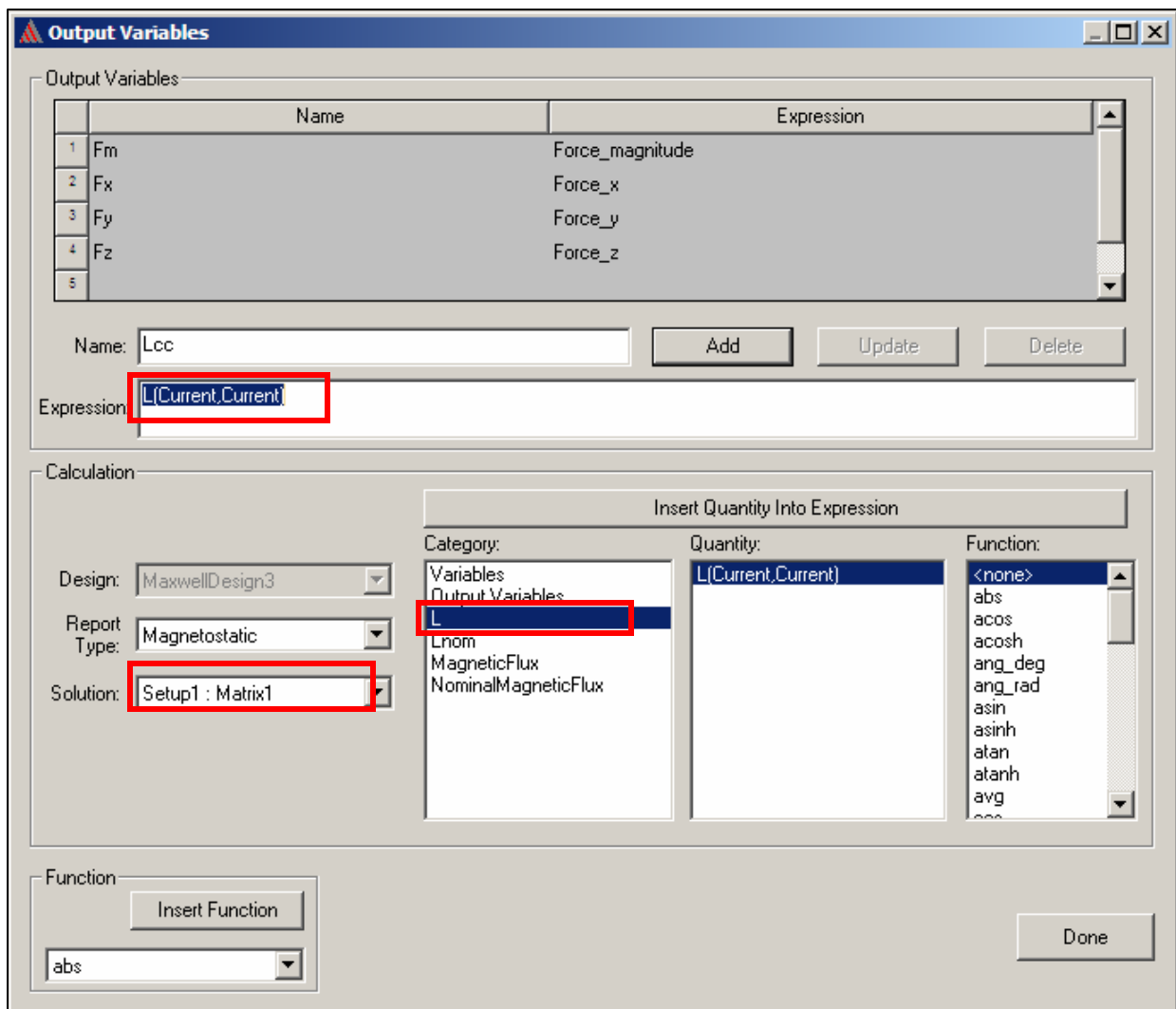
The screenshot shows the 'Output Variables' dialog box in Maxwell 3D. The dialog has a table for 'Output Variables' with columns 'Name' and 'Expression'. Below the table are fields for 'Name' and 'Expression', and buttons 'Add', 'Update', and 'Delete'. At the bottom, there is a 'Calculation' section with dropdowns for 'Design', 'Report Type', and 'Solution'. A 'Function' section at the bottom left has an 'Insert Function' button and a dropdown showing 'abs'. A 'Quantity' list is visible in the center, and a 'Function' list is on the right.

Annotations and steps:

- 1- Select the Executive parameter (points to the 'Solution' dropdown showing 'Setup1 : Force1')
- 2- Select the Quantity (points to the 'Quantity' list where 'Force_magnitude' is selected)
- 3- We do not need to apply any function to the Quantity (points to the 'Function' list)
- 4- Insert the Quantity (points to the 'Insert Quantity Into Expression' button)
- 5- Name the expression (points to the 'Name' field showing 'Fm')
- 6- Add the expression and to the list (points to the 'Add' button)

Create Output Variables (Continued)

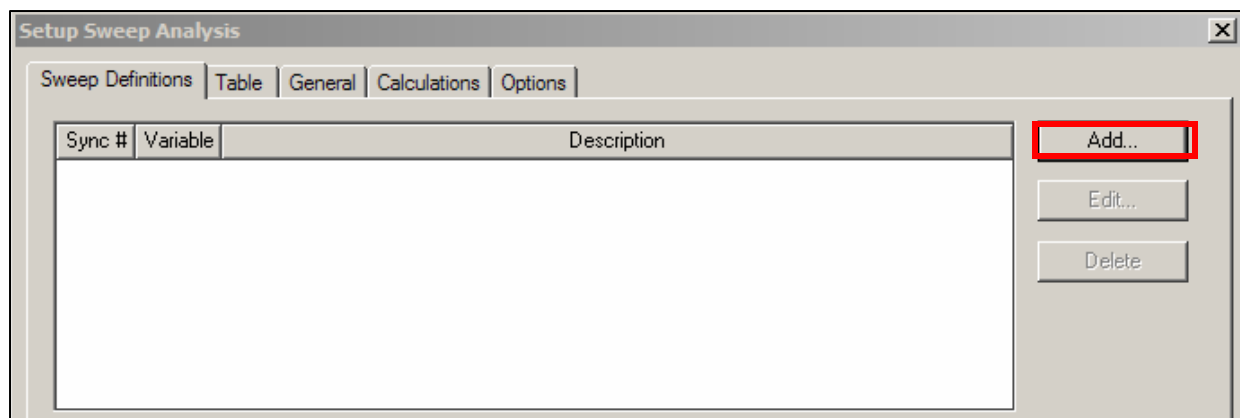
- Proceed the same way to declare Fx, Fy, Fz the components of the Force applied to the Armature.
- We also declare the inductance as an output variable. Follow a similar procedure, but select Matrix1 from the pull-down menu (see below). Lcc stands for the inductance of the coil.



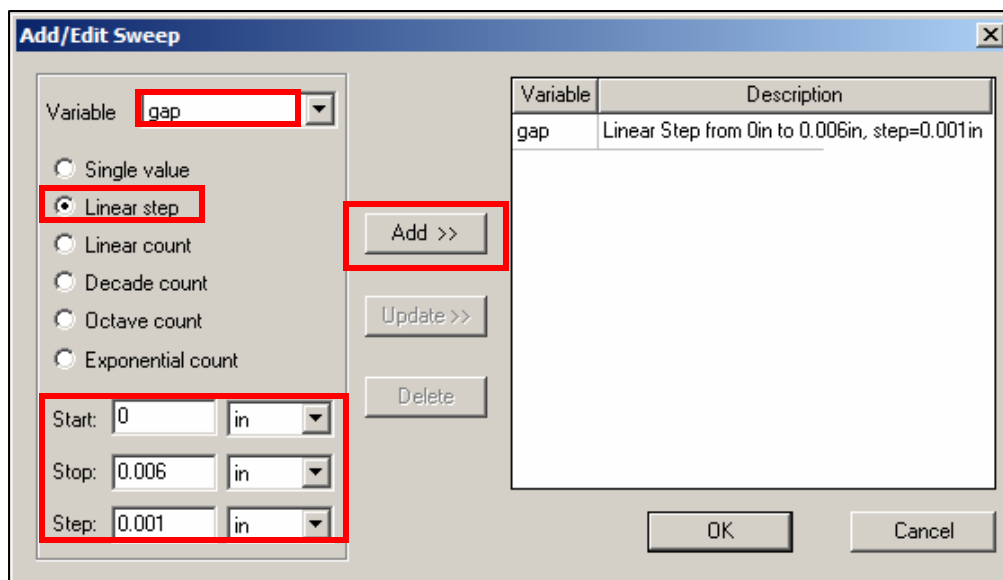
- Finally, you should have 5 output variables declared: Fm, Fx, Fy, Fz and Lcc.

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).

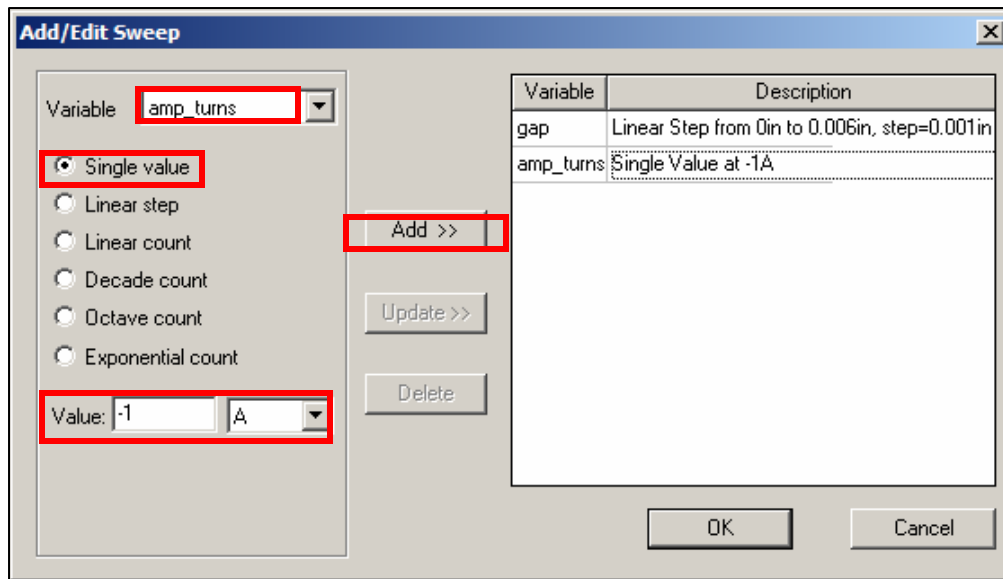


- From the Add/Edit Sweep window, select **gap** in the pull down menu. The air gap will linearly vary from 0.0in to 0.006in with a 0.001in Step. Enter these values in the corresponding boxes. Click Add to proceed.

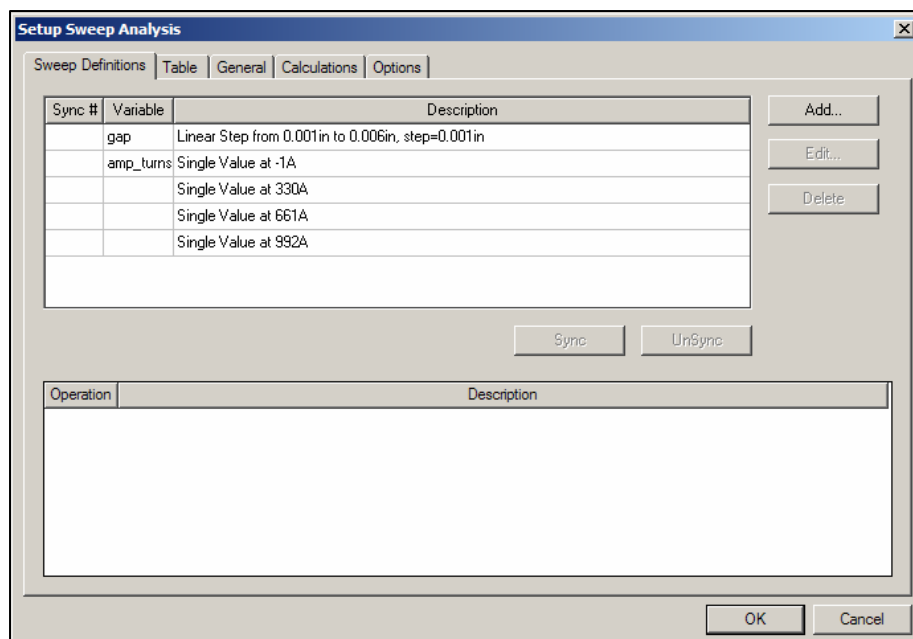


Create Parametric Analysis (Continued)

- 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.

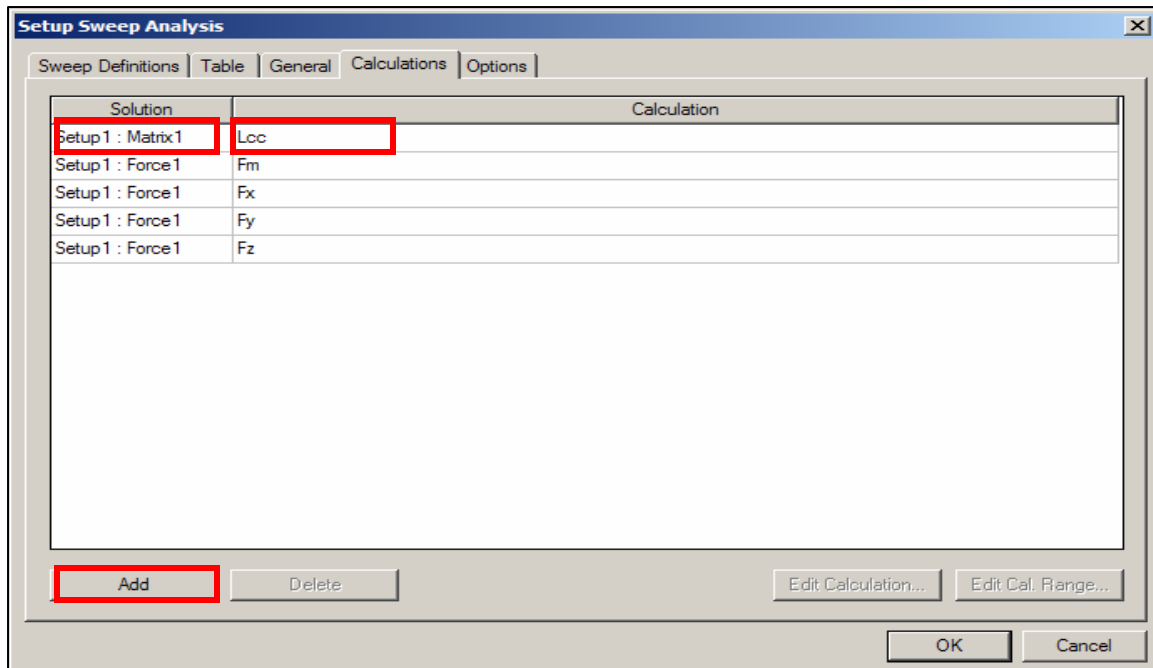


- The Setup Sweep Analysis window has the following definitions:



Create Parametric Analysis (Continued)

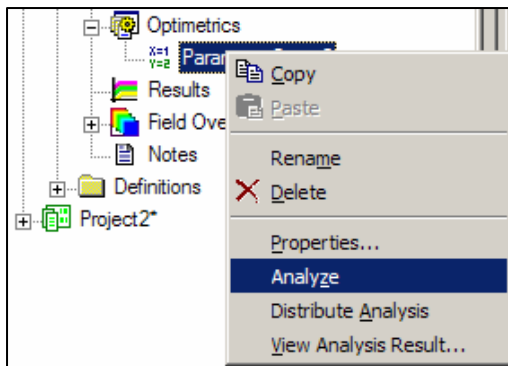
- From the 'Table' tab, the 28 variations are displayed.
- Leave 'General' tab of Setup Sweep Definition unchanged and select the 'Calculations' tab. From this tab, we will define the output variables to include in the sweep.



- 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.

Solve the Parametric Sweep

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

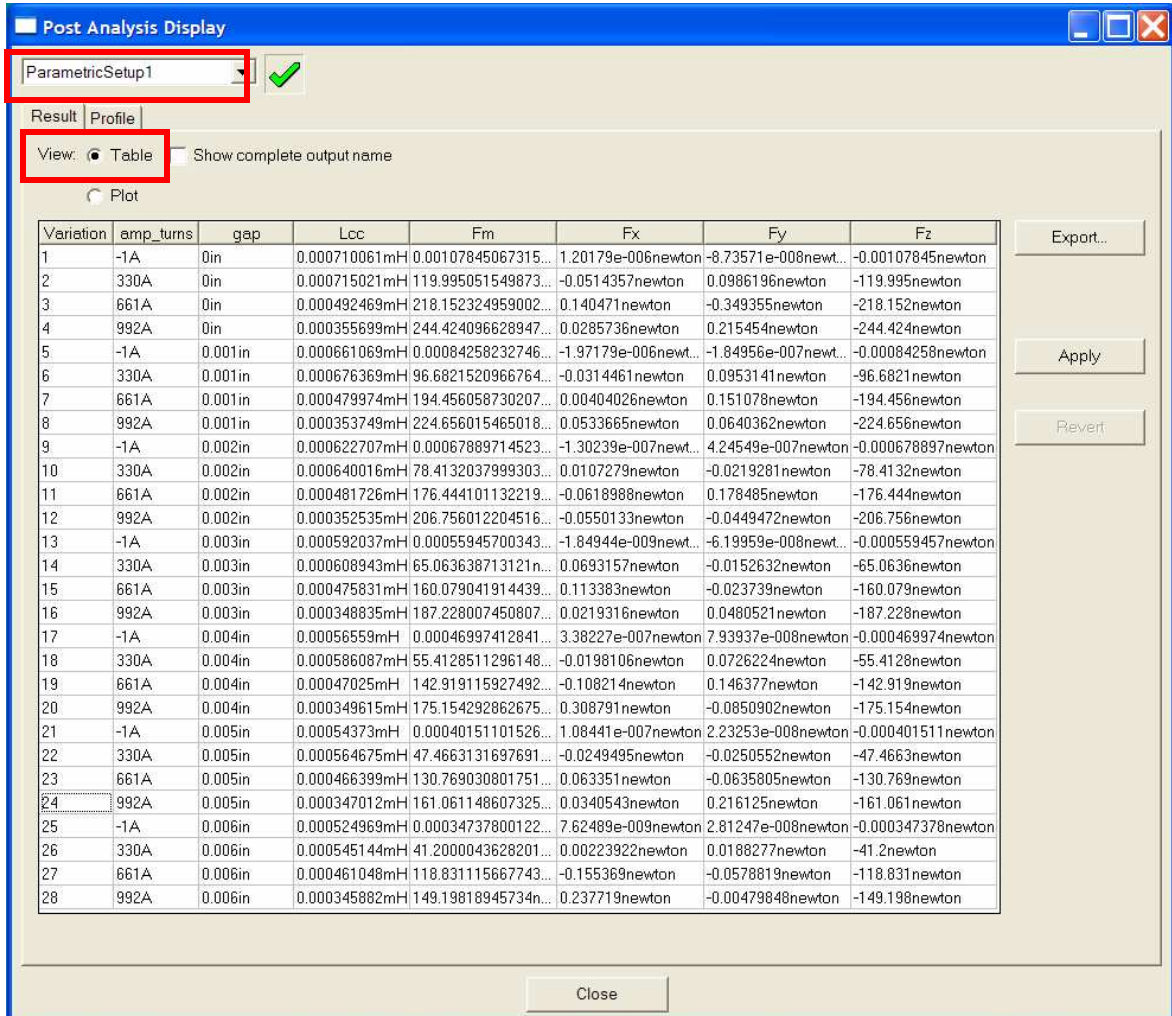


-  Maxwell will start an separated analysis for every row of the table. On a 3GHz PC, this will take approximately 3hours to solve.
-  If you have a Distributive Analysis license, you can run the parametric sweep on up to 10 machines per Distributive Analysis license. Use the **Distributive Analysis** 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.

Topic - ECE: Linear movement

Solution Data of the Parametric Sweep

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



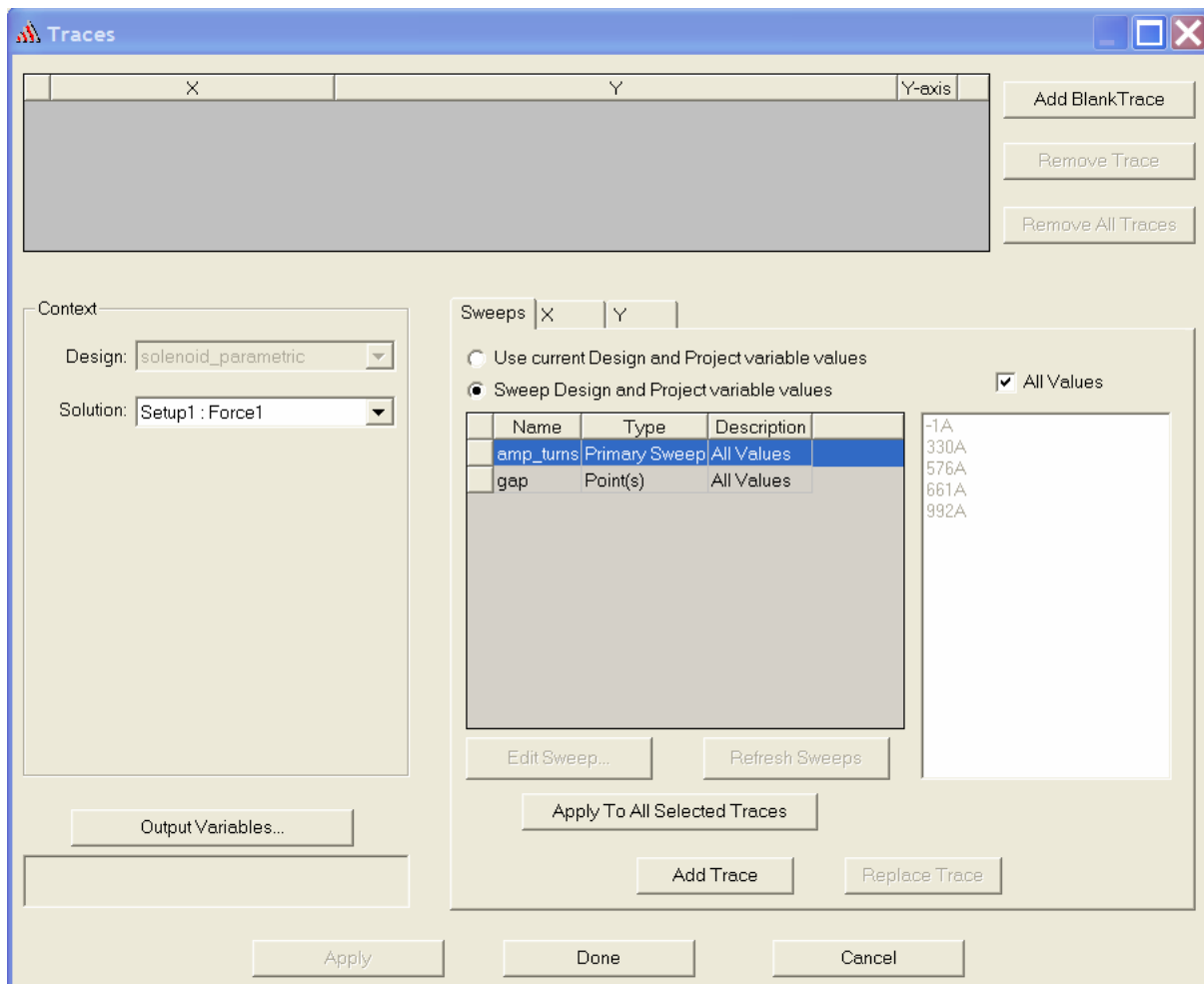
Variation	amp_turns	gap	Lcc	Fm	Fx	Fy	Fz
1	-1A	0in	0.000710061mH	0.00107845067315...	1.20179e-006newton	-8.73571e-008newt...	-0.00107845newton
2	330A	0in	0.000715021mH	119.995051549873...	-0.0514357newton	0.0986196newton	-119.995newton
3	661A	0in	0.000492469mH	218.152324959002...	0.140471newton	-0.349355newton	-218.152newton
4	992A	0in	0.000355699mH	244.424096628947...	0.0285736newton	0.215454newton	-244.424newton
5	-1A	0.001in	0.000661069mH	0.00084258232746...	-1.97179e-006newt...	-1.84956e-007newt...	-0.00084258newton
6	330A	0.001in	0.000676369mH	96.6821520966764...	-0.0314461newton	0.0953141newton	-96.6821newton
7	661A	0.001in	0.000479974mH	194.456058730207...	0.00404026newton	0.151078newton	-194.456newton
8	992A	0.001in	0.000353749mH	224.656015465018...	0.0533665newton	0.0640362newton	-224.656newton
9	-1A	0.002in	0.000622707mH	0.00067889714523...	-1.30239e-007newt...	4.24549e-007newton	-0.000678897newton
10	330A	0.002in	0.000640016mH	78.4132037999303...	0.0107279newton	-0.0219281newton	-78.4132newton
11	661A	0.002in	0.000481726mH	176.444101132219...	-0.0618988newton	0.178485newton	-176.444newton
12	992A	0.002in	0.000352535mH	206.756012204516...	-0.0550133newton	-0.0449472newton	-206.756newton
13	-1A	0.003in	0.000592037mH	0.00055945700343...	-1.84944e-009newt...	-6.19959e-008newt...	-0.000559457newton
14	330A	0.003in	0.000608943mH	65.063638713121n...	0.0693157newton	-0.0152632newton	-65.0636newton
15	661A	0.003in	0.000475831mH	160.079041914439...	0.113383newton	-0.023739newton	-160.079newton
16	992A	0.003in	0.000348835mH	187.228007450807...	0.0219316newton	0.0480521newton	-187.228newton
17	-1A	0.004in	0.00056559mH	0.00046997412841...	3.38227e-007newton	7.93937e-008newton	-0.000469974newton
18	330A	0.004in	0.000586087mH	55.4128511296148...	-0.0198106newton	0.0726224newton	-55.4128newton
19	661A	0.004in	0.00047025mH	142.919115927492...	-0.108214newton	0.146377newton	-142.919newton
20	992A	0.004in	0.000349615mH	175.154292862675...	0.308791newton	-0.0850902newton	-175.154newton
21	-1A	0.005in	0.00054373mH	0.00040151101526...	1.08441e-007newton	2.23253e-008newton	-0.000401511newton
22	330A	0.005in	0.000564675mH	47.4663131697691...	-0.0249495newton	-0.0250552newton	-47.4663newton
23	661A	0.005in	0.000466399mH	130.769030801751...	0.063351newton	-0.0635805newton	-130.769newton
24	992A	0.005in	0.000347012mH	161.061148607325...	0.0340543newton	0.216125newton	-161.061newton
25	-1A	0.006in	0.000524969mH	0.00034737800122...	7.62489e-009newton	2.81247e-008newton	-0.000347378newton
26	330A	0.006in	0.000545144mH	41.2000043628201...	0.00223922newton	0.0188277newton	-41.2newton
27	661A	0.006in	0.000461048mH	118.831115667743...	-0.155369newton	-0.0578819newton	-118.831newton
28	992A	0.006in	0.000345882mH	149.19818945734n...	0.237719newton	-0.00479848newton	-149.198newton

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

Create Plot of Force vs. Current for each gap

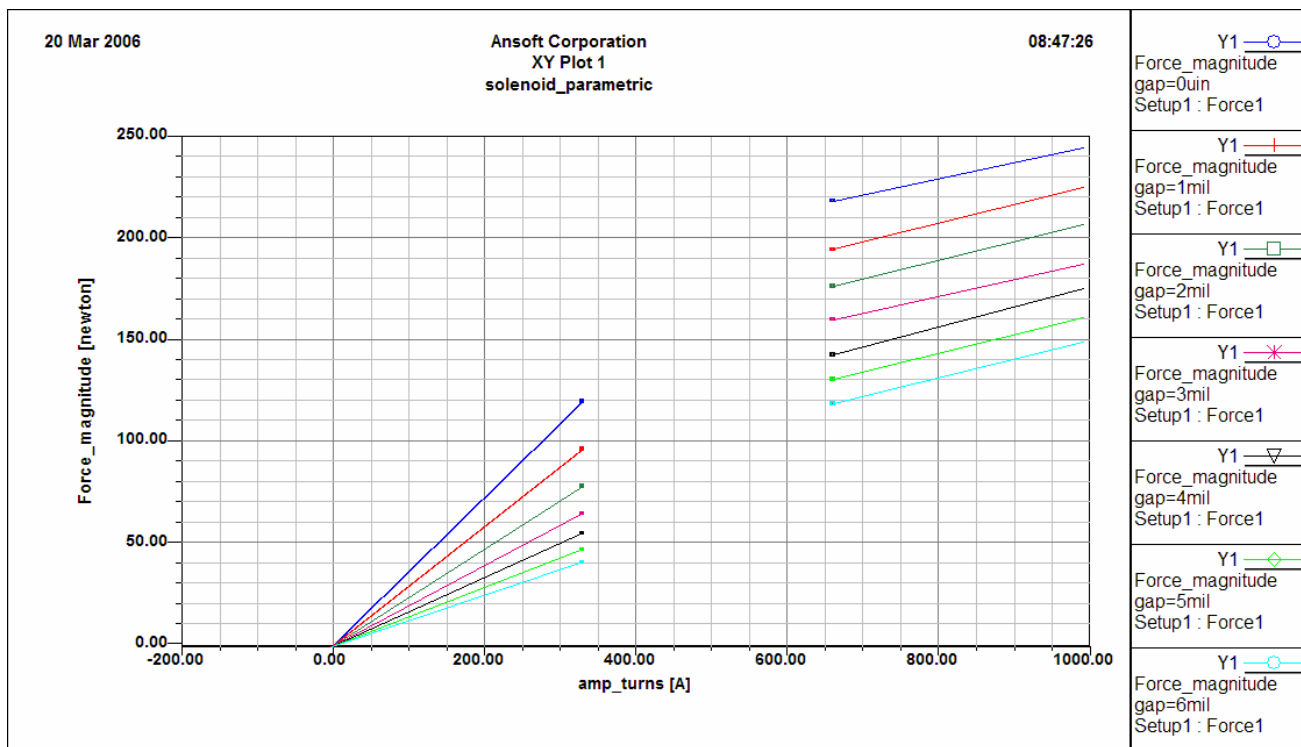
To create a report:

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 **amp_turns** is selected as the primary sweep



Topic - ECE: Linear movement

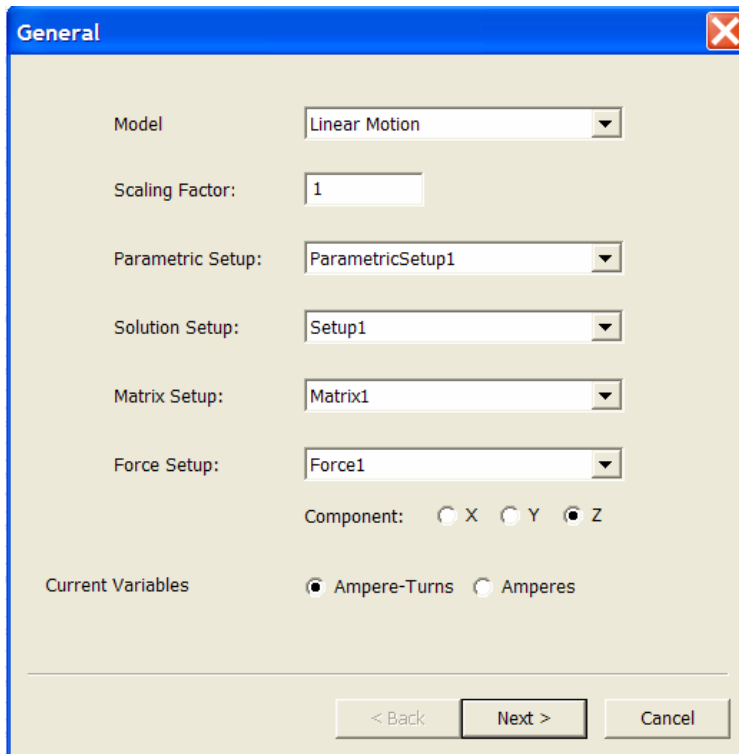
3. Click the **Y** tab
 1. Category: **Force**
 2. Quantity: **Force_magnitude**
 3. Function: **<none>**
 4. Click the **Add Trace** button
4. Click the **Done** button to view the family of curves shown below.



ECE from the Parametric Analysis

In order to ultimately perform a system simulation in Simplorer an equivalent magnetic model must be exported from Maxwell 3D. This model contains input pins for current and gap and outputs for force and flux linkage. The following steps show how to create this equivalent model. (Note: if you do not intend to do a system simulation in Simplorer, then these steps are not necessary.)

- ▲ Select the menu item **Maxwell > Export Equivalent Circuit > 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.

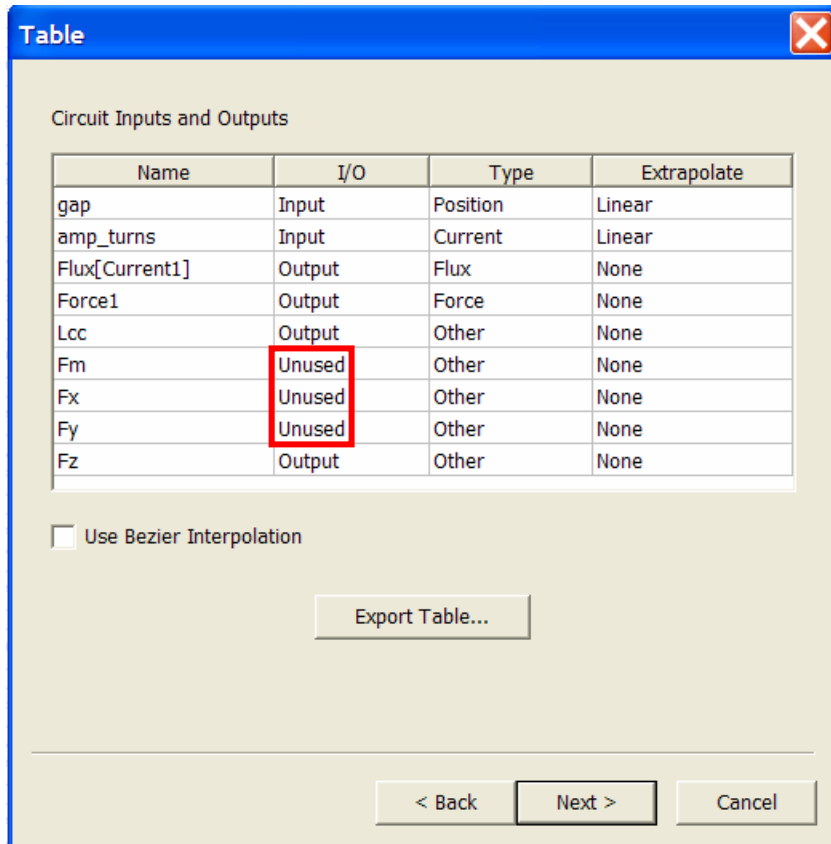


- ▲ Click on Next to continue.

Topic - ECE: Linear movement

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 corresponds to **Position** and amp_turns corresponds to **Current**.
- ▀ Optionally, if you wish to process the data in a table form, click on **Export Table** button and a text file of the parametric table will be created.



Table

Circuit Inputs and Outputs

Name	I/O	Type	Extrapolate
gap	Input	Position	Linear
amp_turns	Input	Current	Linear
Flux[Current1]	Output	Flux	None
Force1	Output	Force	None
Lcc	Output	Other	None
Fm	Unused	Other	None
Fx	Unused	Other	None
Fy	Unused	Other	None
Fz	Output	Other	None

☐ Use Bezier Interpolation

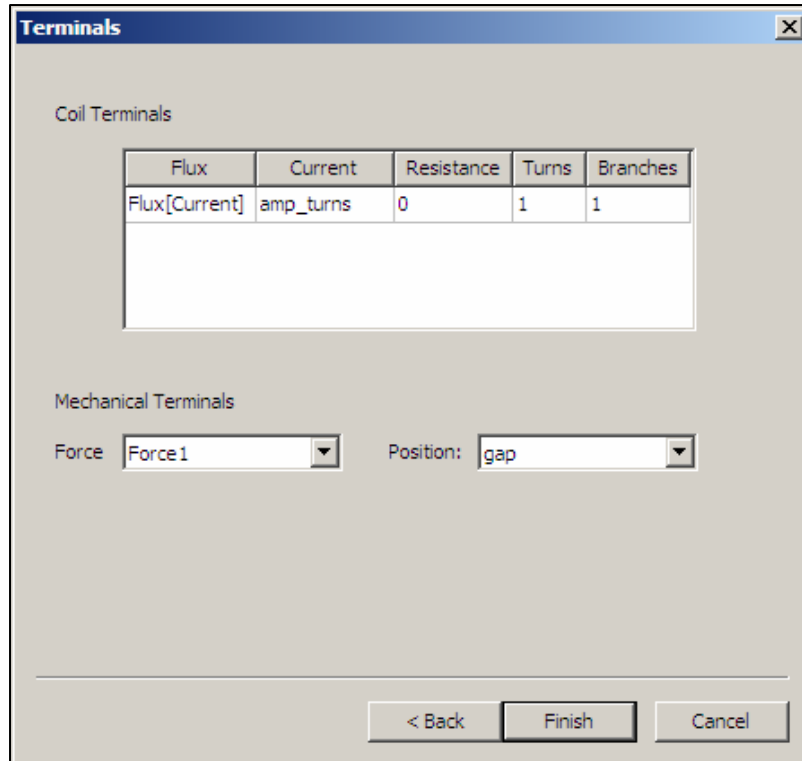
Export Table...

< Back Next > Cancel

- ▀ Click on Next to continue.

▲ 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.



Flux	Current	Resistance	Turns	Branches
Flux[Current]	amp_turns	0	1	1

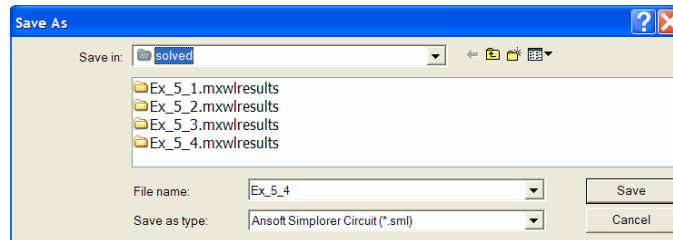
Force: Force1 Position: gap

< Back Finish Cancel

- ▲ Since Maxwell has defined the correct values for the Terminals, simply accept the suggested parameters by clicking on Finish.

ECE from the Parametric Analysis (Continued)

- Give a File name and save the output file.

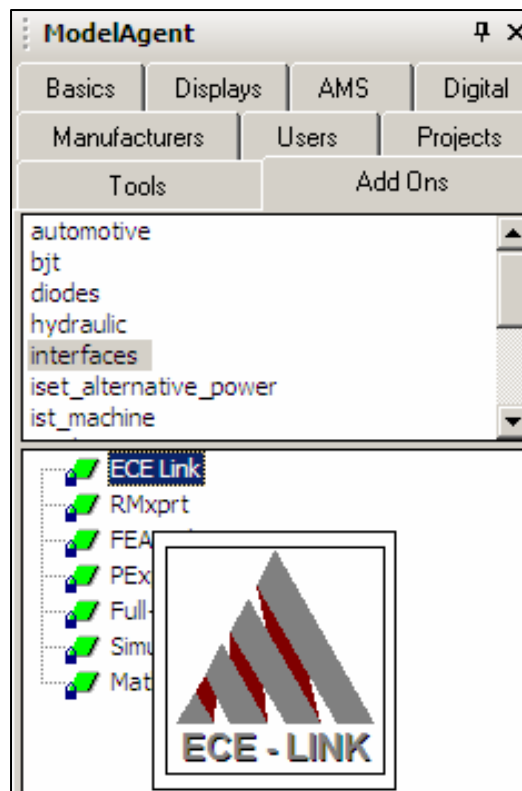


- Maxwell creates the SML file containing the model of the component to be used in Simplorer. SML stands for Simplorer Modeling Language, the internal language of Simplorer. This script file will be imported into Simplorer.
- To improve visibility of the component in Simplorer, it is advised to capture 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.

NOTE: Stop here is not continuing to a Simplorer simulation.

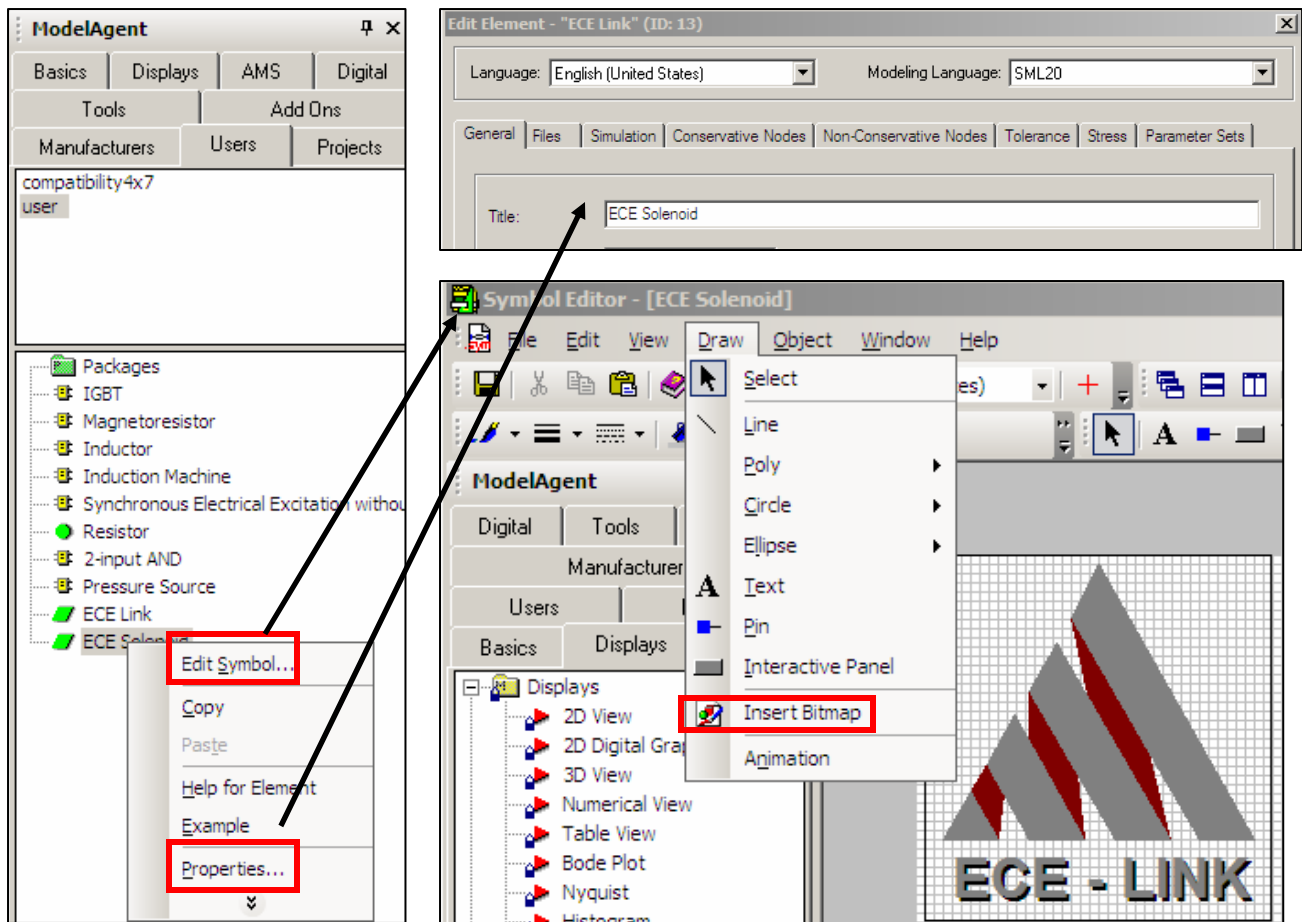
Create the circuit in Simplorer

-  **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.



Create the Component in Simplorer (Continued)

- Select the tab Users (or the Project tab if 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.

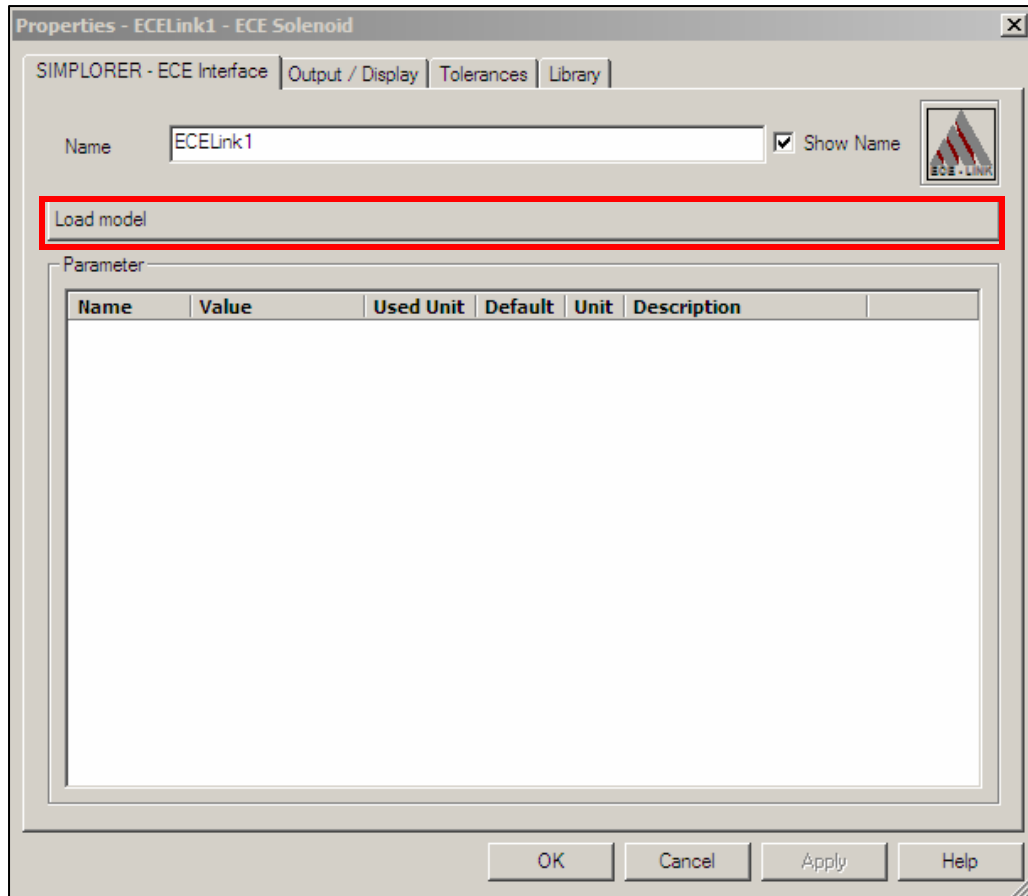


- Right click after having selecting the ECE link component. Choose properties. In the window, select the General tab and give the component name **ECE Solenoid**. Leave the other parameters to their default values and Select OK.
- Right click and select Edit Symbol. From this window, you can draw your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

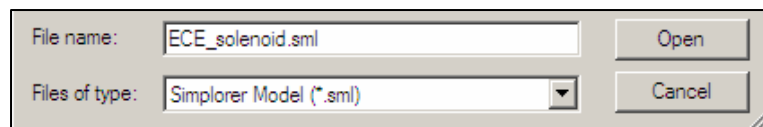
Topic - ECE: Linear movement

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.



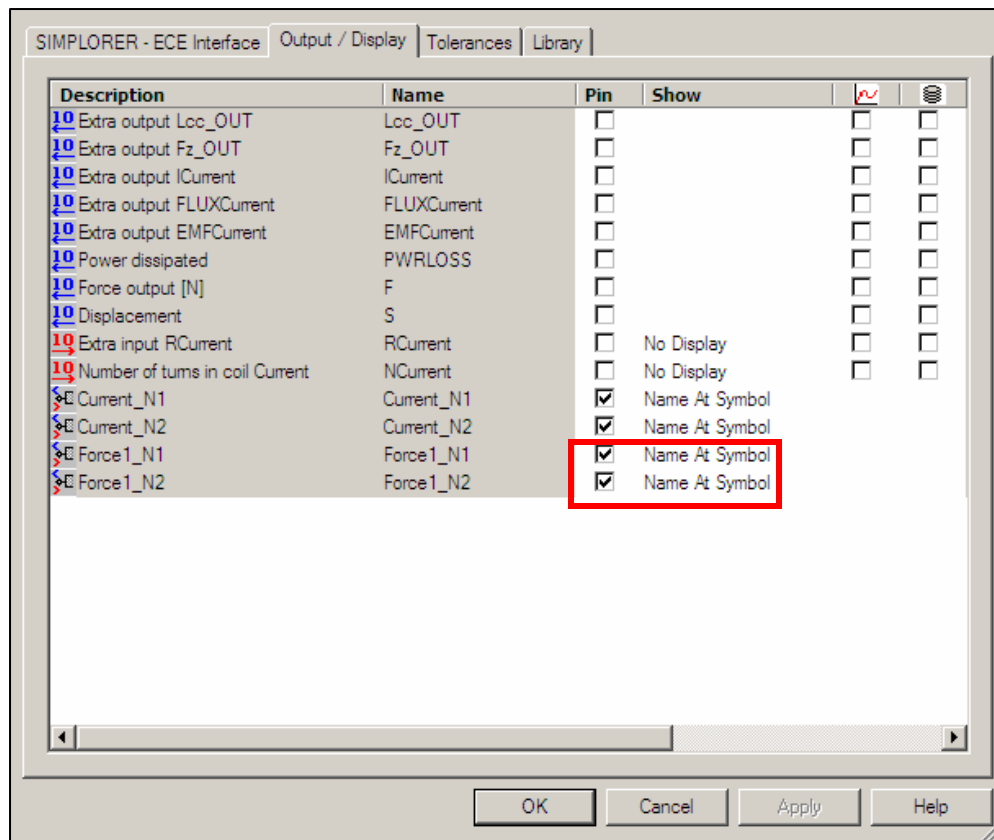
- Browse your disk to recover the output file generated by Maxwell. This file has the extension SML.



- Click on OK. The model is loaded.

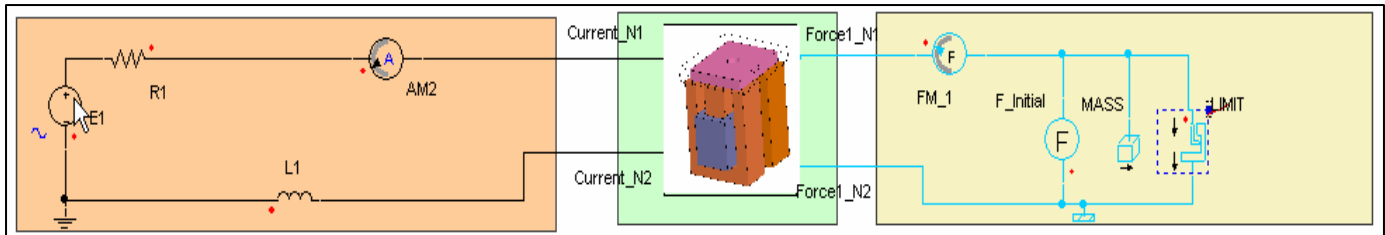
▲ Create the Component in Simplorer (Continued)

- ▲ 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.

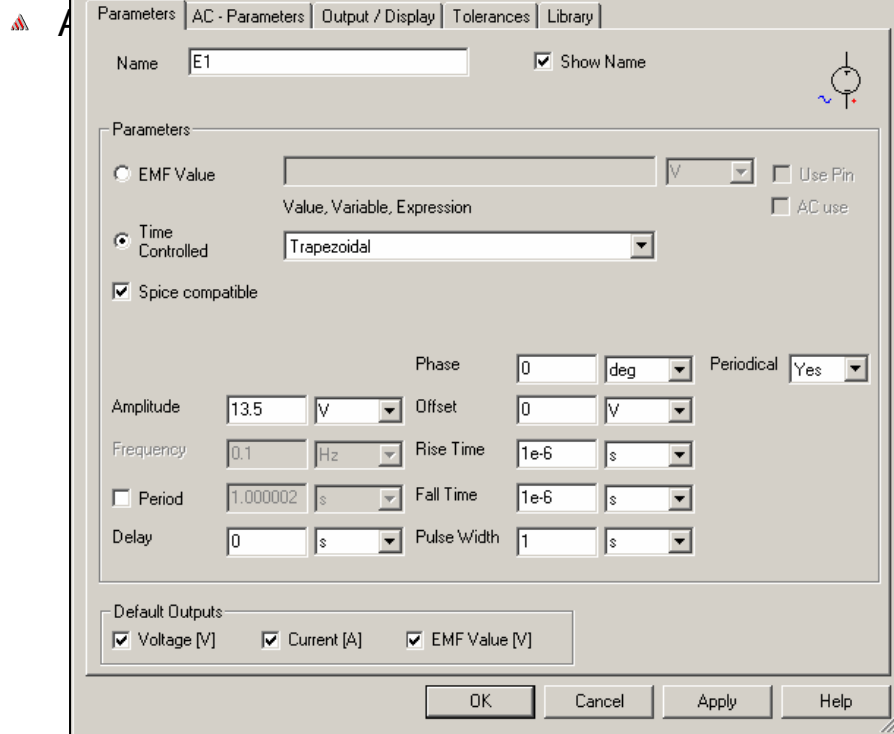


- ▲ As additional input (non conservative node), you can specify the number of turns in the Coil (in Maxwell, we just gave the overall value of the current) as well as give an additional resistance corresponding to the coil in the Solenoid.
- ▲ The component is ready to be used in a Simplorer project.

Create the Schematic sheet in Simplorer



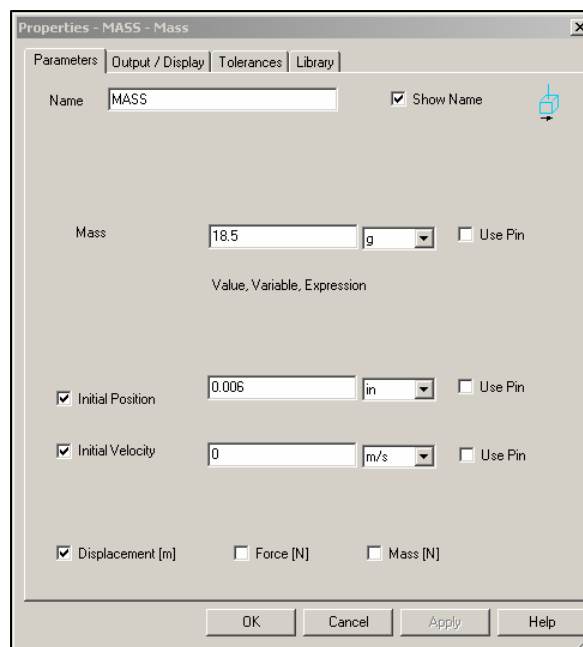
- In this application, the system surrounding the actuator has an electric part and a mechanical part.
- For the electrical part, the following components have been dragged on the



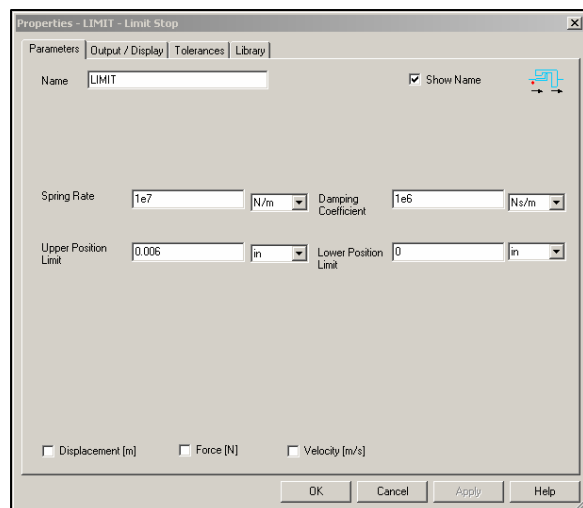
Create the Schematic sheet in Simplorer (Continued)

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 $F=125\text{ N}$
- 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:



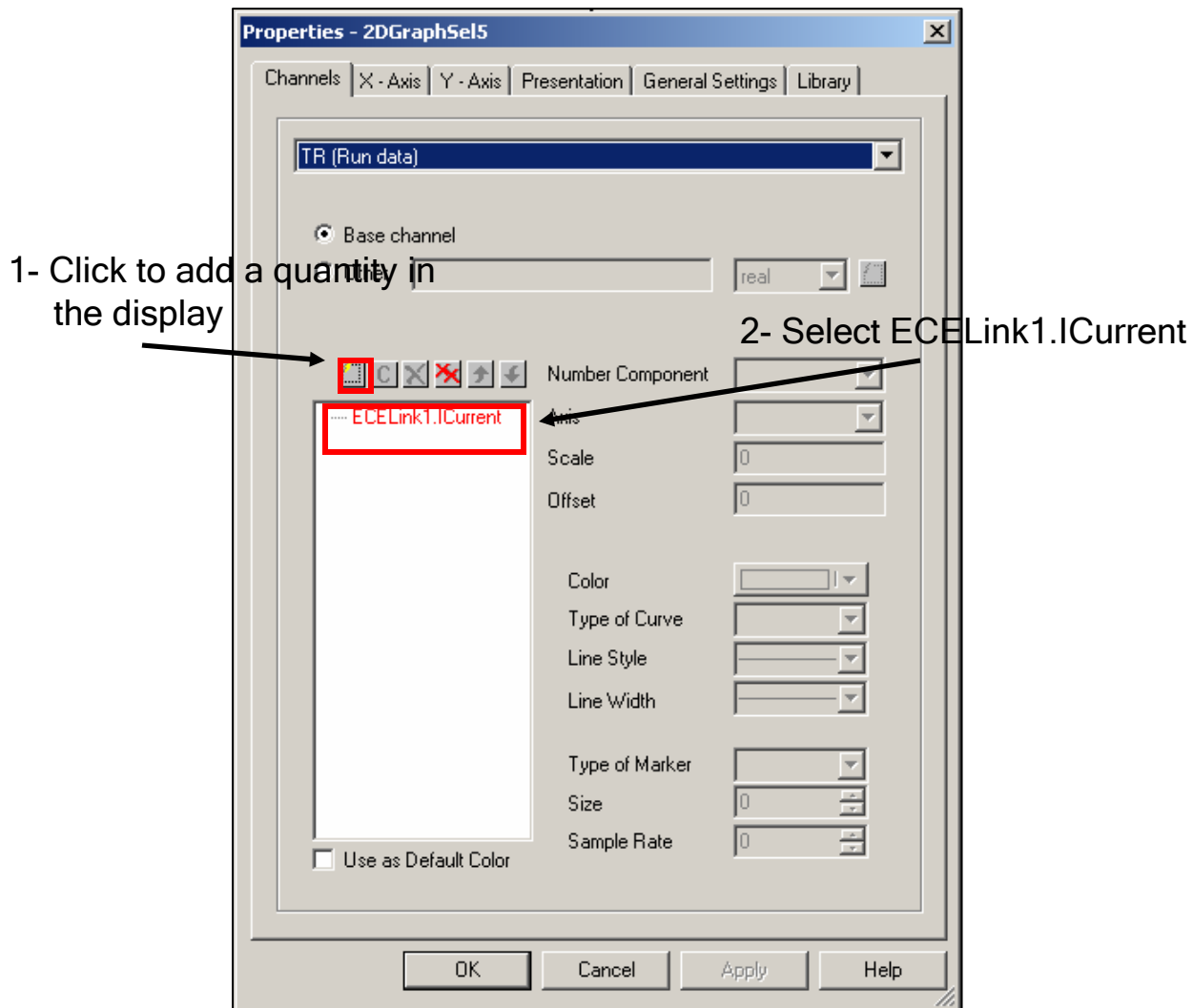
Topic - ECE: Linear movement

▲ Create the Schematic sheet in Simplorer (Continued)

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

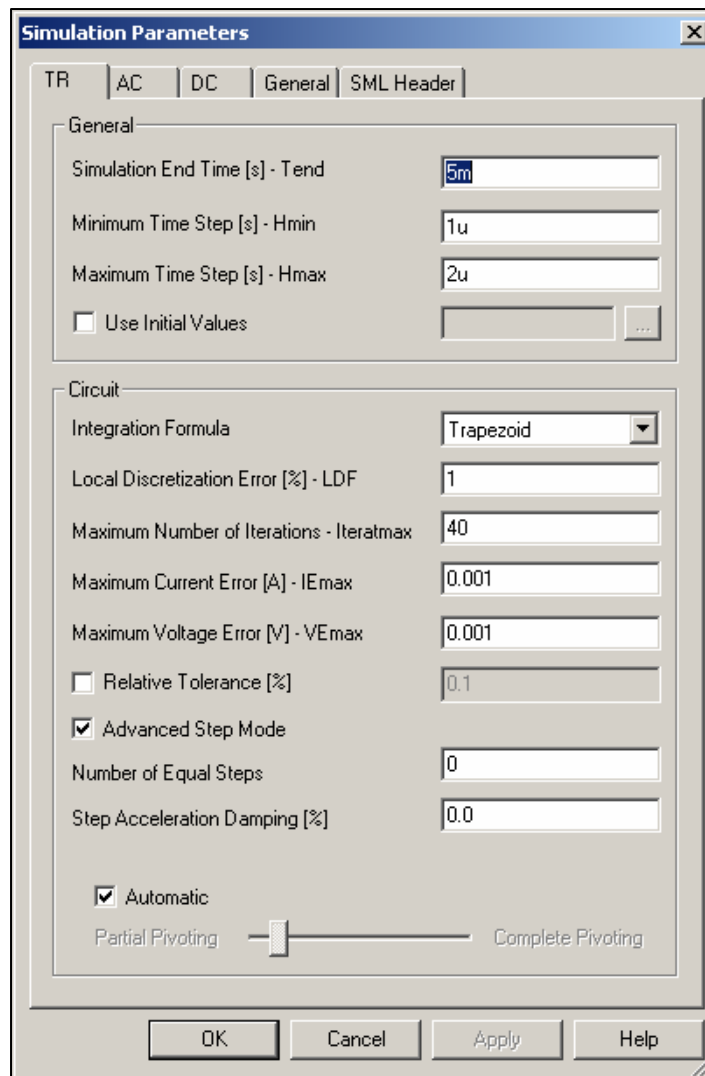
▲ Create the graphical Displays in Simplorer

- ▲ 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).



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

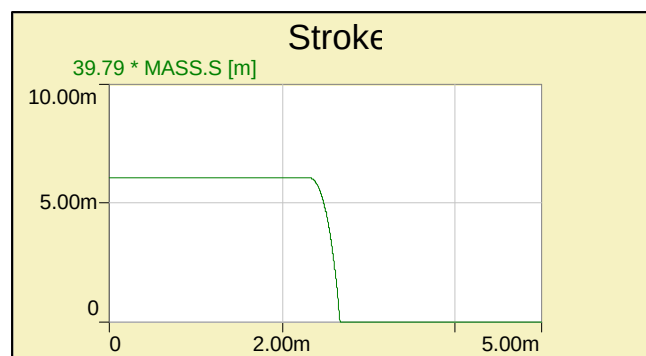
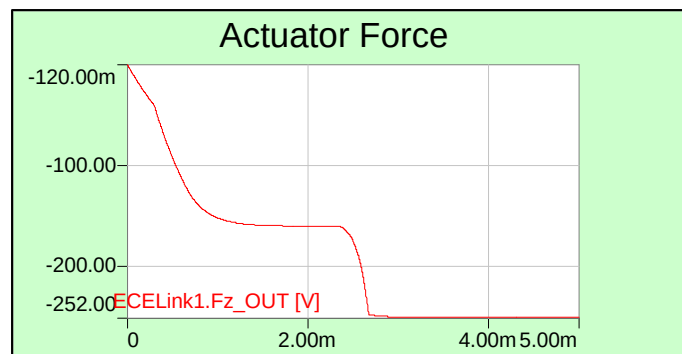
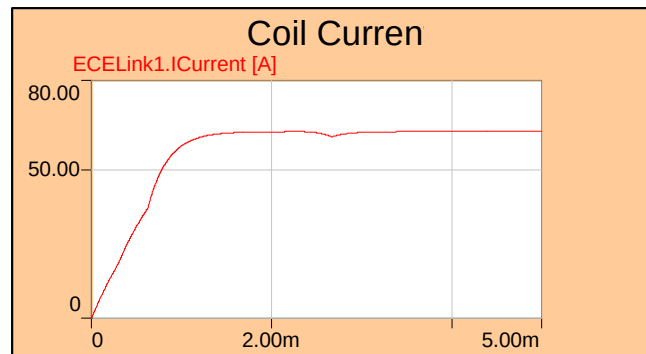
-  **Create the graphical Displays in Simplorer**
 -  Repeat this operation with 2 new **2D view** elements. Add the two following quantities:
 -  The force of the plunger: ECELink1.Fz_OUT
 -  The stroke : $39.79 * \text{MASS.S}$
-  **Create a Transient Analysis**
 -  Select the menu item Simulation > Parameters
 -  Fill the window with these parameters:



Topic - ECE: Linear movement

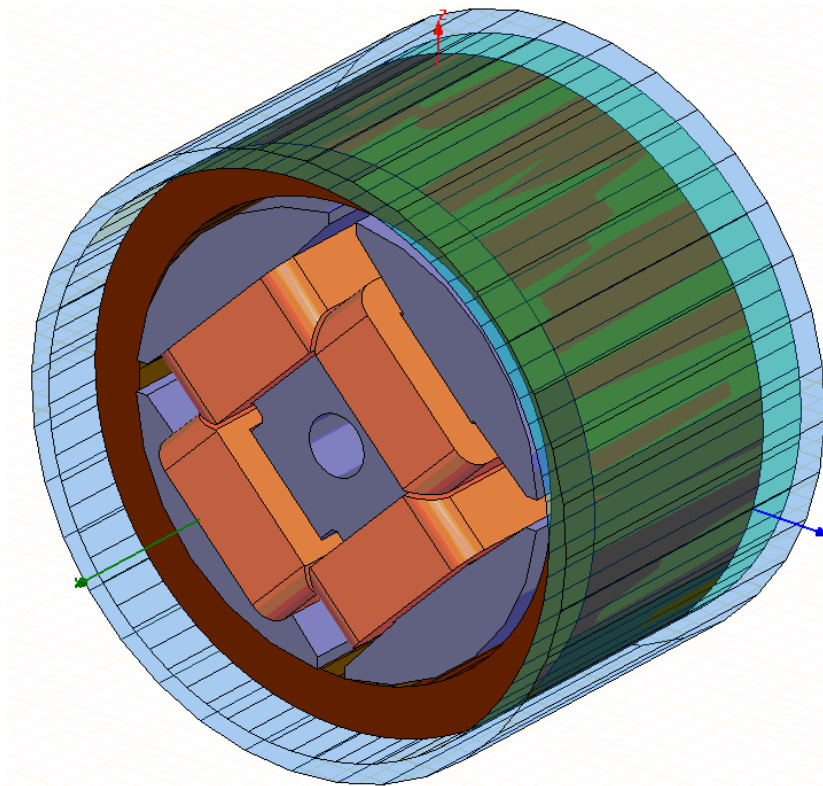
Run the Analysis

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



THIS PAGE INTENTIONALLY LEFT BLANK

- 
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.



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): **RMxprrt/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**

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:

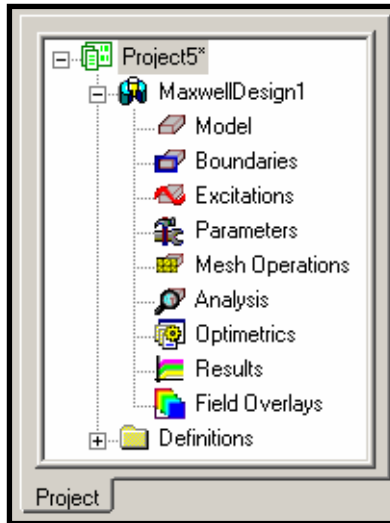
-  **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :

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

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)**

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

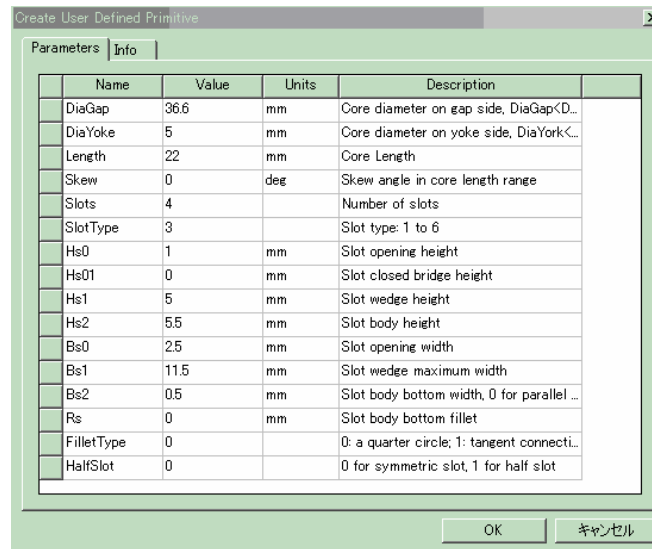
- ▲ Click OK to finish it. Change the name from Box1 to Region by editing the Attribute panel on the left hand side of the interface. Keep the material as vacuum.

Properties: Project1 - MaxwellDesign1 - 3D Modeler

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

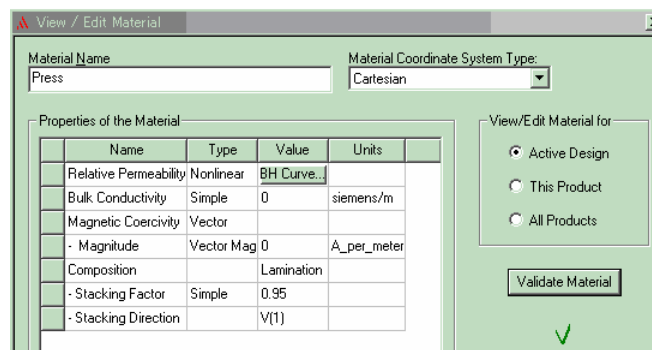
Create the Field Core:

- ▲ 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.



Material Name: Press

Material Coordinate System Type: Cartesian

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)	

View/Edit Material for:

- ☒ Active Design
- ☐ This Product
- ☐ All Products

Validate Material

Topic - Anisotropic Material

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

Properties: GST16 - MaxwellDesign1 - 3D Modeler

Attribute		
Name	Value	Unit
Name	FieldCore	
Material	Press	
Solve Inside	☒	
Orientation	Global	
Model	☒	
Display Wireframe	☐	
Color	Edit	
Transparent	0.8	

- ▲ 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	Value
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**

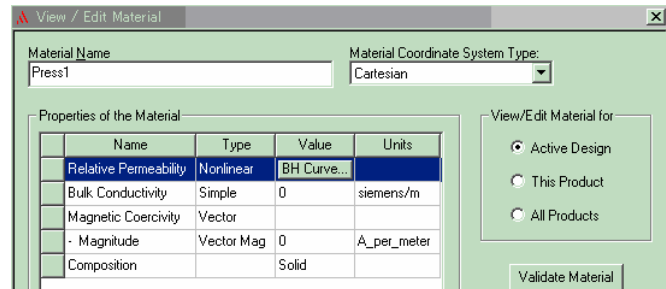
Name	Value
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-15.3 ,0 ,0
Start Position	-15.3 ,0 ,3.325
Axis	X
Height	-1.7
Number of Segme...	16

Topic - Anisotropic Material

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

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

- ▲ **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



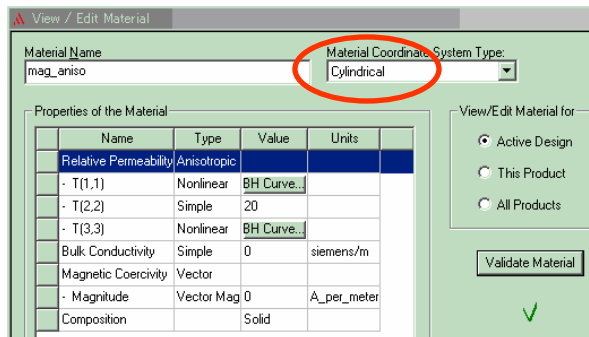
- ▲ **Create Magnet Ring named as *Mag***
 - ▲ **Draw > RegularPolyhedron** then using the coordinate entry field, enter the center position
 - ▲ **X: -10, Y: 0, Z: 0**, Press the **Enter** key
 - ▲ Then enter the height and the radio
 - ▲ **dZ: 22.8** for radius, Press the **Enter** key
 - ▲ **dX: 20** 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

Topic - Anisotropic Material

- ▲ **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 > Subtract** to create final **Mag**
- ▲ 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
Command	CreateRegularPolyhedron
Coordinate System	Global
Center Position	-10 ,0 ,0
Start Position	-10 ,0 ,18.8
Axis	X
Height	20
Number of Segme...	72



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 (highlighted with a red circle)
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.

Topic - Anisotropic Material

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

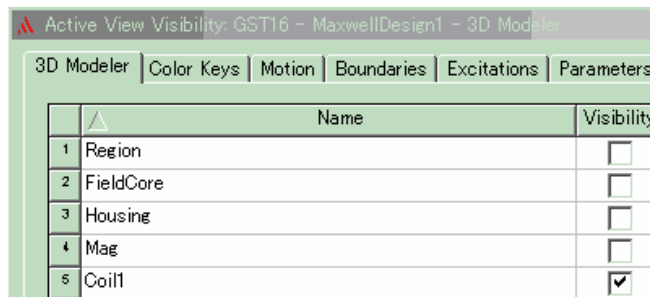
- Pick **Box1** & **Box2**, Using right mouse

Edit > Boolean > Subtract to create final **Box1**

- Change the name of **Box1** to **Coil_1** and set material **Copper** for it

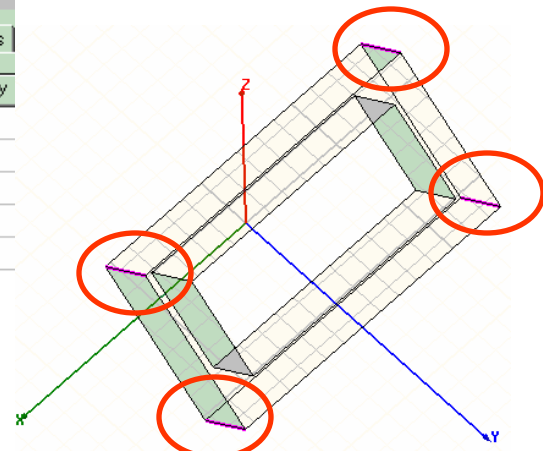
- Select **Coil_1** in History Tree, Using right mouse **Edit > Arrange > Rotate** select **X** as rotating Axis with **45** degree, **OK** to finish it.

- View > Active View Visibility** to erase the tick marks for all except **Coil_1** as below:



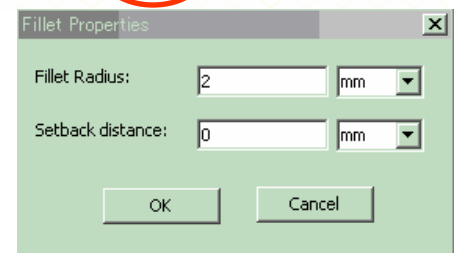
- Change **Object** in Tool Bar to **Edge**

- pick 4 edges of the **Coil_1** as below:



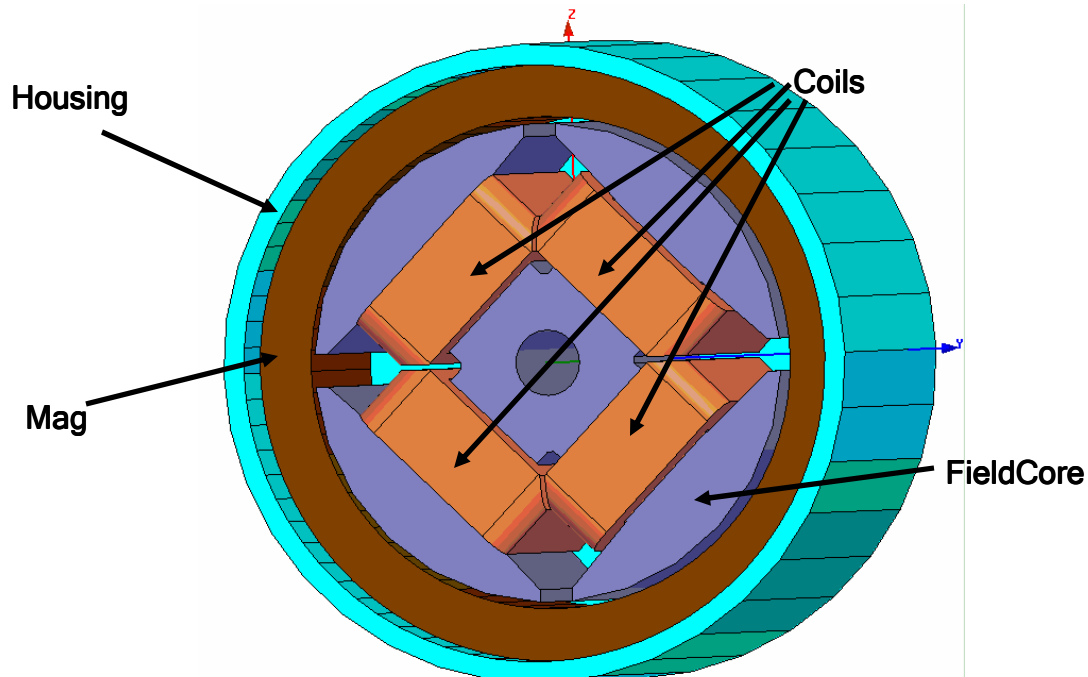
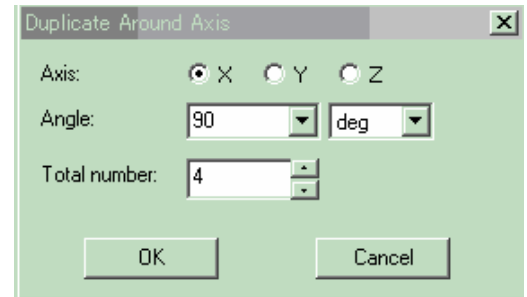
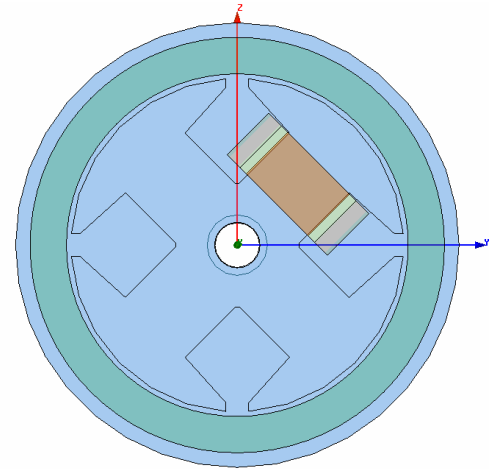
- Select the icon  of Fillet the selected edges in Tool Bar and set 2 mm as Fillet Radius

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



Topic - Anisotropic Material

- ▲ 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 > Offset**, 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:



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

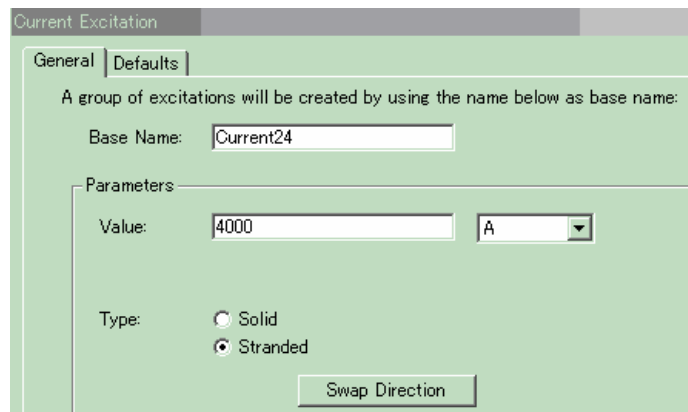
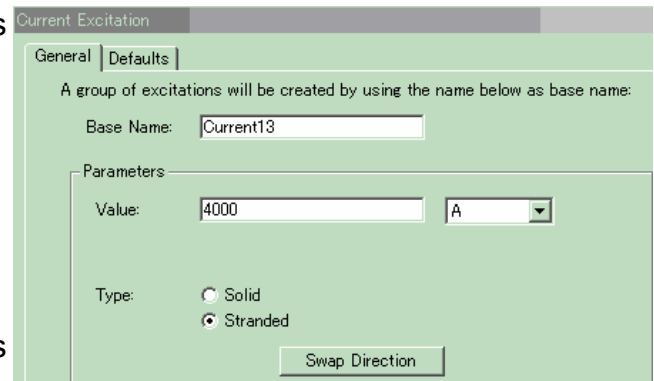
Select the menu item **Maxwell > Excitations > Assign > Current**

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

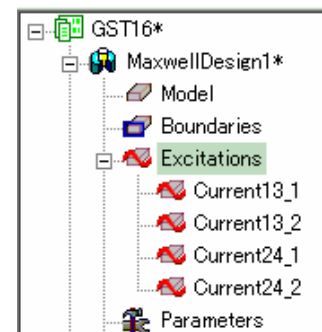
- Select Coil Terminals **Section2 & Section4**

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.



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.

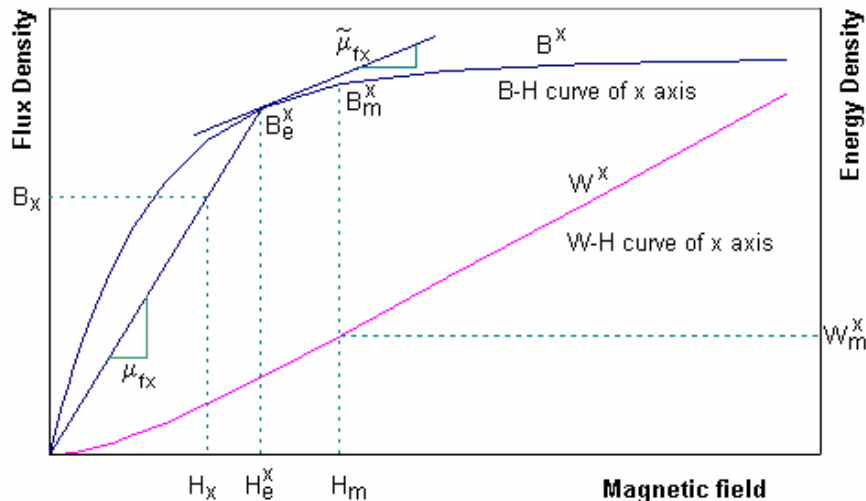
Proposed approach

-  Since applying the same value of $\mathbf{H}$ in different principle directions will cause different level of magnetic saturations, the approach refers all components of $\mathbf{H}$ 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:



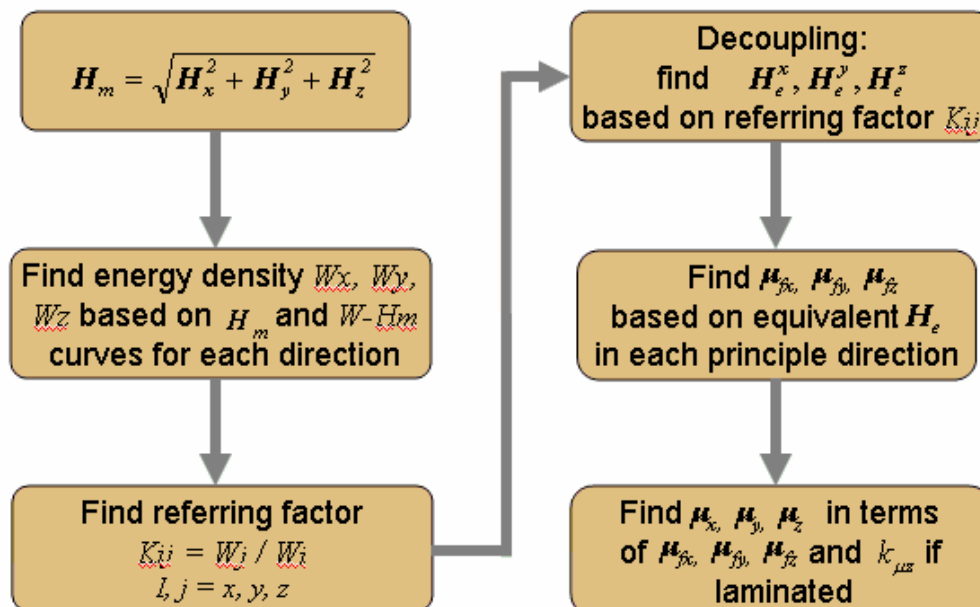
- Taking the lamination into account, the three components of permeability, based on the derived permeability (that is, μ_{fx} , μ_{fy} , μ_{fz}) in iron part, become

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

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:

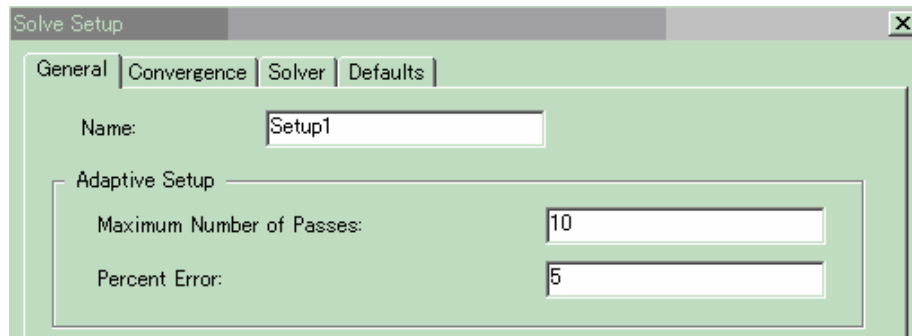


Reference:

- [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

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



Solve Setup

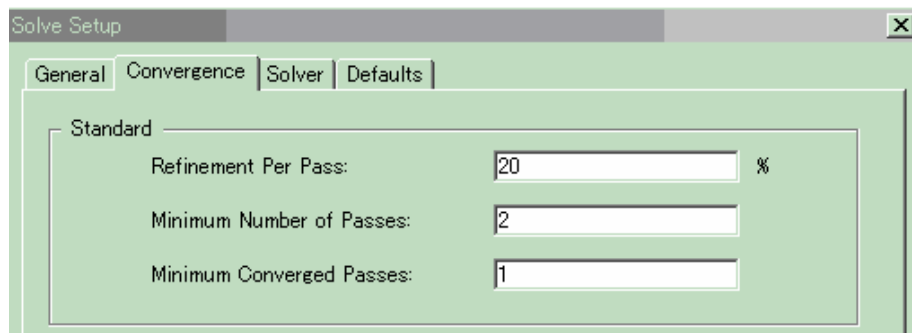
General | Convergence | Solver | Defaults

Name: Setup1

Adaptive Setup

Maximum Number of Passes: 10

Percent Error: 5



Solve Setup

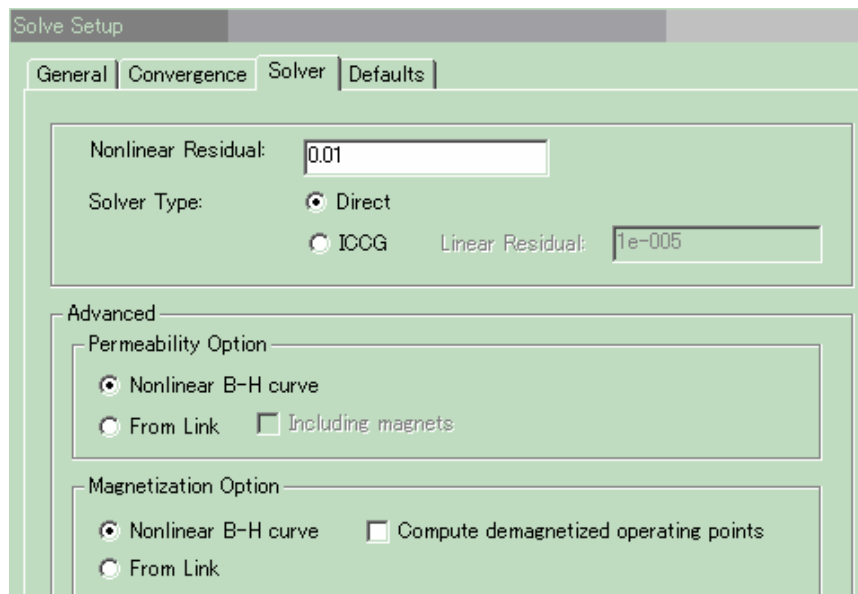
General | Convergence | Solver | Defaults

Standard

Refinement Per Pass: 20 %

Minimum Number of Passes: 2

Minimum Converged Passes: 1



Solve Setup

General | Convergence | Solver | Defaults

Nonlinear Residual: 0.01

Solver Type: ☒ Direct ☐ ICCG Linear Residual: 1e-005

Advanced

Permeability Option

☒ Nonlinear B-H curve ☐ From Link ☐ Including magnets

Magnetization Option

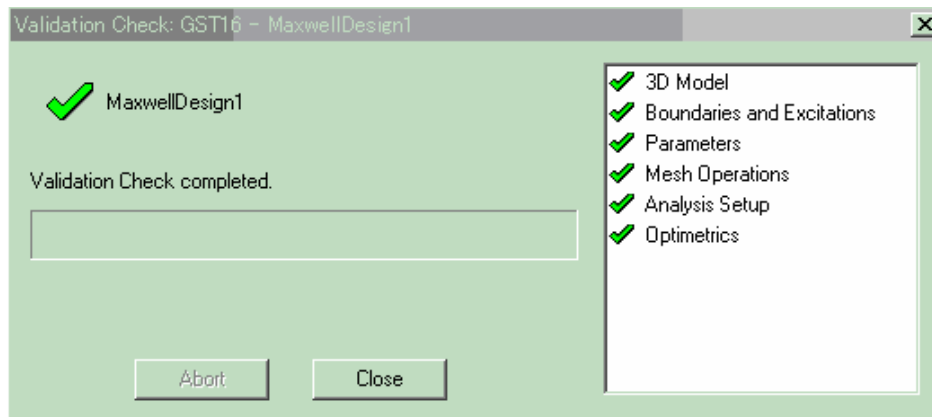
☒ Nonlinear B-H curve ☐ From Link ☐ Compute demagnetized operating points

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

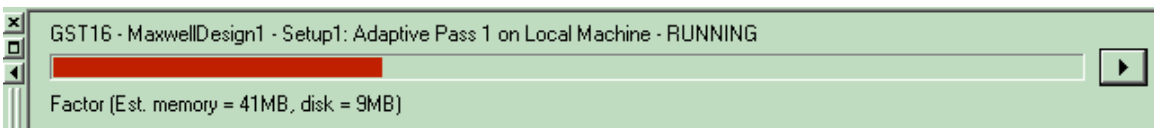
- ▲ Select the menu item **Maxwell > Validation Check**, or click on the  icon



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

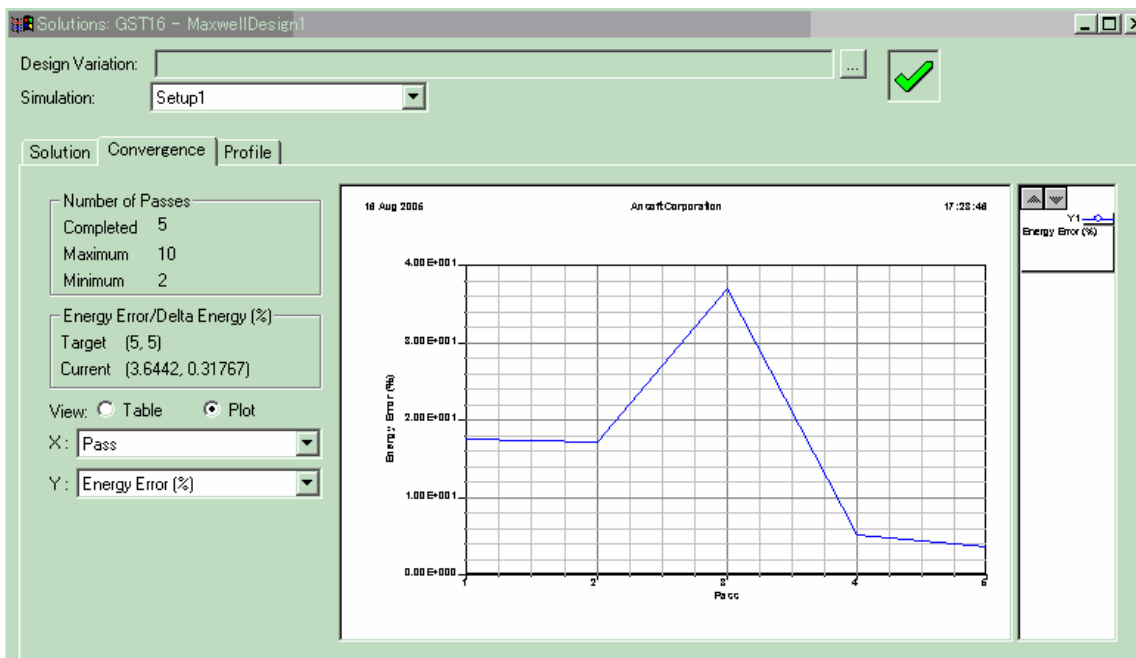
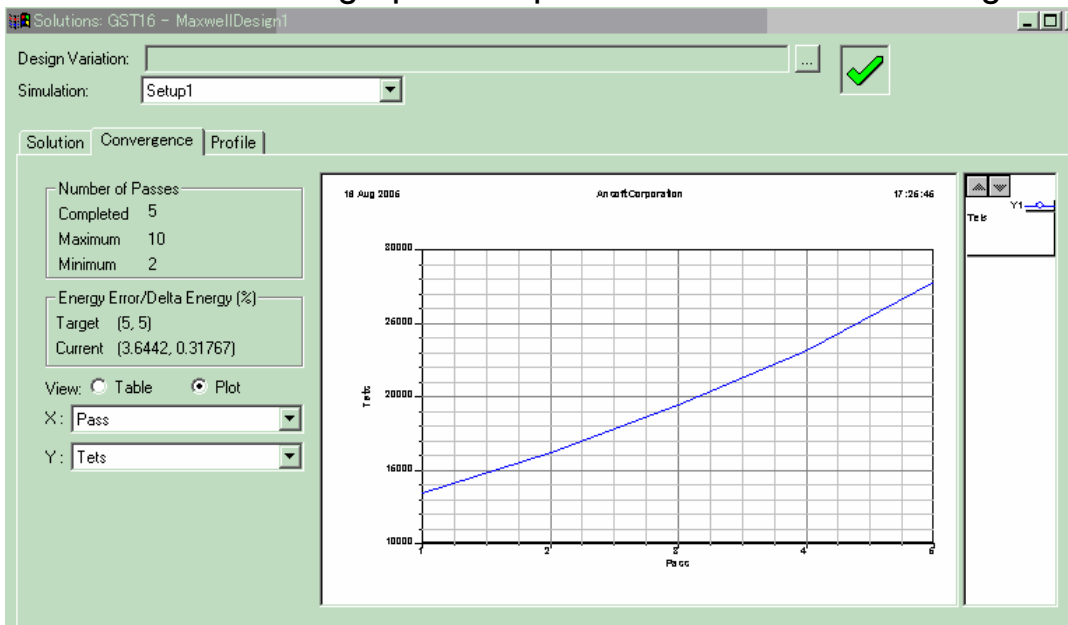
Analyze

- ▲ Select the menu item **Maxwell > Analyze**, or click on the  icon
- The Analysis will be start as below:



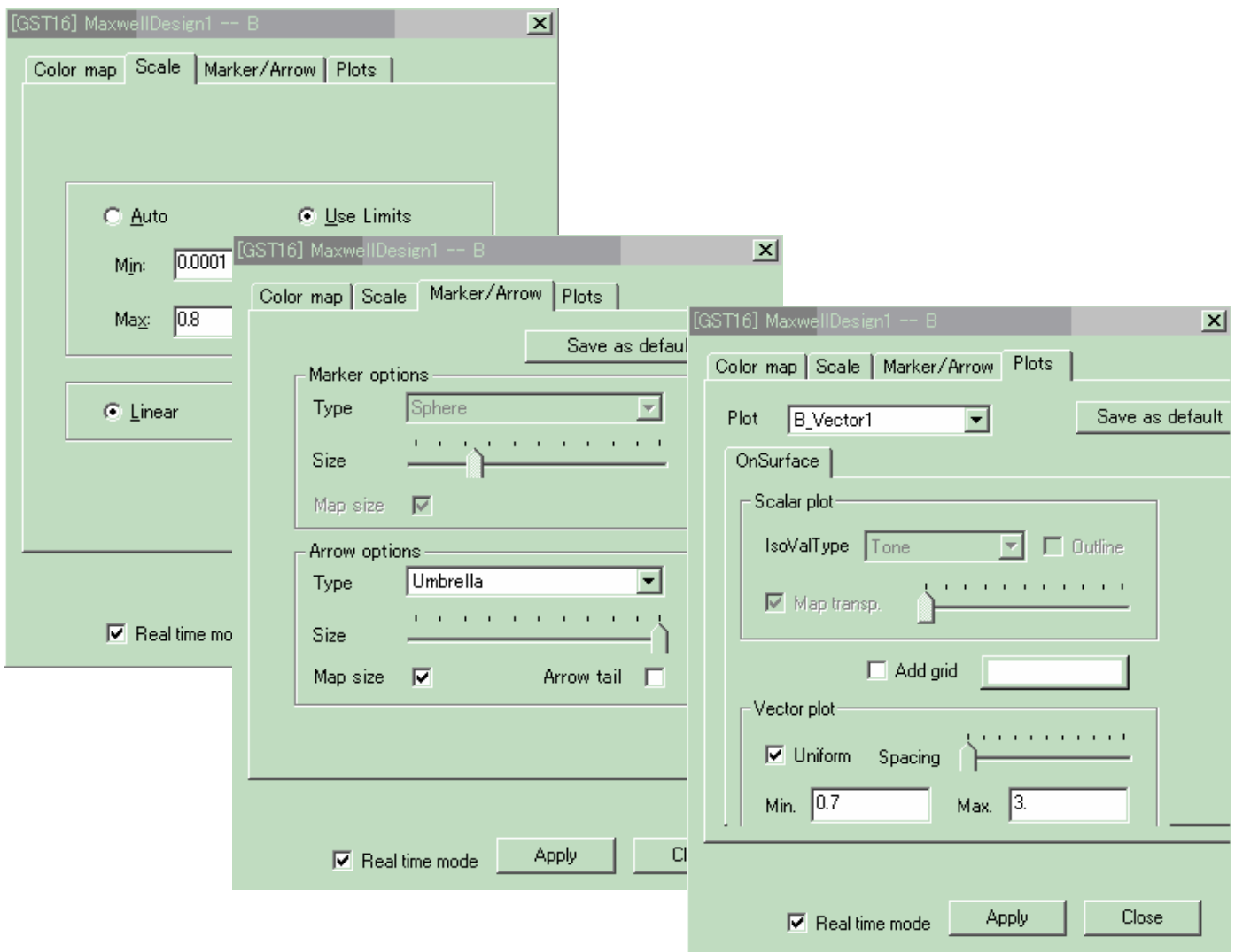
Solution Data

-  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.



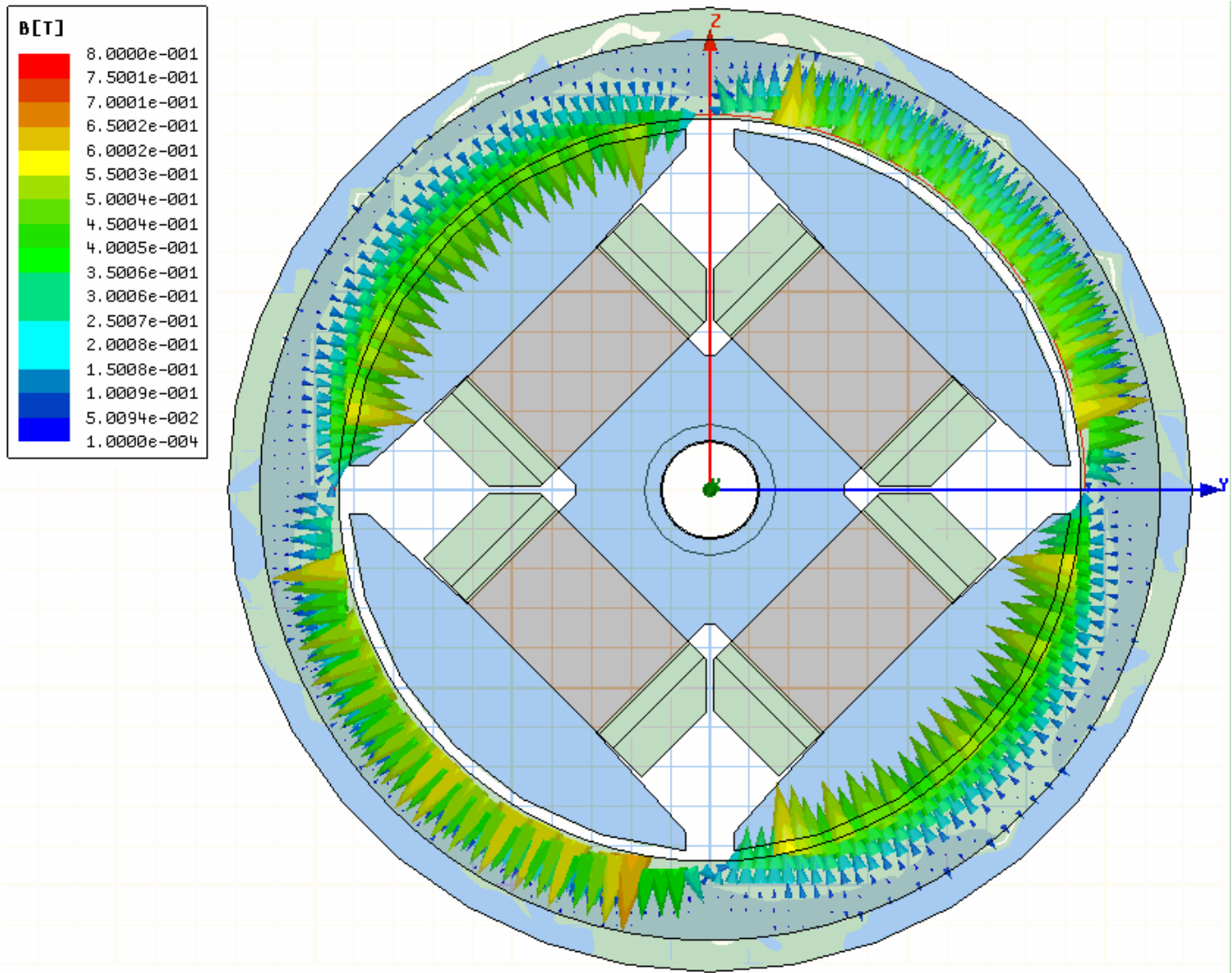
Plot Flux Density on Cross-section YZ of Mag

- ▲ 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**.



Topic - Anisotropic Material

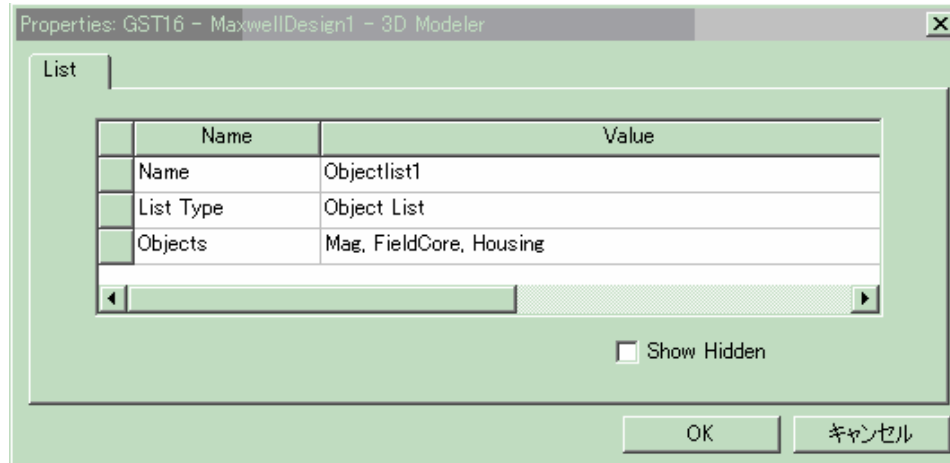
The plot of the flux density on YZ cut plane of the Mag will be as below:



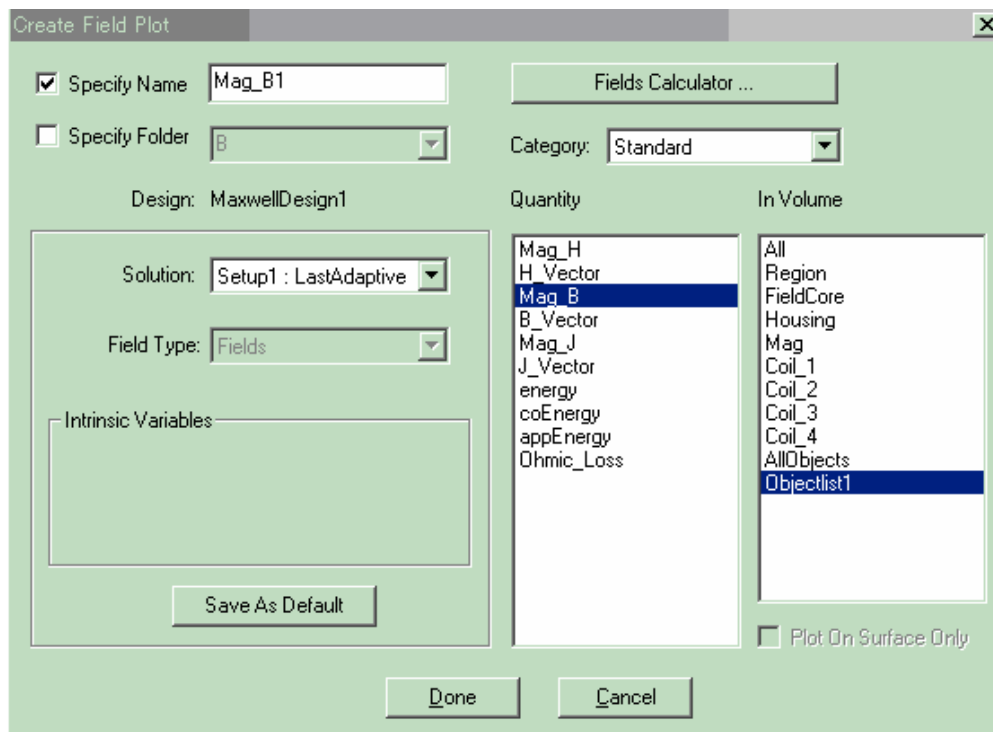
Note: Please note that if the Mag is not set up as anisotropic one (i.e. uniform Nonlinear one) the B field between two magnetic poles will be very large in the theta directions, which will be not as good as user wanted.

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**

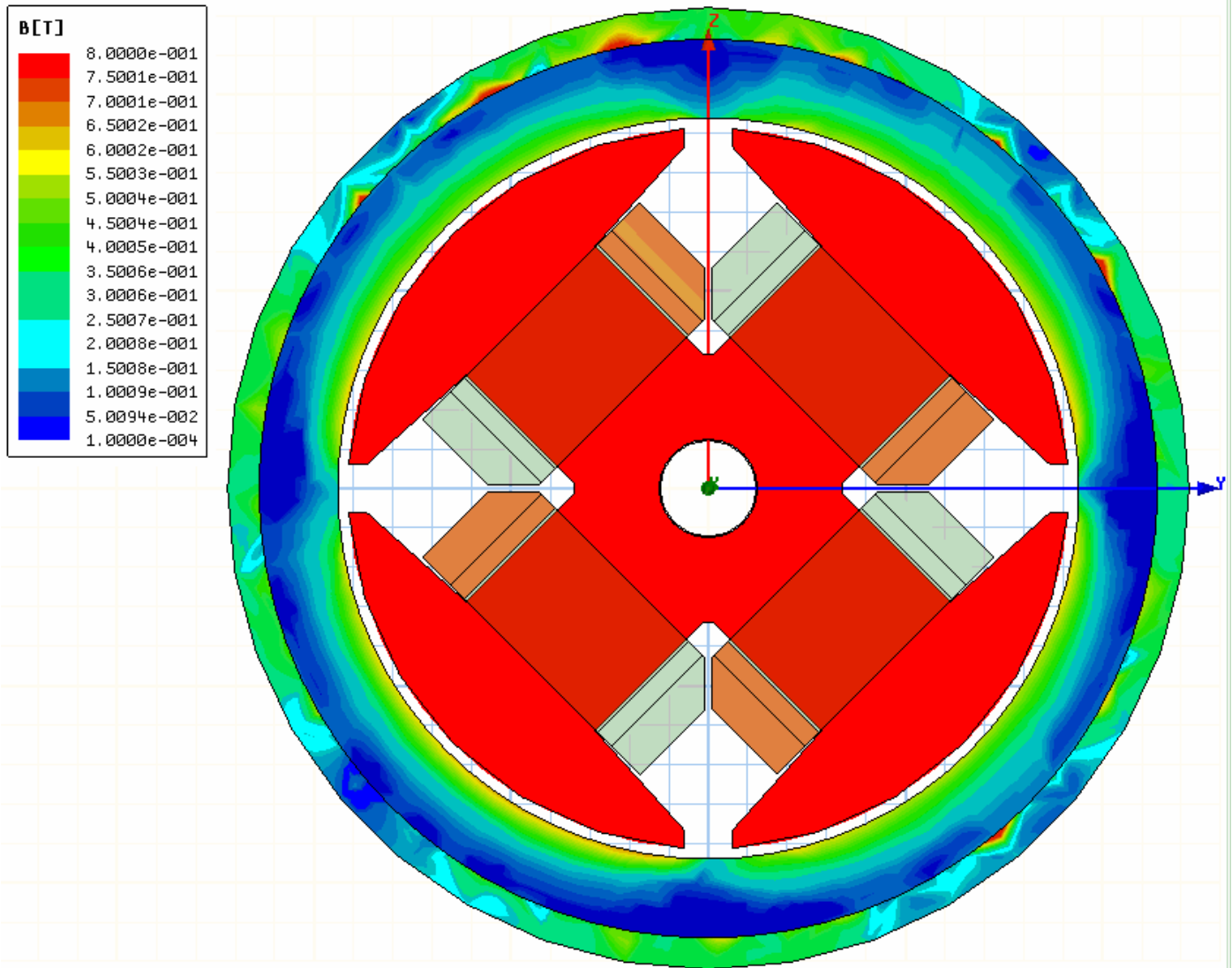


- Select the Global YZ of Planes in History Tree and move the mouse to 3d Modeler Window then right mouse click and select **Fields > B > Mag_B**, pick **Objectlist1** in "In Volume" then press **Done**



Topic - Anisotropic Material

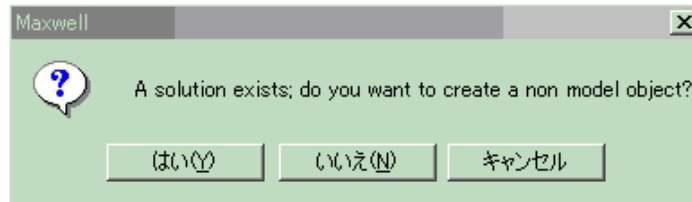
- ▲ The plot of the mag_B on YZ cut plane of the Mag, FieldCore & Housing will be as below:



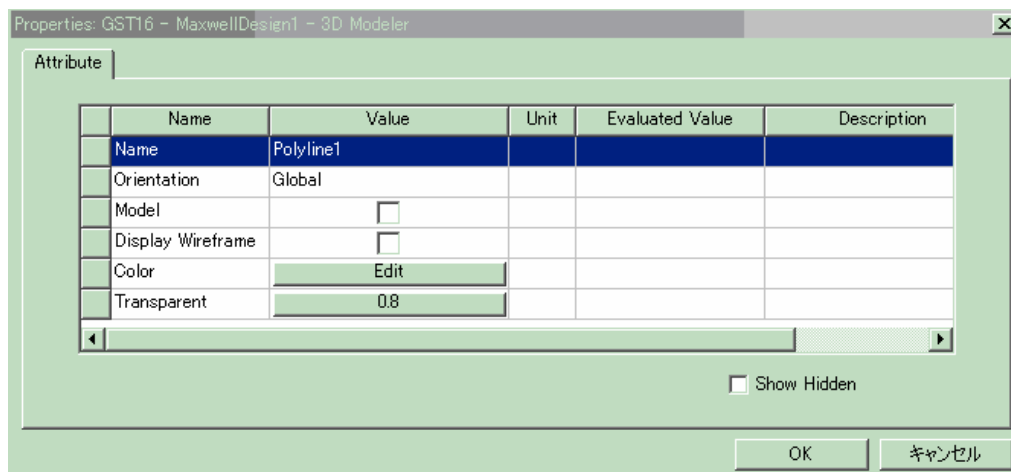
- ▲ User may double click on the legend and change **Scale** to see the detail field distributions

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.

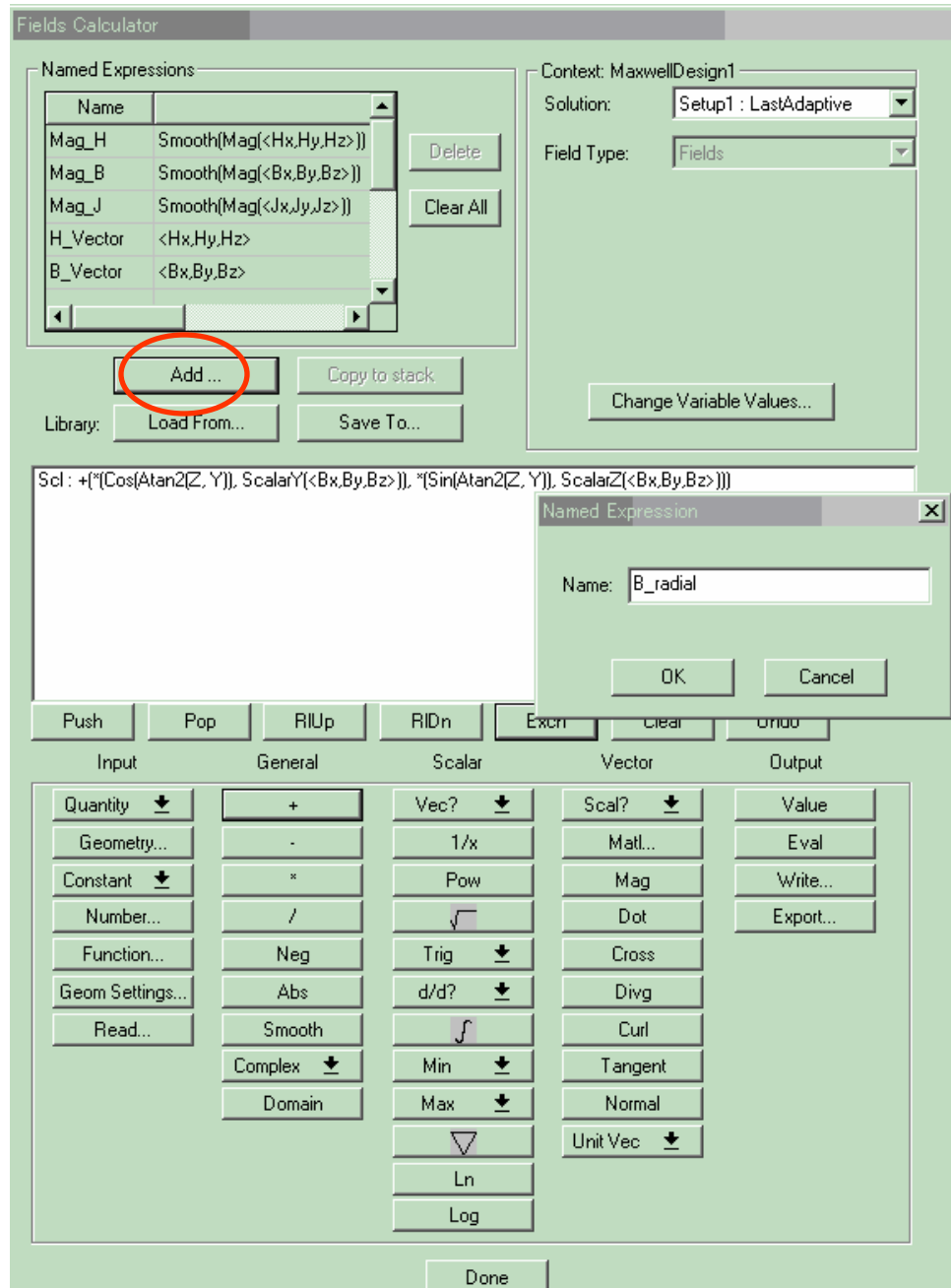


- ▲ Press **OK** to create **Polyline1**.
- ▲ Create B_{radial} in Calculator
 - ▲ Select the Global YZ of Planes in History Tree and move the mouse to 3d Modeler Window then right mouse click and select **Maxwell > Fields > Calculator**
 - Function > Scalar Z > OK**
 - Function > Scalar Y > OK**
 -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

Topic - Anisotropic Material

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

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

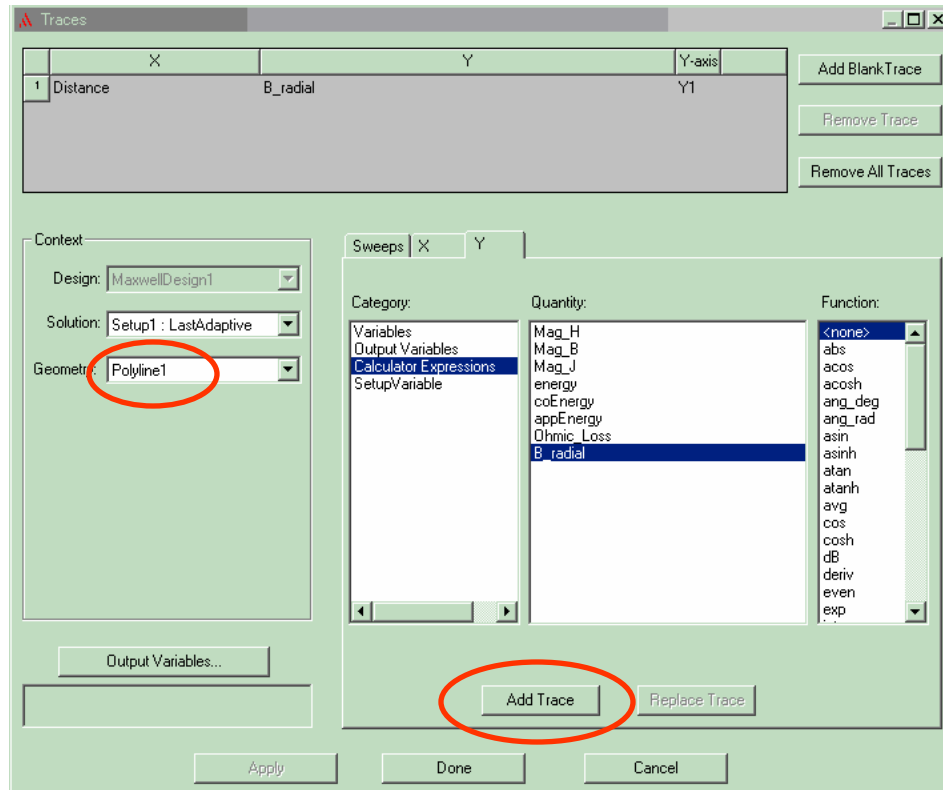


▲ Pick **Add** and type the Name as **B_radial** , pick Ok to finish it.

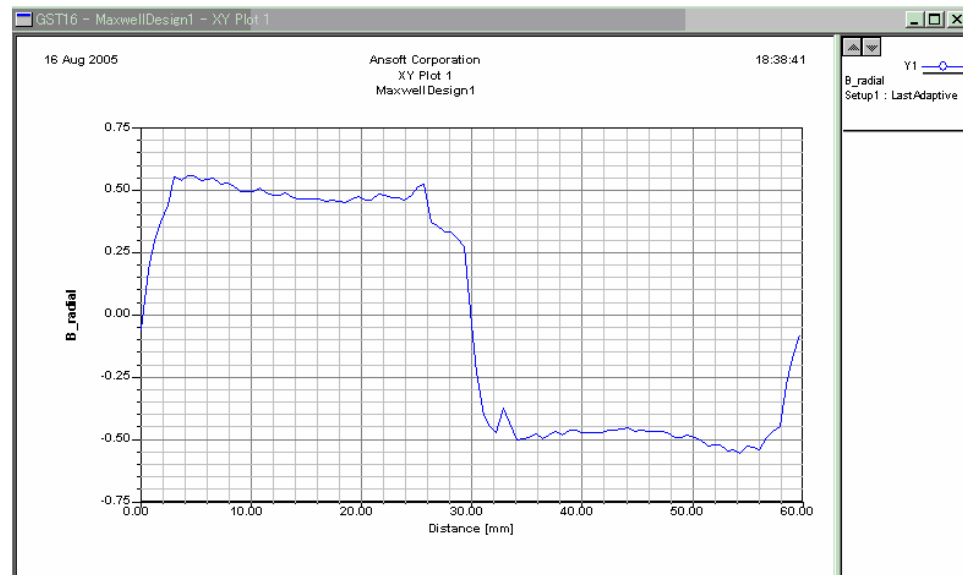
Topic - Anisotropic Material

▲ In Maxwell V11 all 2d plot will be created with **Create Report**

▲ **Maxwell > Result > Create Report > OK**



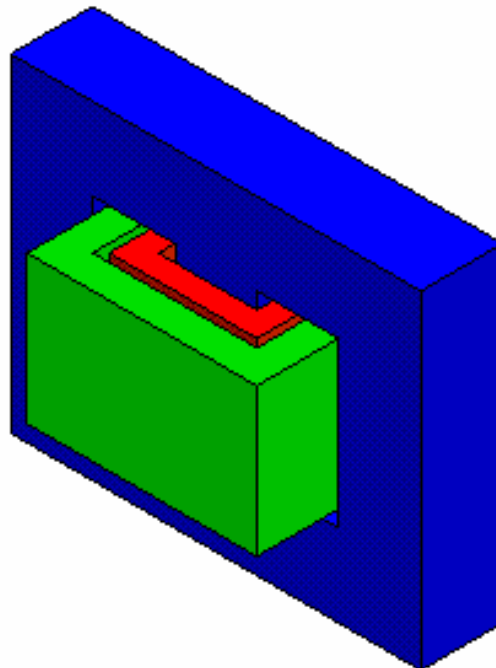
▲ Select **Polyline1** in Geometry, pick **B_radial** in Quantity, click **Add Trace** press **Done**, the result curve on cutline will be se below:



Example (Magnetostatic) - Symmetry Boundaries

The application of Symmetry Boundaries

-  There are many advantages to using Symmetry Boundaries when modeling. Meshes can be decreased greatly by using Symmetry Boundaries when you model the shape of an analytical object. As a result, the saving of the calculation resource and shortening analytical time become possible.
-  In this note, the model of very simple transformer will be analyzed using Magnetostatic solver. At this time, the model size is reduced by using the Symmetry boundaries



Example (Magnetostatic) - Symmetry Boundaries

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: **Line, Rectangle, Box**
-  Surface Operations: **Section, Sweep Along Path**
-  Boolean Operations: **Subtract, Separate Bodies, Split, Duplicate (Mirror)**

- **Boundaries/Excitations**

-  Current: **Stranded**
-  **Symmetry**

- **Analysis**

-  **Magnetostatic**

- **Results**

-  **Solution Data**

- **Field Overlays:**

-  **Flux Density Plots**

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

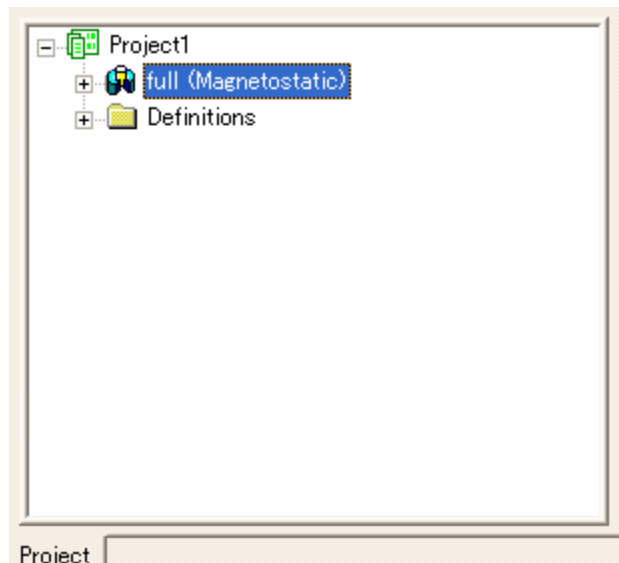
To set the tool options:

-  **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

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.
Select the new design and change the name to **full**.



Set Solution Type

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

Creating the 3D Model of a Symmetry model of a Trans

1. The example that will be used to demonstrate how the Symmetry boundaries is implemented in a Magnetization model of a Transformer. The intent of this write up is not how to simulate the Transformer, it is rather to demonstrate how Symmetry boundaries is implemented within the Magnetostatic solver. A very simple transformer that rolls the winding of coil_in and coil_out in core is used.

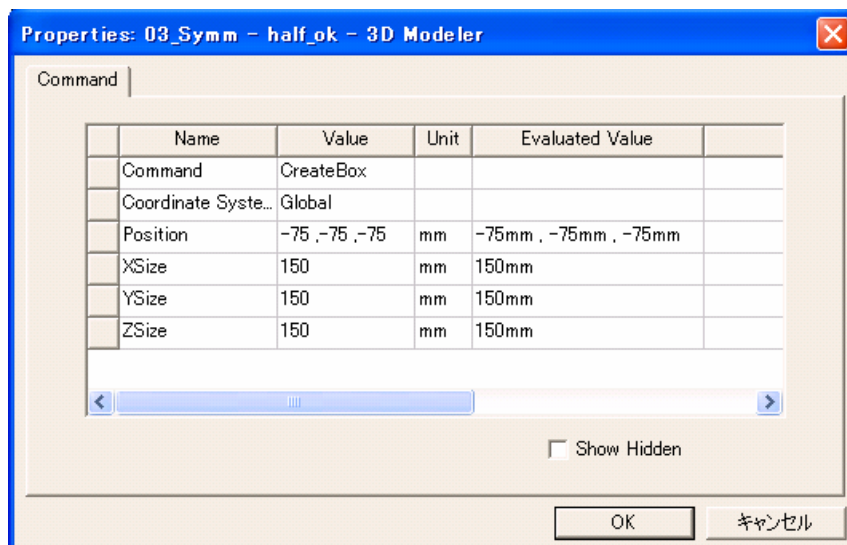
Set Model Units

1. Select the menu item **3D Modeler > Units >**
2. Select Units: **mm (millimeters)**

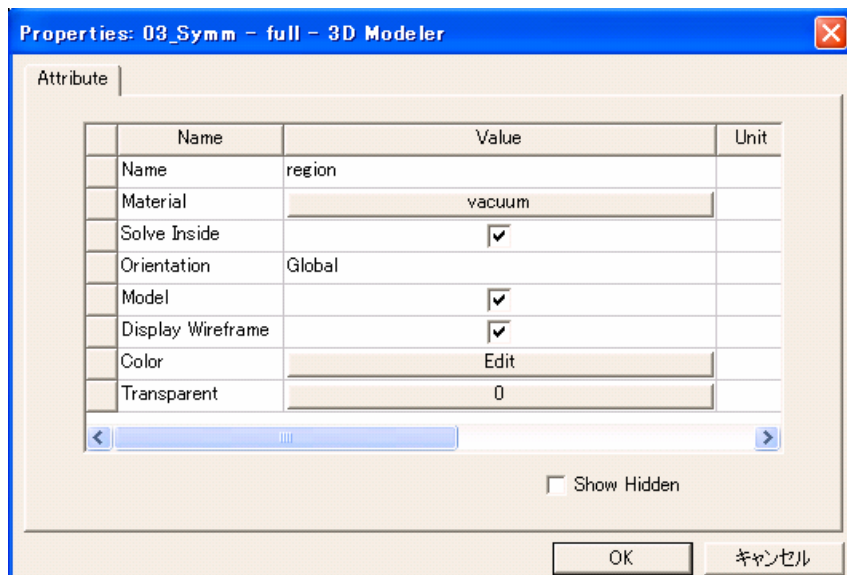
Example (Magnetostatic) - Symmetry Boundaries

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: -75, Y: -75, Z: -75**, Press the **Enter** key
 2. Then enter the opposite corner for the base rectangle with the height
 - ▲ **X: 150, Y: 150, Z: 150**, Press the **Enter** key



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



Example (Magnetostatic) - Symmetry Boundaries

Create the core

▲ **D Modeler > Grid Plane > YZ** then change the grid plane to YZ.

▲ **Draw > Rectangle** then make a rectangle.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen
▲ **X: 5, Y: -25, Z: -25**. Press the **Enter** key.
2. Input the coordinates of opposite corner.
▲ **dY: 50, dZ: 40**, Press the **Enter** key.
3. Change the object name and color.
▲ **Name: core, Color: blue**

Name	Value
Command	CreateRectang...
Coordinate Syste...	Global
Position	5 ,-25 ,-20
Axis	X
YSize	50
ZSize	40

▲ **Draw > Rectangle** then make a rectangle.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen
▲ **X: 5, Y: -15, Z: -10**. Press the **Enter** key.
2. Input the coordinates of opposite corner.
▲ **dY: 10, dZ: -20**, Press the **Enter** key.
3. Change the object name and color.
▲ **Name: slot1, Color: blue**

Name	Value
Command	CreateRectang...
Coordinate Syste...	Global
Position	5 ,-15 ,10
Axis	X
YSize	10
ZSize	-20

▲ Select **slot1** and **Edit > Duplicate > Mirror** then make the copy of **slot1**.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen
▲ **X: 0, Y: 0, Z: 0**. Press the **Enter** key.
2. Input the direction where the copy object will be made.
▲ **.X: 0, dY: 1, dZ: 0**, Press the **Enter** key.

▲ Select **core, slot1, and slot1_1** in this order while pushing the Ctrl key. **3D Modeler > Boolean > Subtract**

▲ Select **core** and **Draw > Sweep > Along Vector**

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen.
▲ **X: 0, Y: 0, Z: 0**. Press the **Enter** key.
2. Input the direction to sweep.
▲ **.X: -10, dY: 0, dZ: 0**, Press the **Enter** key.
3. **Select Draft angle: 0, Draft type: Round**

Name	Value
Command	SweepAlongVe...
Coordinate Syste...	Global
Vector	-10 ,0 ,0
Draft angle	0
Draft type	Round

▲ The material of **core** is **ferrite**.

Example (Magnetostatic) - Symmetry Boundaries

Create the coil_in

▲ **Draw > Box** then make a box.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen

▲ **X: -9, Y: -9, Z: -9.** Press the **Enter** key.

2. Input the coordinates of opposite corner.

▲ **dX: 18, dY: 18, dZ: 18,** Press the **Enter** key.

3. Change the object name and color.

▲ **Name: coil_in, Material: copper, Color: red**

▲ **Draw > Box** then make a box.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen

▲ **X: -6, Y: -6, Z: -9.** Press the **Enter** key.

2. Input the coordinates of opposite corner.

▲ **dX: 12, dY: 12, dZ: 18,** Press the **Enter** key.

3. Change the object name and color.

▲ **Name: coil_in_**

▲ Select **coil_in** and **coil_in_** in this order while pushing the Ctrl key.
3D Modeler > Boolean > Subtract

Name	Value
Command	CreateBox
Coordinate System	Global
Position	-9, -9, -9
XSize	18
YSize	18
ZSize	18

Name	Value
Command	CreateBox
Coordinate System	Global
Position	-6, -6, -9
XSize	12
YSize	12
ZSize	18

Create coil_out

▲ **Draw > Rectangle** then make a rectangle.

1. Input the coordinate of start point in the coordinates input scene in the lower right of the screen

▲ **X: 0, Y: 10, Z: -9.** Press the **Enter** key.

2. Input the coordinates of opposite corner.

▲ **dY: 4, dZ: 18,** Press the **Enter** key.

3. Change the object name and color.

▲ **Name: coil_out, Color: green**

▲ **Draw > Line** then make the path to sweep coil_out

1. Input the following coordinates in the coordinates input scene in the lower right of the screen
In the last point, push Enter key twice

X: 10, Y: 10, Z: 9

X: -10, Y: 10, Z: 9

X: -10, Y: -10, Z: 9

X: 10, Y: -10, Z: 9

2. Change the name to **coil_out_path**

3. Delete **CoverLines** of **coil_out_path** using History window

▲ Select **coil_out** and **coil_out_path** while pushing the Ctrl key.

3D Draw > Sweep > Along Path

Angle of twist: 0deg

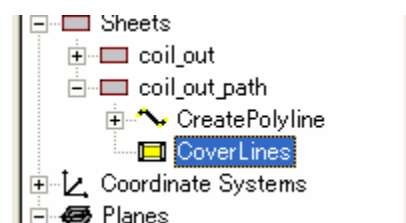
Draft Angle: 0deg

Draft Type: Round

.....

▲ The material of **coil_out** is **copper**.

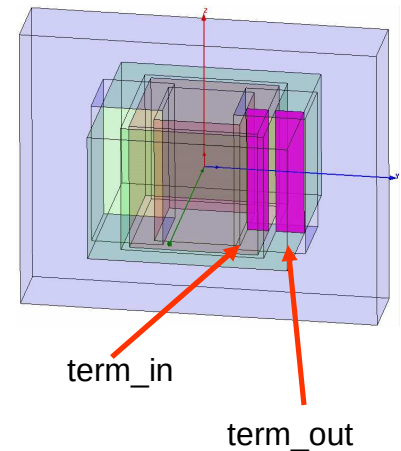
Name	Value
Command	CreateRectangle
Coordinate System	Global
Position	0, 10, -9
Axis	X
YSize	4
ZSize	18



Example (Magnetostatic) - Symmetry Boundaries

Create Coil Terminal

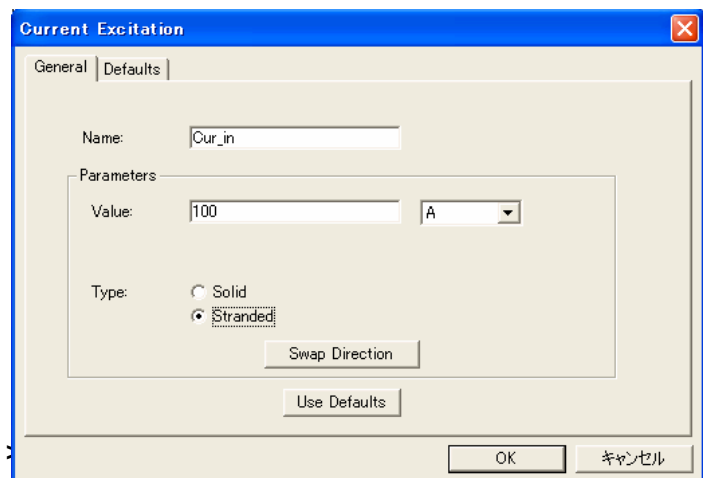
- ▲ Make the terminal for **coil_in** and **coil_out** to pass a current.
 1. Select **coil_in** and **coil_out** while pushing the Ctrl key
3D Modeler > Surface > Section
 - ▲ **Section Plane: YZ**
 2. Select **Section1**, **Section2** that are made now.
3D Modeler > Boolean > Separate Bodies
 3. Delete sections on a minus side of Y axis of each coil as shown in a right figure.
 4. Change each name to **term_in**, **term_out**.



Assign Current Source to the Terminals

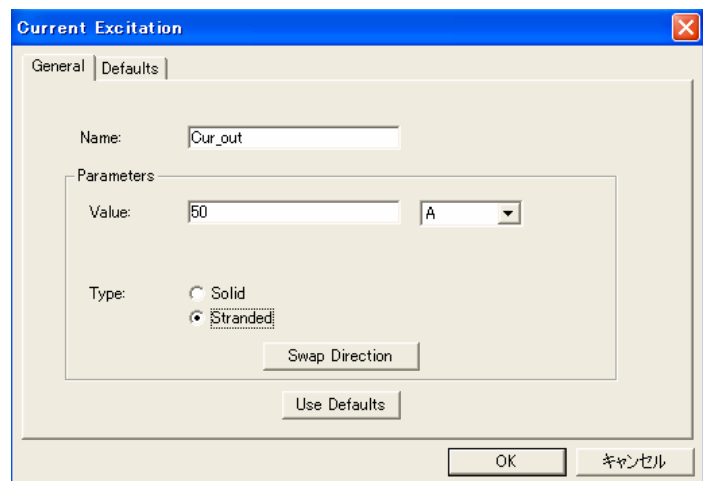
term_in

1. Select **term_in**
2. **Maxwell > Excitations > Assign > Current**
 - Name: Cur_in**
 - Value: 100 A**
 - Type: Stranded**
 - Click **Swap Direction**



term_out

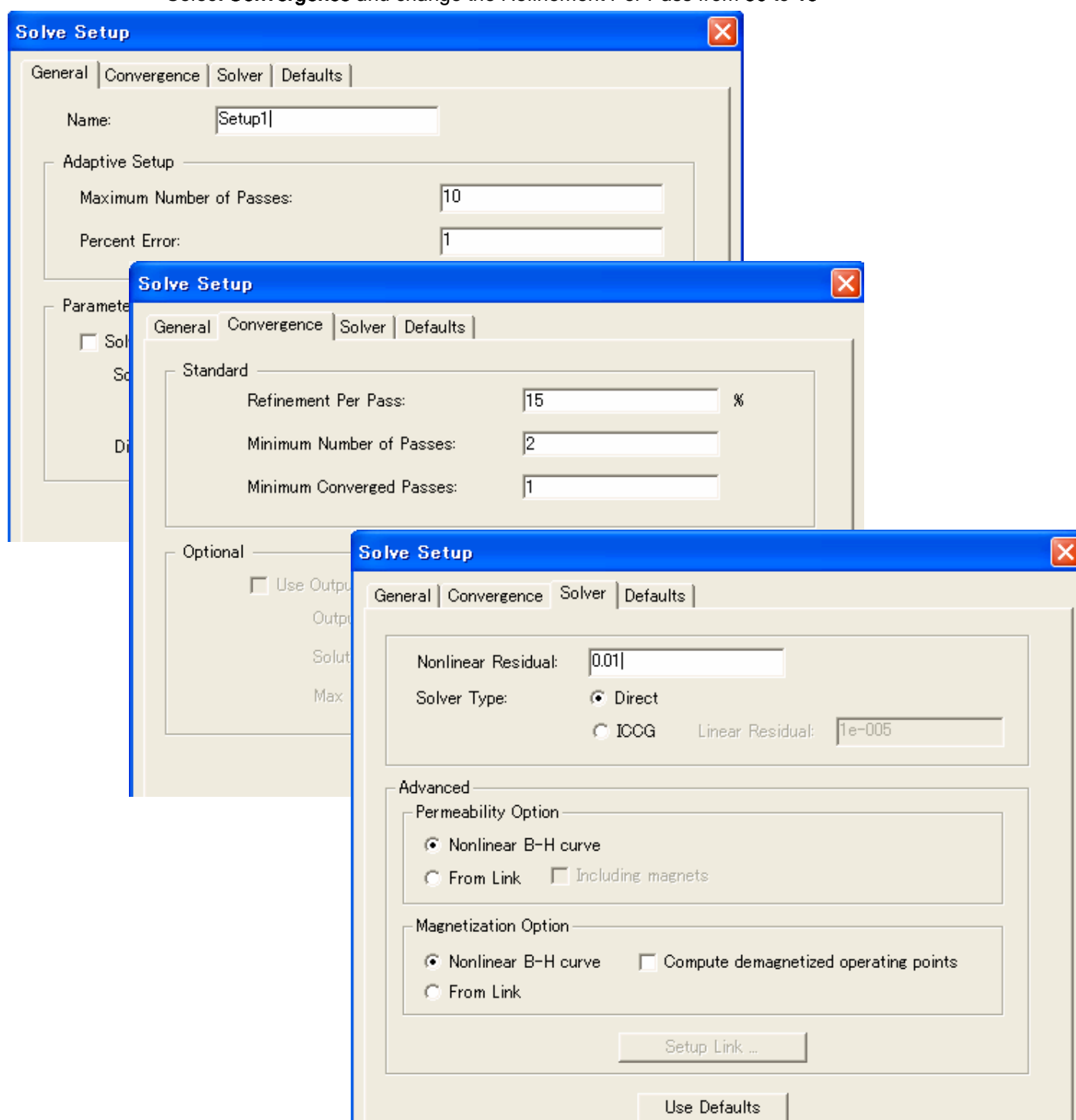
1. Select **term_out**
2. **Maxwell > Excitations > Assign > Current**
 - Name: Cur_out**
 - Value: 50 A**
 - Type: Stranded**



Example (Magnetostatic) - Symmetry Boundaries

Create an Analysis Setup

- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**
- ▲ Select **General** and change the Percent Error to **1**
- ▲ Select **Convergence** and change the Refinement Per Pass from **30** to **15**



Solve Setup

General | Convergence | Solver | Defaults

Name: Setup1

Adaptive Setup

Maximum Number of Passes: 10

Percent Error: 1

Solve Setup

General | Convergence | Solver | Defaults

Standard

Refinement Per Pass: 15 %

Minimum Number of Passes: 2

Minimum Converged Passes: 1

Optional

☐ Use Output

Solve Setup

General | Convergence | Solver | Defaults

Nonlinear Residual: 0.01

Solver Type: ☒ Direct ☐ ICCG Linear Residual: 1e-005

Advanced

Permeability Option

☒ Nonlinear B-H curve ☐ From Link ☐ Including magnets

Magnetization Option

☒ Nonlinear B-H curve ☐ From Link ☐ Compute demagnetized operating points

Setup Link ...

Use Defaults

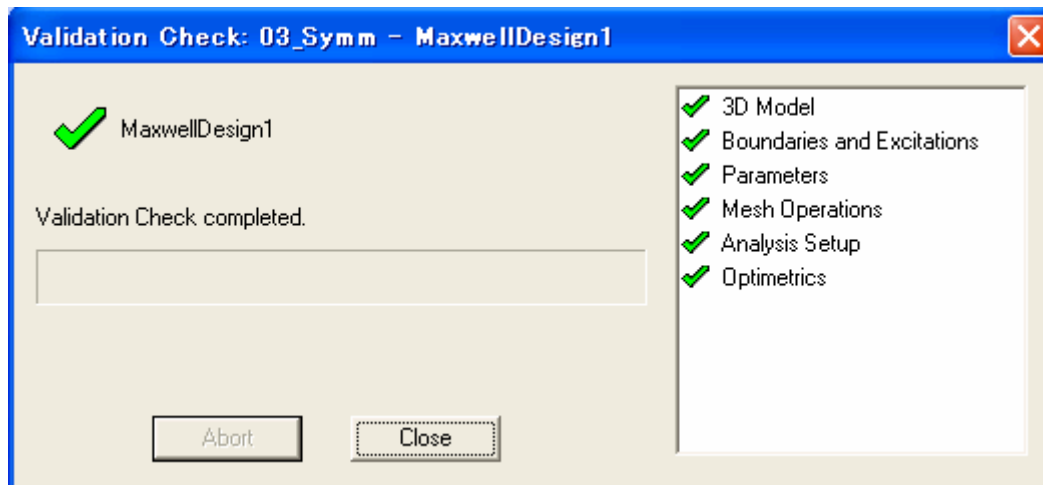
Example (Magnetostatic) - Symmetry Boundaries

Save the Project

- ▲ Select the menu item **File > Save As**
- ▲ From the Save As window, type in **03_Symm**
- ▲ Click on the **Save** button

Check the Validity of the Model

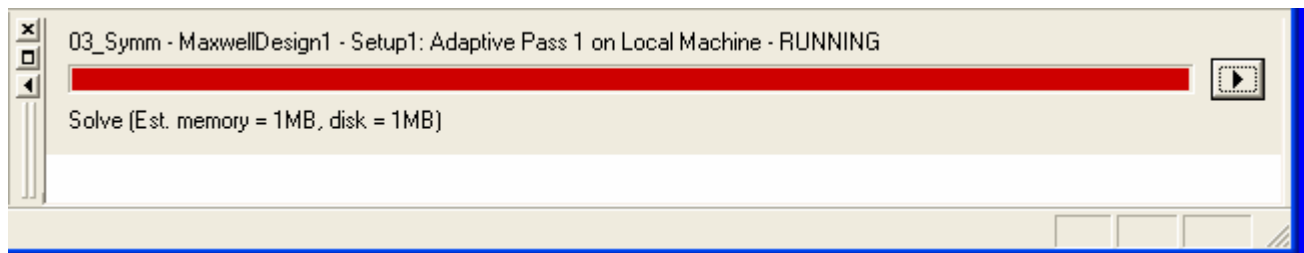
- ▲ Select the menu item **Maxwell > Validation Check**



- ▲ 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:



Example (Magnetostatic) - Symmetry Boundaries

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.

Solutions: 03_Symm - full

Design Variation:

Simulation:

Solution | Convergence | Profile | Mesh Statistics

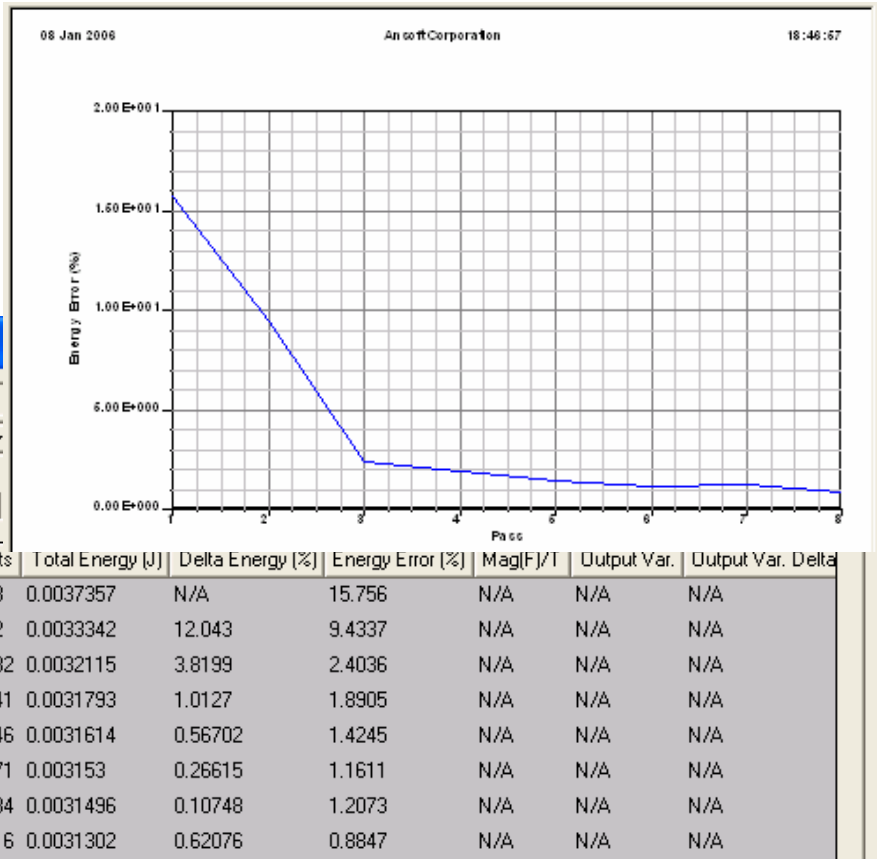
Number of Passes

Completed	8
Maximum	20
Minimum	2

Energy Error/Delta Energy (%)

Target	(1, 1)
Current	(0.8847, 0.62076)

View: ☒ Table ☐ Plot



- You can view mesh information clicking **Mesh Statistics Tab**.

Solutions: 03_Symm - full

Design Variation:

Simulation:

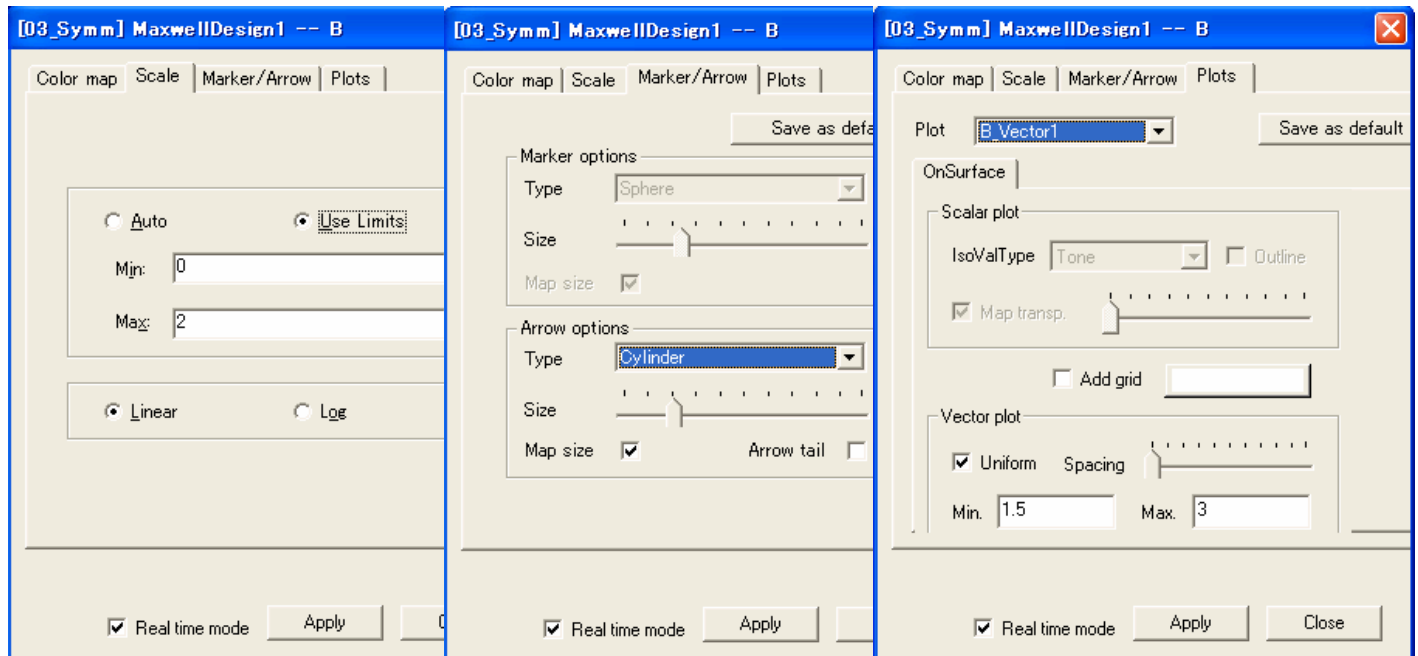
Solution | Convergence | Profile | **Mesh Statistics**

	Num Elements	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn
core	806	2.2295	25	8.51113	0.479441	187.318	19.8511	23.7697
coil_in	101	8.06189	25.4558	14.1044	5.71898	162	32.0792	27.1434
coil_out	99	9.17323	28	17.1173	5.3458	408	69.8182	67.0223
region	1210	2	150	38.0342	0.138184	81529.7	2767.64	9613.54

Example (Magnetostatic) - Symmetry Boundaries

Plot Flux Density Vector on Cross-section YZ

- ▲ 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 **All** 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, the **Max** to 2, click on **Apply**
- ▲ Then select **Marker/Arrow**, in Arrow options Change the Type as **Umbrella** and move Size Arrow to **the third from left**, click on **Apply**.
- ▲ Then select **Plots**, in Vector plot Select **Uniform** and move the **Spacing** size to **smallest**, set Min. as 1.5 and Max. as 3, click on **Apply**.

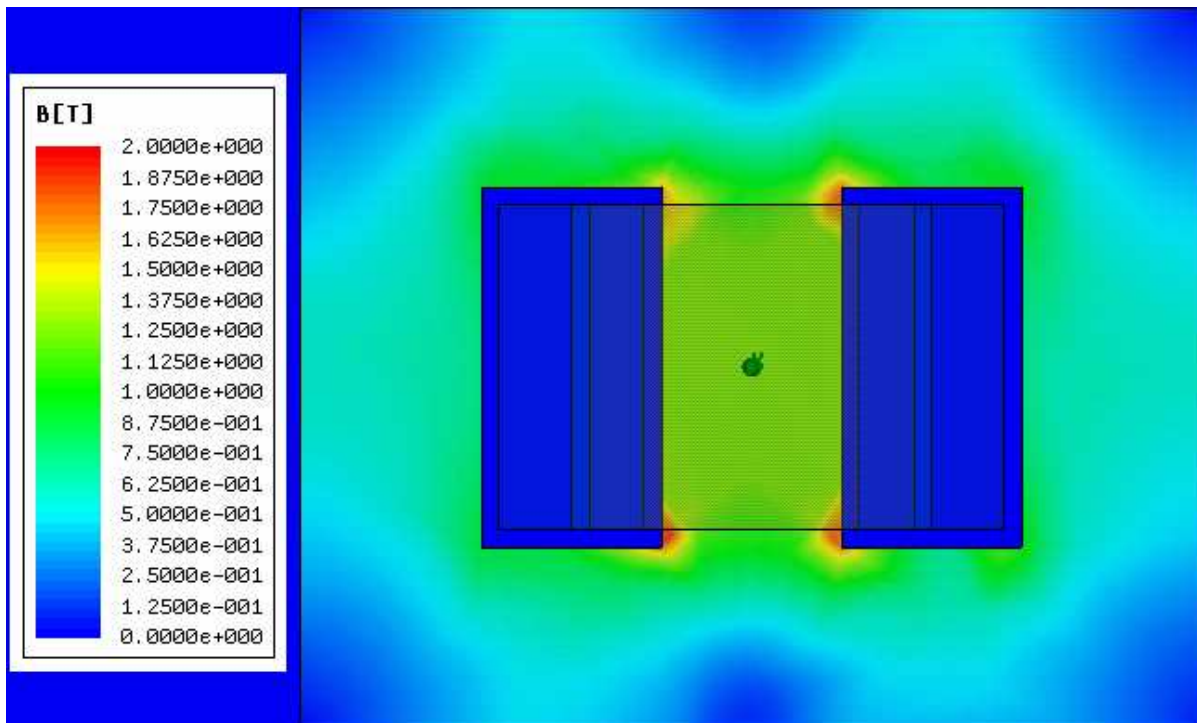
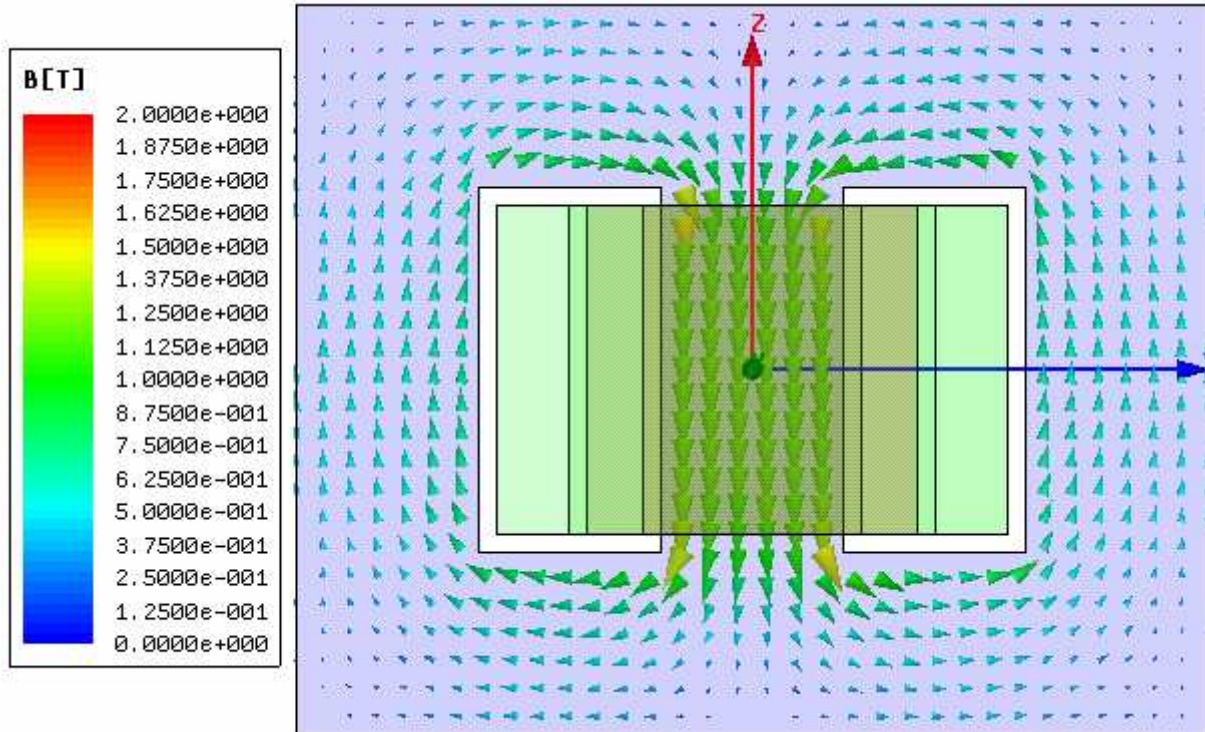


Plot Flux Density Magnitude on Cross-section YZ

- ▲ 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 > Mag_B**, in Create Field Plot Window select **Mag_B** in "Quantity" and **All** in "In Volume", Press **Done**.

Example (Magnetostatic) - Symmetry Boundaries

The plot of the flux density Vector and Magnitude on YZ cut plane will be as below:

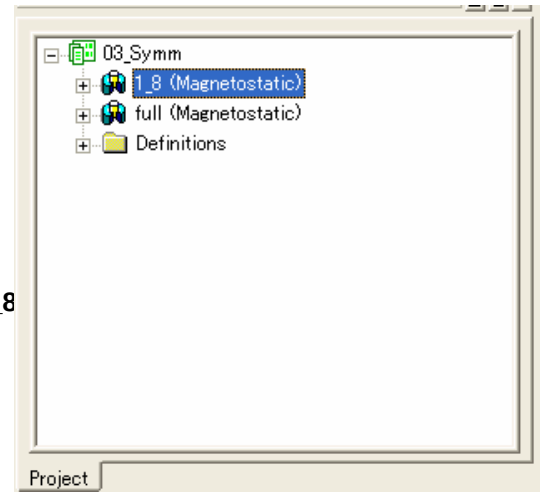


Example (Magnetostatic) - Symmetry Boundaries

Create 1/8 model using Symmetry boundaries

Copy of design

1. Select **full** design in the project manager window
Edit > Copy
2. Select **03_Symm** project in the project manager window
Edit > Paste
3. Change the name of new design from **full_1** to **1_8**



Divide the model

Divide by XY plane

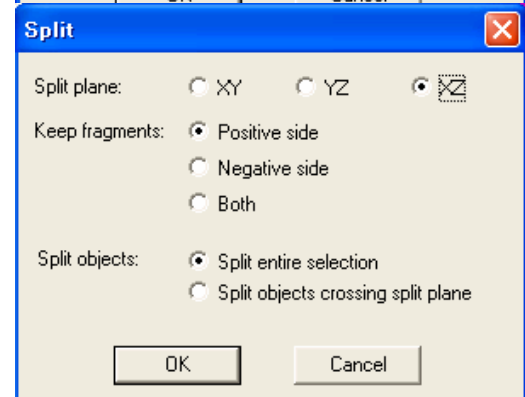
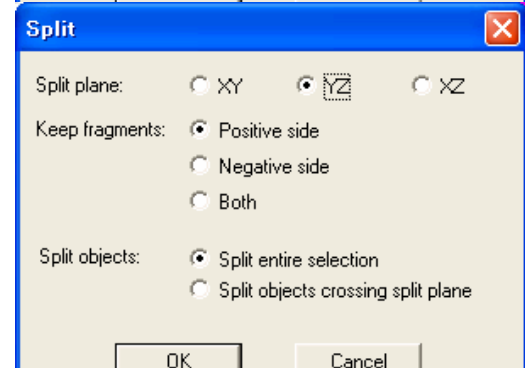
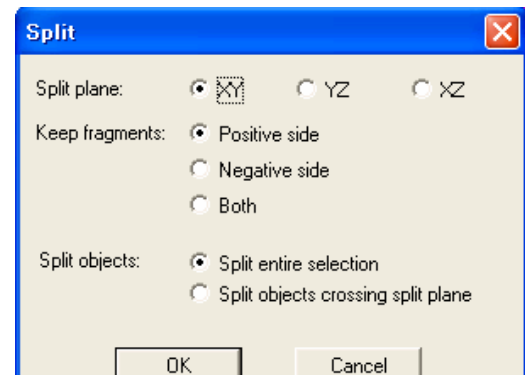
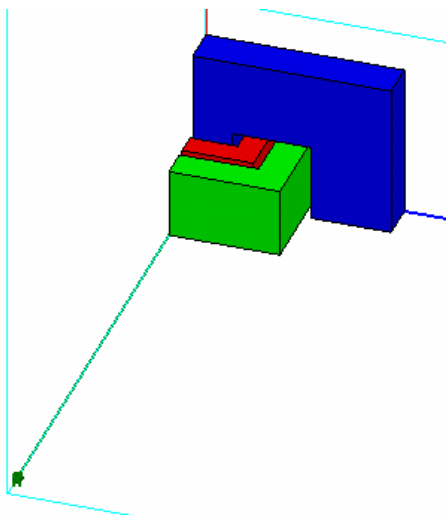
1. **Edit > Select All**
2. **3D Modeler > Boolean > Split**
Split Plane: XY
Keep fragments: Positive side
Split objects: Split entire selection

Divide by YZ plane

1. **Edit > Select All**
2. **3D Modeler > Boolean > Split**
Split Plane: YZ
Keep fragments: Positive side
Split objects: Split entire selection

Divide by XZ plane

1. **Edit > Select All**
2. **3D Modeler > Boolean > Split**
Split Plane: XZ
Keep fragments: Positive side
Split objects: Split entire selection



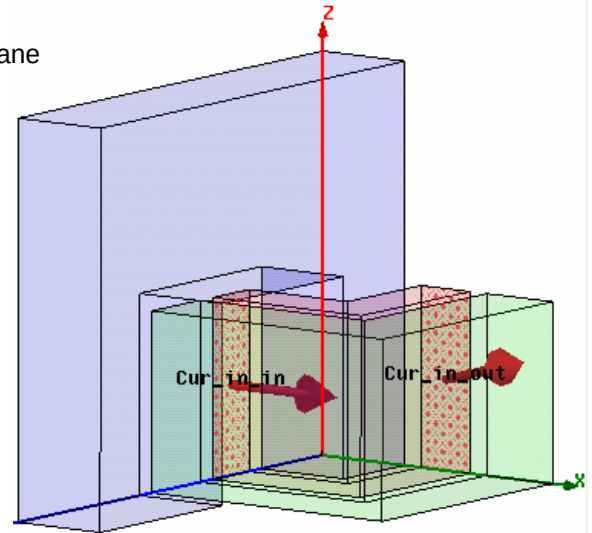
Example (Magnetostatic) - Symmetry Boundaries

Assign Current Source to the Terminals

Edit > Select > Faces then change the selection mode of object.

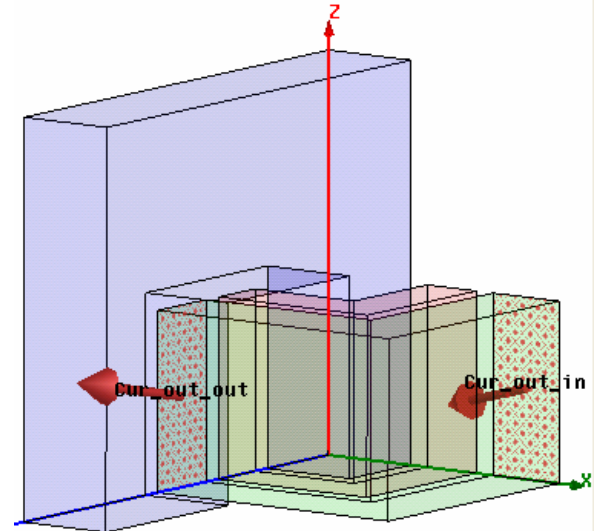
coil_in

1. Select the face of **coil_in** which touches YZ plane
2. **Maxwell > Excitations > Assign > Current**
 Name: Cur_in_in
 Value: 50 A
 Type: Stranded
3. Select the face of **coil_in** which touches XZ plane
4. **Maxwell > Excitations > Assign > Current**
 Name: Cur_in_out
 Value: 50 A
 Type: Stranded
 Click **Swap Direction**



coil_out

1. Select the face of **coil_out** which touches YZ plane
2. **Maxwell > Excitations > Assign > Current**
 Name: Cur_out_out
 Value: 25 A
 Type: Stranded
3. Select the face of **coil_out** which touches XZ plane
4. **Maxwell > Excitations > Assign > Current**
 Name: Cur_out_in
 Value: in A
 Type: Stranded
 Click **Swap Direction**

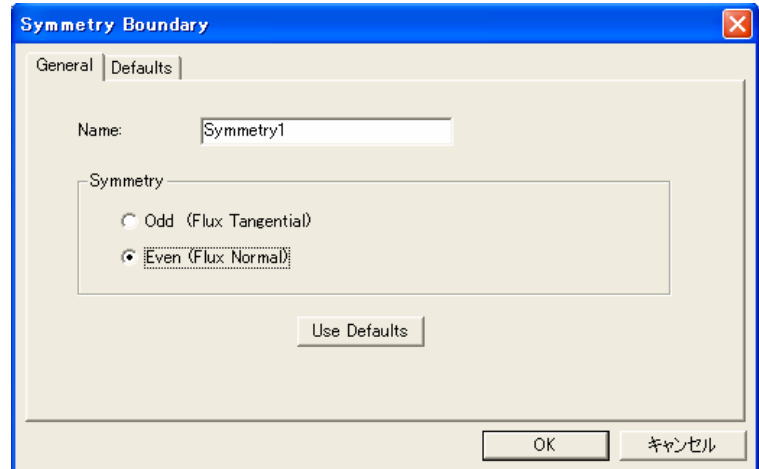


Note: The current value of coil_in was 100A and coil_out was 50A in full model. The sectional area of the coil is 1/2 in 1/8 models because it is divided by XY plane. Therefore, the current value becomes 1/2, too.

Example (Magnetostatic) - Symmetry Boundaries

Setup boundaries

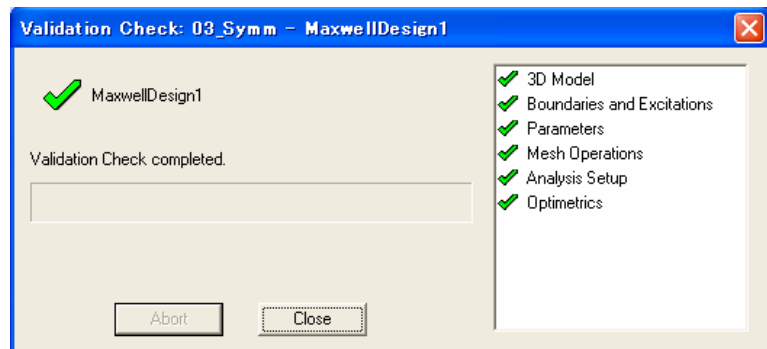
- Select the face of **region** which touch XY plane.
Maxwell > Boundaries > Assign > Symmetry
 Name: Symmetry1
 Symmetry: Even (Flux Normal)



Note: The surface of boundary area (region) is analyzed as a natural boundary condition (Flux is the horizontal compared with respect). Because Flux becomes the horizontal for the YZ plane and the XZ plane in this model, the Symmetry boundaries is not necessary.

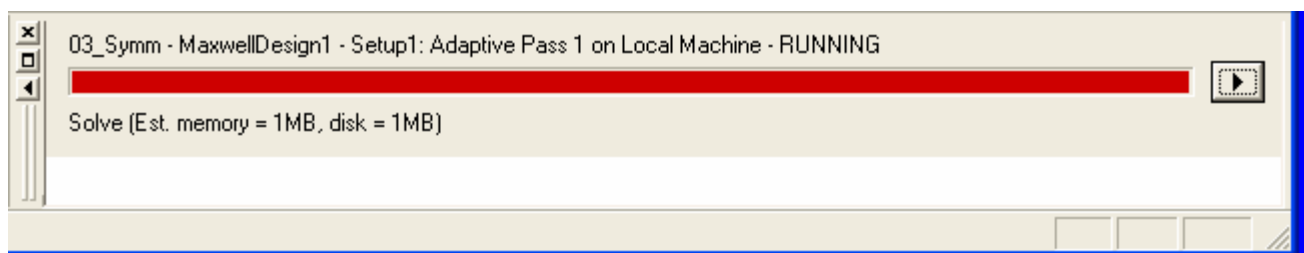
Check the Validity of the Model

- Select the menu item
Maxwell > Validation Check



Analyze

- Select the menu item **Maxwell > Analyze**, or click on the  icon
 The Analysis will be start as below:



Example (Magnetostatic) - Symmetry Boundaries

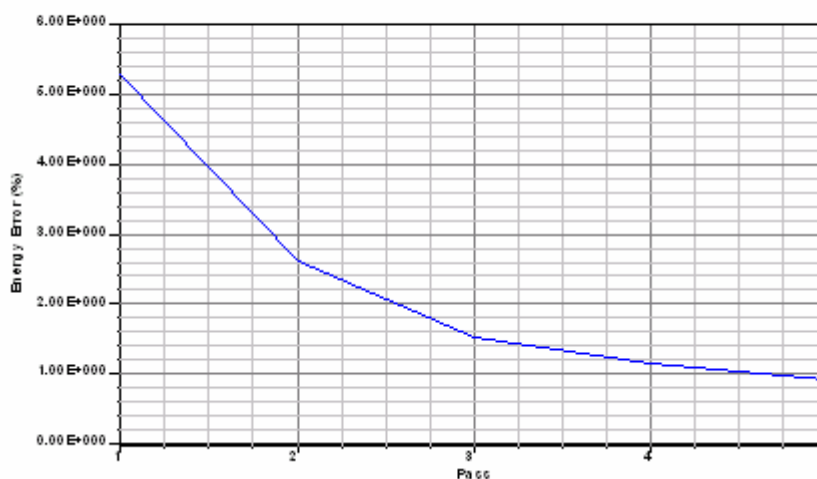
Solution Data

- 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.

08 Jan 2006

Ansoft Corporation

18:51:18



Solutions: 03_Symm - 1_8

Design Variation:

Simulation:

Solution **Convergence** Profile Mesh Statistics

Number of Passes:

Completed 5

Maximum 10

Minimum 2

Energy Error/Delta Energy (%)

Target (1.1)

Current (0.95793, 0.15055)

View: ☒ Table ☐ Plot

Pass							
1	222	0.00040545	N/A	5.2944	N/A	N/A	N/A
2	245	0.00039764	1.9637	2.6459	N/A	N/A	N/A
3	284	0.00039251	1.308	1.5366	N/A	N/A	N/A
4	319	0.00039086	0.42281	1.1866	N/A	N/A	N/A
5	372	0.00039027	0.15055	0.95793	N/A	N/A	N/A

- You can view mesh information clicking **Mesh Statistics Tab**.

Solutions: 03_Symm - 1_8

Design Variation:

Simulation:

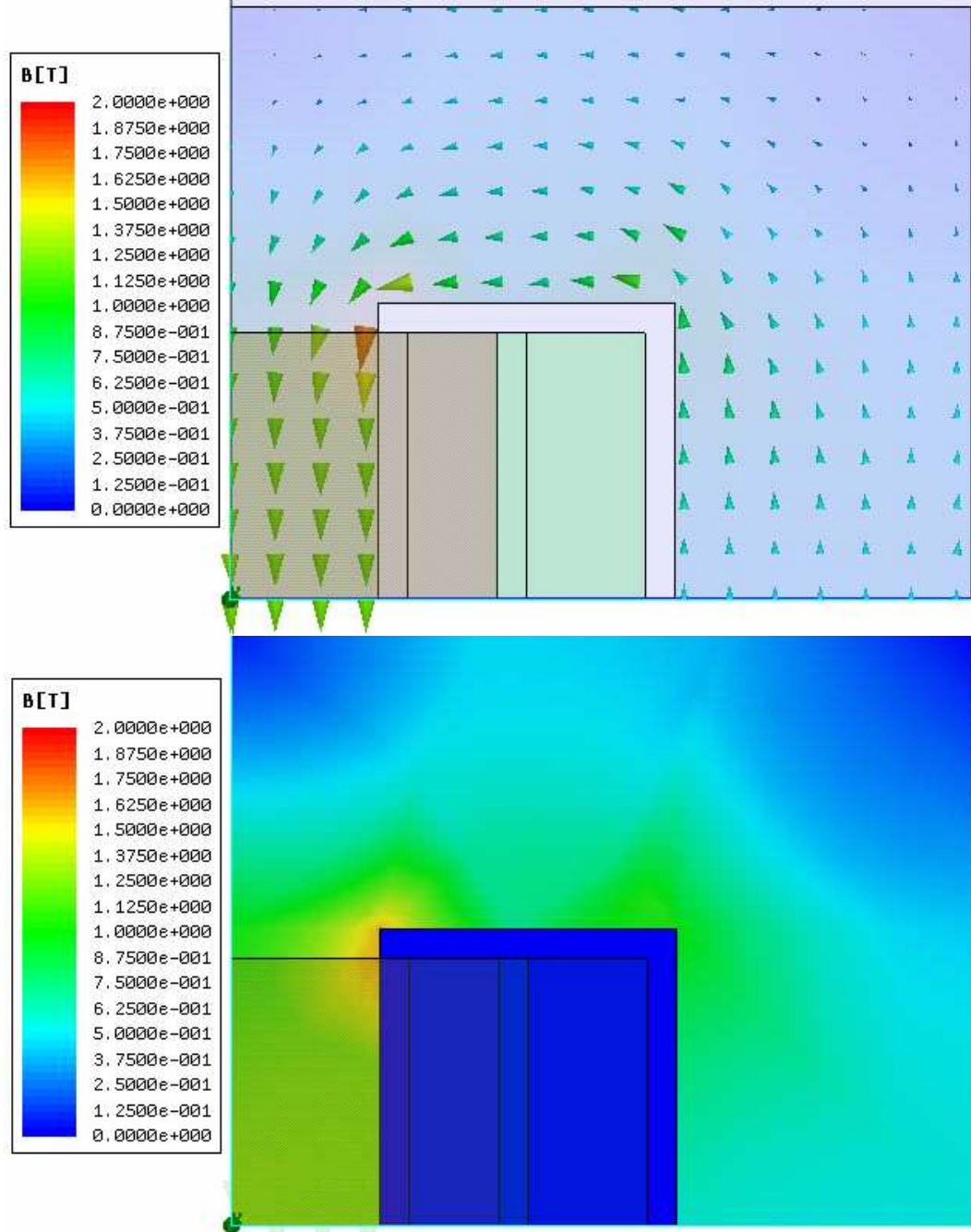
Solution Convergence Profile **Mesh Statistics**

	Num Elements	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn
core	108	3.57071	14.1421	8.33517	1.04167	83.3333	18.5185	17.7378
coil_in	25	9.48683	10.5034	10.0544	4.5	27	16.2	7.54996
coil_out	36	5.22614	16.6433	10.6764	5.8125	84	24	20.1851
region	203	1.41421	106.066	33.6047	0.166667	43554	2062.1	6746.59

- In the full model, the number of mesh was 2216, now it becomes 372 meshes in 1/8 models. That means the number of mesh become about 1/6. Moreover, analytical accuracy is almost same because it is specified by Energy Error.

Example (Magnetostatic) - Symmetry Boundaries

▲ The plot of the flux density Vector and Magnitude on YZ cut plane will be as below:

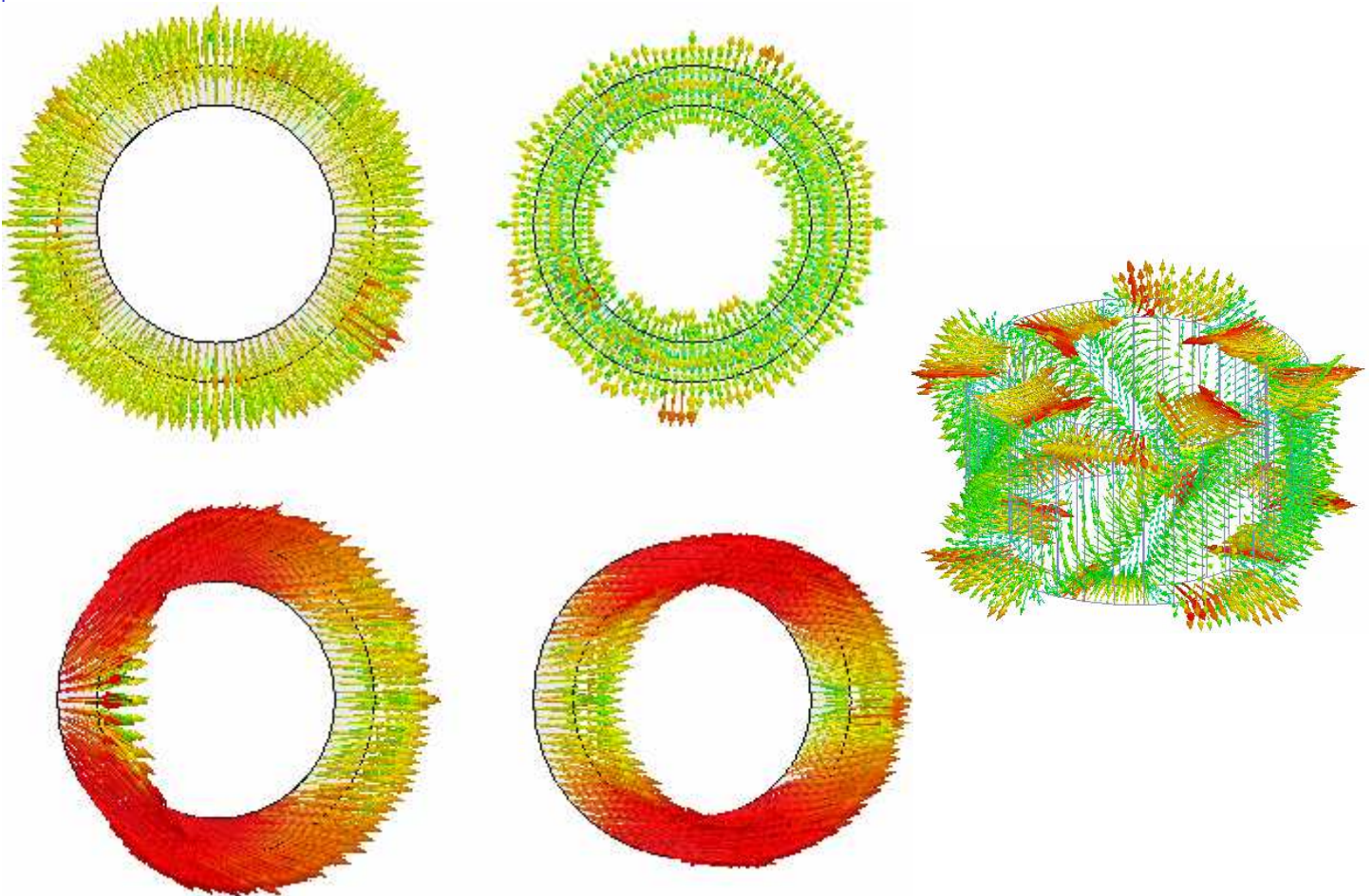


▲ The result is almost same as that of full model

Example (Magnetostatic) - Permanent Magnet Magnetization

Permanent Magnet Magnetization

- ▲ Each object can have the coordinate system that is independent and is arbitrary in Maxwell v11. The magnetization setting can be done in an arbitrary direction for using this coordinate system.
- ▲ Moreover, because it is also possible to do the magnetization setting by using the equation and the numeric character data, the user can do a variety of magnetization settings.
- ▲ This chapter introduces the magnetization setting as shown in the figure below as an example of an easy toroidal magnet.





Example (Magnetostatic) - Permanent Magnet Magnetization

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: **Regular Polyhedron Box**
-  Coordinate System: **Relative CS**
-  Boolean Operations: **Subtract**

- Analysis

-  Magnetostatic

- Field Overlays:

-  **B Vector Plots.B Magnitude Plots**



Example (Magnetostatic) - Permanent Magnet Magnetization

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 :

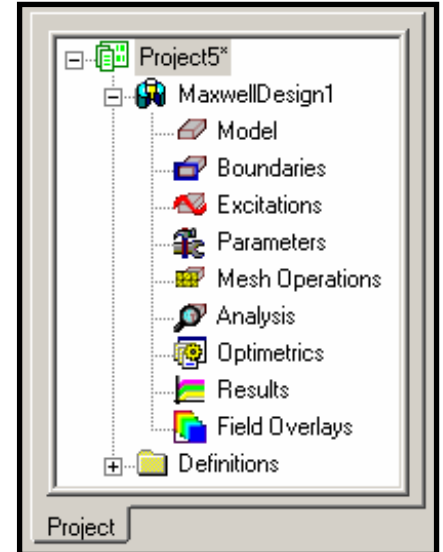
1. Select the menu item **Tools > Options > Maxwell Options**
2. Maxwell Options Window:
 1. Click the **General Options** tab
 -  Use Wizards for data entry when creating new boundaries: ☒ **Checked**
 -  Duplicate boundaries with geometry: ☒ **Checked**
 2. Click the **OK** button
3. Select the menu 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

Example (Magnetostatic) - Permanent Magnet Magnetization

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 ...**

Change Design name

- Right-click **MaxwellDesign1** at the project manager window , change the Design Name to **Ring01**.

Set Model Units

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

Saving of project

- Select the menu item **File > Save as**
- Enter the folder name that you want to save and change the Project name to **Magnet**.

Example (Magnetostatic) - Permanent Magnet Magnetization

Example..Coordinate system of object and setting for each axis

In this example, it is introduced the magnetization of using Coordinate system of object and setting for each axis. Material : **Mag01** and **Mag02** is newly made in this Example.

- ▲ 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: -200, Y: -200, Z: -200.** Press the **Enter** key
 2. Then enter the opposite corner for the base rectangle with the height
 - ▲ **dX: 400, dY: 400, dZ: 400,** Press the **Enter** key

Command Attribute			
	Name	Value	Unit
	Command	CreateBox	
	Coordinate System	Global	
	Position	-200 ,-200 ,-200	mm
	XSize	400	mm
	YSize	400	mm
	ZSize	400	mm

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

Command Attribute	
	Name Value
	Name Region
	Material vacuum
	Solve Inside ☒
	Orientation Global
	Model ☒
	Display Wireframe ☒
	Color Edit
	Transparent 0

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create Ring01_1

- ▲ **Draw > RegularPolyhedron** then using the coordinate entry field, enter the center position
 - ▲ X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
- ▲ Then enter the height and the radio
 - ▲ dX: **40** for radius, Press the **Enter** key
 - ▲ dZ: **8** for height, Press the **Enter** key
 - ▲ Number of segments = **36**, **OK**
- ▲ Change the name and color from **RegularPolyhedron1** to **Ring01_1** and **blue**

Command	Attribute

Command	Attribute
Name	Value
Name	Ring01_1
Material	vacuum
Solve Inside	☒
Orientation	Global
Model	☒
Display Wireframe	☐
Color	Edit
Transparent	0

- ▲ **Draw > RegularPolyhedron** then using the coordinate entry field, enter the center position
 - ▲ X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
- ▲ Then enter the height and the radio
 - ▲ dX: **30** for radius, Press the **Enter** key
 - ▲ dZ: **8** for height, Press the **Enter** key
 - ▲ Number of segments = **36**, **OK**
- ▲ Change the name from **RegularPolyhedron1** to **Hole**

Command	Attribute

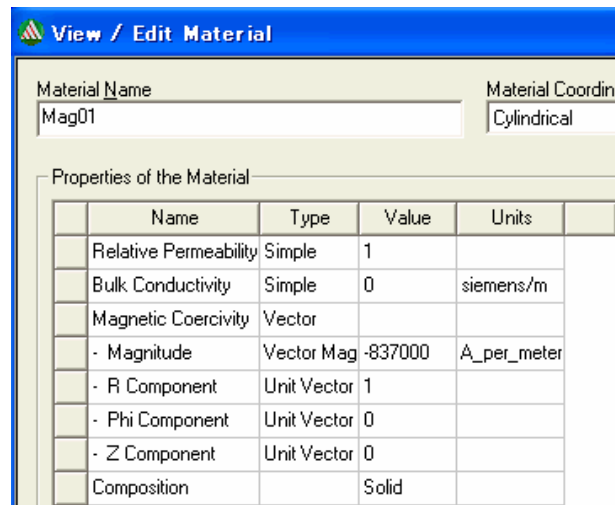
Command	Attribute
Name	Value
Name	Hole
Material	vacuum
Solve Inside	☒
Orientation	Global
Model	☒
Display Wireframe	☐
Color	Edit
Transparent	0

- ▲ Select **Ring01_1** and **Hole** in History Tree, Use right mouse **Edit > Boolean > Subtract** to create new **Ring01_1**

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create new material .Mag01

- ▲ Click **Ring01_1**, then pick Material.**Vacuum** in it's Attribute to get in the Material **Select Definition** window.
- ▲ Select **Add Material**
- ▲ Change properties as follows.
- ▲ Material Name.**Mag01**.
Material Coordinate System.**Cylindrical**
Magnitude.**-837000**.
R Component.**1**
- ▲ Click **OK** to exit. **Mag01** will be appeared in Material Name Listing
- ▲ Click **OK** to exit **Select Definition**. Make sure **Mag01** will be the Material of **Ring01_1**



Note. Please note that Orientation of **Ring01_1** is Global. It will be radially magnetized from (0,0,0) by this setting.

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create Ring01_2

Create new coordinate system for Ring01_2

- ▲ **3D Modeler > Coordinate System > Create > Relative CS > Offset**
then using the coordinate entry field, enter the origin position
 - ▲ X: **120**, Y: **-50**, Z: **0**, Press the **Enter** key
- ▲ Select **Coordinate Systems > RelativeCS1** in History Tree, change the name from **RelativeCS1** to **RelativeCS_Mag01_2** in Property window.
- ▲ **3D Modeler > Coordinate System > Set Working** then select **Global**

Command			
	Name	Value	Unit
	Command	DuplicateAlongLi...	
	Coordinate System	Global	
	Vector	120 ,0 ,0	mm
	Total Number	2	

- ▲ Select **Ring01_1** in History Tree, Use right mouse **Edit > Duplicate > Along Line** to create **Ring01_2**
- ▲ Then enter the start point, end point, the number of duplicated objects
 - ▲ X: **0**, Y: **0**, Z: **0** for start point, Press the **Enter** key
 - ▲ dX:**120**, dY: **0**, dZ: **0** for end point, Press the **Enter** key
 - ▲ Total Number.**2** for the number of duplicated objects, Press the **Enter** key
- ▲ Select **Ring01_1_1** in History Tree, change the name from **Ring01_1_1** to **Ring01_2** and Orientation from **Global** to **RelativeCS_Mag01_2**.

Attribute	
Name	Value
Name	Ring01_2
Material	Mag01
Solve Inside	☒
Orientation	RelativeCS_Mag01_2
Model	☒
Display Wireframe	☐
Color	Edit
Transparent	0

- ▲ **Note.** Please note that Orientation of **Ring01_2** is **RelativeCS_Mag01_2** . It is radially magnetized from (0,0,0) of **RelativeCS_Mag01_2** by this setting.

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create Ring02_1 and Ring02_2

Create new coordinate system for Ring02_2

- ▲ **3D Modeler > Coordinate System > Create > Relative CS > Both** then using the coordinate entry field, enter the origin position, X axis, and the point of XY plane.
 - ▲ X: **120**, Y: **120**, Z: **0** for origin, Press the **Enter** key
 - ▲ X: **1**, Y: **0**, Z: **0** for X axis, Press the **Enter** key
 - ▲ X: **0**, Y: **0**, Z: **-1** for the point of XY plane, Press the **Enter** key
- ▲ Select **RelativeCS1** in History Tree, change the name from **RelativeCS1** to **RelativeCS_Mag02_2** in Property window.
- ▲ **3D Modeler > Coordinate System > Set Working** then select **Global**
- ▲ Select **Ring01_1** and **Ring01_2** in History Tree, Use right mouse **Edit > Duplicate > Along Line** to create **Ring02_1** and **Ring02_2**
- ▲ Then enter the start point, end point, the number of duplicated objects
 - ▲ X: **0**, Y: **0**, Z: **0** for start point, Press the **Enter** key
 - ▲ dX:**0**, dY: **120**, dZ: **0** for end point, Press the **Enter** key
 - ▲ Total Number.**2** for the number of duplicated objects, Press the **Enter** key
- ▲ Select **Ring01_1_1** in History Tree, change the name from **Ring01_1_1** to **Ring02_1**.
- ▲ Select **Ring01_1_2** in History Tree, change the name from **Ring01_1_2** to **Ring02_2** and Orientation from **Global** to **RelativeCS_Mag02_2**.

Attribute	
Name	Value
Name	Ring02_1
Material	Mag01
Solve Inside	☒
Orientation	Global
Model	☒
Display Wireframe	☐
Color	Edit
Transparent	0

Attribute	
Name	Value
Name	Ring02_2
Material	Mag01
Solve Inside	☒
Orientation	RelativeCS_Mag02_2
Model	☒
Display Wireframe	☐
Color	Edit
Transparent	0

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create new material .Mag02

- ▲ Click **Ring02_1** and **Ring02_2**, then pick Material. **Mag01** in it's Attribute to get in the Material **Select Definition** window.
- ▲ Select **Clone Material**
- ▲ Change properties as follow.
- ▲ Material Name.**Mag02**.
Material Coordinate System.**Cylindrical**
Magnitude.**-837000**.
R Component.**0**
Z Component.**1**
- ▲ Click OK to exit. **Mag02** will be appeared in Material Name Listing
- ▲ Click OK to exit **Select Definition**. Make sure **Mag01** will be the Material of **Ring02_1** and **Ring02_2**

Material Name		Material Coordinate Sy		
Mag02		Cylindrical		

Properties of the Material				
	Name	Type	Value	Units
	Relative Permeability	Simple	1	
	Bulk Conductivity	Simple	0	siemens/m
	Magnetic Coercivity	Vector		
	- Magnitude	Vector Mag	-837000	A_per_meter
	- R Component	Unit Vector	0	
	- Phi Component	Unit Vector	0	
	- Z Component	Unit Vector	1	
	Composition		Solid	

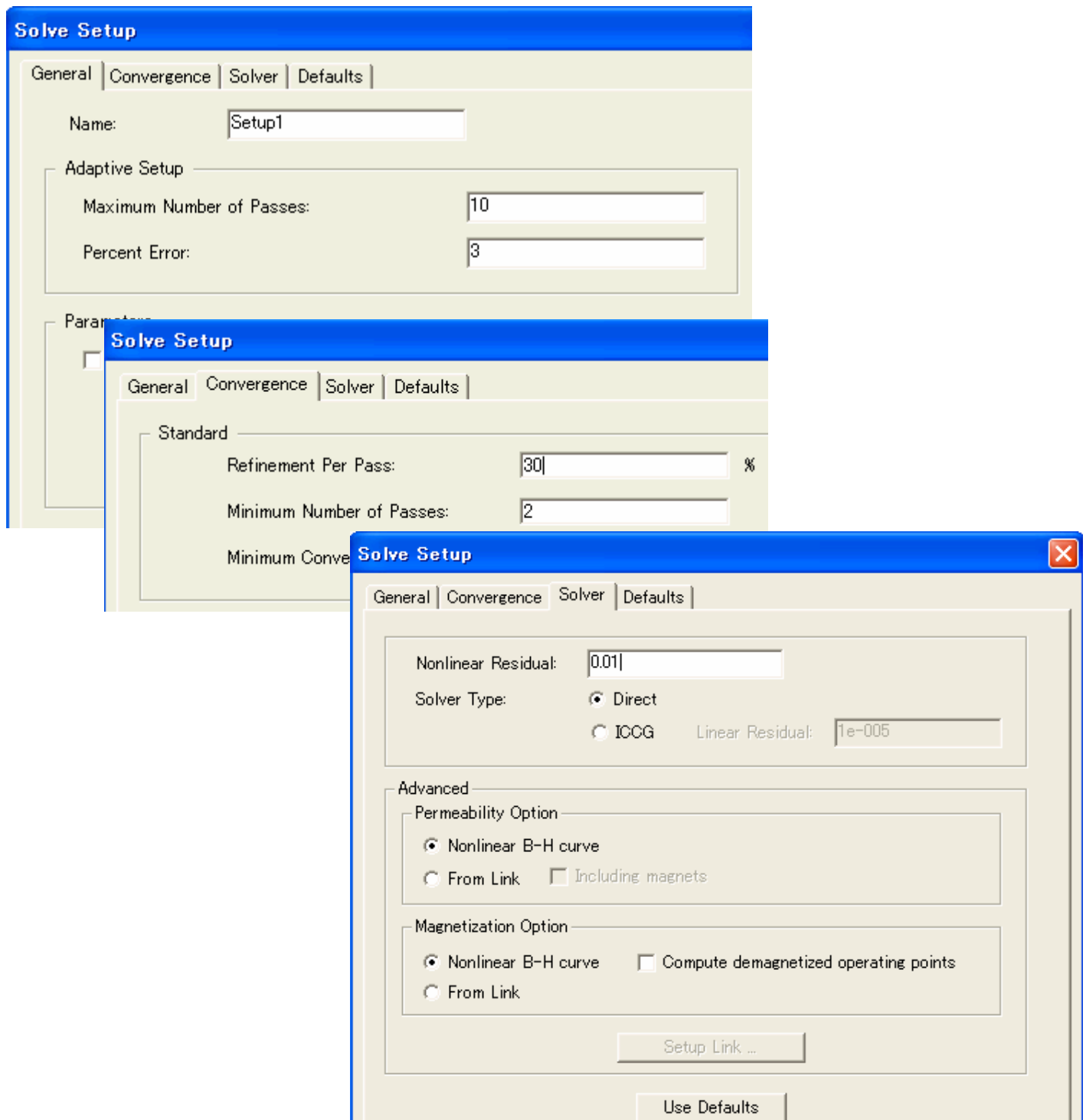
Note. Please note that Orientation of **Ring02_1** is Global. It is magnetized in parallel to Z axis of Global.

Note. Please note that Orientation of **Ring02_2** is **RelativeCS_Mag02_2**. It is magnetized in parallel to Z axis of **RelativeCS_Mag02_2**.

Example (Magnetostatic) - Permanent Magnet Magnetization

Create an Analysis Setup

- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**
- ▲ Select **General** and change the Percent Error from **1** to **3**



Solve Setup

General | Convergence | Solver | Defaults

Name: Setup1

Adaptive Setup

Maximum Number of Passes: 10

Percent Error: 3

Parameters

Solve Setup

General | Convergence | Solver | Defaults

Standard

Refinement Per Pass: 30 %

Minimum Number of Passes: 2

Minimum Convergence: 0.01

Solve Setup

General | Convergence | Solver | Defaults

Nonlinear Residual: 0.01

Solver Type: ☒ Direct ☐ ICCG Linear Residual: 1e-005

Advanced

Permeability Option

☒ Nonlinear B-H curve ☐ From Link ☐ Including magnets

Magnetization Option

☒ Nonlinear B-H curve ☐ From Link ☐ Compute demagnetized operating points

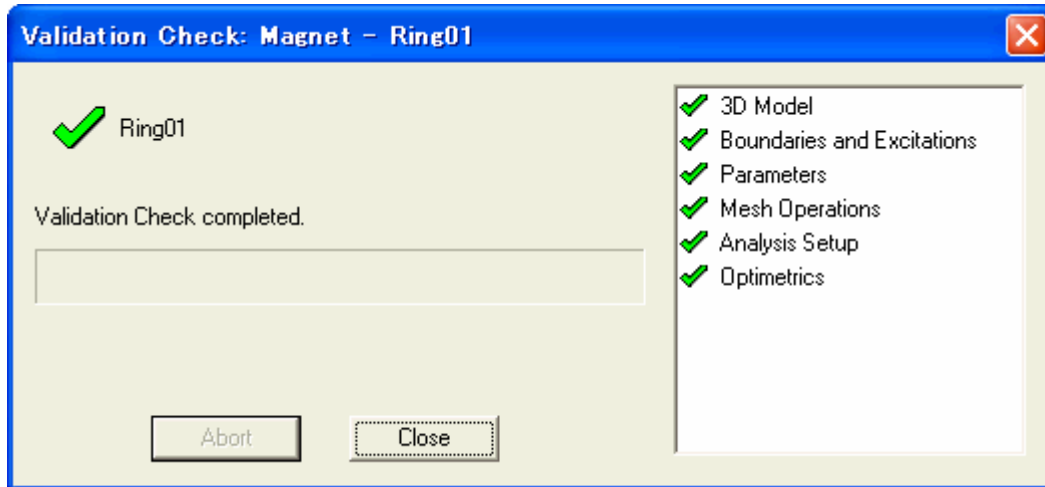
Setup Link ...

Use Defaults

Example (Magnetostatic) - Permanent Magnet Magnetization

Check the Validity of the Model

- Select the menu item **Maxwell > Validation Check**.

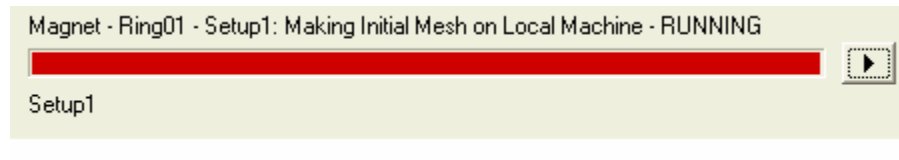


- The problem won't solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**.

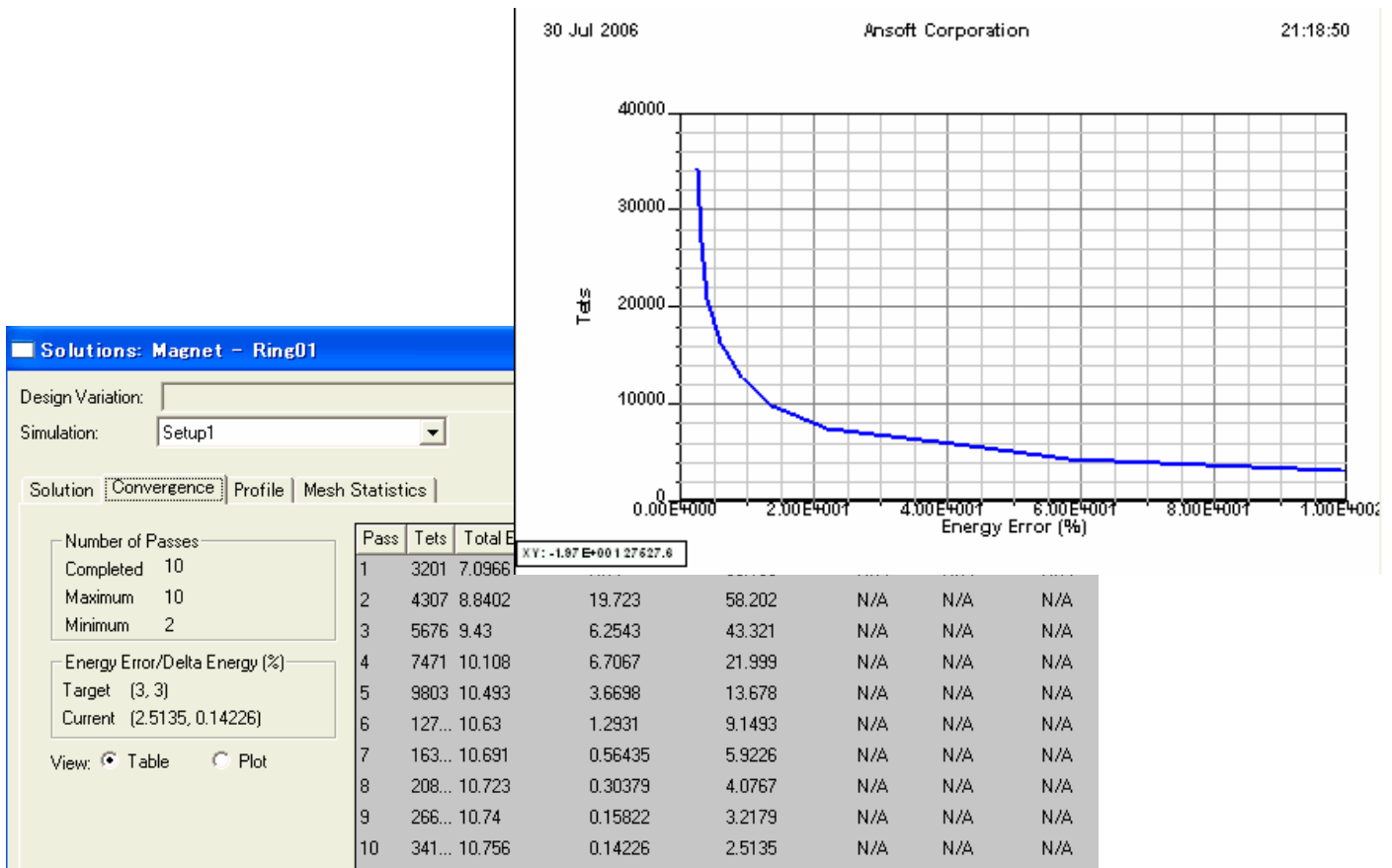
The Analysis will be start as below:



Example (Magnetostatic) - Permanent Magnet Magnetization

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.



- Mesh information can be confirmed by clicking the Mesh Statistics tab.

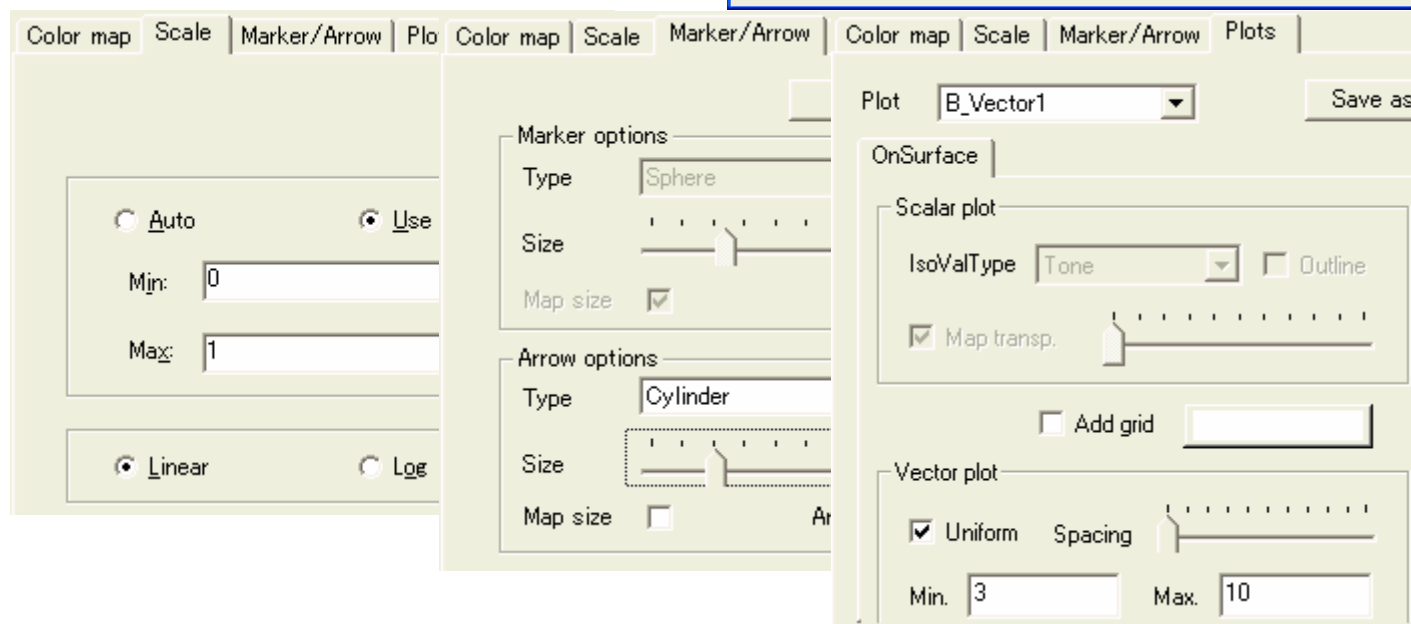
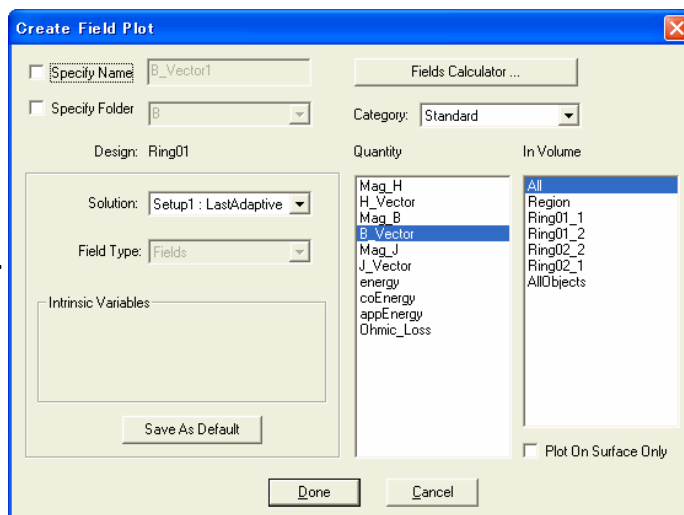
The screenshot shows the 'Solutions: Magnet - Ring01' window with the 'Mesh Statistics' tab selected. It displays a table with mesh information for various regions.

	Num Elements	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Dev
Region	28921	2.85618	423.748	26.8208	0.539723	1.64543e+00...	2210.5	26233.1
Ring01_1	1757	3.23418	11.6817	6.78386	2.05146	36.847	9.96229	5.42713
Ring01_2	1161	4.05979	11.6817	7.68963	1.89185	66.1475	15.0764	10.6472
Ring02_2	1131	3.18415	14.1584	7.7468	1.70093	95.1576	15.4763	12.5937
Ring02_1	1194	3.62363	10.612	7.06239	1.58177	48.8964	14.6597	7.93996

Example (Magnetostatic) - Permanent Magnet Magnetization

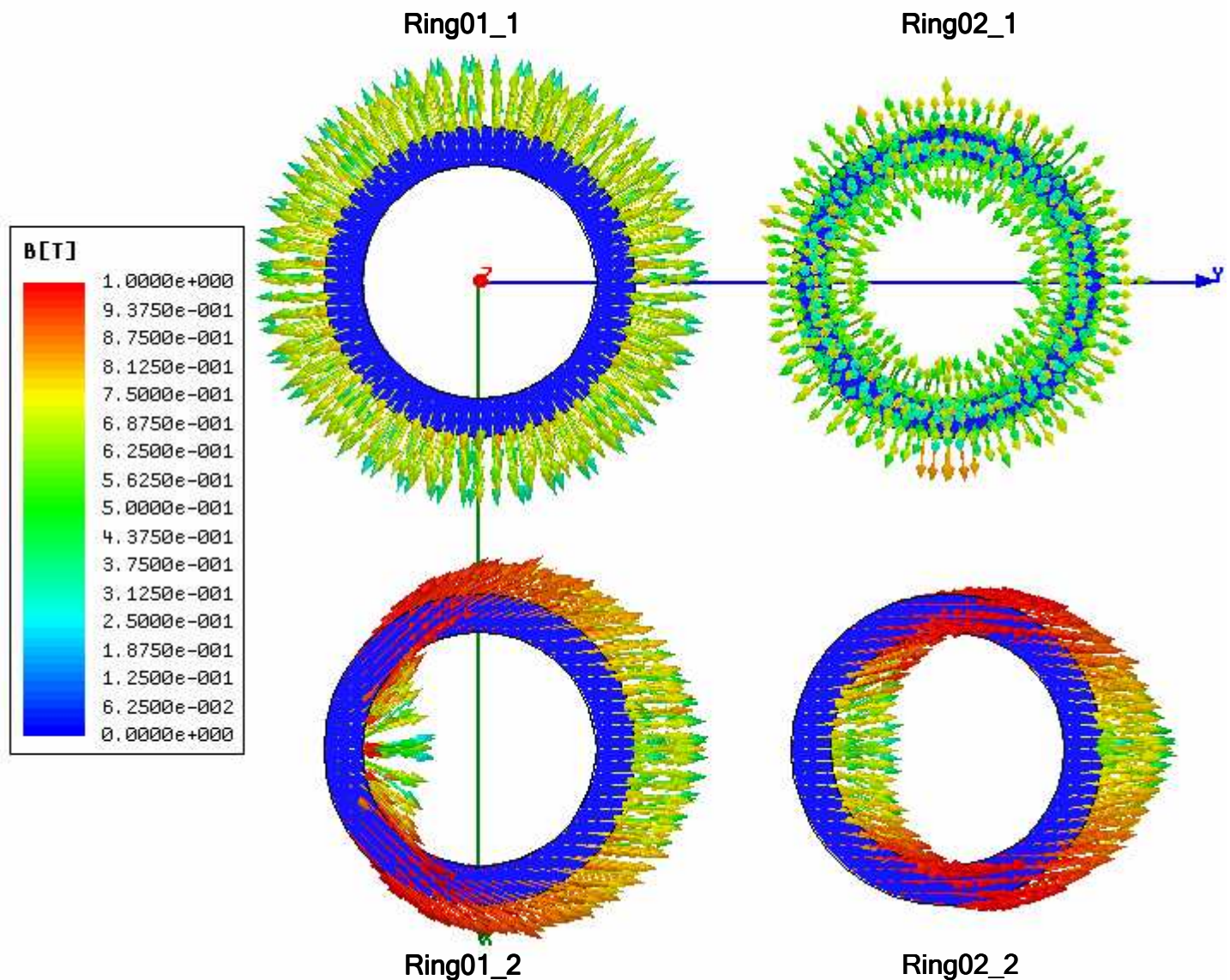
Plot Flux Density

- To plot the flux density on **Ring01_1.Ring01_2.Ring02_1.Ring02_2** of the magnetic, select the **Ring01_1.Ring01_2.Ring02_1.Ring02_2** 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 **All** in “In Volume”, and **Check Plot Surface Only**, Press **Done**.
- The field plot is a little misleading since it show different colors, 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**, the **Max** to **1**, click on **Apply**
- Then select **Marker/Arrow**, in Arrow options Change the Type as **Cylinder**, move Size Arrow to be third from the left, remove the **check** of Map Size and click on **Apply**.
- Then select **Plots**, in Vector plot Select **Uniform** and move the **Spacing** size to smallest, set **Min.** as **3** and **Max.** as **10**, click on **Apply**.



Example (Magnetostatic) - Permanent Magnet Magnetization

- Each object has been magnetized in a different direction. Thus, when the coordinate system of the object is different, it is possible to magnetize it in a different direction even by the same material characteristic.



Example (Magnetostatic) - Permanent Magnet Magnetization

Example2.Magnetization using the equation

In this example, it is introduced the magnetization using the equation.
Material : Mag03 is newly made in this example.

Design copy

Select **Ring01** design in project window, use right mouse **Copy**, select **Magnet** project in project window, use right mouse **Paste** to create **Ring2** design.

Correction of object

- ▲ Select **Ring01_2.Ring02_1.Ring02_2** ,Press the **Del** key.
- ▲ Open the history of **Ring01_1** in history tree, double click **CreateRegularPolyhedron**
- ▲ Change to
 - ▲ Center Position..**0,0,-20**..
 - Start Position..**40,0,-20**..
 - Height..**40**

Command		
	Name	Value
	Command	CreateRegularPo...
	Coordinate System	Global
	Center Position	0 ,0 ,-20
	Start Position	40 ,0 ,-20
	Axis	Z
	Height	40
	Number of Segments	36

- ▲ In addition, double click **Subtract > Hole > CreateRegularPolyhedron** in the history.
- ▲ Change to
 - ▲ Center Position..**0,0,-20**..
 - Start Position..**30,0,-20**..
 - Height..**40**

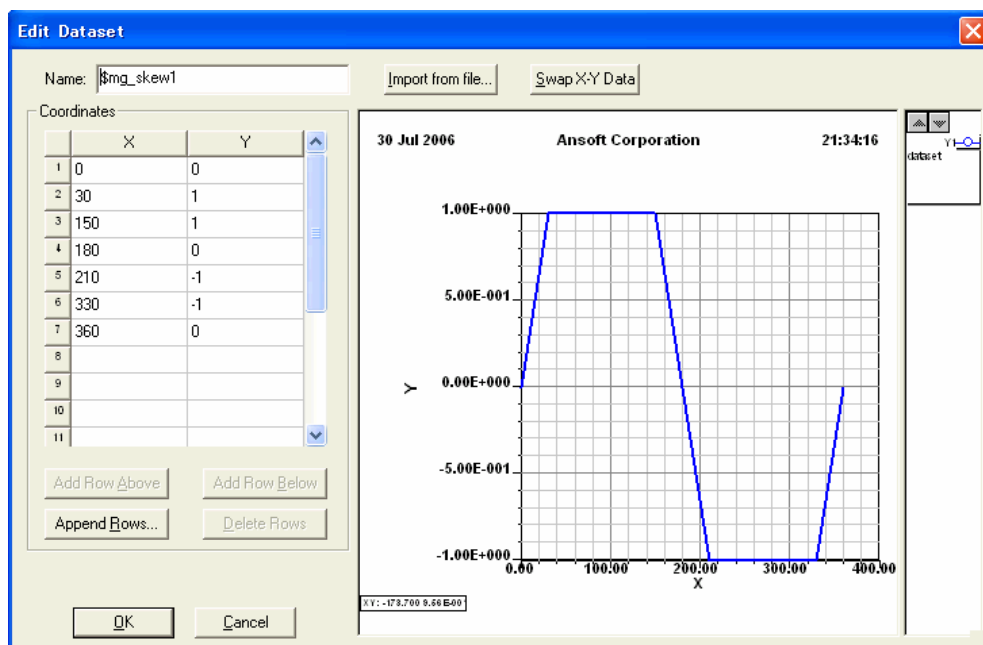
Command		
	Name	Value
	Command	CreateRegularPo...
	Coordinate System	Global
	Center Position	0 ,0 ,-20
	Start Position	30 ,0 ,-20
	Axis	Z
	Height	40
	Number of Segments	36

- ▲ Double click **Ring01_1** in history tree, change the name from **Ring01_1** to **Ring03**
- ▲ Click Material.**Mag01** and make new Material:**Mag03** as follows.

Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create new material .Mag03

- ▲ Select **Clone Material**
- ▲ Change properties as follow.
- ▲ Material Name. **Mag03**.
Magnitude. $-837000 * \text{pwl\_periodic}(\$mg\_skew1, 4 * (\Phi * 180 / \text{PI} - 45 * (Z / 40 \text{ mm})))$.
- ▲ Because the **Add Dataset window** is displayed, data is input as follows.
Click **OK** to exit.



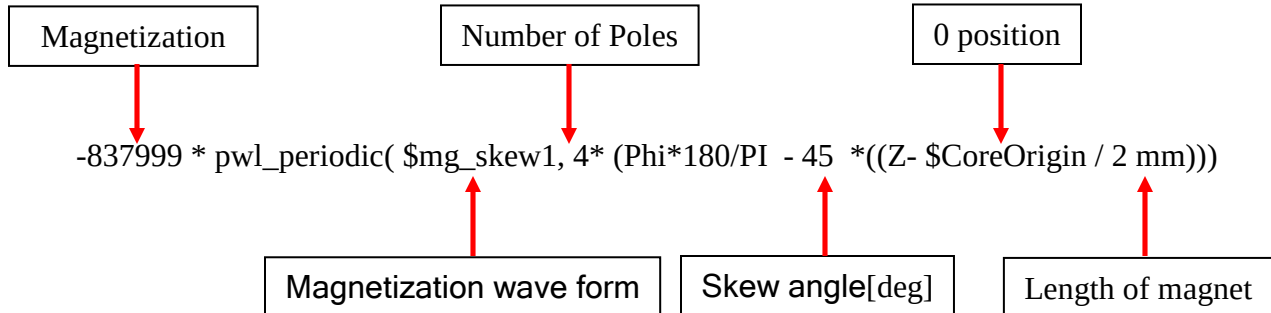
- ▲ Click **OK** to exit. **Mag03** will be appeared in Material Name Listing
- ▲ Click **OK** to exit **Select Definition**. Make sure **Mag03** will be the Material of **Ring03**

Attribute		
	Name	Value
	Name	Ring03
	Material	Mag03
	Solve Inside	☒
	Orientation	Global
	Model	☒
	Display Wireframe	☐
	Color	Edit
	Transparent	0

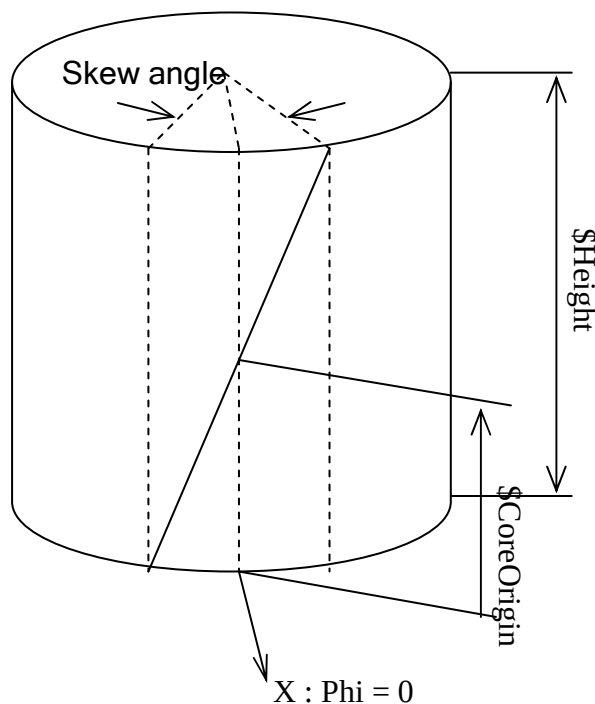
Example (Magnetostatic) - Permanent Magnet Magnetization

Note About the skew magnetization

- The function used in this case defines the magnetization of the direction of Phi with \$mg_skew1. Moreover, to apply the skew, magnetic data is rotated according to Z axis coordinates.



The magnetization wave form is defined at one cycle of 0 - 360deg. $\$CoreOrigin$ is Z axis value that becomes 0deg of the magnetization wave form by the $\Phi=0$ degree. The unit is meter.



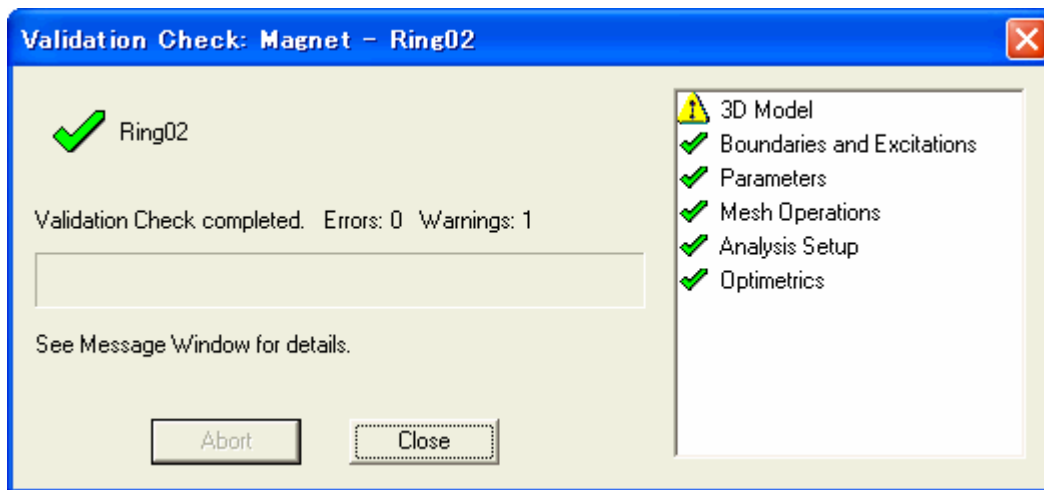
Example (Magnetostatic) - Permanent Magnet Magnetization

▲ Create an Analysis Setup

- ▲ There is already **Setup1** because it copied from the **Ring01** design. Analyze this design by the untouched setting.

▲ Check the Validity of the Model

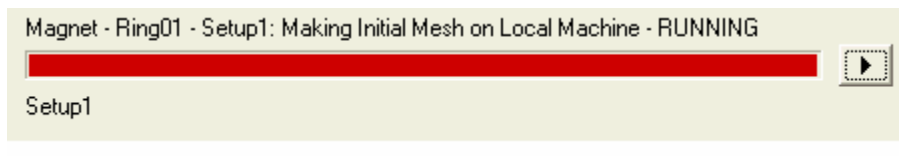
- ▲ Select the menu item **Maxwell > Validation Check**.



- ▲ To change the sign of the magnetization, attention is displayed. Ignore it because it intentionally sets it this time.
- ▲ The problem won't solve unless each object has a check mark.

▲ Analyze

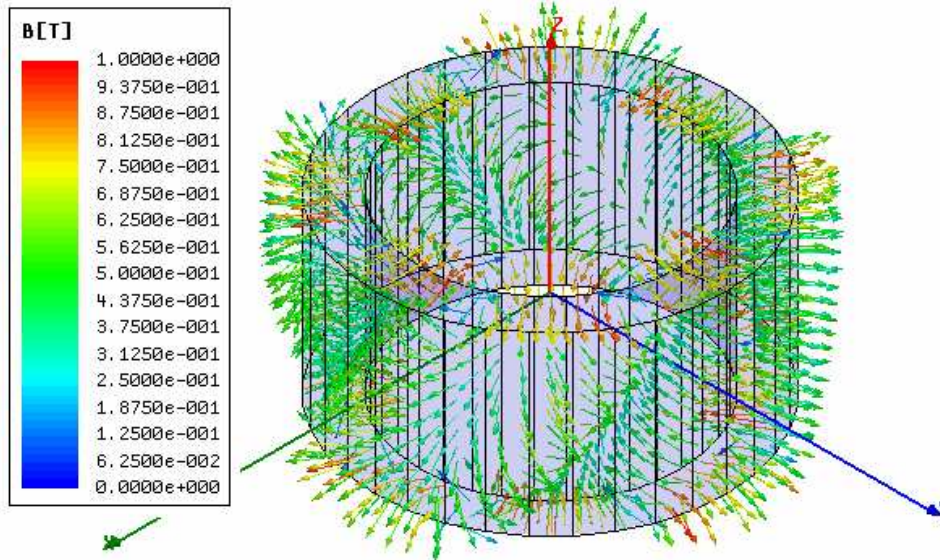
- ▲ Select the menu item **Maxwell > Analyze**.
The Analysis will be start as below:



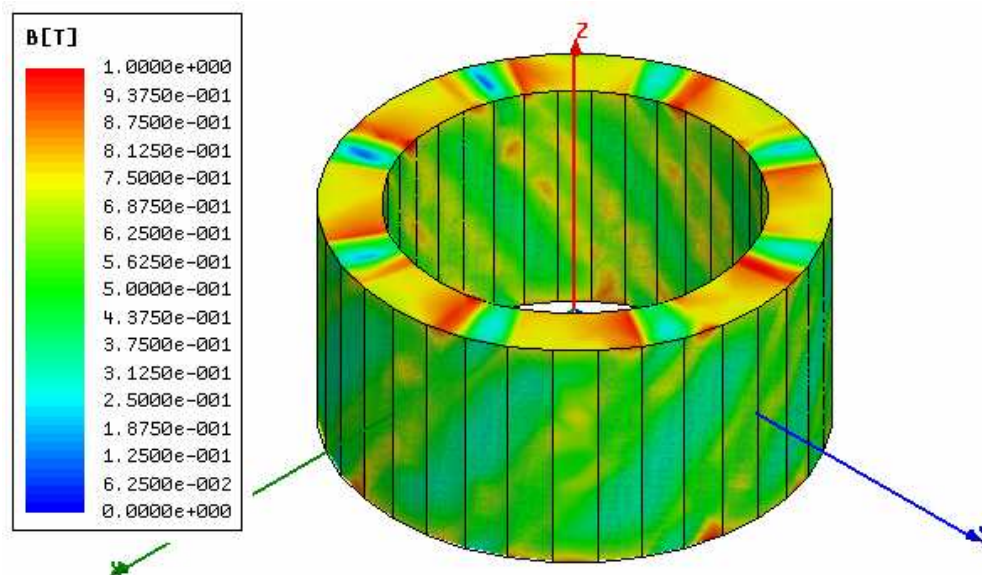
Example (Magnetostatic) - Permanent Magnet Magnetization

Plot Flux Density

- Double click **Field Overlays > B > B_Vector** in Project window.
- Because the set up information made by **Ring01** design remains, it is possible to display easily.



- Select the **Ring03** in History Tree by clicking on it, and move the mouse to 3D Modeler Window then right mouse click and select **Fields > B > Mag_B**, in Create Field Plot Window select **Mag_B** in “Quantity” and **All** in “In Volume”, and **Check Plot Surface Only**, Press **Done**.



- The magnetic induction changes into the direction of Φ , and the skew magnetization.

Assigning Master/Slave Boundary Conditions

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.

Master/Slave

-  Master and slave boundaries enable you to model planes of periodicity where the H-field (E-field for Electrostatic Solver) at every point on the slave boundary surface is forced to match the H-field of every corresponding point on the master boundary surface. The transformation used to map the H-field from the master to the slave is determined by specifying a coordinate system on both the master and slave boundaries.

Considerations

-  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.

Why They are Important

-  When used properly, Master/Slave boundary conditions can be successfully utilized to reduce the model complexity.
-  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.

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): **SRMCore**
-  Surface Operations: **Section**
-  Boolean Operations: **Separate Bodies**

Boundaries/Excitations

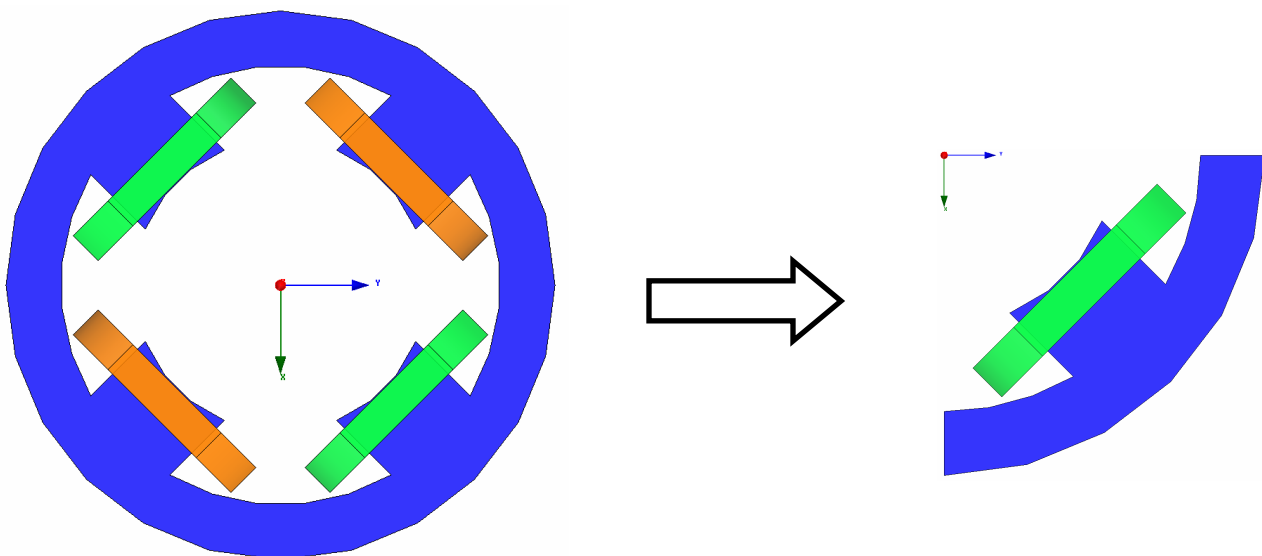
-  Current: **Stranded**
-  Boundaries: **Master/Slave**

Analysis

-  Magnetostatic

Field Overlays:

-  **H Vector**



Getting Started

Launching Maxwell

To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select

Ansoft and then **Maxwell 11**. Or double click the  icon on the desktop.

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**

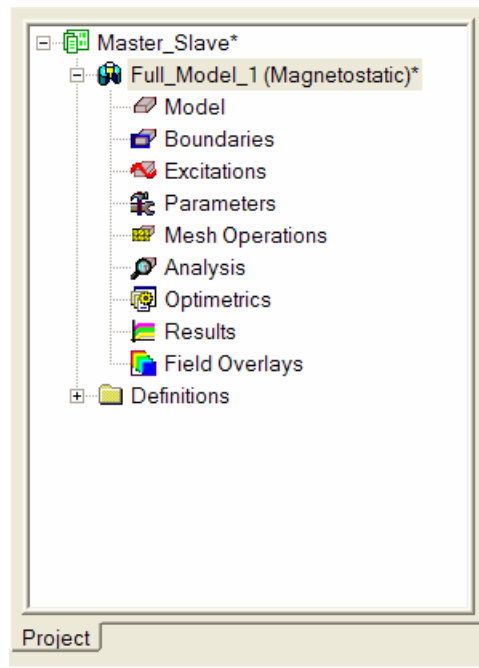
3. Click the **OK** button

Topic - Master/Slave Boundaries

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.
3. Save the project with name "Master_Slave" to your own folder. Change the name of the design from "MaxwellDesign1" to "Full_Model_1".



Set Solution Type

1. Select the menu item: **Maxwell > Solution Type > Magnetostatic**, or right mouse click on **Full_Model_1** and select **Solution Type**.

Creating the 3D Model

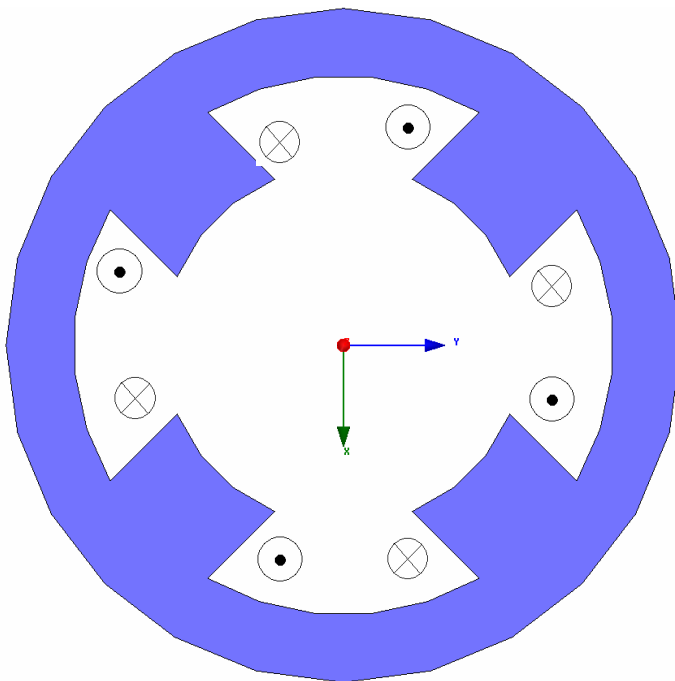
1. The example that will be used to demonstrate how to assign Master/Slave boundary conditions does not represent any real-world product. The intent of this write up is rather to demonstrate how Master/Slave boundary conditions are implemented.

Set Model Units

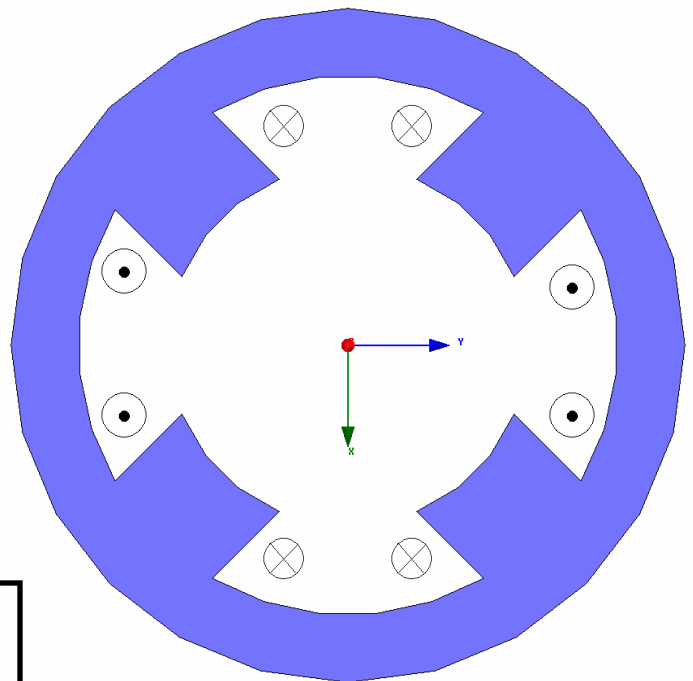
1. Select the menu item **3D Modeler > Units > Select Units: mm (millimeters)**

Topic - Master/Slave Boundaries

- There will be four designs in this example, as shown below. The objective is to demonstrate how to assign Master/Slave boundary conditions and how they can help reduce model complexity and hence solution time.



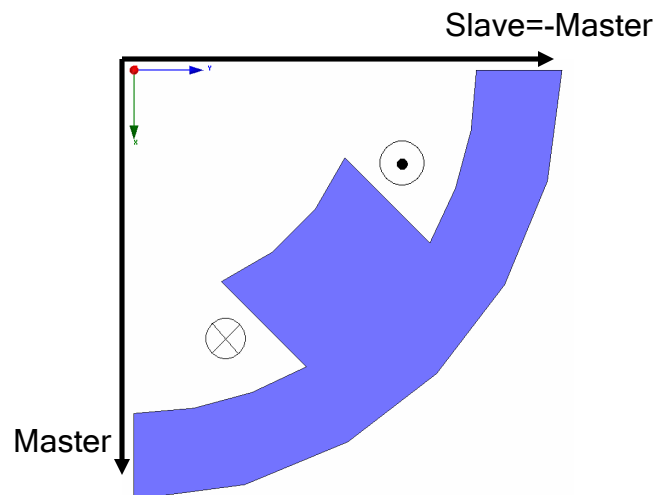
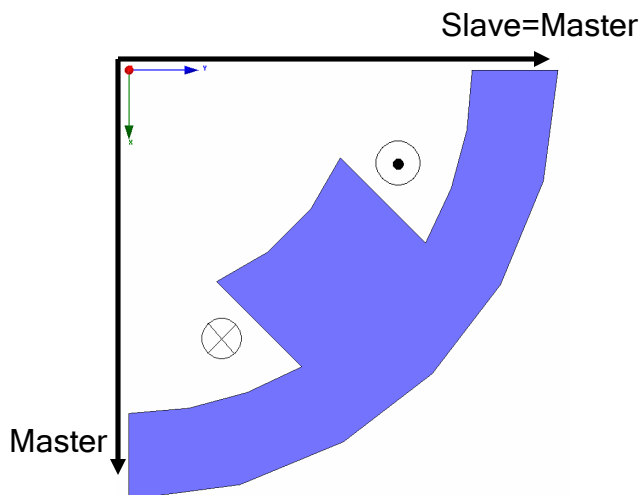
Design1: Full_Model_1



Design2: Full_Model_2

Design3: Quarter_Model_1, Slave=Master

Design4: Quarter_Model_2, Slave=-Master



Topic - Master/Slave Boundaries

Create Stator and Coils:

- ▲ A User Defined Primitive will be used to create the Stator and Coils
 - ▲ **Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore**
 - ▲ Use the values given in the panel below to create the Stator and Coils

Create User Defined Primitive
✕

Parameters
Info

Name	Value	Units	Description
DiaGap	80	mm	Core diameter on gap side, DiaGap<DiaYoke for outer cores
DiaYoke	150	mm	Core diameter on yoke side, DiaYoke<diaGap for inner cores
Length	10	mm	Core length
Poles	4		Number of poles
ThkYoke	15	mm	Yoke thickness
Embrace	0.5		Pole embrace (the ratio of pole arc to pole pitch)
EndExt	5	mm	Coil one-side end extended length
infoCore	1		0: core only; 1: core & all coils; 2: one coil only.

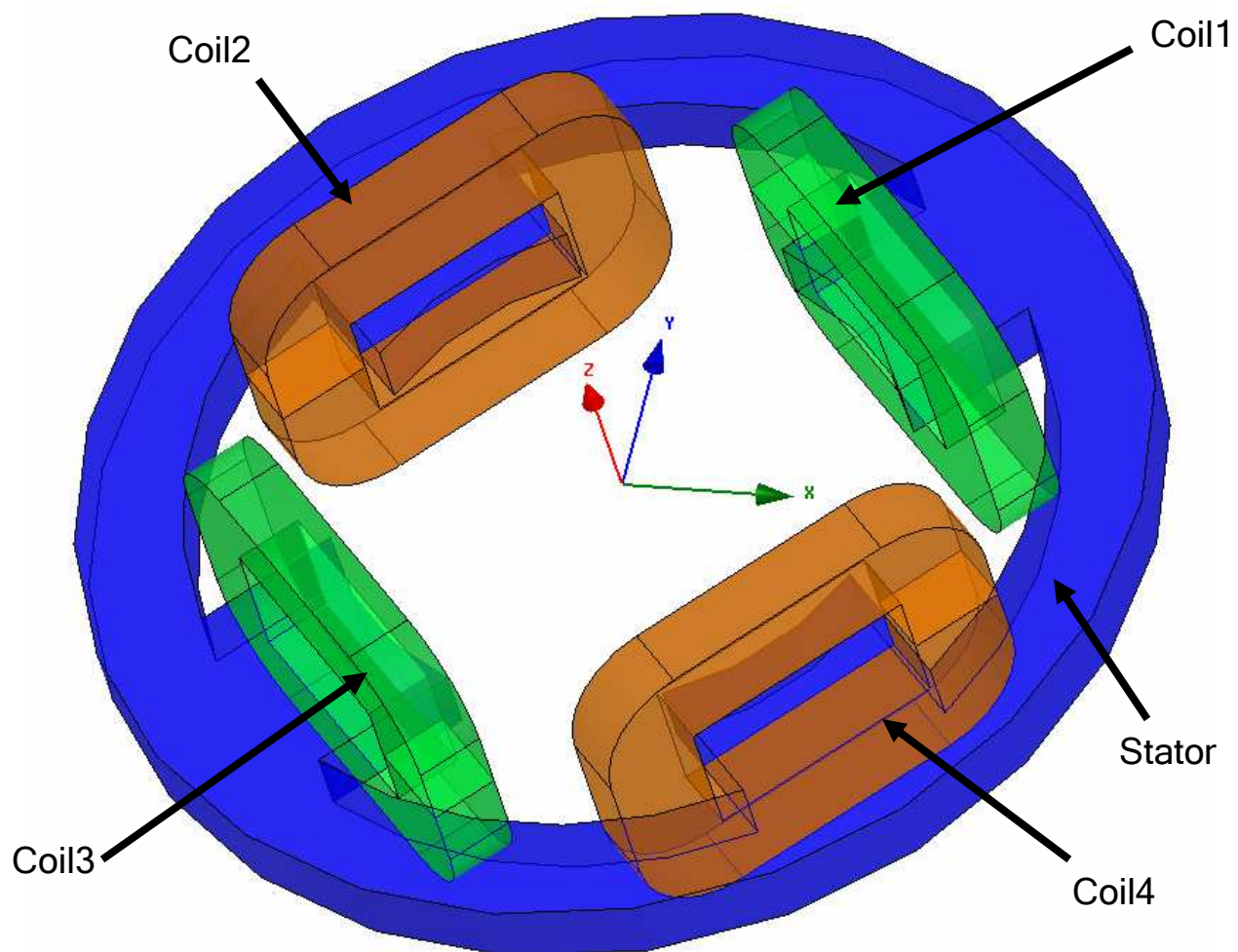
OK
Cancel

- ▲ Click on the just created object in the drawing window and in the panel on the left change its name from **SRMCore1** to **Stator**.
- ▲ Change the Material from **vacuum** to **nickel**.

	Name	Value	Unit
	Name	Stator	
	Material	nickel	
	Solve Inside	☒	
	Orientation	Global	
	Model	☒	
	Display Wireframe	☐	
	Color	Edit	
	Transparent	0.4	

Topic - Master/Slave Boundaries

- ▲ The stator and coils were created as one entity and they need to be separated.
 1. Click on the Stator-Coil group so that they are selected
 2. Select the menu item **3D Modeler > Boolean > Separate Bodies**, the result will be a single stator and four coils.
- ▲ As was done with the Stator, change the name, materials, and color for the Coils. The material property for the Stator will be nickel, and the material property for the Coils will be copper. The name and color for each object is given below.



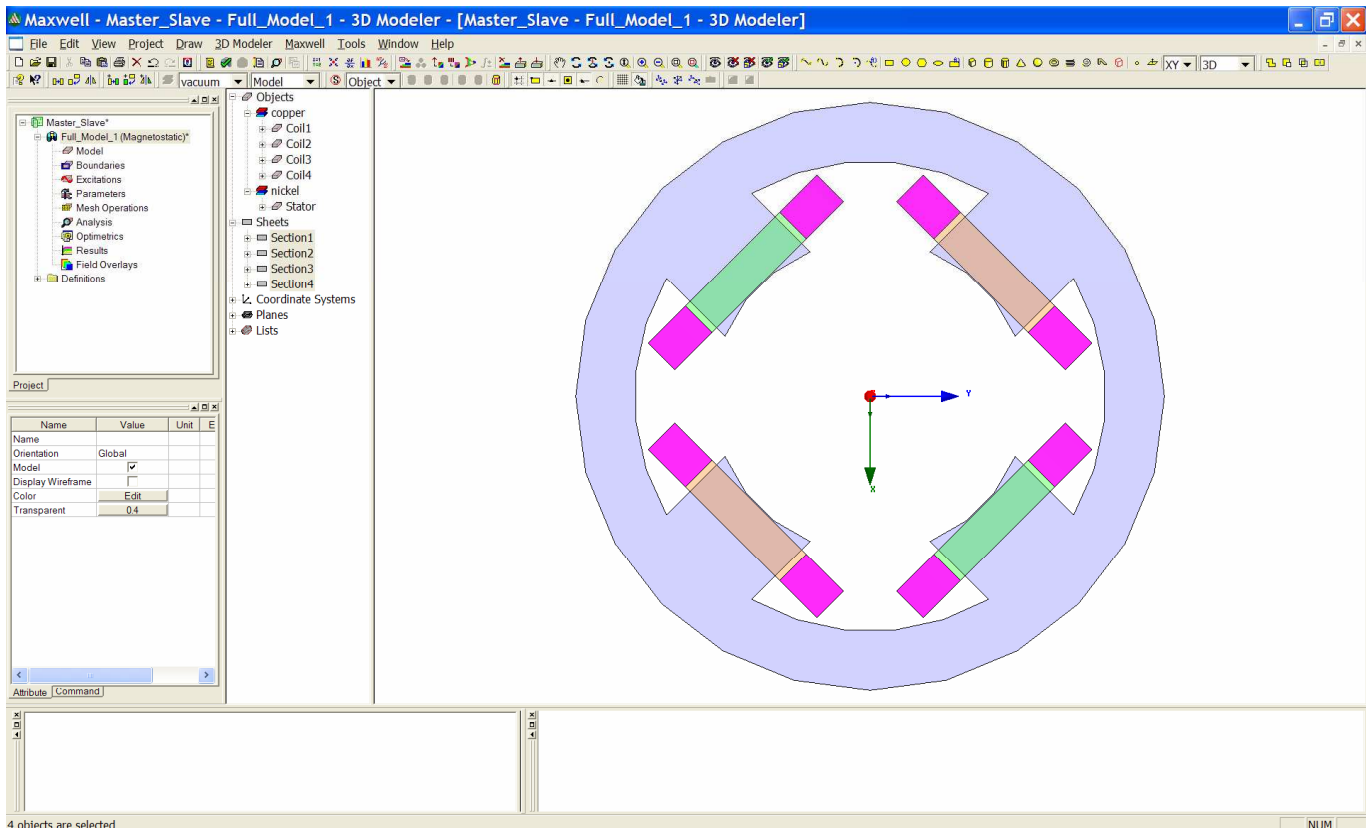
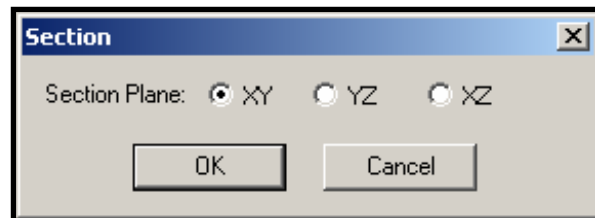
Topic - Master/Slave Boundaries

To fit the view:

- Select the menu item **View > Fit All > Active View**, or press the **CTRL+D** key

Create Coil Terminals

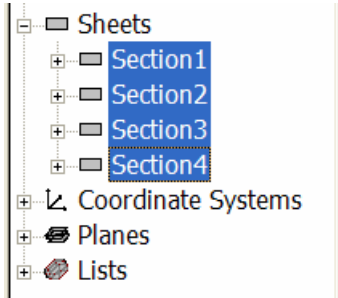
- Coil terminals need to be created in order to assign the current excitation
- Holding the Control key, click on the four coils so that they are selected, or select them from the History tree on the left of the 3D Modeler window
- Select the menu item **3D Modeler > Surface > Section > XY Plane**, or **right click > Surface > Section > XY Plane**.



Separate the Grouped Sections

- The result of the **Surface > Section** command, yields two terminals for each coil that are united. Since only one terminal per coil is needed, half of these terminals need to be deleted, but first they need to be separated.

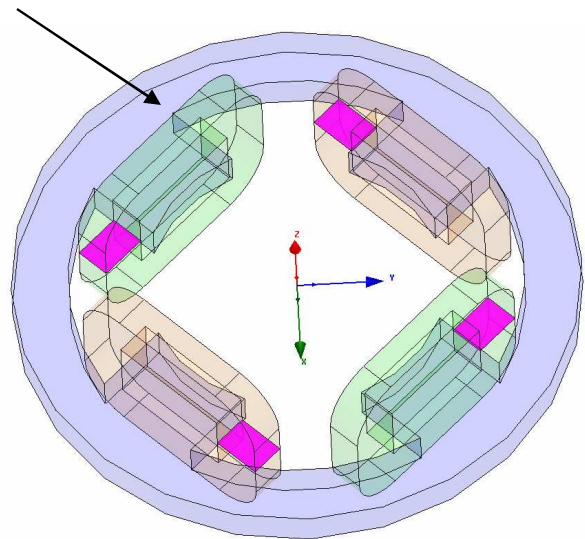
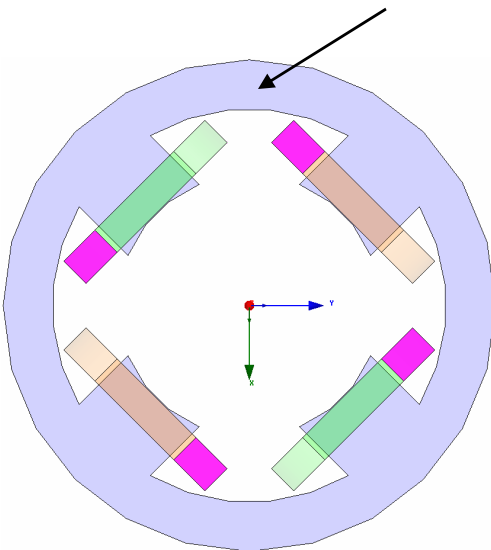
- Select all of the sheet objects:



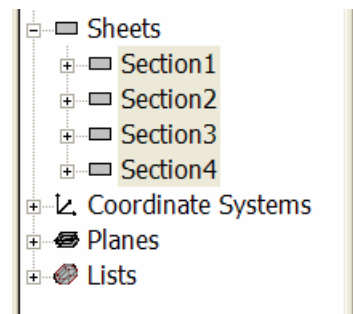
- Select the menu item **3D Modeler > Boolean > Separate Bodies** or **right click > Edit > Boolean > Separate Bodies**

- There are now 8 separate terminals. Delete half of them so that each coil has one terminal.

One Section for each Coil

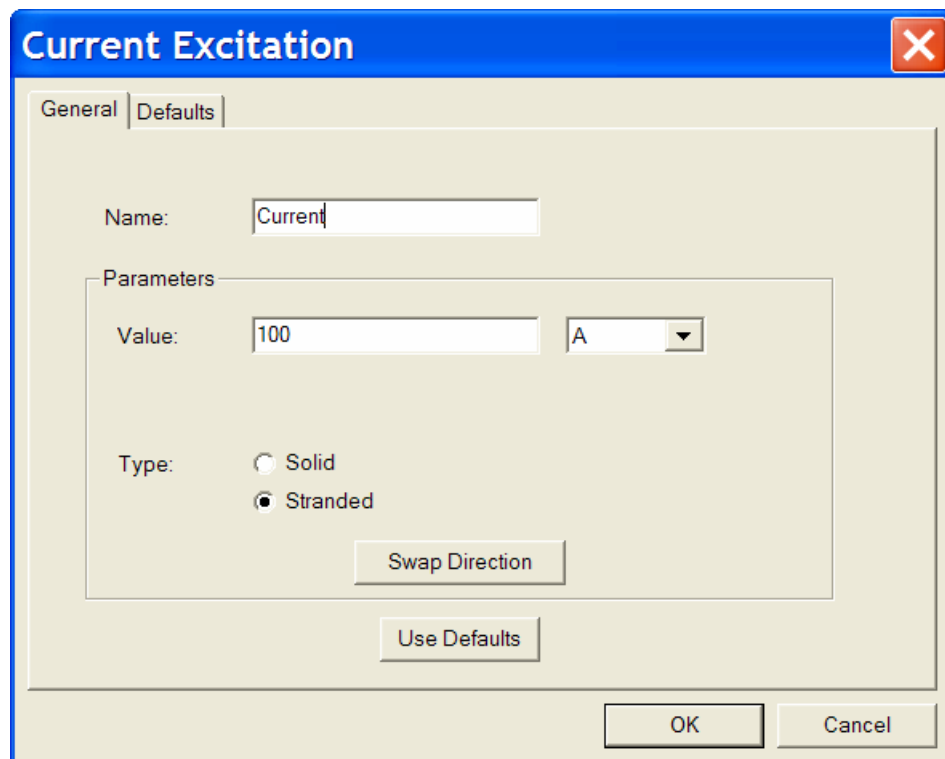


- Make sure the section numbers correspond to coil numbers correctly



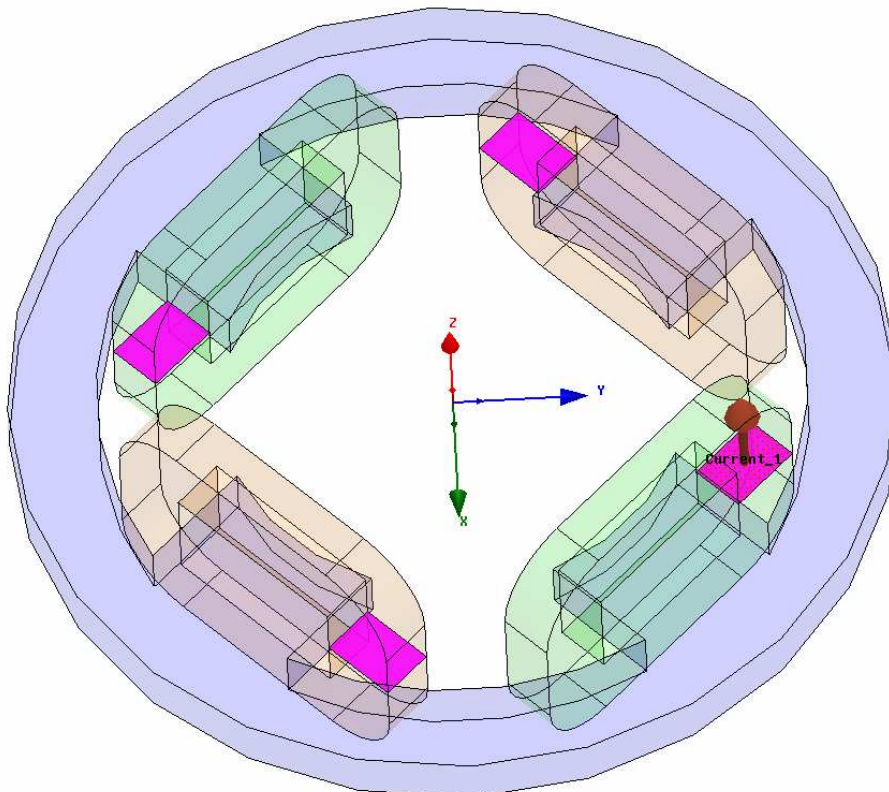
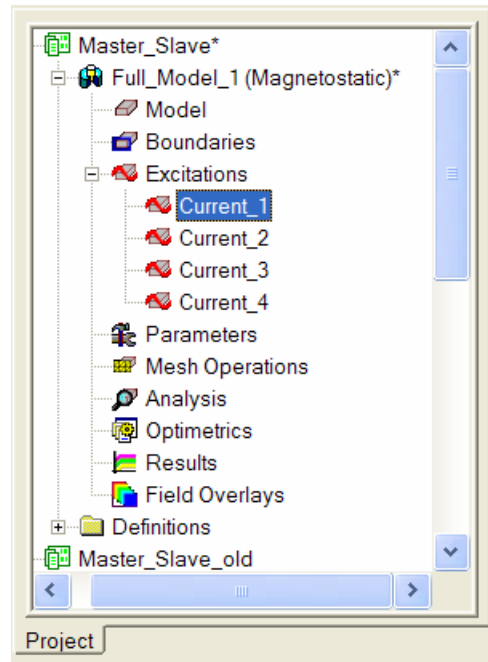
Assign Current Source to the Terminals

-  Select all four coil sections 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 > Current** or *right click > Assign Excitations > Current*
 1. Change the Base Name to **Current**
 2. Change the value to **100 Amps**
 3. Change the type to **Stranded**



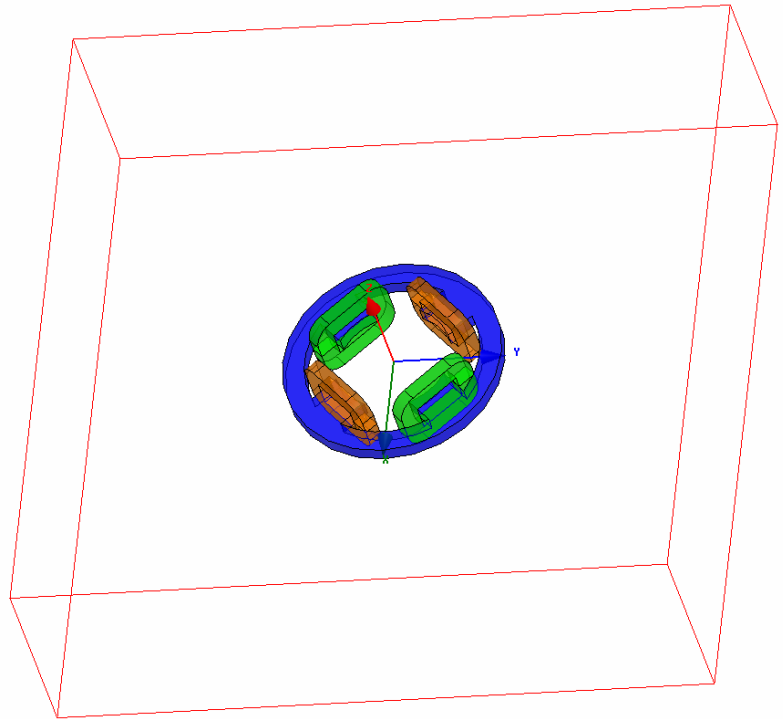
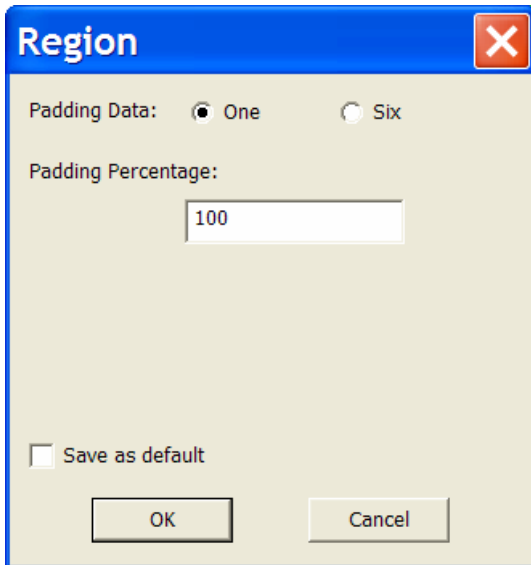
Topic - Master/Slave Boundaries

- ▲ The project tree now shows four separate Excitations:



Create the Problem Region

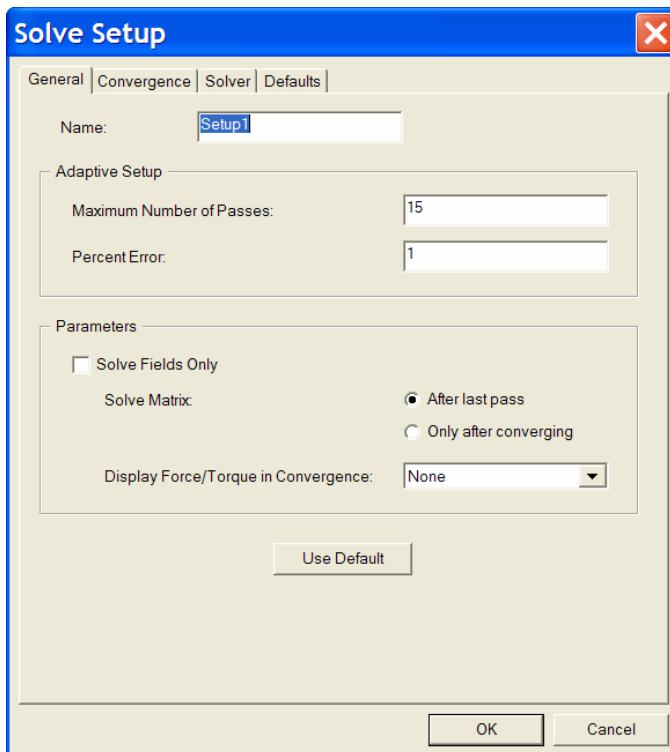
-  One of the main differences between Maxwell V10 and V11 is that a Background Region is not automatically created when a project is started. A separate object needs to be specifically created.
-  To create a rectangular region simply select **Draw > Region**, or click the  icon from standard toolbar. The size of this rectangular region is based on dimensions of the existing objects. Change Padding Percentage to 100.



-  Click **View > Fit All > All Views**.

Create an Analysis Setup

- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup** or **right click on the Analysis in the project window > Add Solution Setup**
- ▲ Select **General** and change the setting as follows
- ▲ Select **Convergence** and change the setting as follows



Solve Setup

General | Convergence | Solver | Defaults

Name:

Adaptive Setup

Maximum Number of Passes:

Percent Error:

Parameters

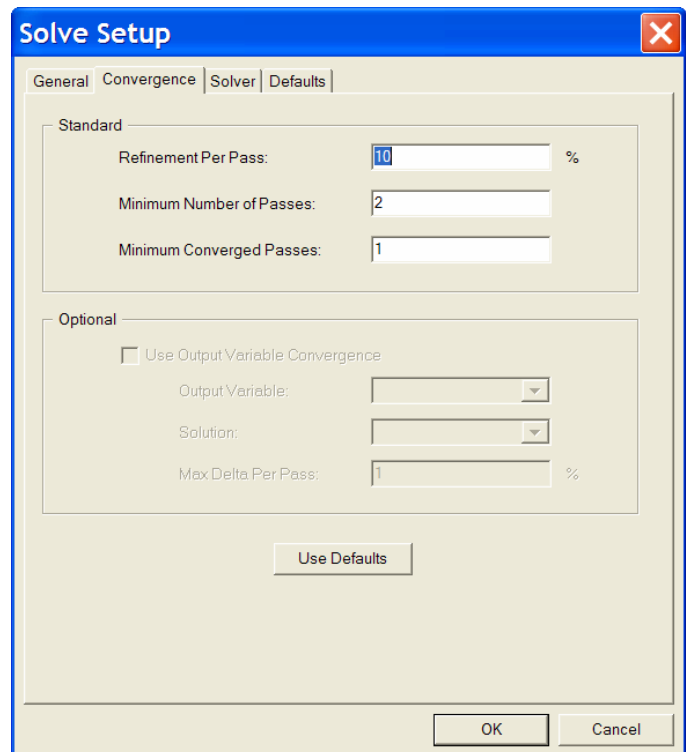
☐ Solve Fields Only

Solve Matrix: ☒ After last pass ☐ Only after converging

Display Force/Torque in Convergence:

Use Default

OK Cancel



Solve Setup

General | Convergence | Solver | Defaults

Standard

Refinement Per Pass: %

Minimum Number of Passes:

Minimum Converged Passes:

Optional

☐ Use Output Variable Convergence

Output Variable:

Solution:

Max Delta Per Pass: %

Use Defaults

OK Cancel

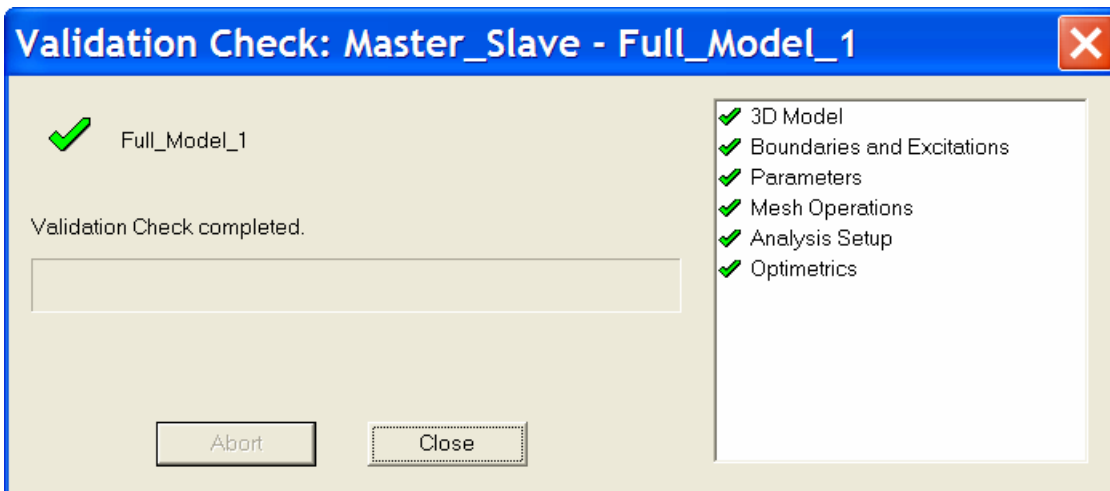
Topic - Master/Slave Boundaries

Save the Project

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

Check the Validity of the Model

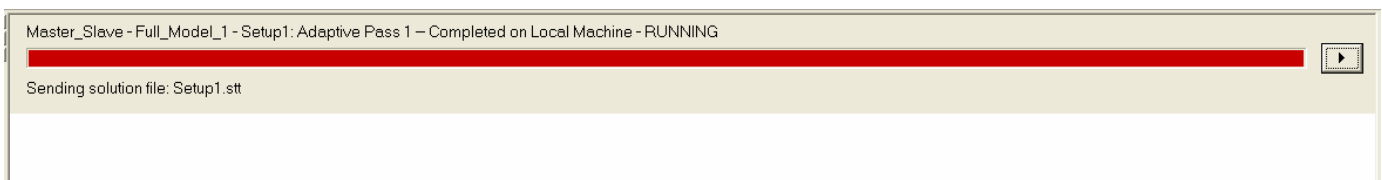
- ▲ Select the menu item **Maxwell > Validation Check**, or click on the  icon



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

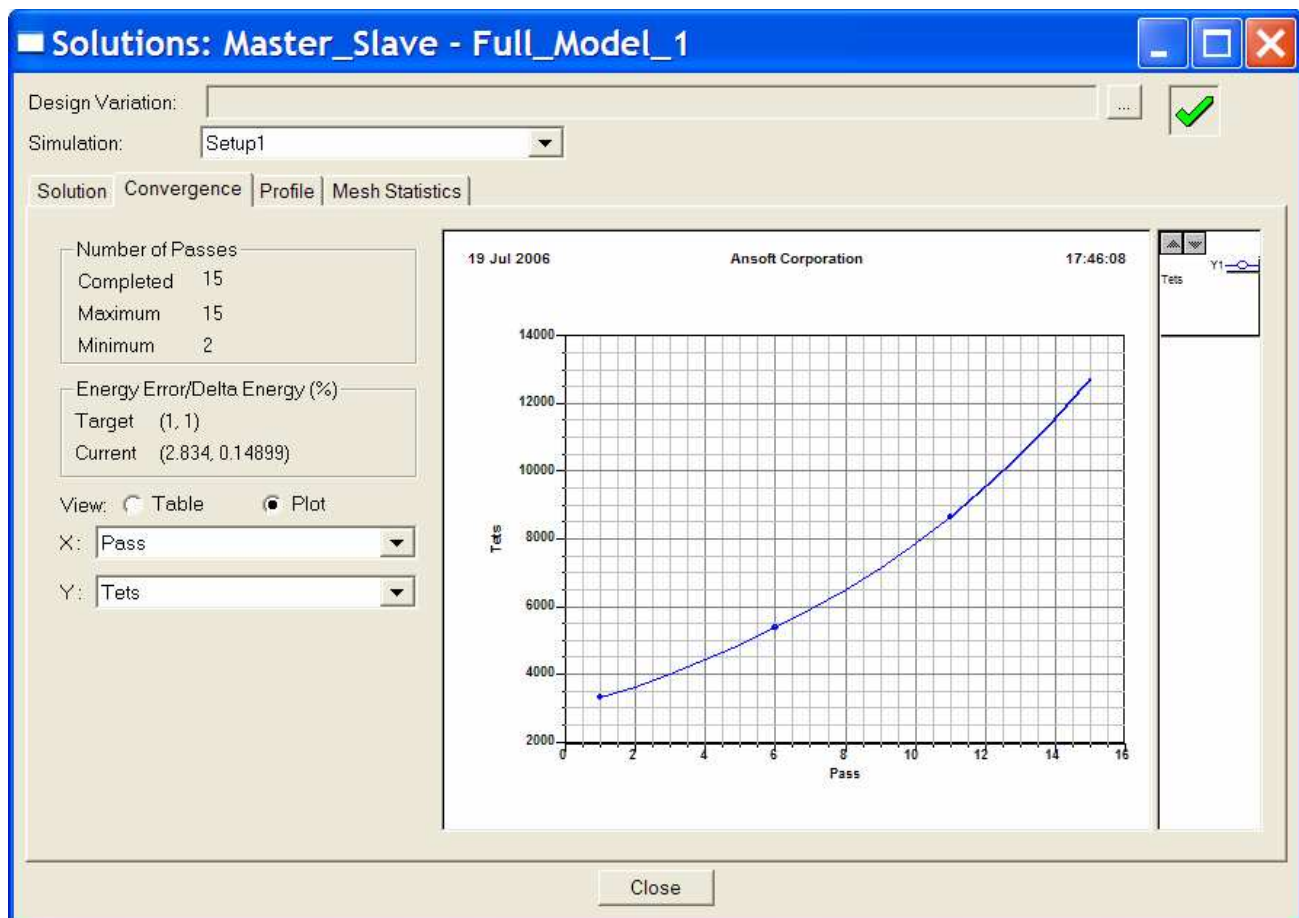
Analyze

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



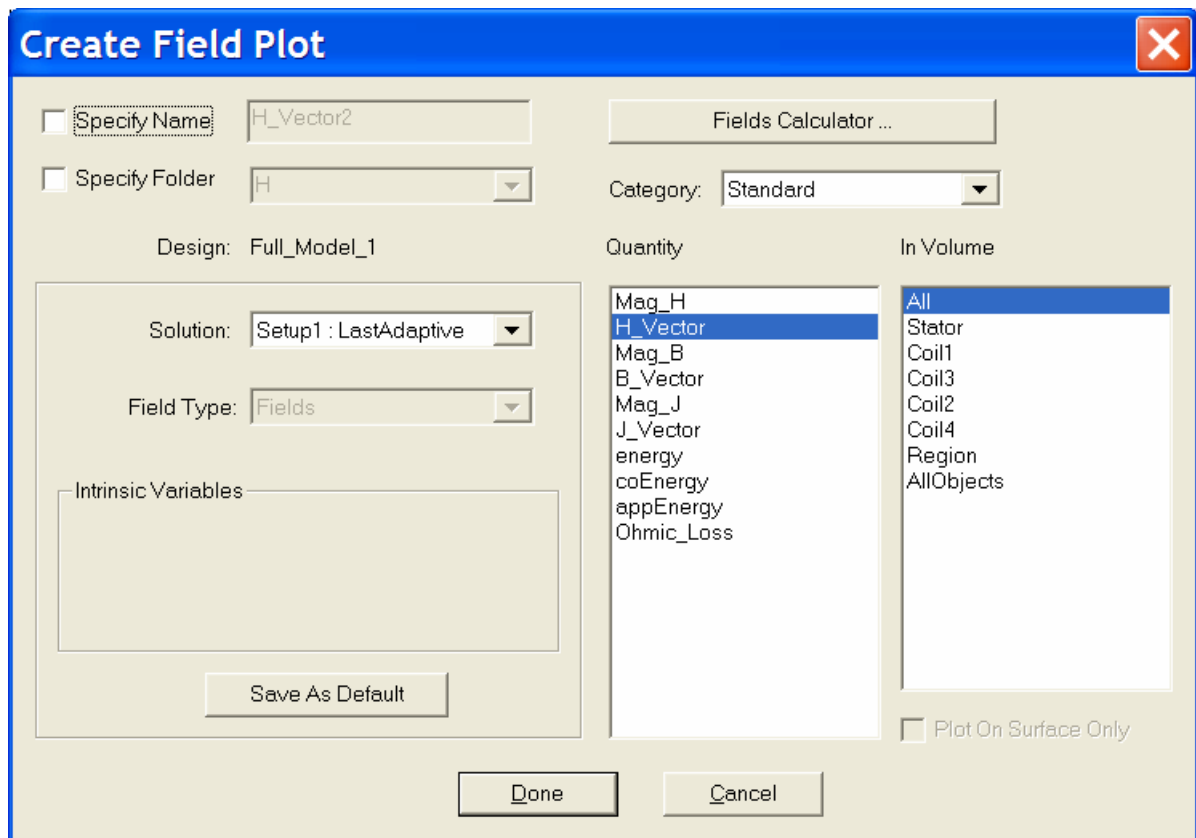
Solution Data

-  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 for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
-  **Note:** You don't have to wait for the solution to be done to do this. You can do this while the simulation is running, all information will update automatically after each pass is done.



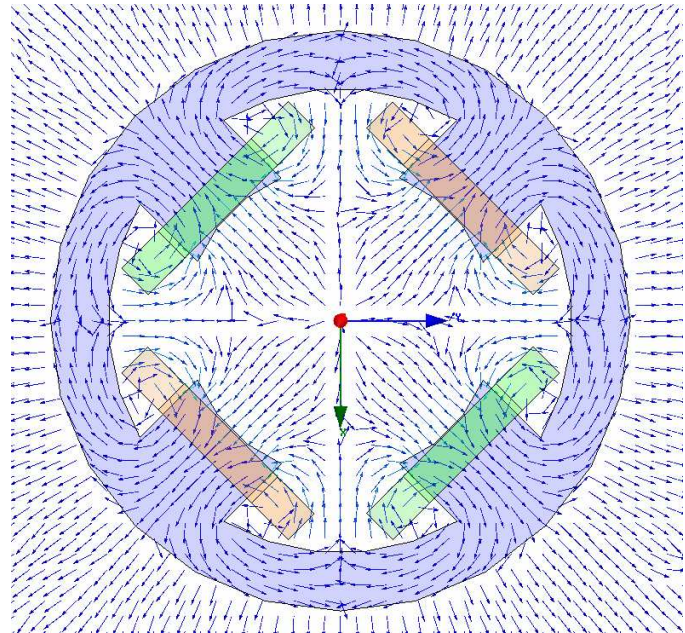
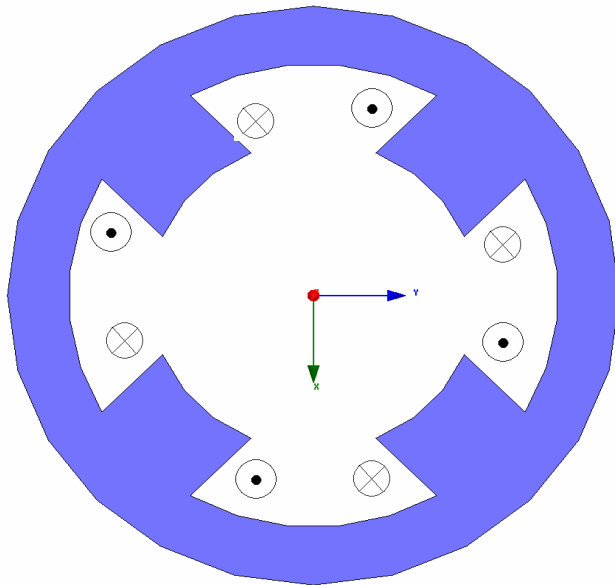
Plot H-Field Vector on X-Y Plane

- To plot the H-field vector on the X-Y plane, click **Planes > Global: XY** from the history tree window, then right click on **Field Overlays > Fields > H > H_Vector > Done** in the project window.

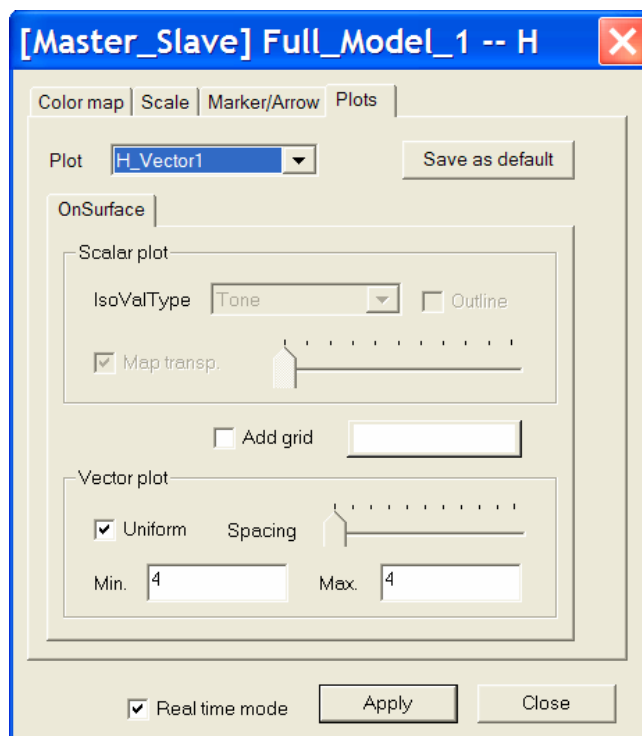
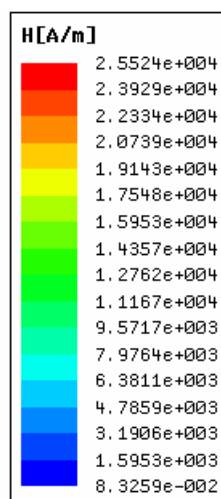


Topic - Master/Slave Boundaries

- ▲ The following H-field vector plot will appear, which is the result of the current excitation on the left side.

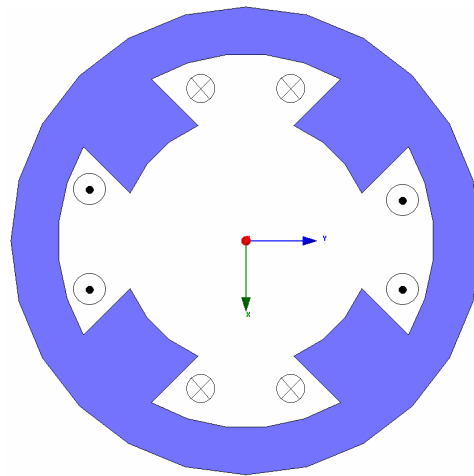
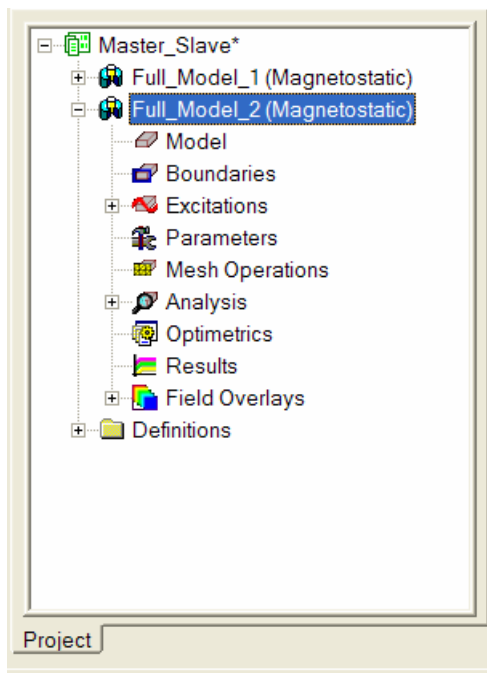


- ▲ If the plot is not as nice as you may want to see, you can double click the legend bar, then change various settings under Color map/Scale/Marker/Arrow/Plot tabs.



Create Design2: Full_Model_2

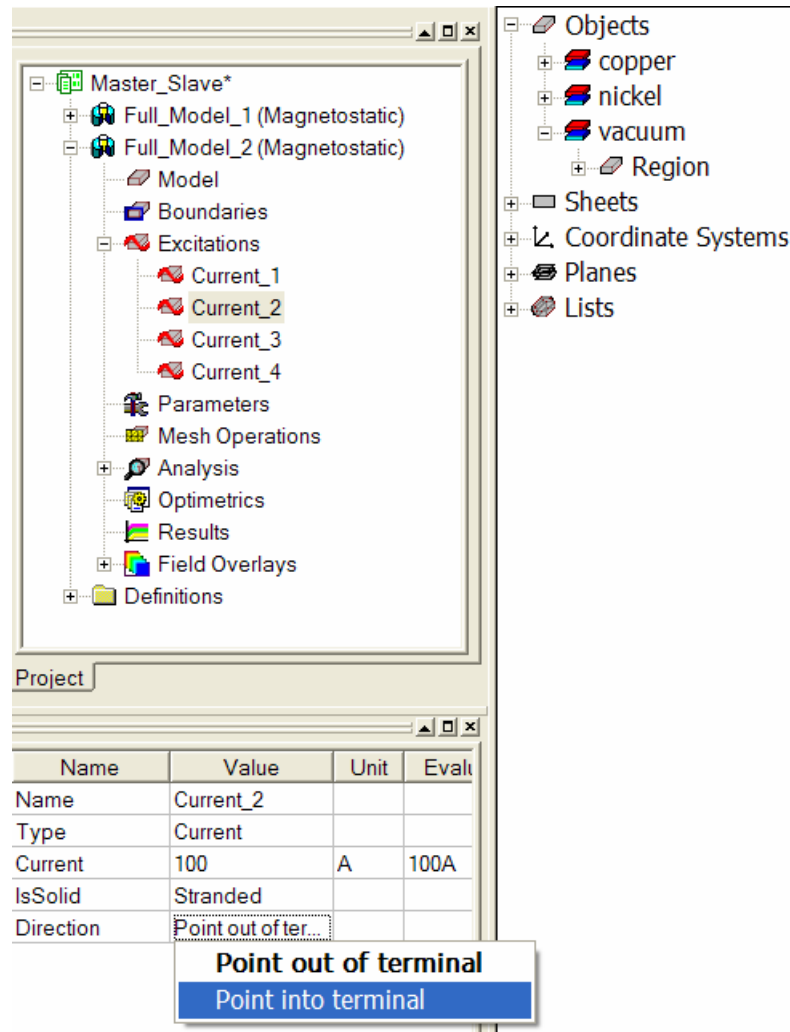
- Click on **Full_Model_1** in the Project Tree and then right mouse click and select **Copy**
- Click on the project name **Master_Slave**, right click and select **Paste**, a new design called **Full_Model_2** has been created



Modify Current Excitation

- Click on **Excitations > Current_2** in the **Project** window, you will see a **MaxwellTab** window under **Project** window, change from **Point out of terminal** to **Point into terminal** in the **Direction** row.
- Do the same for **Current_4**. The purpose is to change the current excitation as shown in the graph above.

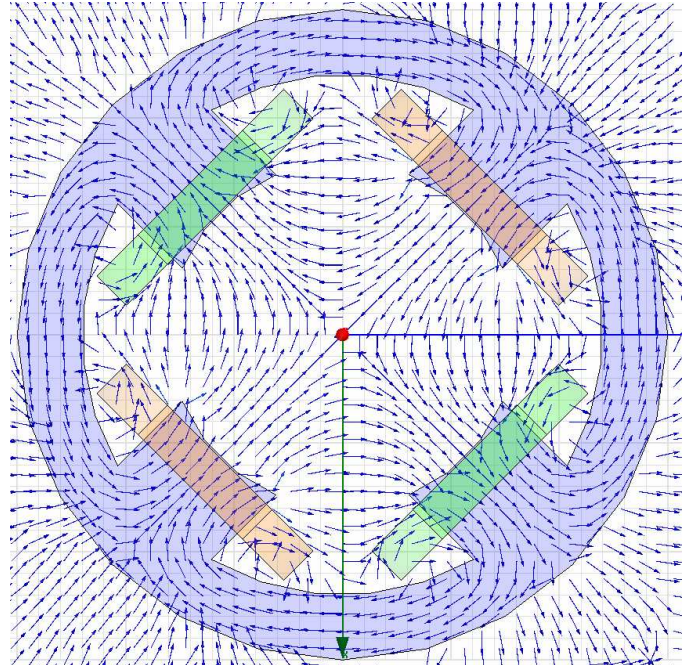
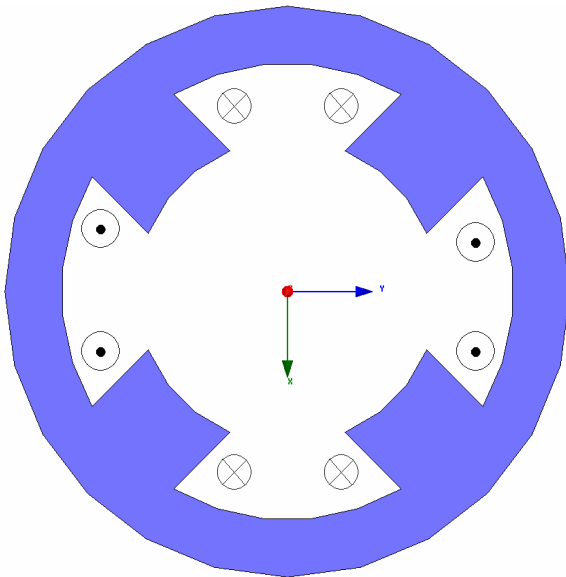
Topic - Master/Slave Boundaries



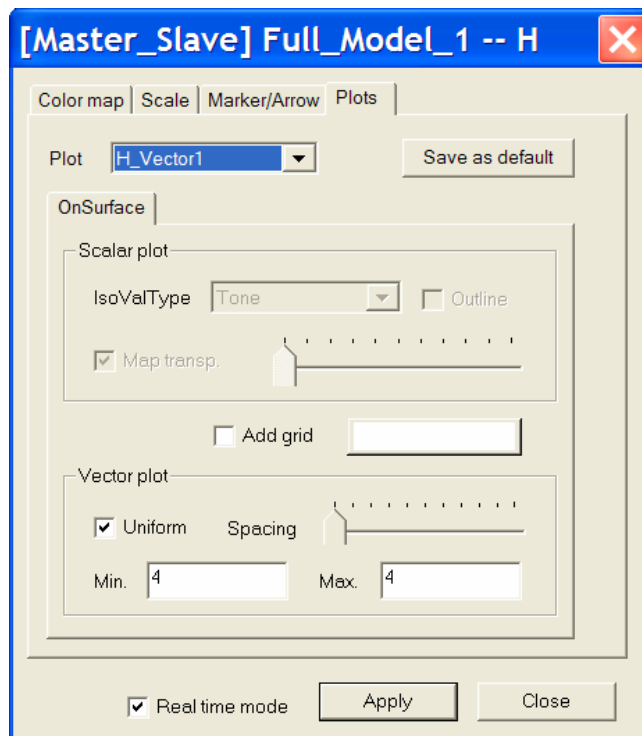
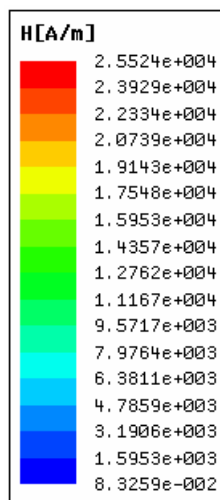
- Right click on **Setup1** of **Analysis**, select **Analyze**.
- Double click on **Field Overlays > H > H_Vector1**. You can see the vector plot is updating automatically after each pass.

Topic - Master/Slave Boundaries

- ▲ The following H-field vector plot will appear, which is the result of the current excitation on the left side.



- ▲ If the plot is not as nice as you may want to see, you can double click the legend bar, then change various settings under Color map/Scale/Marker/Arrow/Plot tabs.

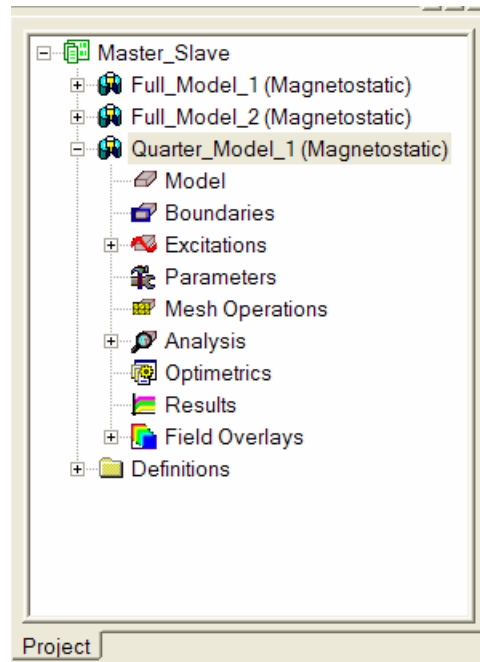


Create Quarter Symmetry Geometry

- So far we've been working with full geometry. Often it is useful to use symmetry in order to reduce the problem size and thus increase the solution time. In this section, we'll show how to create the symmetric model and how to assign Master/Slave boundary conditions.

Copy the Existing Design

- Select the design **Full_Model_1**, and right mouse click and select **Copy**.
- Click on project name **Master_Slave**, and right mouse click and select **Paste**; this will create a third design called **Full_Model_3**. Change its name to **Quarter_Model_1**. Double click on this design to make it active.

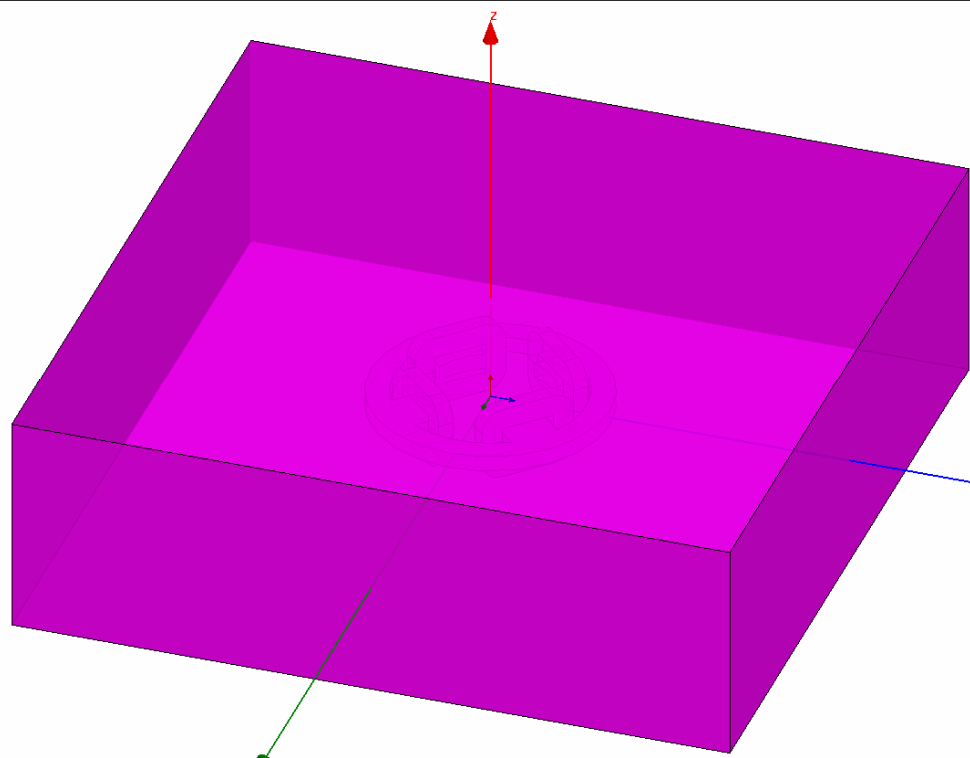


Modify the Geometry to Create Quarter Symmetry

- Make all of the objects visible and then select them. Next select the menu item **3D Modeler > Boolean > Split** and choose
 - Split Plane > YZ
 - Keep Fragments > Positive Side
 - Split Objects > Split Objects Crossing Split Plane

Topic - Master/Slave Boundaries

- Objects
 - copper
 - Coil1
 - Coil2
 - Coil3
 - Coil4
 - nickel
 - Stator
 - vacuum
 - Region
- Sheets
 - Section1
 - Section2
 - Section3
 - Section4
- Coordinate Systems
 - Planes
 - Lists



Split

Split plane:

☐ XY
 ☒ YZ
 ☐ XZ

Keep fragments:

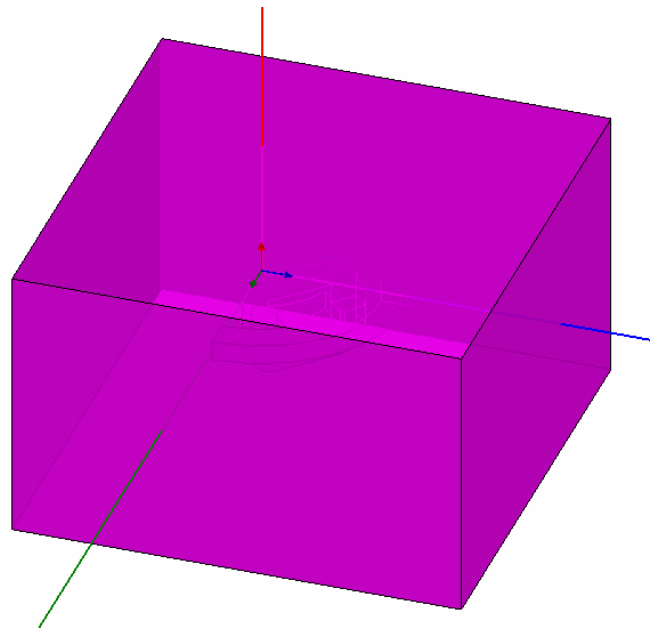
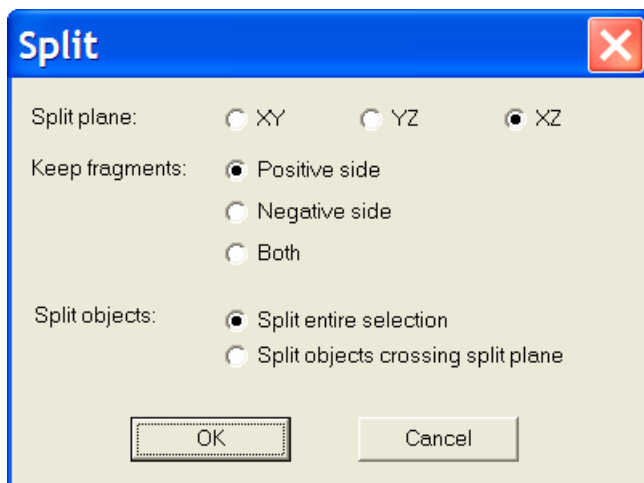
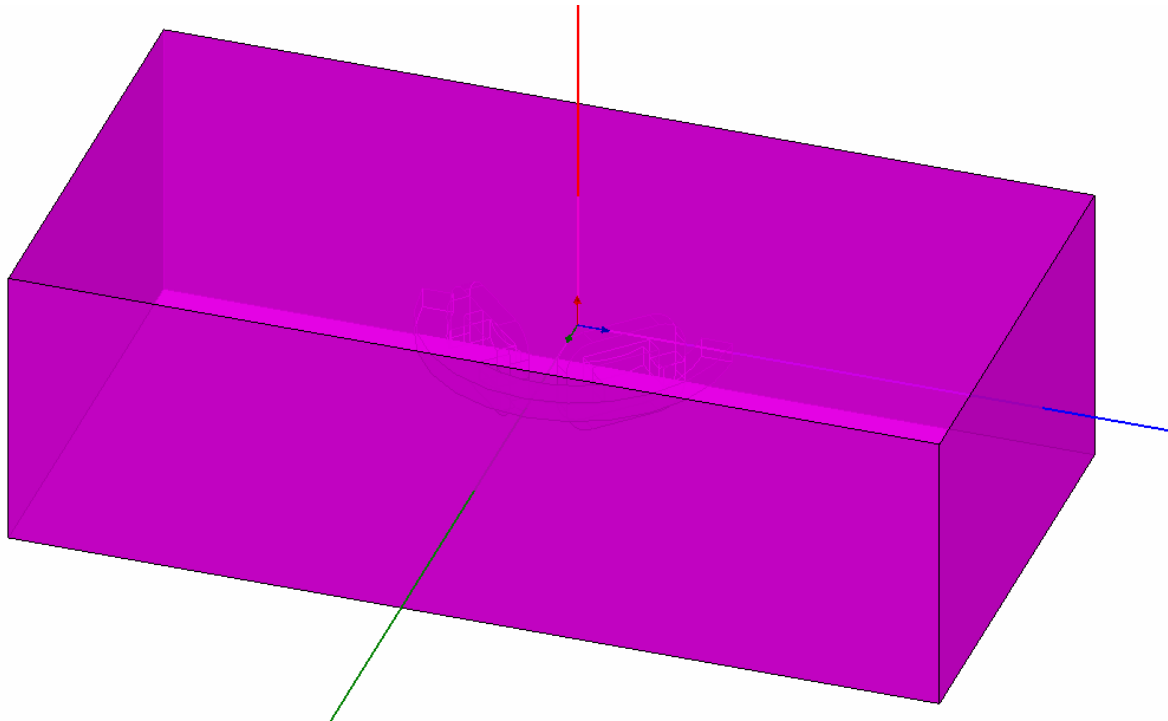
☒ Positive side
 ☐ Negative side
 ☐ Both

Split objects:

☒ Split entire selection
 ☐ Split objects crossing split plane

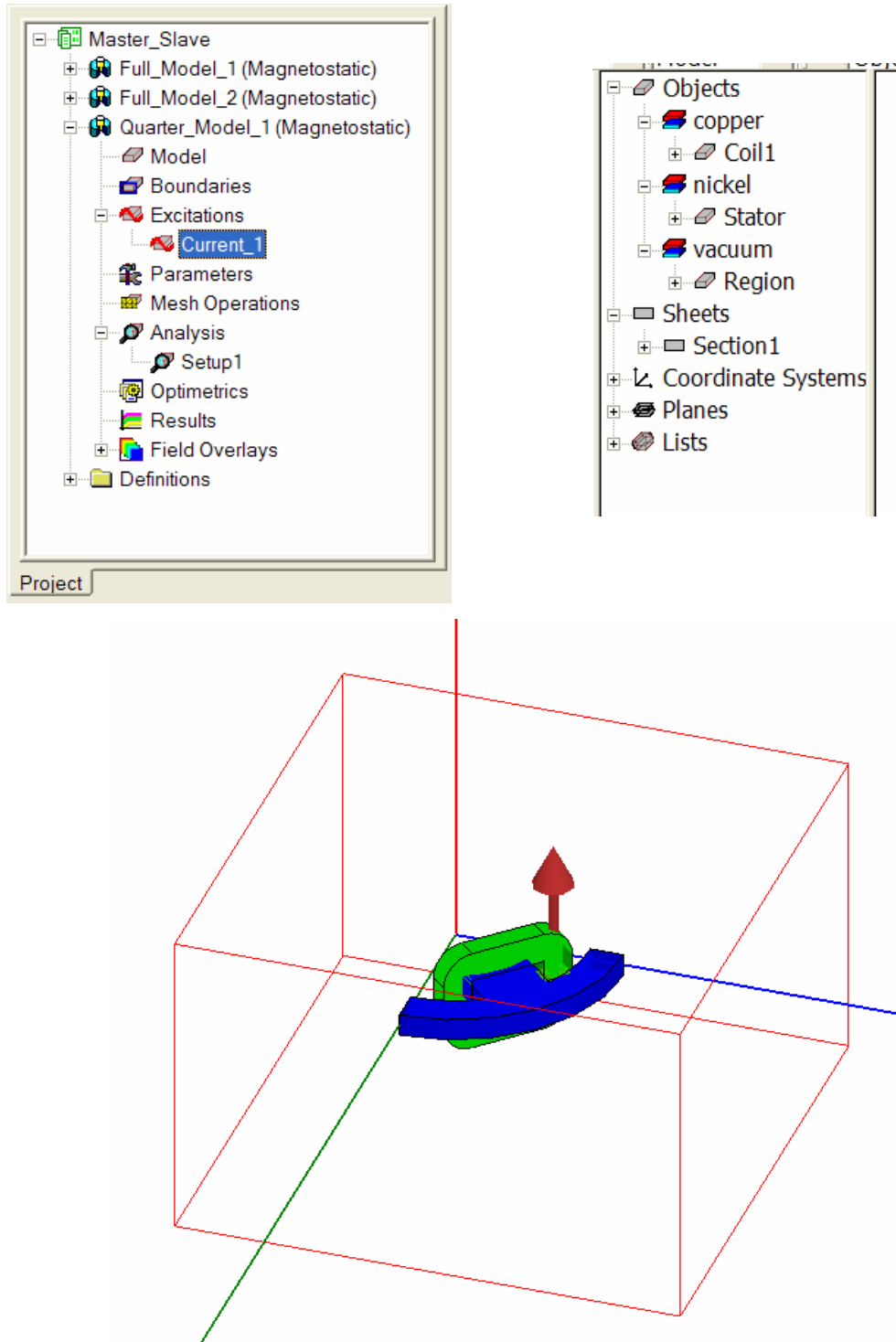
OK

Cancel



Topic - Master/Slave Boundaries

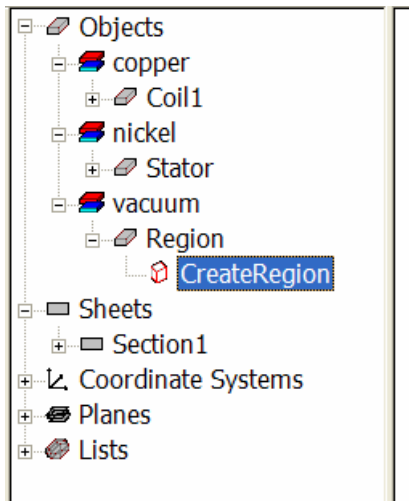
- ▲ The resulting geometry, Project window and History Tree window should look like this:



Topic - Master/Slave Boundaries

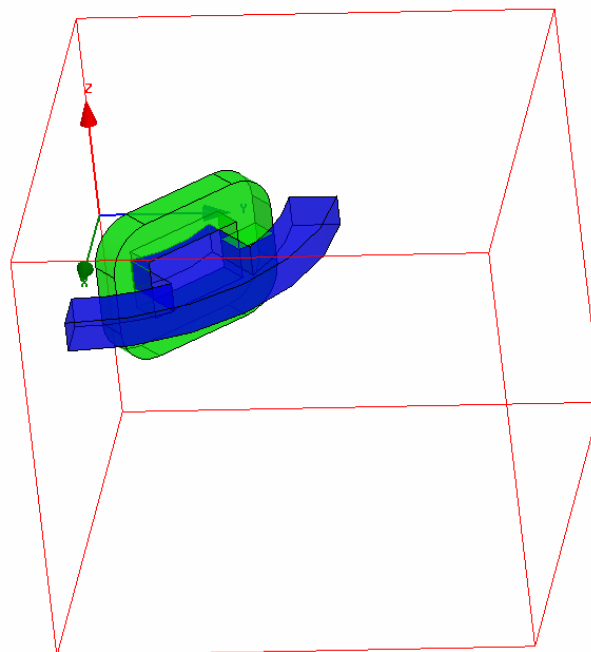
Assigning Master/Slave Boundary Conditions

- Click on **Objects > Vacuum > Region > CreateRegion** in the History Tree window, you will see the following information window:



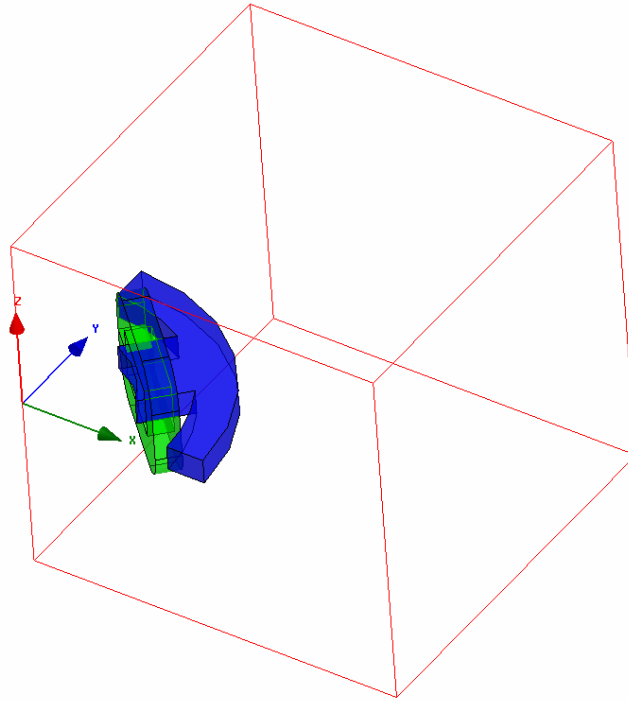
Name	Value
Command	CreateRegion
Coordinate System	Global
+X Padding Percentage	100
-X Padding Percentage	0
+Y Padding Percentage	100
-Y Padding Percentage	0
+Z Padding Percentage	100
-Z Padding Percentage	100

- Change **Padding Percentage** of **-X** and **-Y** to be 0. The geometry has become this:

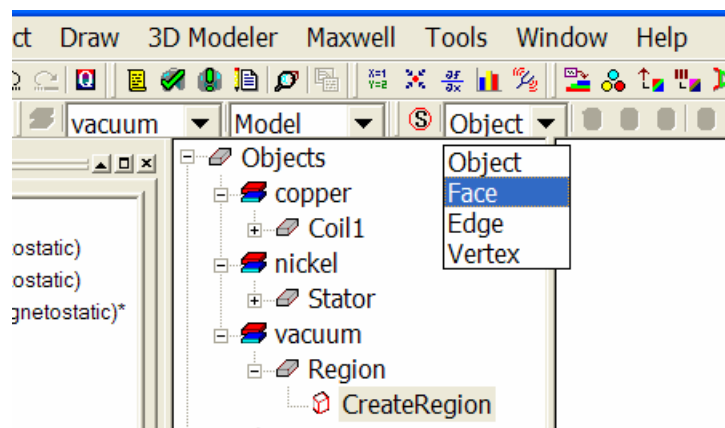


Topic - Master/Slave Boundaries

- Spin the model so that you can see the face on the XZ plane. Like this:

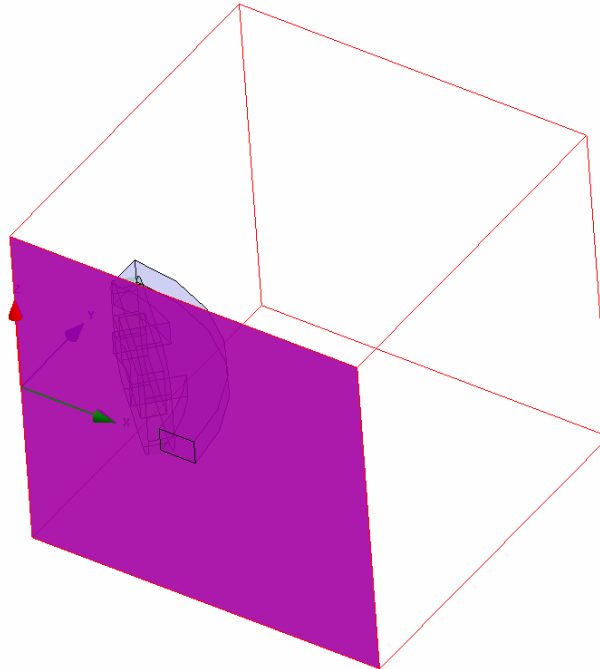


- Click on **Object** from the Standard toolbar, select **Face**. Now we are ready to select faces instead of objects.

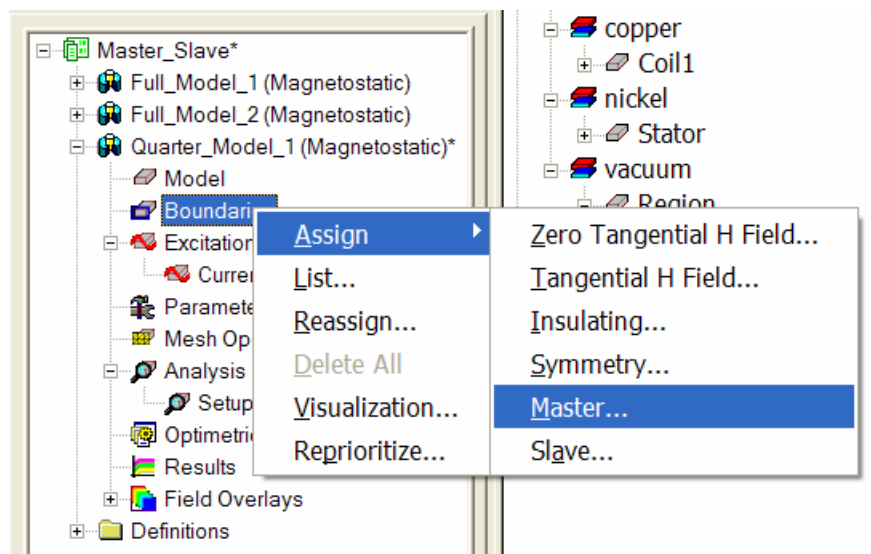


Topic - Master/Slave Boundaries

- Click on the face on the XZ plane

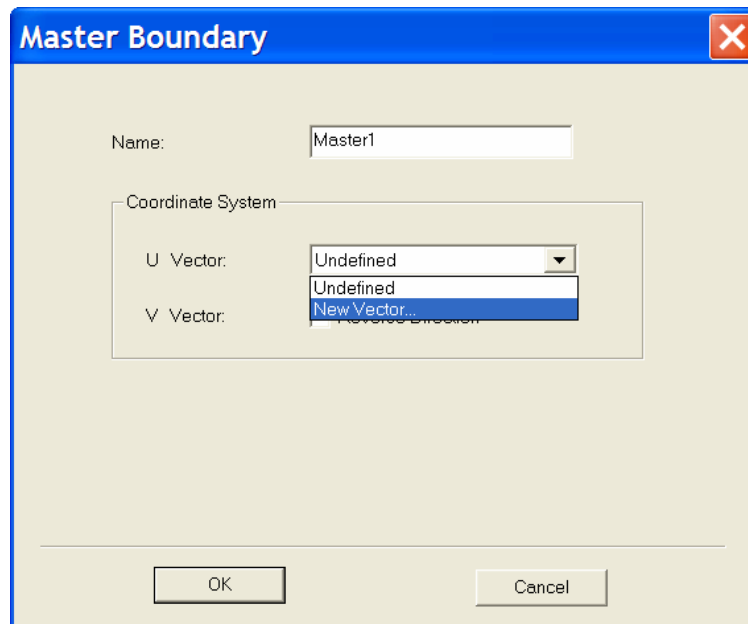


- Right click on Boundaries > Assign > Master...

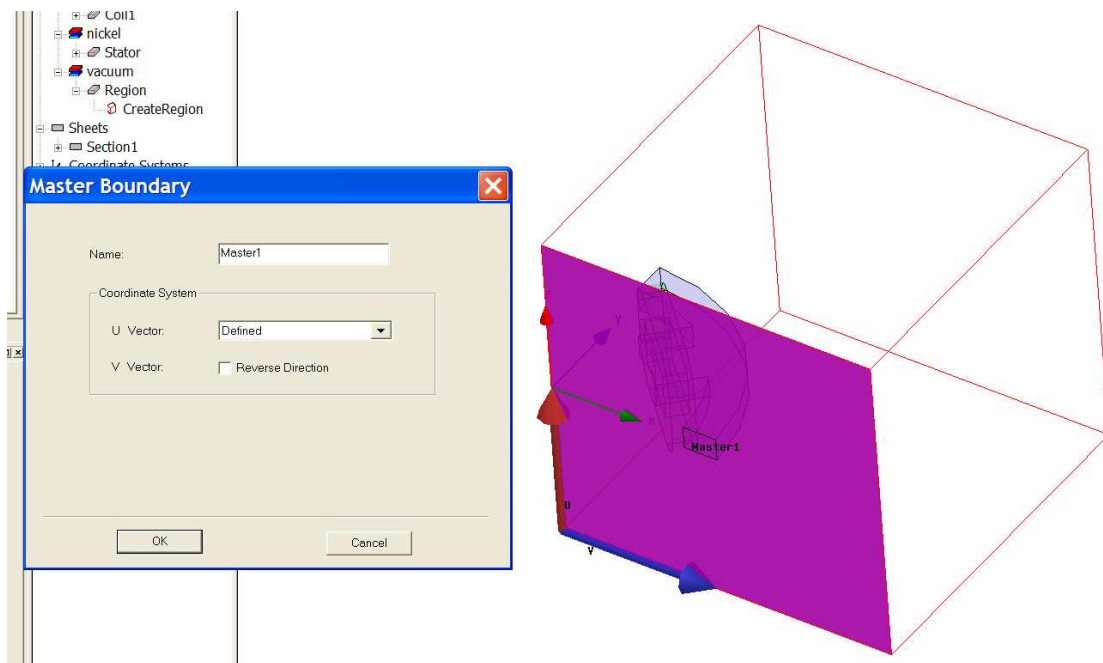


Topic - Master/Slave Boundaries

- ▲ The following window pops up, select **New Vector...**

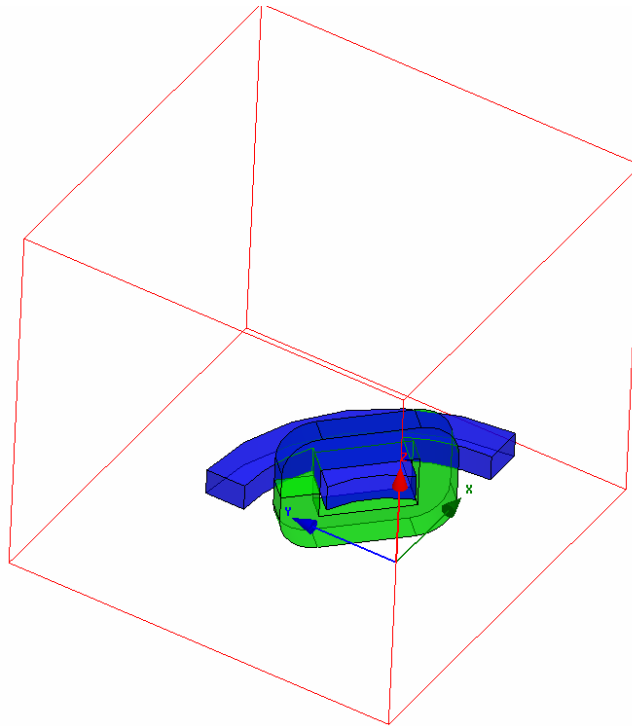


- ▲ Draw a straight line along Z direction starting from the left-most vertex, ending at the top-left corner of the box. Master boundary has been assigned, as shown in the following. Click **OK**.

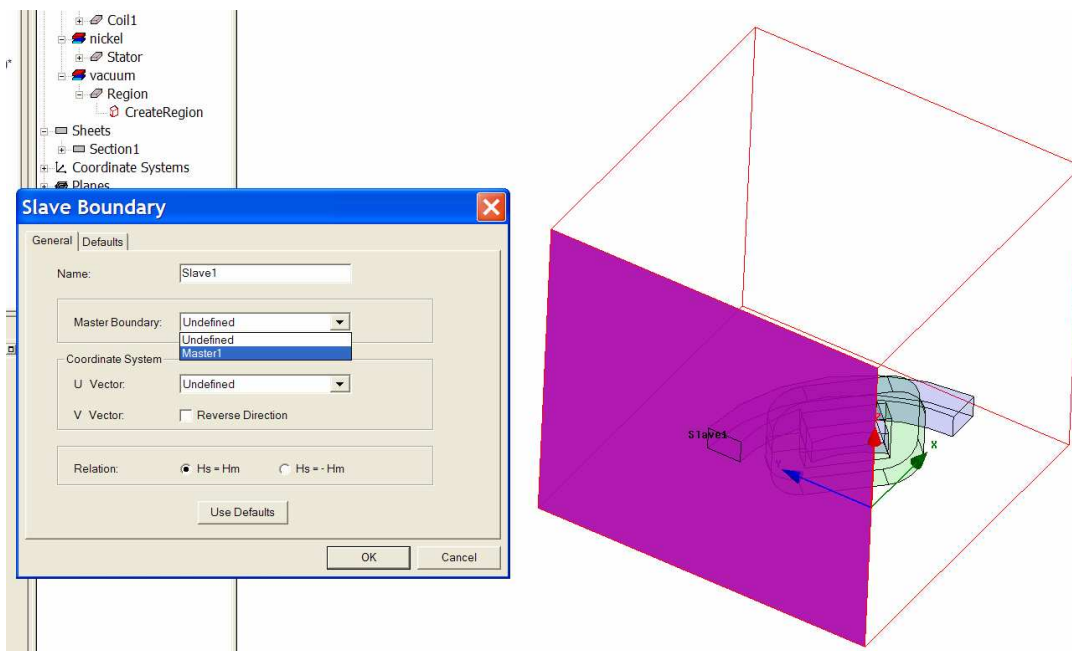


Topic - Master/Slave Boundaries

- Spin the model so that you can see the face in the YZ plane, like this:

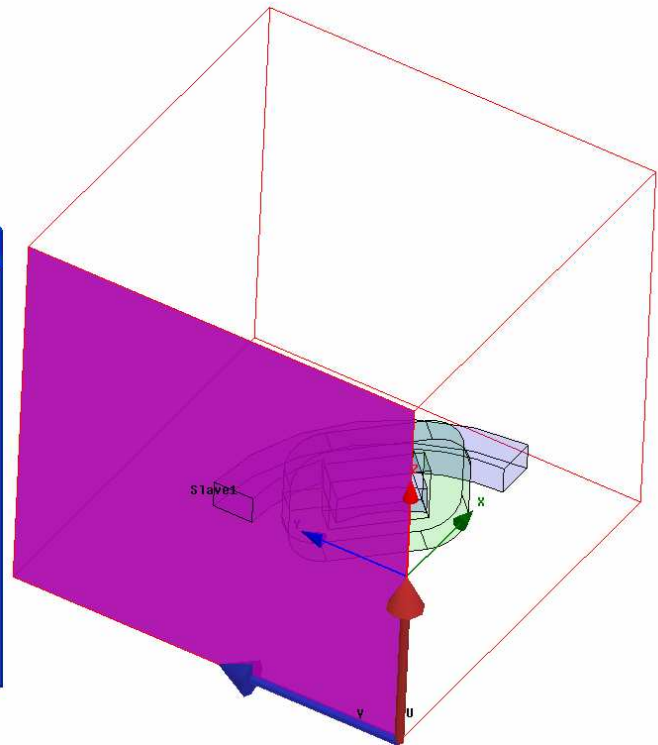
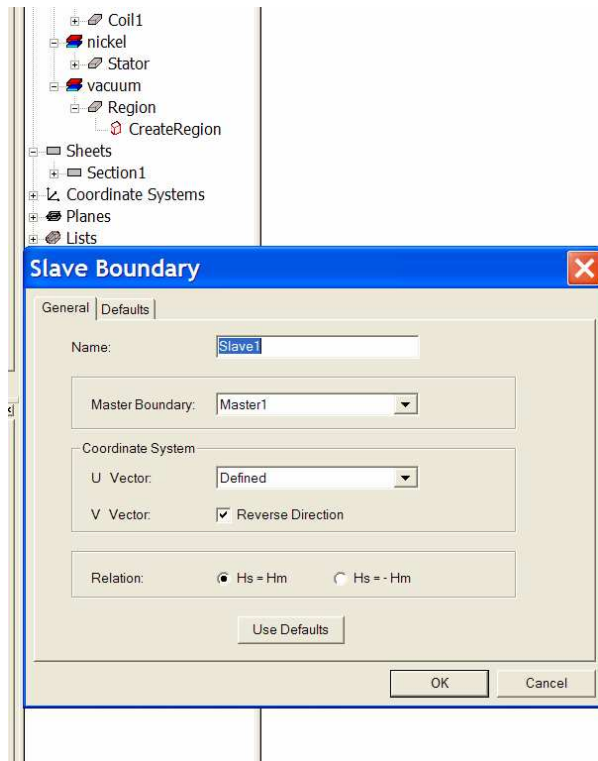


- Select the face on the YZ plane, right click on Boundaries > Assign > Slave.

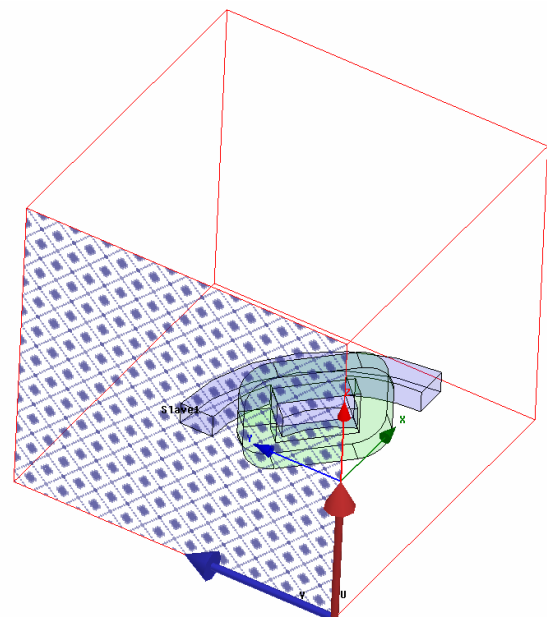
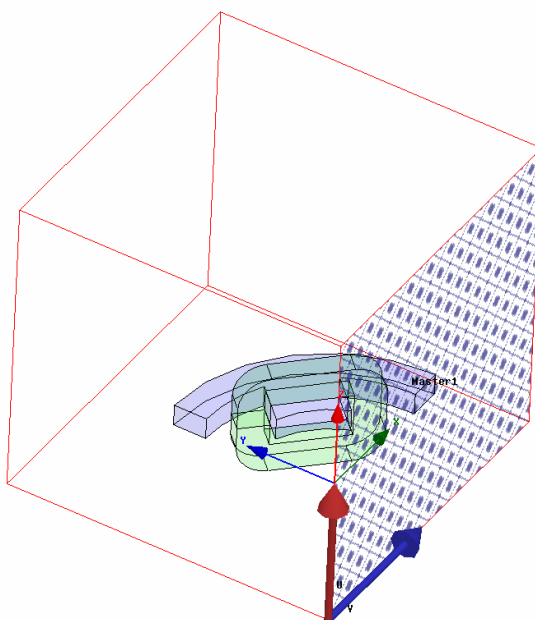


Topic - Master/Slave Boundaries

- Select **Master1** for **Master Boundary**, **New Vector** for **U Vector**, then draw the same straight line along the Z direction. Select **Hs=Hm** for **Relation**. The **Slave Boundary** has been assigned, as shown below. Click **OK**.

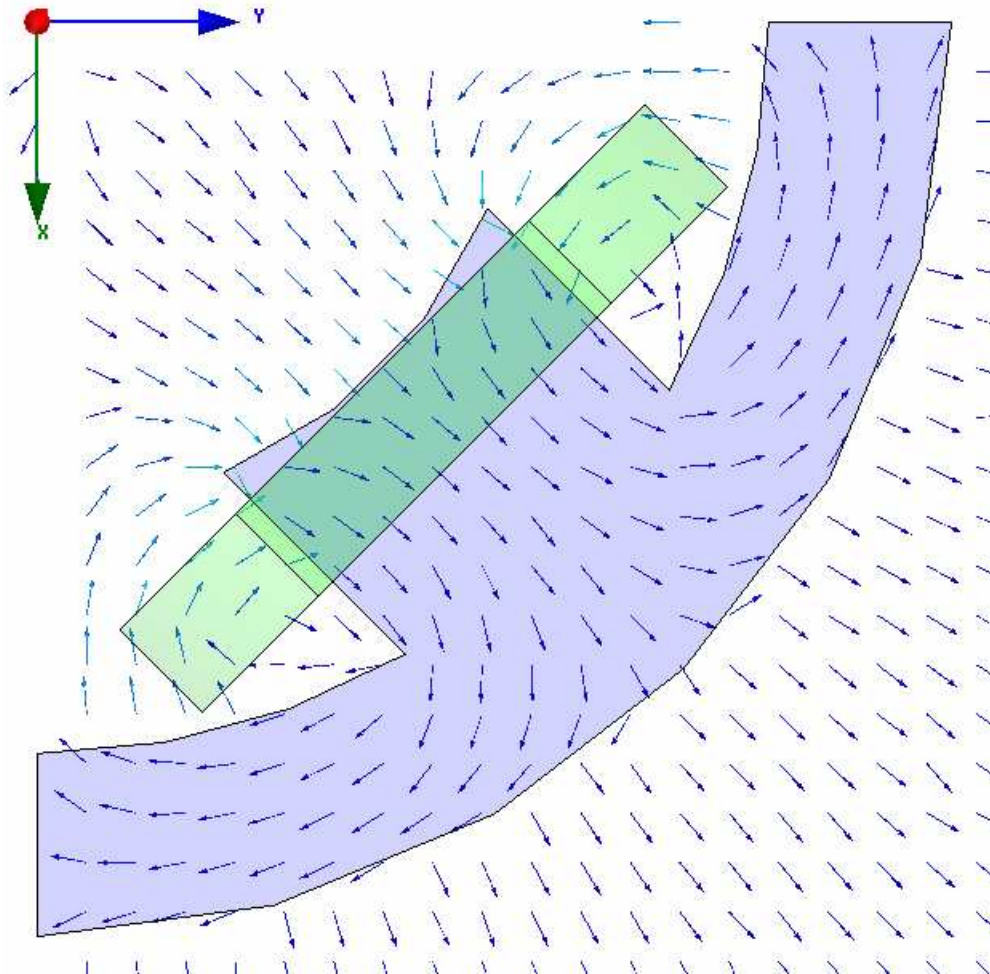


- Click on **Boundaries > Master1** and **Slave1**, you should see these:



Topic - Master/Slave Boundaries

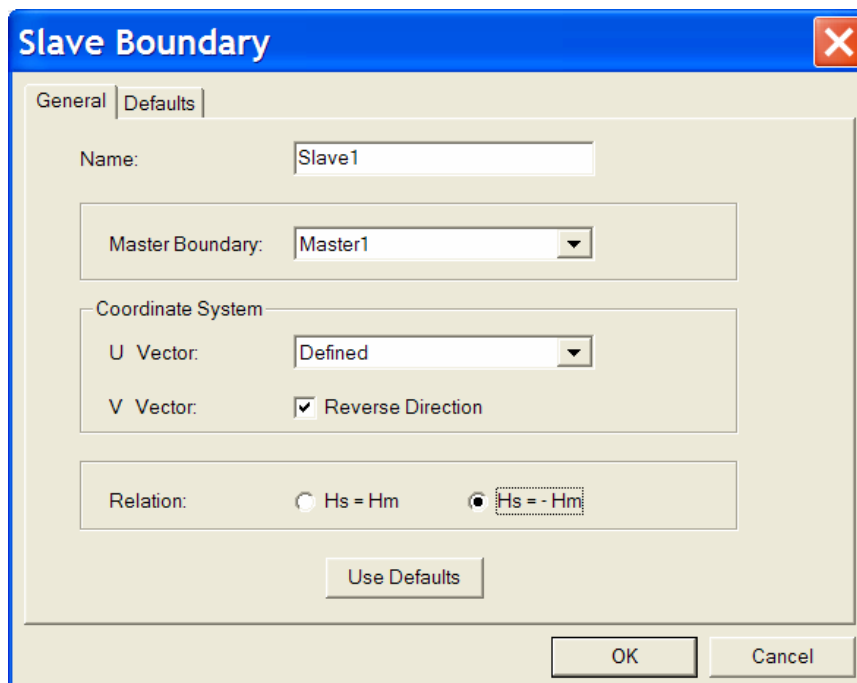
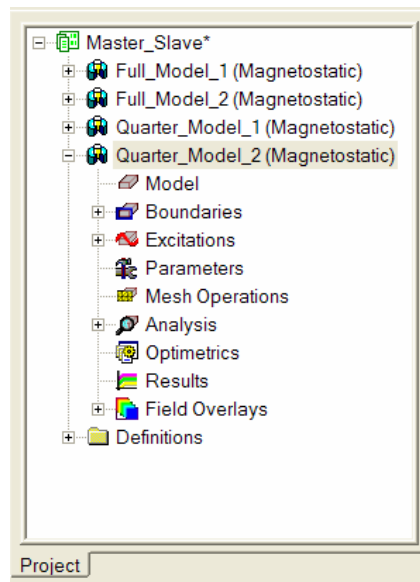
- ▲ Right click on **Analysis > Setup1**, select **Analyze**.
- ▲ Select **Region**, **Hide** it for the view.
- ▲ Double click **Field Overlays > Field > H > H_Vector1**. You will see the following vector plot. This is a quarter view of design **Quarter_Model_1**.



Topic - Master/Slave Boundaries

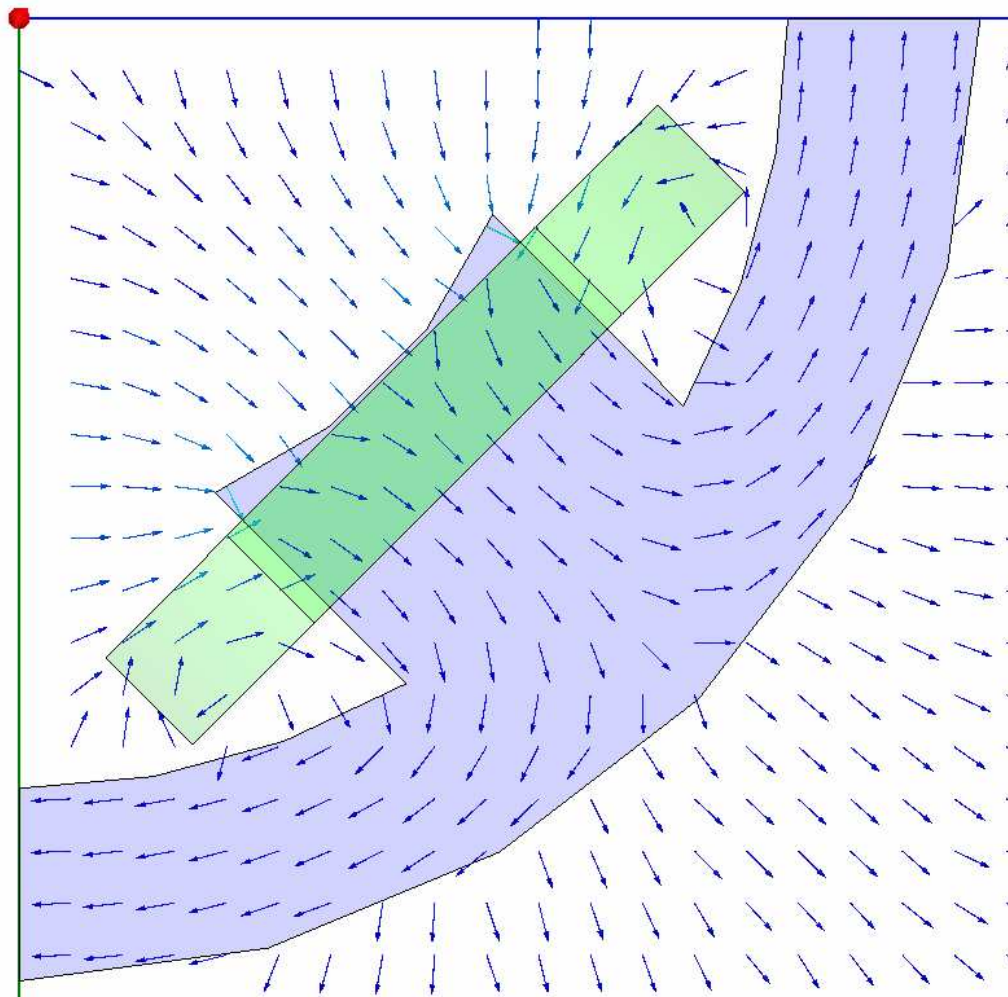
Create a New Design and Assign Different Master/Slave Boundary Conditions

- Select the design **Quarter_Model_1**, and right mouse click and select **Copy**.
- Click on project name **Master_Slave**, and right mouse click and select **Paste**; this will create a fourth design called **Quarter_Model_2**. Double click on this design to make it active.

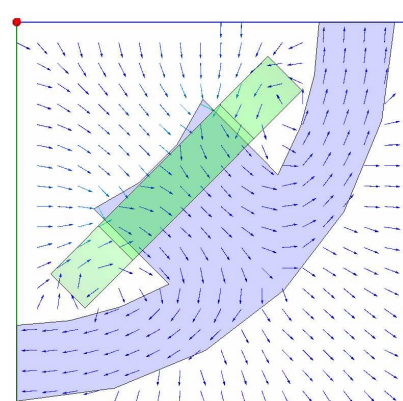
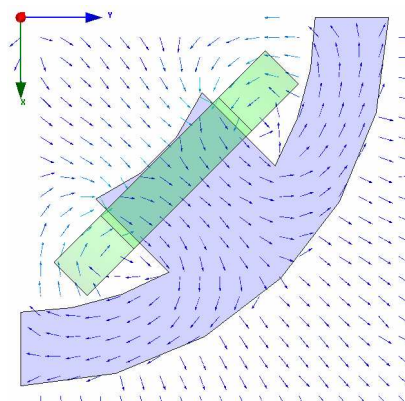
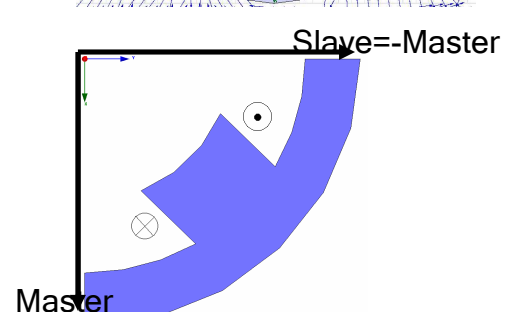
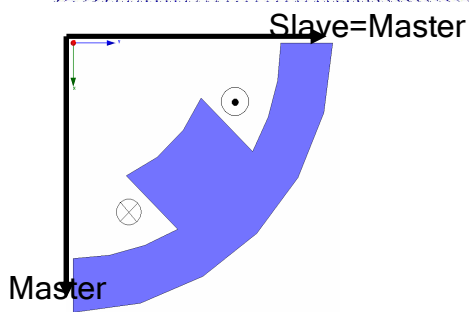
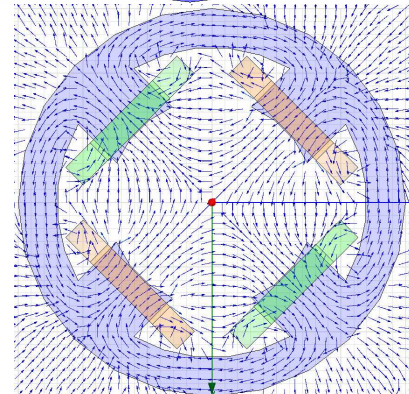
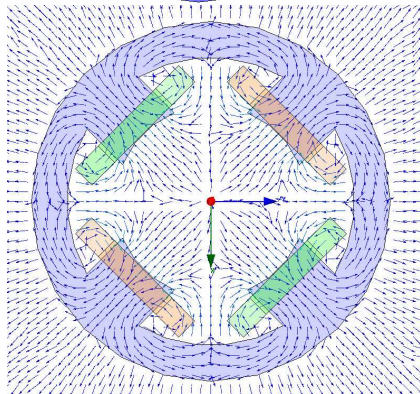
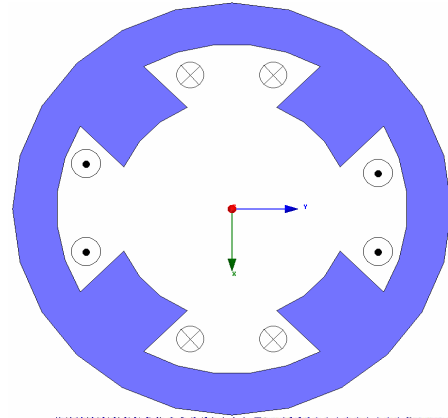
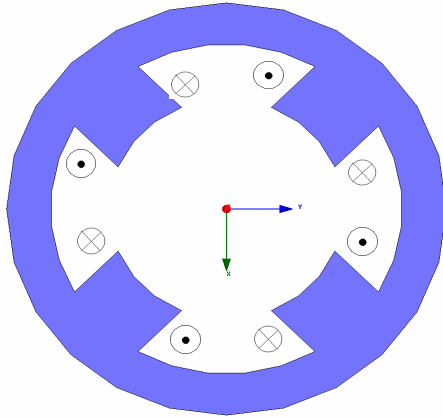


Modify the Slave Boundary

- ▶ Double click **Boundaries > Slave1**, the following window pops up. Change **Relation** to $H_s = -H_m$.
- ▶ Right click **Analysis > Setup1 > Analyze**.
- ▶ Double click **Field Overlays > Field > H > H_Vector_1**. You will see the H vector plot like this.



Topic Summary





Chapter 6.0 - Eddy Current Examples

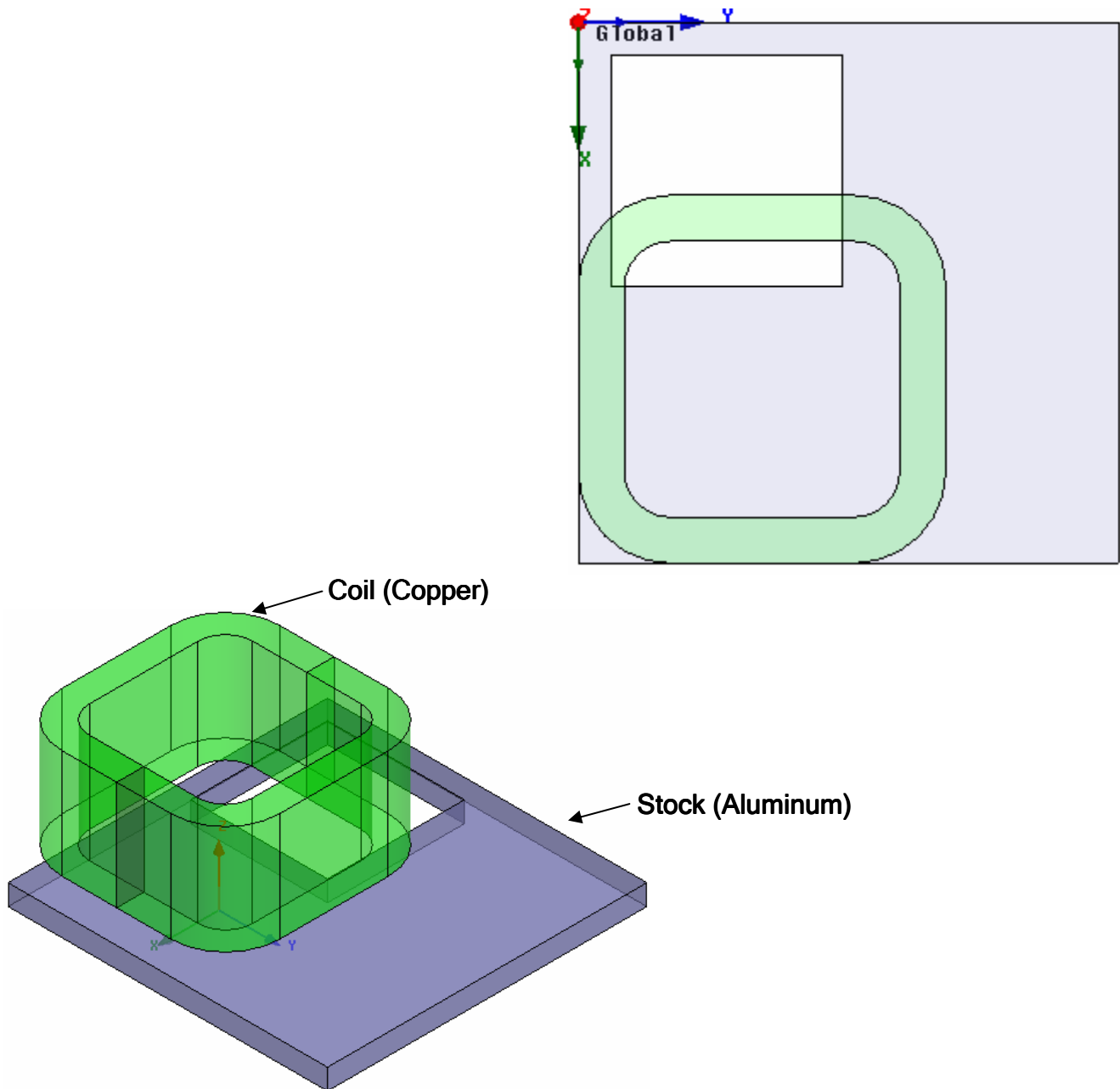
-  6.1 - Asymmetrical Conductor with a Hole
-  6.2 - Radiation Boundary

THIS PAGE INTENTIONALLY LEFT BLANK

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](#).



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

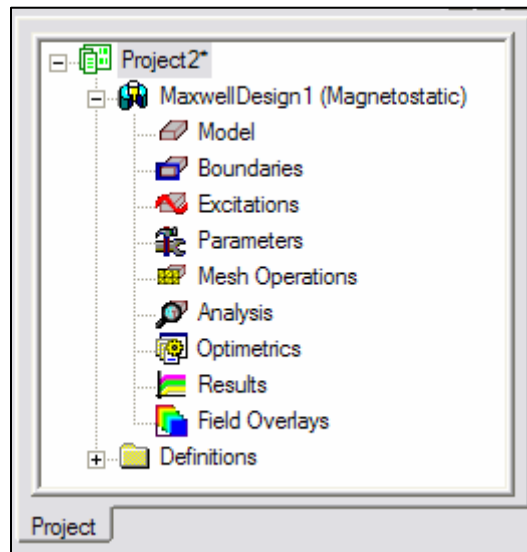
-  Edit property of new primitives: ☒ **Checked**

3. Click the **OK** button

Opening a New Project

To open a new project:

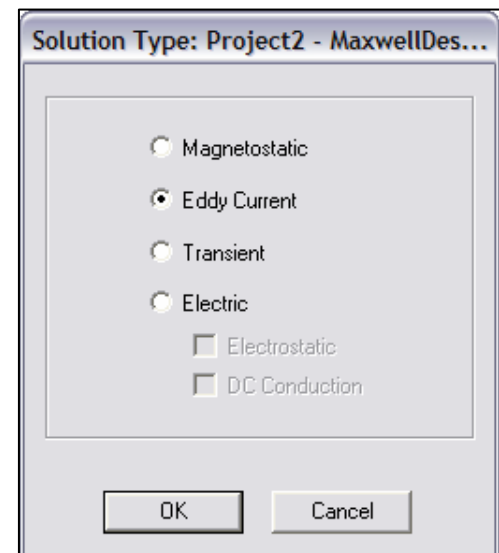
- ▲ In an Ansoft Maxwell window, click the  On the Standard toolbar, or select the menu item **File > New**.
- ▲ From the **Project** menu, select **Insert Maxwell Design**.



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

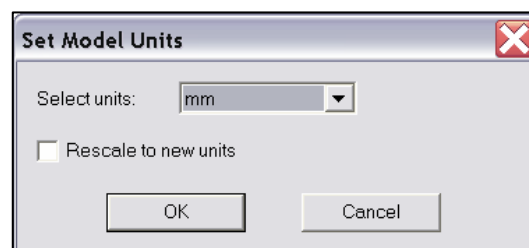


Creating the 3D Model

Set Model Units

To set the units:

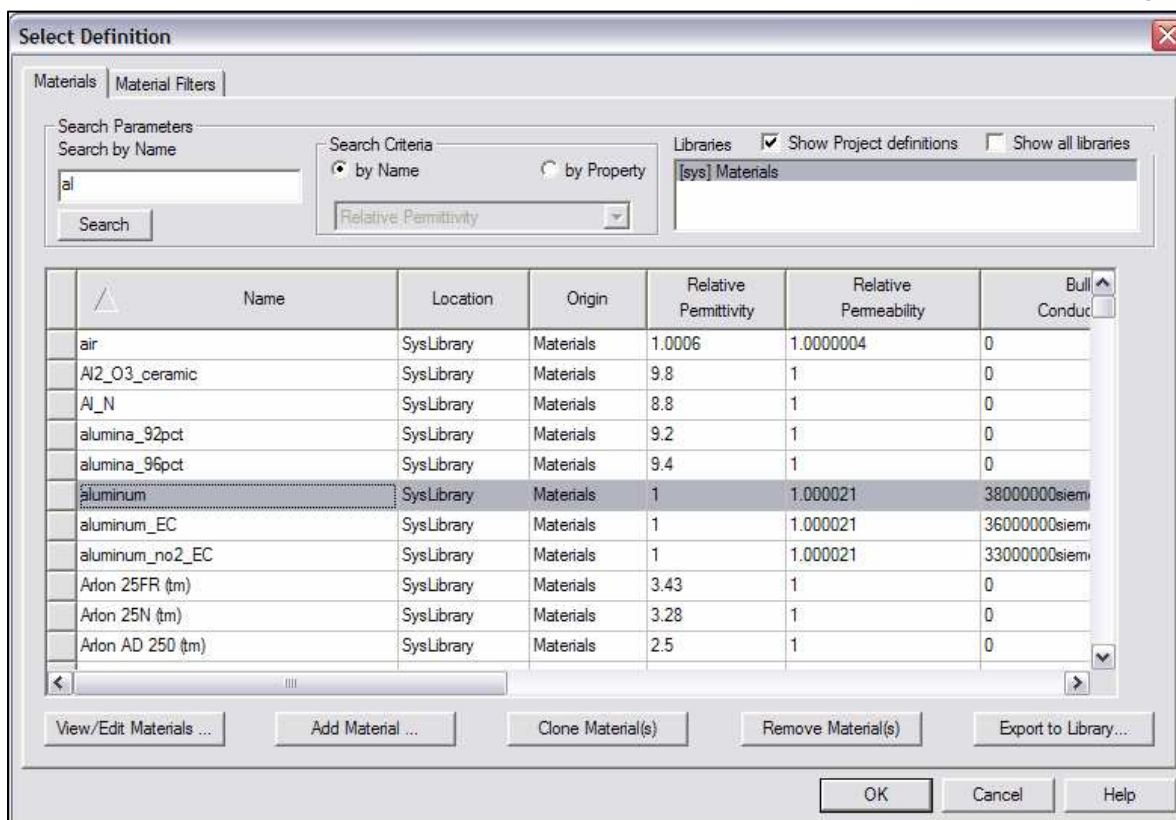
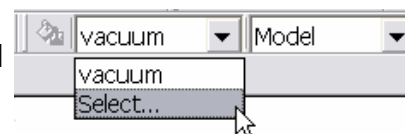
1. Select the menu item **3D Modeler > Units**
2. Set Model Units:
 1. Select Units: **mm**
 2. Click the **OK** button



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



Create Stock

To create the stock:

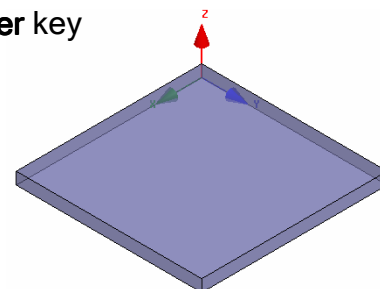
1. Select the menu item **Draw > Box**
2. Using the coordinate entry fields, enter the box position
 - ▲ X: **0.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
 - ▲ dX: **294.0**, dY: **294.0**, dZ: **19.0**, Press the **Enter**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

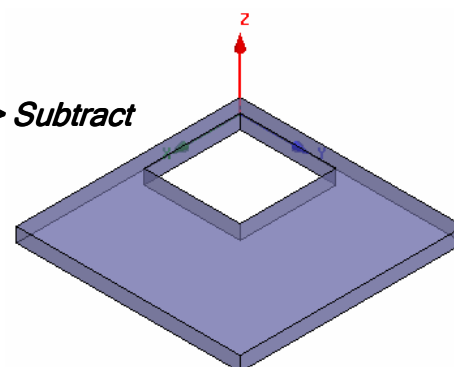
1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **hole**
3. Click the **OK** button

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



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:

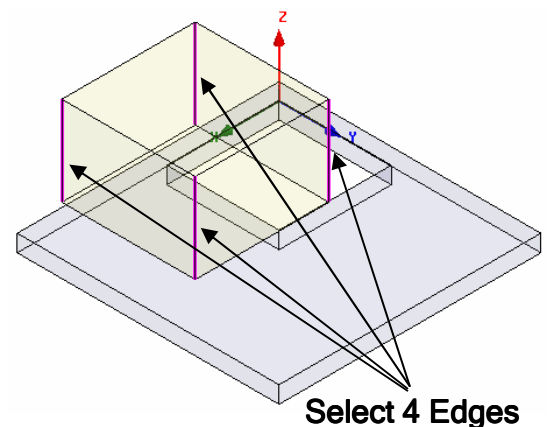
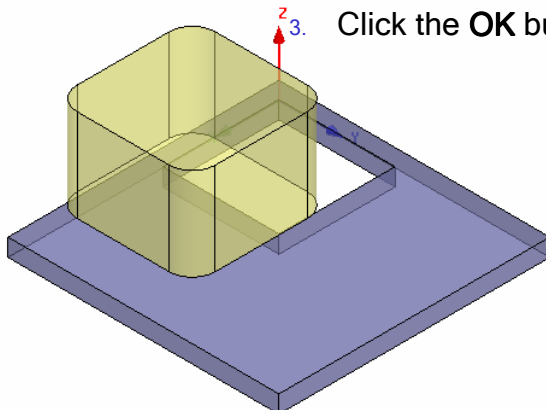
1. Select the menu item **Draw > Box**
2. Using the coordinate entry fields, enter the box position
 - ▀ X: **119.0**, Y: **25.0**, Z: **49.0**, Press the **Enter** key
3. Using the coordinate entry fields, enter the opposite corner of the box:
 - ▀ dX: **150.0**, dY: **150.0**, dZ: **100.0**, Press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

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



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 fillets:

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

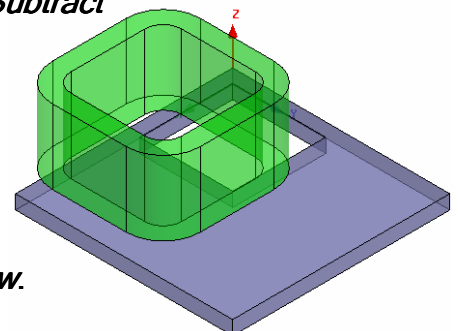
1. Select the menu item **Edit > Select > Objects**
2. Select the menu item **Edit > Select > By Name**
3. Select Object Dialog,
 1. Select the objects named: **coil**, **coil_hole**
 2. Click the **OK** button

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:

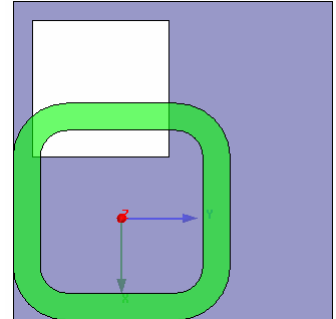
1. Select the menu item **View > Fit All > Active View**.



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

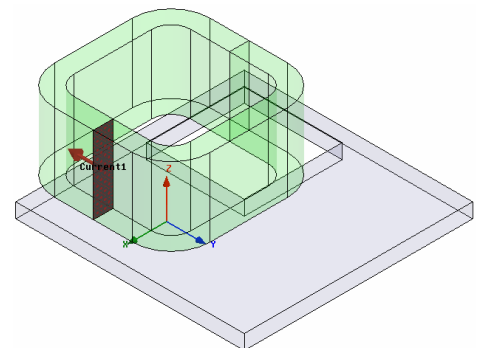
1. Select the menu item **3D Modeler > Surface > Section**
 1. Section Plane: **XZ**
 2. Click the **OK** button

Separate Bodies

1. Select the menu item **3D Modeler > Boolean > Separate Bodies**

Assign Excitation

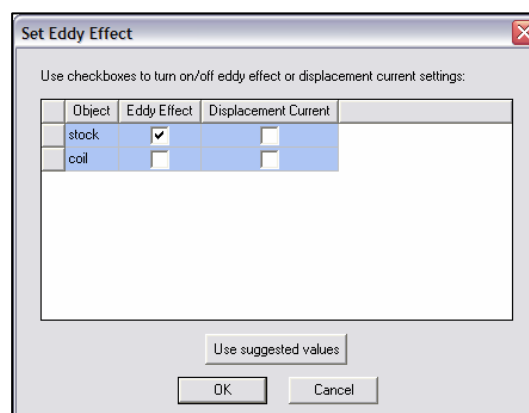
1. Select the menu item **Maxwell > Excitations > Assign > Current**
2. Current Excitation : General
 1. Name: **Current1**
 2. Value: **2742 A**
 3. Type: **Stranded**
 - ▲ **Note:** The current flow should be clockwise when viewing the coil from above. Use the **Swap Direction** button to change the direction.
3. Click the **OK** button



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

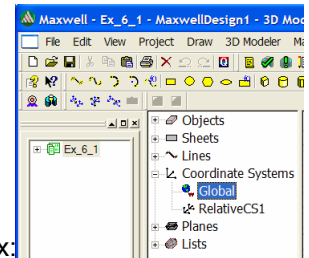
Example (Eddy Current) - Asymmetrical Conductor

Create a dummy object

- As a post-processing results we are interested to calculate the flux density on a line with the following end points: [0.0; 72.0; 34.0] and [288.0; 72.0; 34.0]. In order to get as accurate results as possible, the mesh around the line has to be fine enough. It is generally a good practice to create dummy objects around regions of interest so that the mesh quality can be better controlled in those regions. For this particular problem we create a box around the line of interest.

To create a dummy object:

1. Be sure that the global coordinate system is active by clicking on it in the history tree
2. Select the menu item **Draw > Box**
3. Using the coordinate entry fields, enter the box position
 - X: **-3.0**, Y: **68.0**, Z: **30.0**, Press the **Enter** key
4. Using the coordinate entry fields, enter the opposite corner of the box:
 - X: **297.0**, Y: **76.0**, Z: **38.0**, Press the **Enter** key
 - (Note that the coordinates of the opposite corner are absolute values and that the size of the box will be 300 x 8 x 8mm.)



To set the name:

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

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

Example (Eddy Current) - Asymmetrical Conductor

Assign and apply mesh operations

-  Select the **dummy** object.
-  Select the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 -  Deselect **Restrict Length of Elements** check box
 -  Select **Restrict the Number of Elements** check box and change the **Maximum Number of Elements** to be 1000.
-  Select the menu item **Maxwell > Analysis Setup > Apply Mesh Operations**

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*

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

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

CVc : <Bx,By,Bz>

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

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

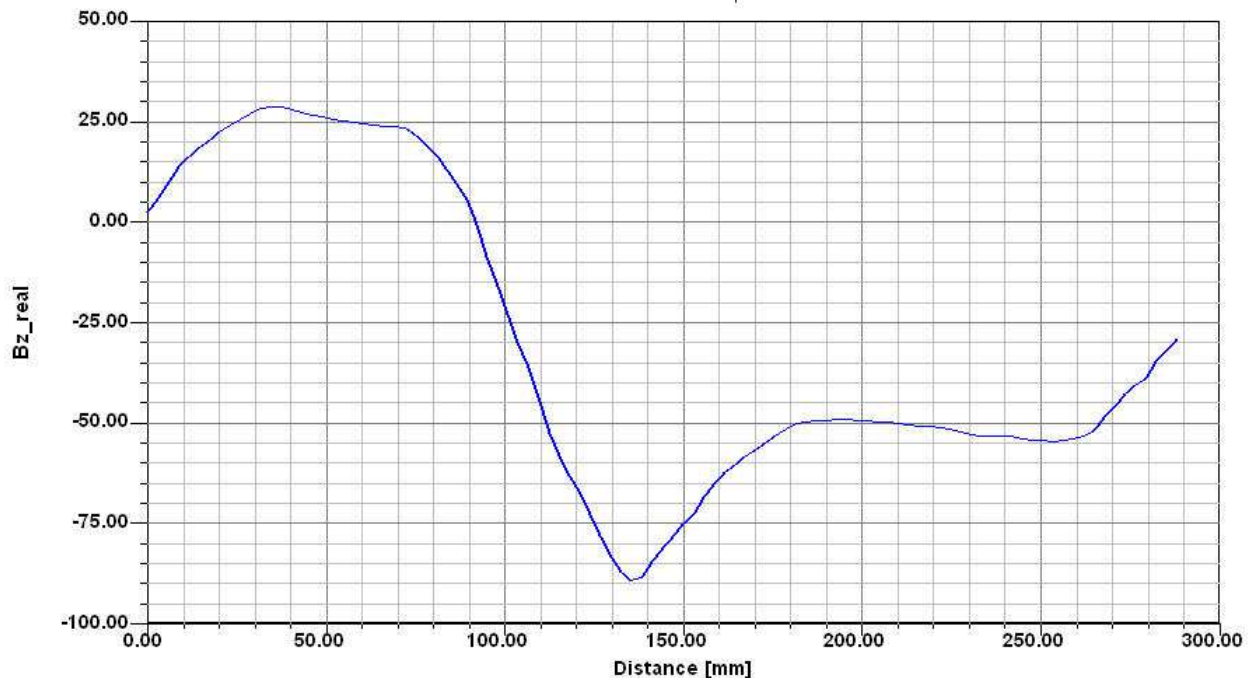
(The flux density will be displayed by default in the units of Tesla [T]. If you wish to see the results in Gaussian units perform steps 6 and 7, otherwise jump to step 8.

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

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

Create Report

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



Field Overlays

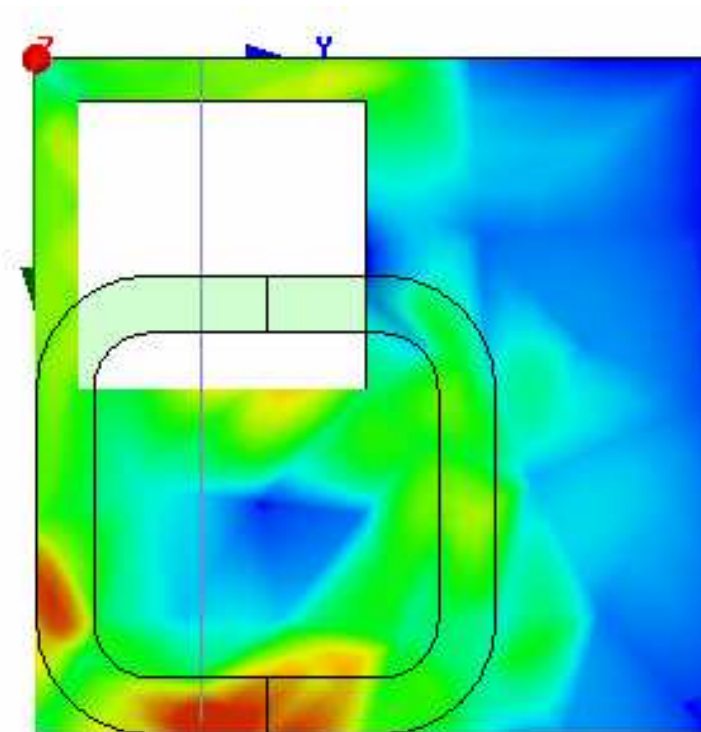
Create Field Overlay

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



Create Field Overlay

To select an object

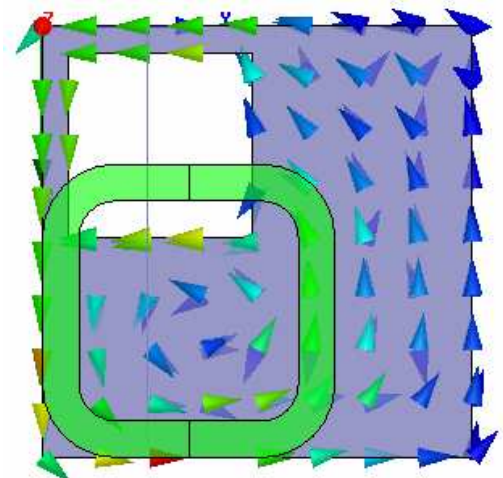
- ▲ Select the menu item *Edit > Select > By Name*
- ▲ Select Object Dialog,
 - ▲ Select the objects named: **stock**
 - ▲ Click the **OK** button

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

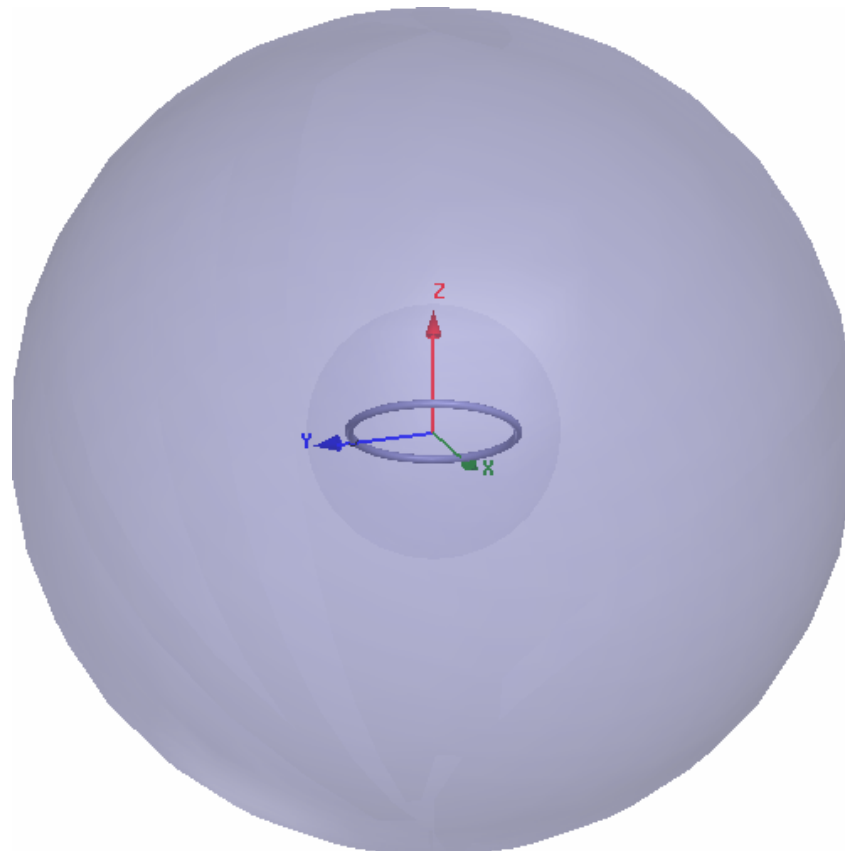


THIS PAGE INTENTIONALLY LEFT BLANK

Example (Eddy Current) -- Radiation Boundary

The implementation and application of a Radiation Boundary in the Eddy Current Solver

-  Radiation Boundaries are used when one wants to simulate the free emission of fields into space. The main distinction of this boundary from all other boundary conditions is that the radiation boundary does not enforce any restrictions on the field behavior. This is used specifically to model unbounded eddy currents and propagating waves without reflection. This boundary has several specific uses which will be discussed in detail in this document.
-  Note that Maxwell3D is a low-frequency simulator, so it is not recommended to model standard antennas and wave propagation with this tool. Maxwell is ideal for near-field simulations for the investigation of interactions between low-frequency inductors and RFID applications.
-  The example that will be used to demonstrate how Radiation Boundaries are implemented is a Magnetic Dipole. The intent of this write up is not how to simulate a dipole antenna, it is rather to demonstrate how Radiation Boundaries are implemented within the Eddy Current solver.





Example (Eddy Current) -- Radiation Boundary

▲ 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: **Torus**

- ▲ **Sphere**

- ▲ Surface Operations: **Section**

- ▲ Boolean Operations: **Separate Bodies**

- ▲ Boundaries/Excitations

- ▲ Current: **Solid**

- ▲ Radiation Boundary

- ▲ Symmetry Boundary

- ▲ Eddy Effects

- ▲ Analysis

- ▲ Eddy Current

- ▲ Results

- ▲ Fields Calculator: **Named Expression**

- ▲ Field Overlays:

- ▲ Field Plots



Example (Eddy Current) -- Radiation Boundary

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:

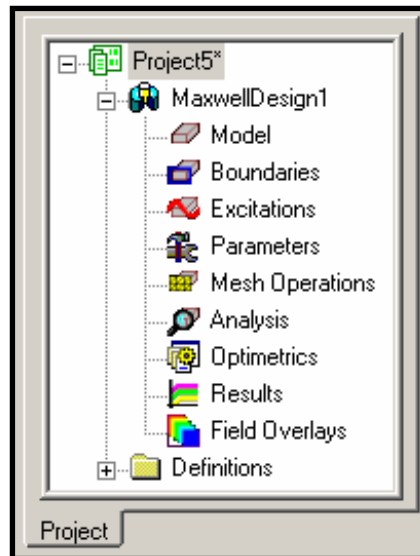
-  **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
 1. Select the menu item ***Tools > Options > Maxwell Options***
 2. Maxwell Options Window:
 1. Click the **General Options** tab
 -  Use Wizards for data entry when creating new boundaries: ☒ **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

Example (Eddy Current) -- Radiation Boundary

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 > Eddy Current**, or right mouse click on MaxwellDesign1 and select **Solution Type ...**

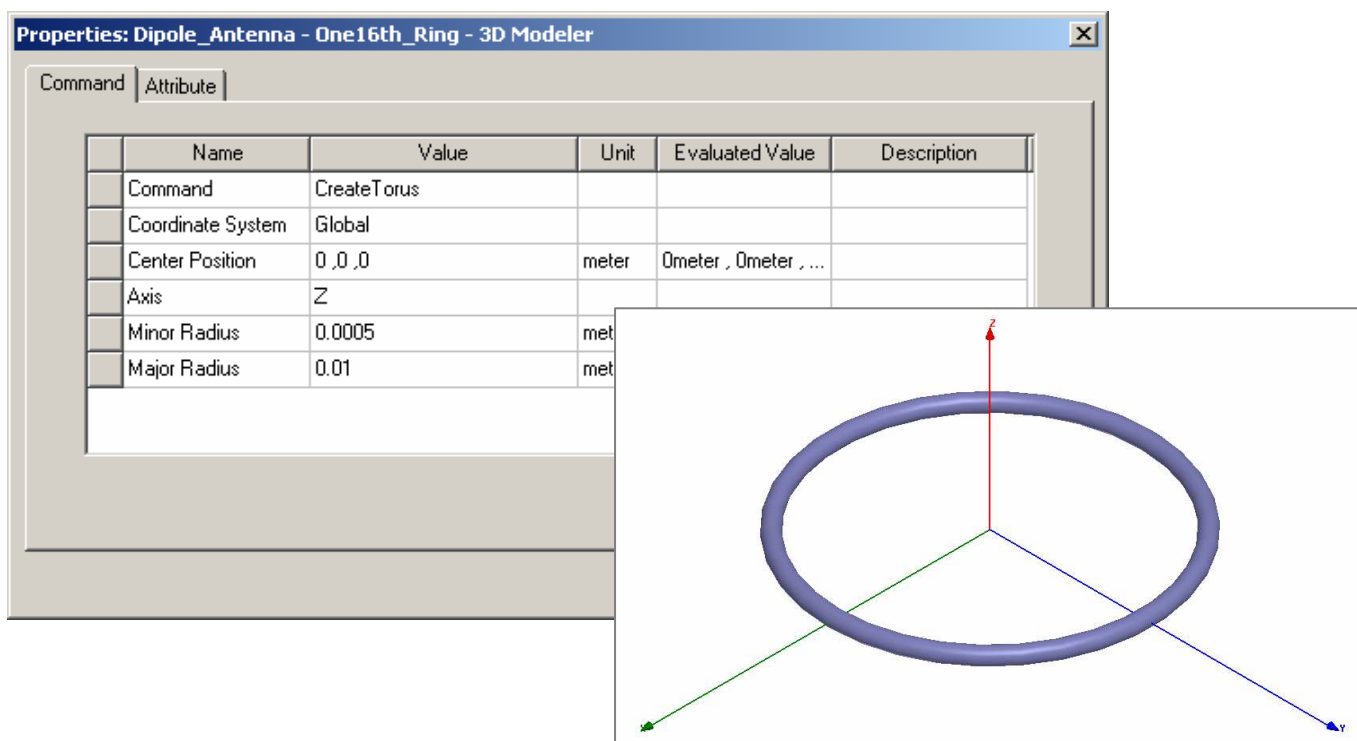
Set Model Units

- Select the menu item **3D Modeler > Units > Select Units: m (meters)**

Example (Eddy Current) -- Radiation Boundary

Create the Dipole Ring

1. Select **Draw > Torus** to start the drawing procedure:
 1. Enter 0, 0, 0 as the center point of the torus
 2. Enter 0.0095, 0, 0 as the inner radius
 3. Enter 0.001, 0, 0 as the outer radius
2. A Properties dialog appears:
 1. In the Command Tab it should be as follows:
 1. Center: 0, 0, 0
 2. Axis: Z
 3. Minor Radius: 0.0005m
 4. Major Radius: 0.01m
 2. In the Attribute Tab make the following changes:
 1. Name: Ring
 2. Material: copper
 1. Select **vacuum** in the Material Value area
 2. Type **copper** to select copper from the Material list
 3. Select **OK**
3. Select **OK**

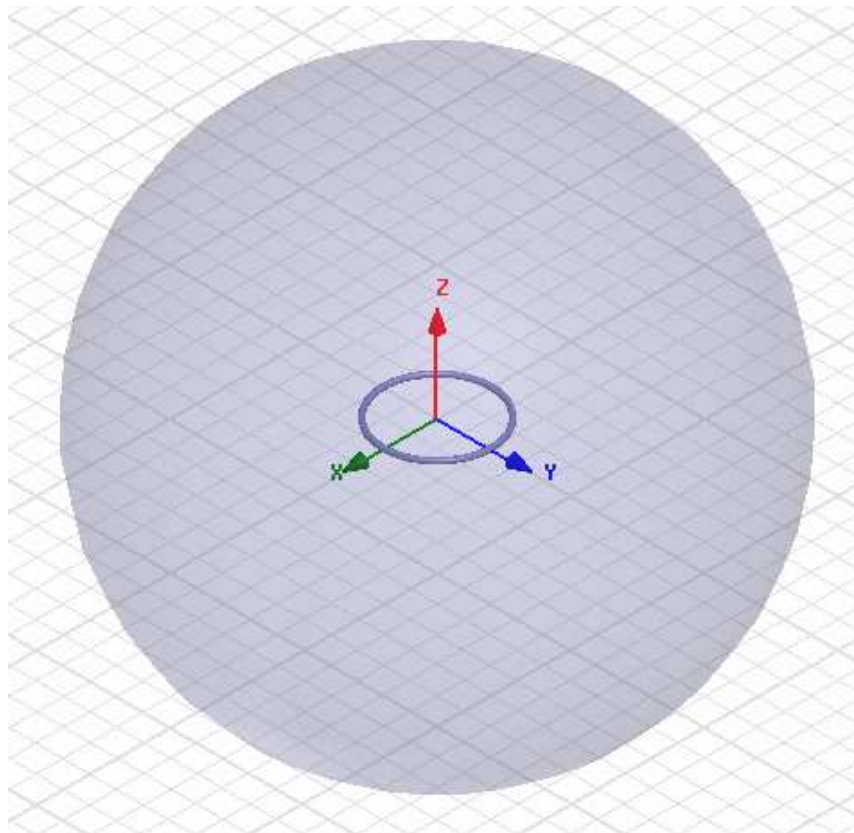


Example (Eddy Current) -- Radiation Boundary



Create the Bounding Region

1. Select **Draw > Sphere** to start the drawing procedure:
 1. Enter 0, 0, 0 as the center point of the sphere
 2. Enter 0.06, 0, 0 as the radius of the sphere
2. A Properties dialog appears:
 1. In the Attribute Tab make the following changes:
 1. Name: Region
 2. Transparent: 0.8
 2. Select **OK**
3. Check that the Material of the **Region** object is **vacuum** by selecting the **Region** object from the history tree. The Material shown in the Properties window should be **vacuum**.



Example (Eddy Current) -- Radiation Boundary

Create a Surface for Current Excitation

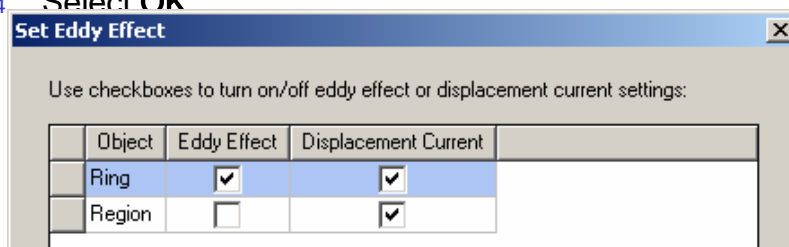
1. Select the **Ring** object from the history tree
2. Select **3D Modeler > Surface > Section...**
 1. Select **YZ** and then **OK**
 2. This creates a new sheet object with two separate parts
3. Select the new sheet object
4. Select **3D Modeler > Boolean > Separate Bodies**
 - ▀ This creates two separate sheet objects from the one previous object
5. Select and Delete the sheet that lies along the negative Y axis
6. Select and Rename the remaining sheet as **Current**

Assign a Current Excitation

1. Select the sheet named **Current**
2. Select **Maxwell > Excitations > Assign > Current ...**
 1. Input 1.414 as the Value of the current. Note that in the eddy current solver, peak current is always specified instead of RMS.
 2. Keep Solid selected
 3. Select **OK**

Assign Eddy Effects

1. Select **Maxwell > Excitations > Set Eddy Effects ...**
 1. Note that conductors will be high-lighted in blue, and eddy effects can only be assigned to conductors.
 2. Keep both Eddy Effect and Displacement Current set for Ring
 3. Set Displacement Current for Region
 4. Select **OK**



Example (Eddy Current) -- Radiation Boundary

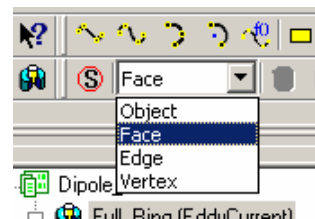
Radiation Boundary Overview

-  The radiation boundary is only available in the Eddy Current solver - it is only applicable for frequency domain solutions.
-  The radiation boundary should be used in conjunction with displacement current in the background object - implying that the fields in the background can propagate to the edge of the model. This means that if displacement currents are simulated in the background, then a radiation boundary should be used on the outside surface of the background.
-  A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space, such as antenna designs. Maxwell3D absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure.
-  A radiation surface does not have to be spherical, but it must be exposed to the background, convex with regard to the radiation source, and located at least a quarter wavelength from the radiating source. In some cases the radiation boundary may be located closer than one-quarter wavelength, such as portions of the radiation boundary where little radiated energy is expected.
-  Remember that Maxwell3D is a near-field simulator, so the dimension of components should be much smaller than the wavelength (i.e. $\lambda/10$)
-  The wavelength at 1.5GHz is 0.2m. Therefore, our 0.06m radius sphere is exactly one quarter wavelength from the ring. In order to make the boundary placement more automated, we can use variables and equations to define the size of the bounding object.
-  **Make the size of the sphere a function of speed of light “c0” and frequency**
 1. Expand **Region** in the history tree
 2. Select **CreateSphere**
 3. Change the radius to **$\lambda/4 + 0.01\text{m}$**
 4. Add Variable **λ** defined by **$c0/\text{frequ}$**
 -  Click the **OK** button
 5. Add Variable **frequ** defined by **1.5GHz**
 -  Click the **OK** button

Example (Eddy Current) -- Radiation Boundary

Assign the Radiation Boundary

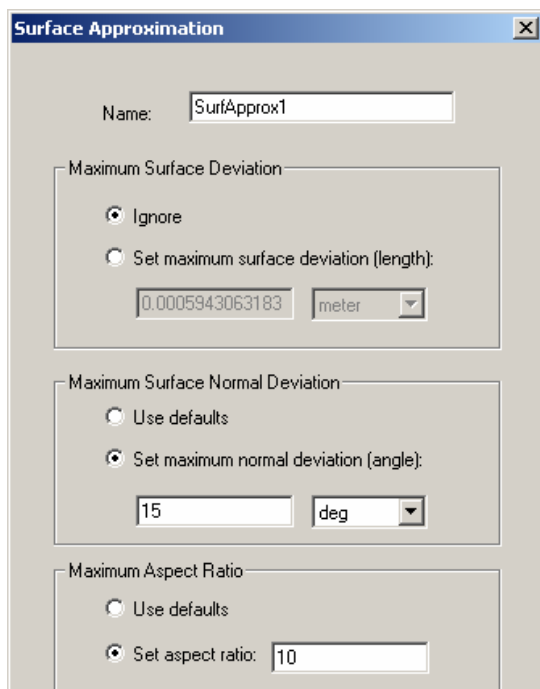
1. Type **f** to enter face selection mode or change the selection mode to **Face** in the toolbar buttons
2. Select the face of the Region
3. Select **Maxwell > Boundaries > Assign > Radiation ...**
 Select OK



- This assigns a radiation boundary to the outside of the sphere, allowing the waves to propagate outside of the sphere. If no radiation boundary is assigned, the interactions of the components may be correct, but the fields near the boundaries will certainly be incorrect. The Poynting vector on the outer surface is a clear indicator of this, which we will explore later.

Assign a Surface Approximation

1. Select the Ring object
2. Select **Maxwell > Mesh Operations > Assign > Surface Approximation ...**
 1. Choose **Set maximum normal deviation (angle)** and keep it at 15 deg
 2. Choose **Set aspect ratio** and keep it at 10
 3. Select OK



- *Maximum surface deviation (length) is the difference between the shortest and longest edge of a triangle on a true surface. This encourages equilateral triangles on the surface.*
- *Maximum Surface Normal Deviation (degrees) which allows you to limit the maximum angle between normal vectors to adjacent tetrahedrons which break up a true surface. This can force more tetrahedron to break up a curved true surface like a cylinder.*
- *Maximum Aspect Ratio is the ratio between the largest and shortest edges of triangles on a given face. A ratio = 1 would attempt to create equilateral triangles.*

Example (Eddy Current) -- Radiation Boundary

Create Outer Surface

-  One important part of creating a geometry is to consider the results needed in the post-processor. We will wish to use the outer surface of the sphere to compute the radiated power and the radiation resistance. Therefore, we will need to create a surface to integrate over that defines precisely the area that we need.

1. Type **f** to enable face-selection mode
2. Select the outer face of the sphere **Region**
3. Select **3D Modeler > Surface > Create Object From Face**
4. A sheet object called **ObjectFromFace1** appears in the history tree (expand the **Sheets** category if this is not visible)
5. Change the name of the new sheet object
 1. Double-click on **ObjectFromFace1** in the history tree
 2. Change the name from **ObjectFromFace1** to **Outside**

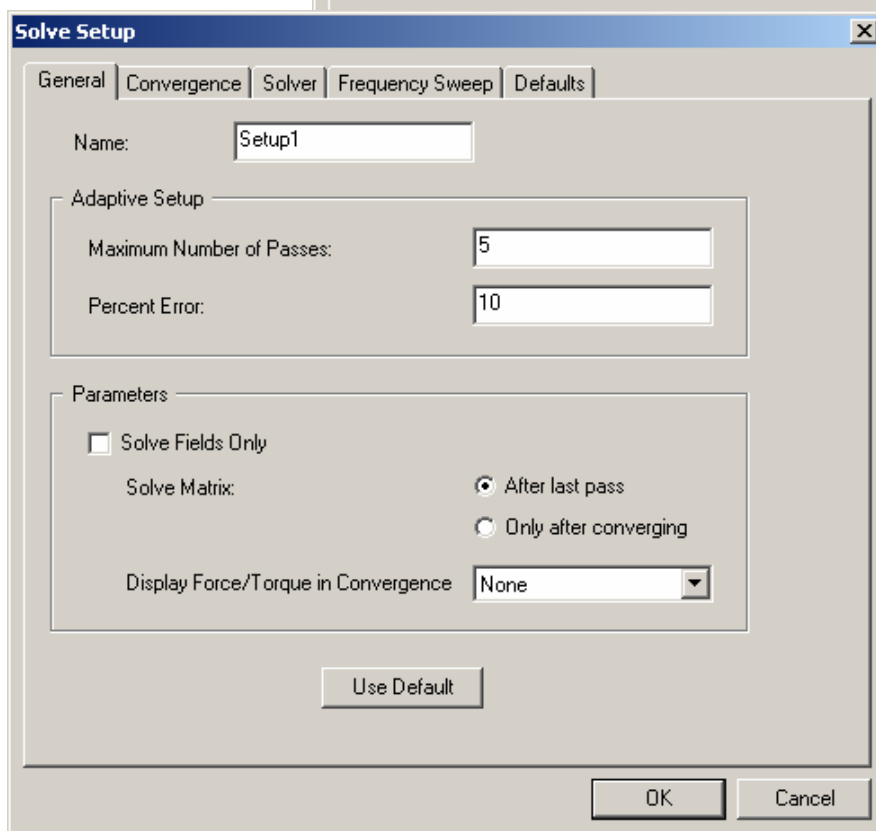
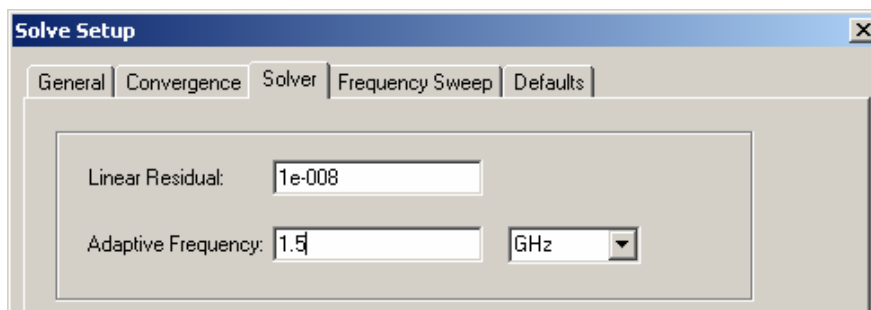
Add Matrix Parameter

1. Select **Maxwell > Parameters > Assign > Matrix ...**
 1. The program will check conduction paths before proceeding.
 2. The Matrix window appears, allowing selection of excitations for use in automatic inductance and impedance calculations.
 3. Check the box next to **Current** to include the ring in the inductance calculations.
 4. Select **OK**

Example (Eddy Current) -- Radiation Boundary

Create a Solution Setup

1. Select **Maxwell > Analysis Setup > Add Solution Setup ...**
 1. In the General tab:
 1. Set Maximum Number of Passes to 5
 2. Set Percent Error to 10
 2. In the Solver tab:
 1. Set Adaptive Frequency to 1.5 GHz
 3. Select **OK**



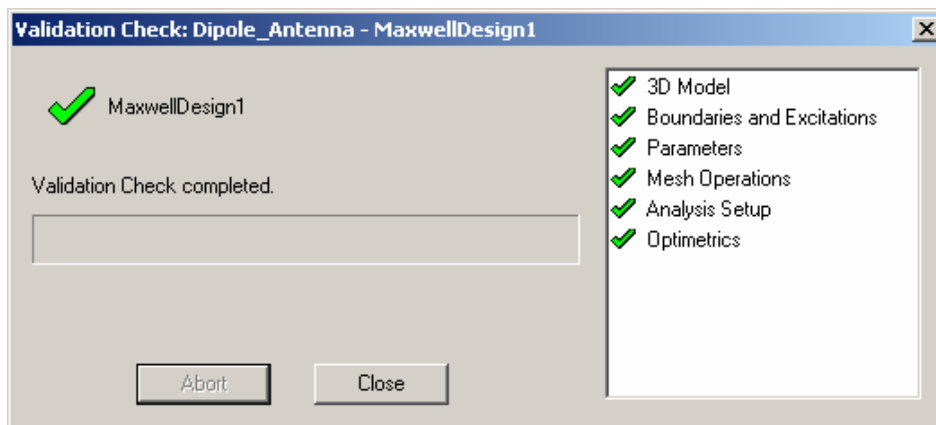
Example (Eddy Current) -- Radiation Boundary

Save the Project

1. Select the menu item **File > Save As**
2. From the Save As window, type in **Dipole_Antenna**
3. Click on the **Save** button

Check the Validity of the Model

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

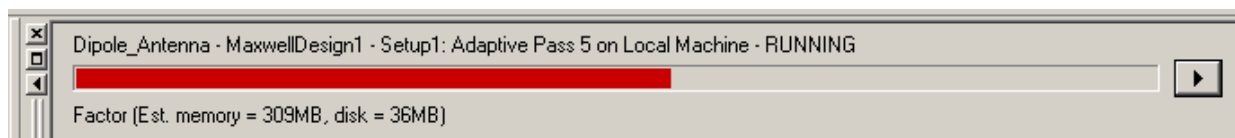


- The problem won't solve unless each object has a check mark.

Analyze

- NOTE: This problem will solve, but it probably will not converge within 5 passes. This will be discussed more later. To proceed you do not need a fully converged simulation - a rough solution will work fine (only 3 or 4 passes are necessary).

- Select the menu item **Maxwell > Analyze**, or click on the  icon
- This should take about 10-20 minutes to solve



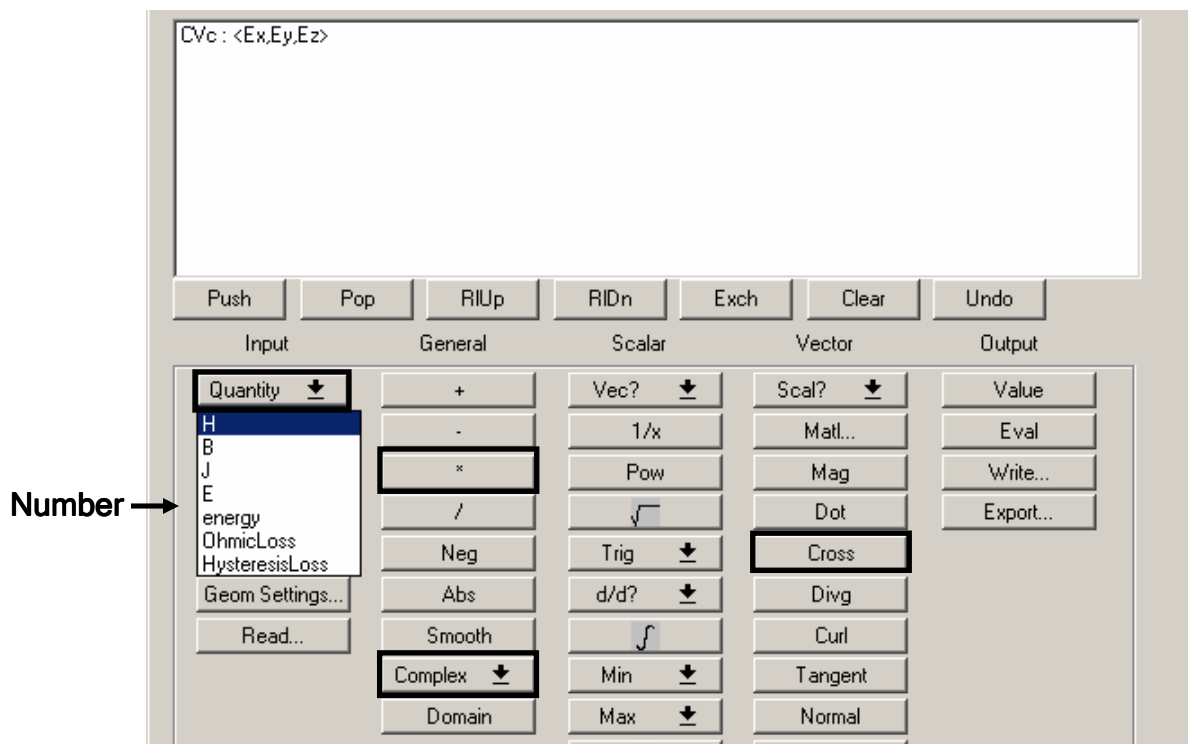
Example (Eddy Current) -- Radiation Boundary

Create Named Calculator Expression

- We will create a Poynting vector expression in the calculator that is accessible as a field plot and for power calculations. The expression will be:

$$\vec{P} = \frac{1}{2} \text{Re}\{\vec{E} \times \vec{H}^*\}$$

- Open the calculator by selecting **Maxwell > Fields > Calculator ...**
 - The fields calculator window appears.
 - Select **Quantity > E** to add the complex E field to the stack
 - Select **Quantity > H** to add the complex H field to the stack
 - Select **Complex > Conj** to obtain the complex conjugate of H
 - Select **Cross** to obtain the cross product of E and H*
 - Select **Complex > Real** to obtain the real part of the complex field
 - Select **Number...** and enter **0.5** for the value. Select **OK**
 - Select ***** to multiply the vector field by 0.5

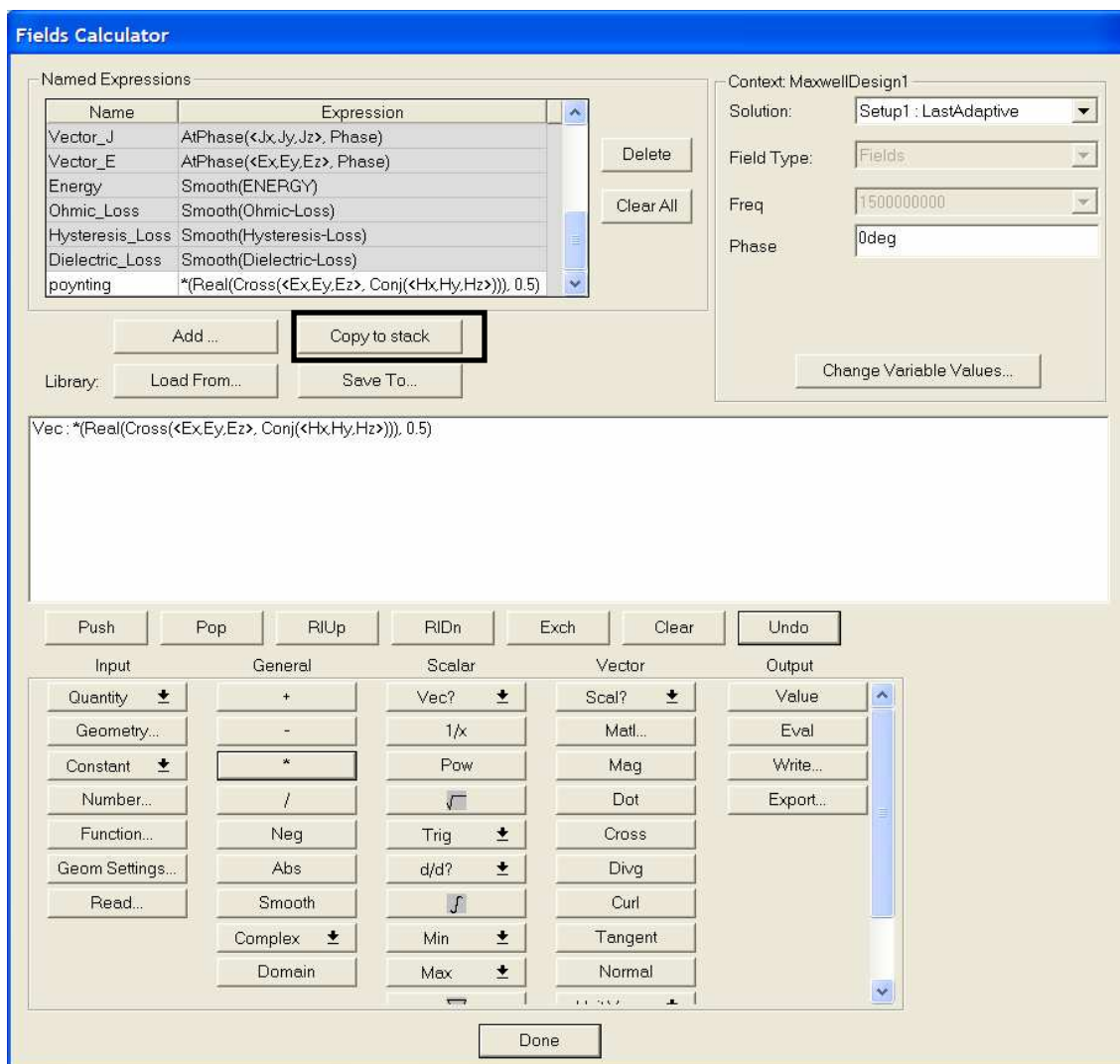


Example (Eddy Current) -- Radiation Boundary

Create Named Calculator Expression (Continued)

1. Select **Add...** (above the calculator stack) to add the top entry of the stack to the list of named expressions.

▀ Type **Poynting** and select **OK**



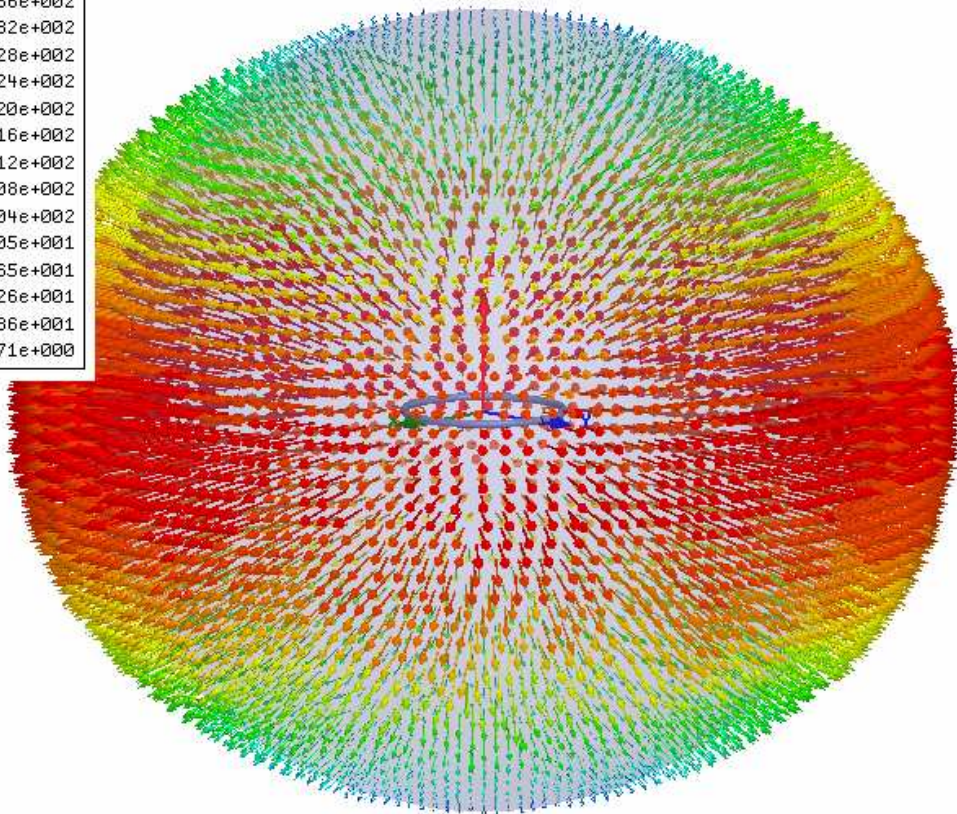
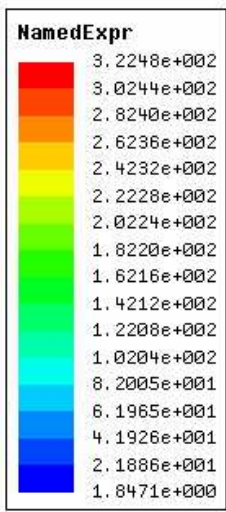
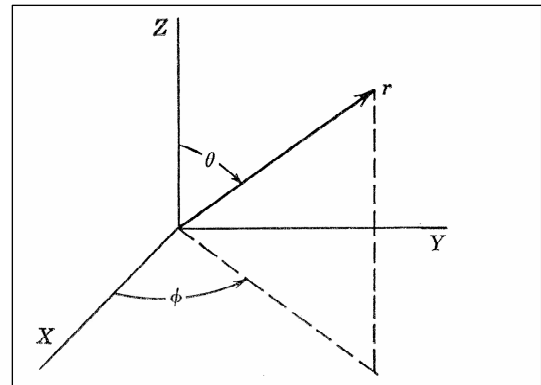
2. Select **Done** to exit the calculator

Example (Eddy Current) -- Radiation Boundary

Plot the Power flow from the Dipole

1. Type **f** to enter face selection mode
2. Select the outside face of the Region
3. Select **Maxwell > Fields > Fields > Named Expression ...**
 1. Select Poynting from the list
 2. Select OK
4. Select Done

-  A plot similar to the following should appear, with strong fields near $\theta=90^\circ$.



Example (Eddy Current) -- Radiation Boundary

Calculate the Radiation Resistance

- The radiation resistance of an antenna is determined by the following formula:

$$R_r = \frac{P_{av}}{I_{rms}^2}$$

- Where the radiated Power is defined as the integral of the Poynting vector over the surface S (outer surface of the region).

$$P_{av} = \frac{1}{2} \oint_S \text{Re}(\vec{E} \times \vec{H}^*) dS$$

- Since the rms current is 1 A (the peak current is defined as 1.414 A), the radiation resistance is simply the result of this integral.

- The sequence of computing the resistance is as follows:

- Select **Maxwell > Fields > Calculator** to enter the fields calculator
- Select **Poynting** from the list of Named Expressions
- Select **Copy to stack** to add this quantity to the stack
- Select **Geometry...**
 - Select **Surface**, then **Outside**
 - Select **OK** to add the geometry to the stack
- Select **Normal** to take the normal component of the Poynting vector
- Select **∫** (integrate)
- Select **Eval**

- The result should be about 2.2 W or 2.2 ohms.

- According to the literature, the radiation resistance of a Magnetic Dipole with no thickness is:

$$R_{rad\_air} = 20\pi^2 \beta^4 a^4$$

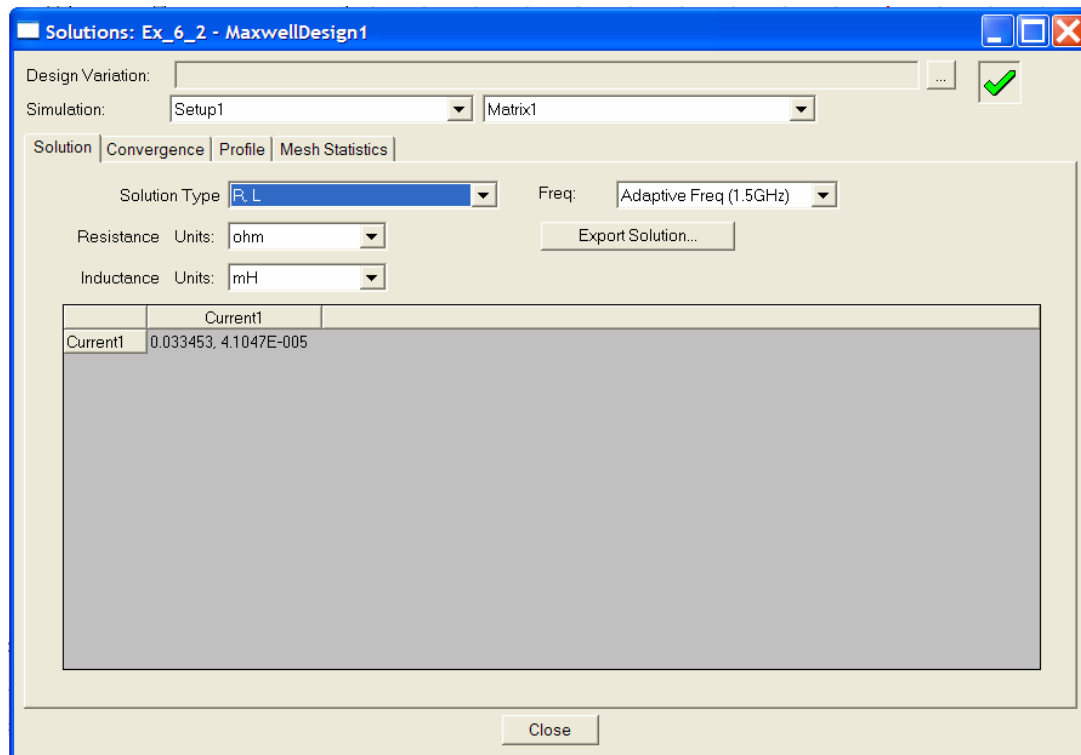
where β is the wave number and a is the radius of the loop

- This equation results in 1.923 ohms for a radius of 0.01m and a wavelength of 0.2m.

Example (Eddy Current) -- Radiation Boundary

View the Impedance Matrix

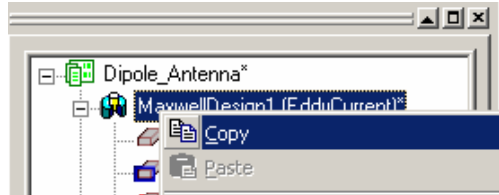
1. Now, view the results of the impedance matrix:
 1. Select **Maxwell > Results > Solution Data**
 2. Change the solution type to R,L to see resistance and inductance.



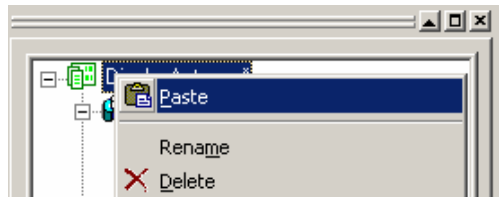
Example (Eddy Current) -- Radiation Boundary

PART 2: Create a 1/16th Section Model

1. Copy the design by right clicking on MaxwellDesign1 and selecting **Copy**

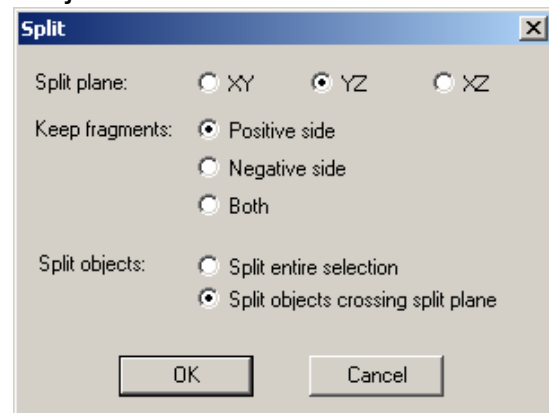


2. Paste the design by right clicking on Dipole_Antenna and selecting **Paste**



3. Rename MaxwellDesign1 to Full_Ring
 1. Right click on MaxwellDesign1 and select **Rename**
4. Rename MaxwellDesign2 to One16th_Ring
5. Double-click on One16th_Ring in order to make this the active design
6. Select all objects by typing **ctrl + a** while in the 3D Modeler
7. Select **3D Modeler > Boolean > Split ...**
 1. Keep **XY**, **Positive Side**, and **Split entire selection** selected
 2. Select **OK**
8. With all objects still selected, **Split** all objects again.

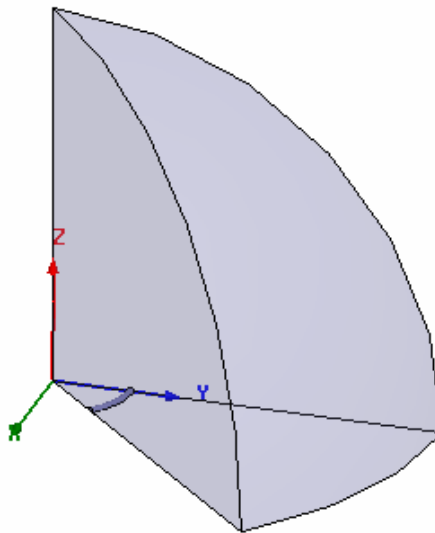
1. Change split plane to **YZ**, and split objects to **Split objects crossing split plane**
2. Keep **Positive Side** selected
3. Select **OK**



Example (Eddy Current) -- Radiation Boundary

Create a 1/16th Section Model (Continued)

9. Create a rotated Coordinate system:
 1. Select **3D Modeler > Coordinate System > Create > Relative CS > Rotated**
 1. Select 0.01, 0.01, 0 for the x-axis
 2. Select -0.01, 0.01, 0 for the y-axis
10. Select all objects again by typing **ctrl + a**
11. Select **3D Modeler > Boolean > Split ...**
 1. Choose **XZ** for the split plane, **Positive side** for the fragments, and **Split objects crossing split plane** for the split objects
 2. Select **OK**



12. Now the geometry is all set and only a couple changes are required to the boundaries/sources.

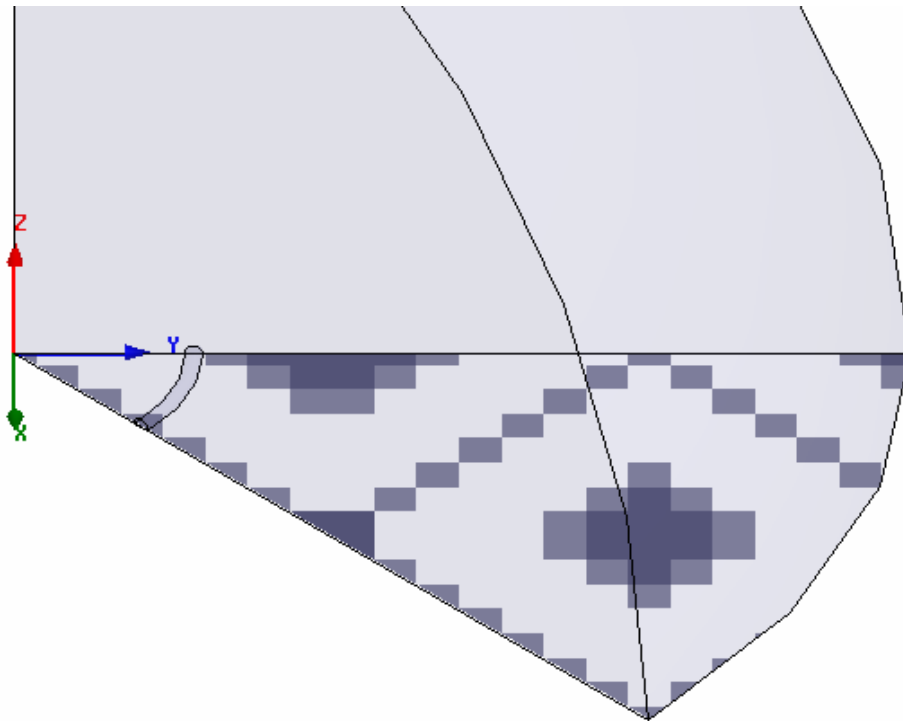
Example (Eddy Current) -- Radiation Boundary

Create a Symmetry Boundary

-  Only one symmetry boundary is required even though three surfaces define distinct symmetries. The two surfaces perpendicular to the ring have a symmetry that is accounted for by the default boundary condition of Zero Tangential H Field - i.e. this is a special case of symmetry, where the symmetric tangential fields are zero. The surface that lies in the plane of the ring has a Flux Normal symmetry, so we need to define this separately.

- 1. Select the face of the region that lies in the XY plane (type **f** to select faces)
- 2. Select **Maxwell > Boundaries > Assign > Symmetry ...**
- 3. Choose **Even (Flux Normal)** to define this as a Flux Normal symmetry
- 4. Select **OK**

-  The squared pattern (seen below) appears when the boundary is selected, and indicates on which surfaces the boundary is defined.

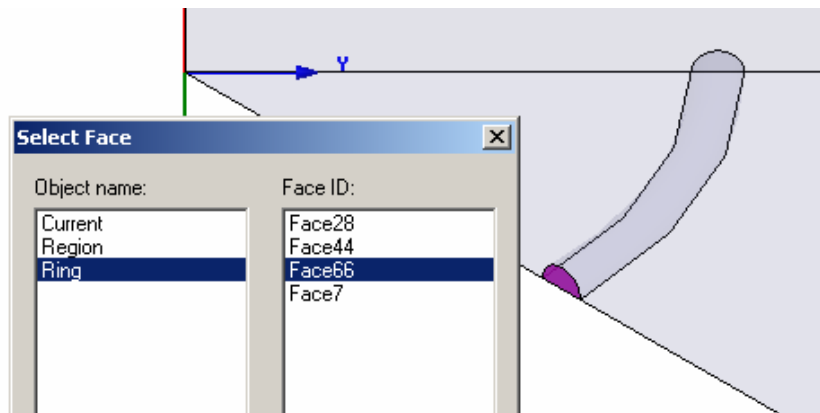


Example (Eddy Current) -- Radiation Boundary

Adjust the Current Excitation

- ▲ In the last simulation, the current excitation was internal to the simulation. That is, the conduction path formed a loop and the current excitation defined the current throughout the entire loop. The one current excitation that remains in this one sixteenth model lies on the edge of the simulation, and the conduction path does not form a loop. Therefore, this current excitation is considered an external excitation, and it defines how the current flows out of the model. We need a second external current excitation to define how the current flows into the model. Also, the conduction path was split in half, so half the current is needed in the excitations to define the same current density - this will be dealt with now.

1. Go to **Edit > Select >** and make sure that **Faces** is selected (has a checkmark)
2. Select **Edit > Select > By Name ...**
 1. Select **Ring** and the half-circle face that does not contain the Current sheet
 2. Select **OK**

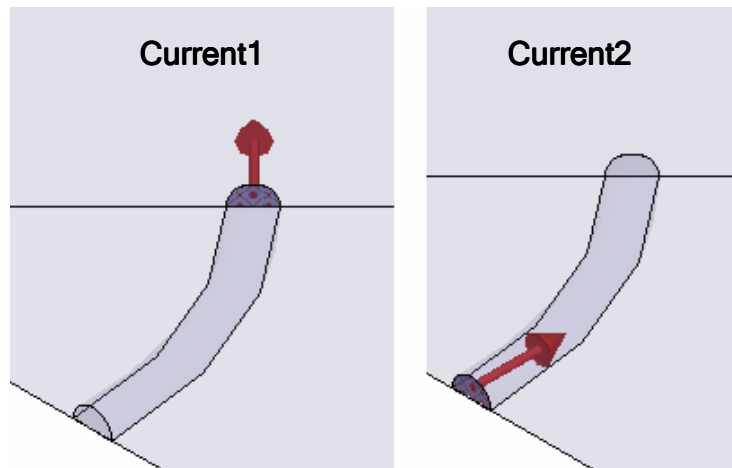


3. Select **3D Modeler > Surface > Create Object From Face**
4. Rename the object by double clicking **ObjectFromFace1** and entering **Current2** as the new name. Then select **OK**.
5. Select the sheet **Current2**
6. Select **Maxwell > Excitations > Assign > Current ...**
 1. Enter 0.707 A for the value
 2. Select **OK**

Example (Eddy Current) -- Radiation Boundary

Adjust the Current Excitation (Continued)

6. In the Project Manager, expand Excitations and double-click **Current1**
 1. Change the value from 1.414 A to 0.707 A
 2. Select **OK**
7. In the Project Manager, select Current1 and note the direction of the arrow
8. Select Current2 and note the direction of this arrow
9. These arrows should be on opposite ends of the copper segment, and they should be in the same direction (choose Swap Direction in the excitation definition window if both current excitations are not in the same direction).



Adjust the Matrix Parameter

1. In the Project Manager, expand the **Parameters** category in the **One16th_Ring** design
2. Double-click on **Matrix1**
 1. Check the box next to **Current2**
 2. Select **OK**

- The excitations available to the Matrix are only those excitations that define the current entering the conduction path. The **Current1** excitation defines current exiting the conduction path, and so is not available to the matrix. **Current1** can be made available by swapping the direction of both current excitations. This switches which excitation defines the current entering and exiting the conduction path.

Example (Eddy Current) -- Radiation Boundary

Save the Project

- Select the menu item **File > Save**

Check the Validity of the Model

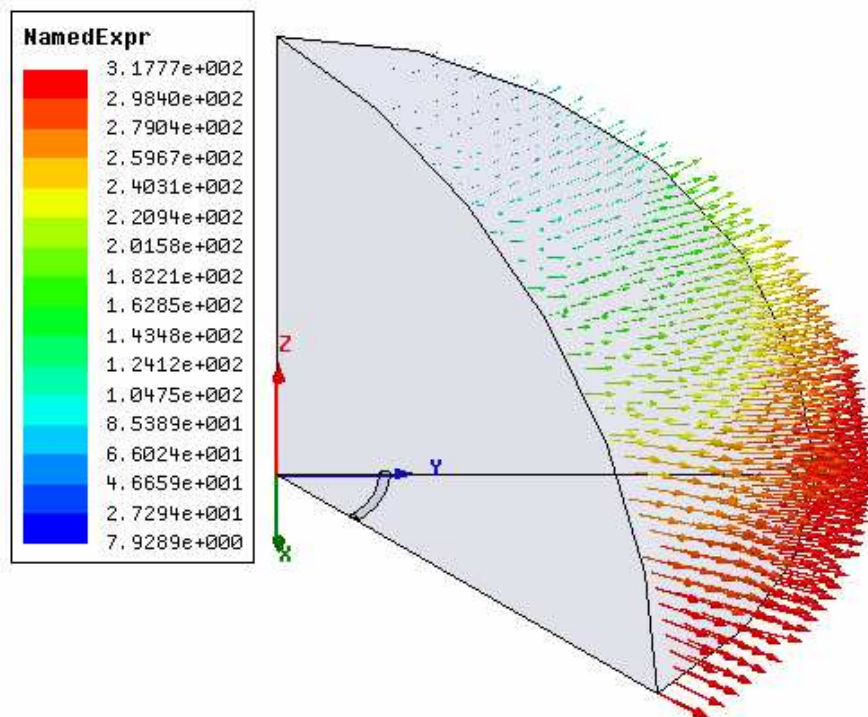
- Select the menu item **Maxwell > Validation Check**, or click on the  icon
- The problem won't solve unless each object has a check mark.

Analyze

- Select the menu item **Maxwell > Analyze**, or click on the  icon
- This should take 1 minute or less to solve, and it may or may not converge in 5 passes.

Plot Power Flow

- In the Project Manager, expand Field Overlays and NamedExpr to find Poynting1
- Double click on Poynting1 to refresh the plot
- The plot should look similar to the following



Example (Eddy Current) -- Radiation Boundary

Calculate the Radiation Resistance

1. Select **Maxwell > Fields > Calculator** to enter the fields calculator
2. Select **Poynting** from the list of Named Expressions
3. Select **Copy to stack** to add this quantity to the stack
4. Select **Geometry...**
 1. Select **Surface**, then **Outside**
 2. Select **OK** to add the geometry to the stack
5. Select **Normal** to take the normal component of the Poynting vector
6. Select  (integrate)
7. Select Eval

-  The result should be about 0.138 W or 0.138 ohms - which is one-sixteenth of 2.2.

Find the Impedance of this Antenna

-  The impedance (both resistive and reactive) of the antenna (as seen from the current source) is computed automatically through the inclusion of the excitation in the matrix calculation.
-  These results computed at the end of the simulation and are reported in the solution data.
-  Select **Maxwell > Results > Solution Data ...**
-  The results are reported both as $\text{Re}\{Z\}$, $\text{Im}\{Z\}$, and as R, L. The Inductive Coupling Coefficient is related to mutual inductance with multiple conductors, so this is not relevant now.



Chapter 7.0 - Transient Examples

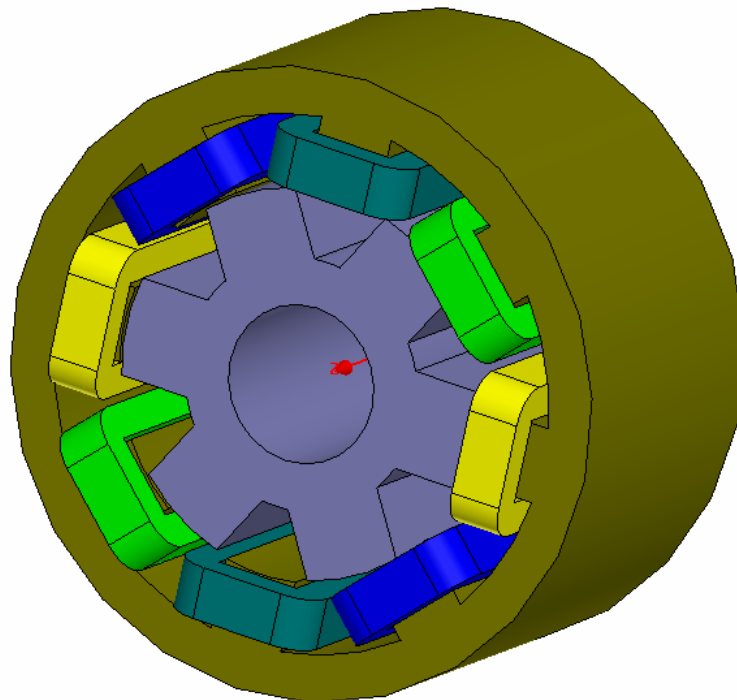
-  7.1 - Switched Reluctance Motor (Stranded Conductors)
-  7.2 - Rotational Motion
-  7.3 - Translational Motion

THIS PAGE INTENTIONALLY LEFT BLANK

Example (Transient) - Stranded Conductors

Stranded Conductors

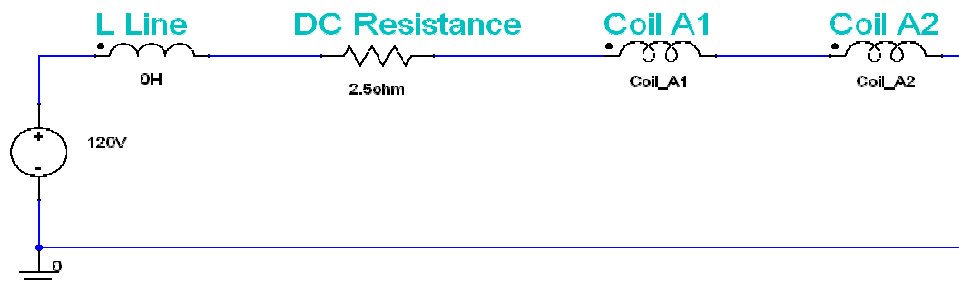
- ▲ This example is intended to show you how to create and analyze a transient problem on a Switched Reluctance Motor geometry using the Transient solver in the [Ansoft Maxwell 3D Design Environment](#).
- ▲ Within the Maxwell 3D Design Environment, solid coils can be modeled as Stranded Conductors. 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.
- ▲ **Magnetostatic:** See **Example 5.3** for setup instructions for this design.



Example (Transient) - Stranded Conductors

Theory - Transient Solver

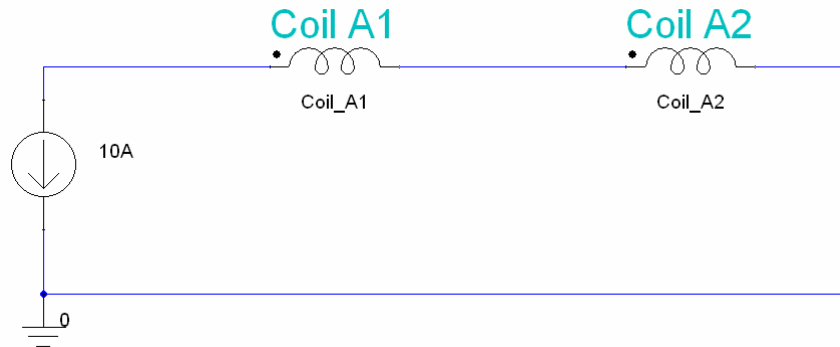
- ▲ When creating Windings in the Transient solver, it is assumed that all of the coils used to make up that winding are connected in series.
- ▲ When creating a Winding and using voltage sources, the **Winding Panel** asks for the **Initial Current**, **Resistance**, **Inductance**, and **Voltage**.
 - ▲ **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:



Example (Transient) - Stranded Conductors

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**.

Example (Transient) - Stranded Conductors

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**

Example (Transient) - Stranded Conductors

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:

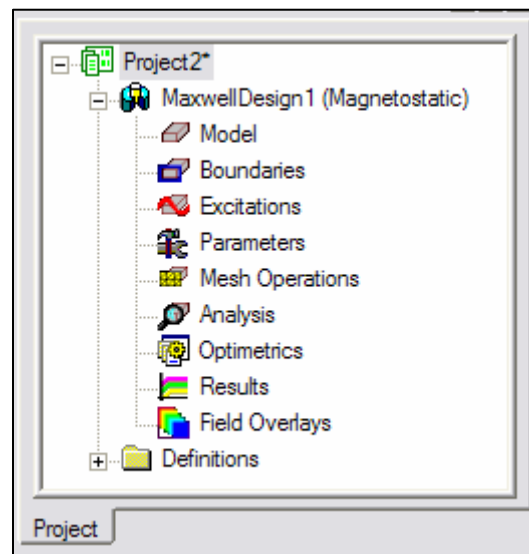
-  **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
 1. Select the menu item ***Tools > Options > Maxwell Options***
 2. Maxwell Options Window:
 1. Click the **General Options** tab
 -  Use Wizards for data entry when creating new boundaries: ☒ **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

Example (Transient) - Stranded Conductors

Opening a New Project

To open a new project:

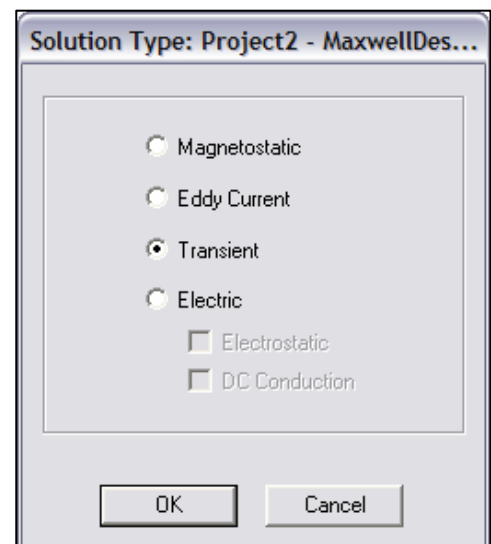
- ▲ In an Ansoft Maxwell window, click the  On the Standard toolbar, or select the menu item **File > New**.
- ▲ From the **Project** menu, select **Insert Maxwell Design**.



Set Solution Type

To set the solution type:

- ▲ Select the menu item **Maxwell > Solution Type**
- ▲ Solution Type Window:
 - ▲ Choose **Transient**
 - ▲ Click the **OK** button



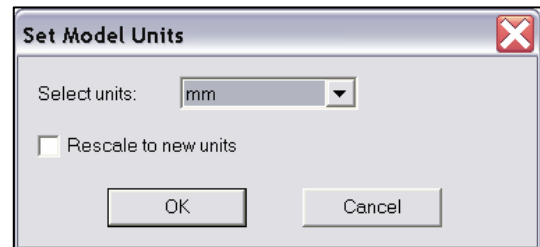
Example (Transient) - Stranded Conductors

Creating the 3D Model

Set Model Units

To set the units:

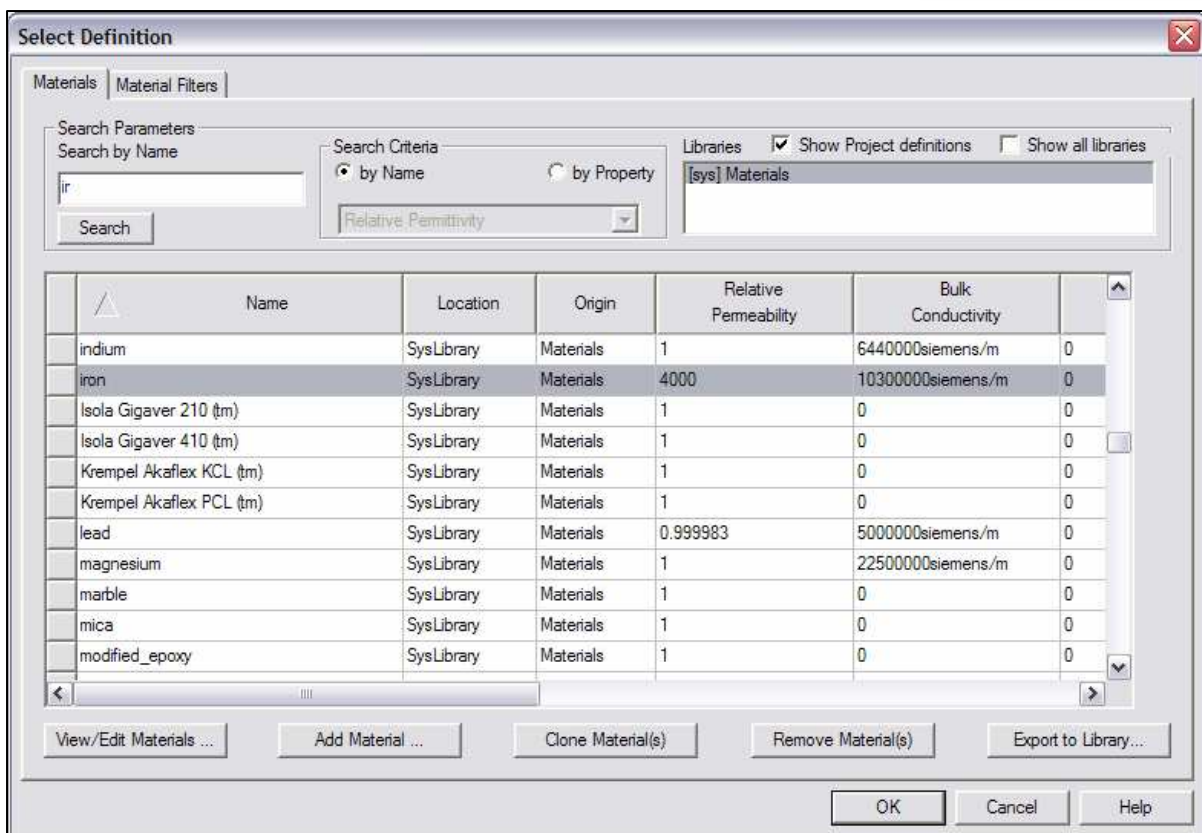
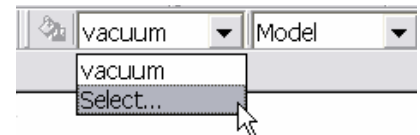
1. Select the menu item **3D Modeler > Units**
2. Set Model Units:
 1. Select Units: **mm**
 2. Click the **OK** button



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



Example (Transient) - Stranded Conductors

Create Rotor

To create the rotor:

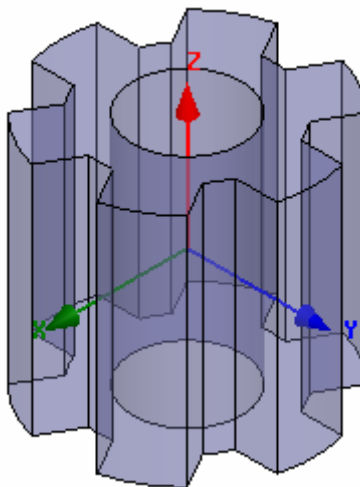
1. Select the menu item **Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore**
2. From the **Create User Defined Primitive** dialog
 1. For the value of **DiaGap**, type: **70**, Click the **Tab** key to accept
 2. For the value of **DiaYoke**, type: **30**, Click the **Tab** key to accept
 3. 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**.



Example (Transient) - Stranded Conductors

Create Stator and Coils:

To create the Stator and Coils

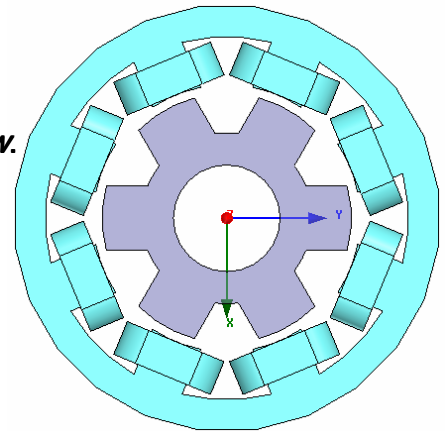
1. Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
2. From the **Create User Defined Primitive** dialog
 1. For the value of **DiaGap**, type: **75**, Click the **Tab** key to accept
 2. For the value of **DiaYoke**, type: **120**, Click the **Tab** key to accept
 3. 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*

Example (Transient) - Stranded Conductors

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

Example (Transient) - Stranded Conductors

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

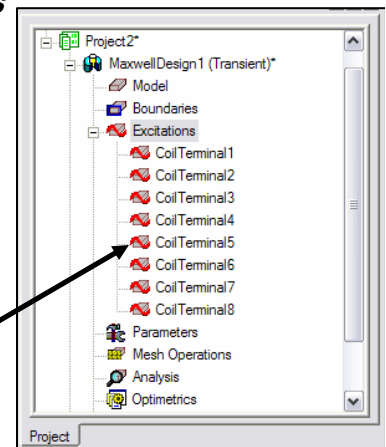
To select objects

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



Example (Transient) - Stranded Conductors

▲ Add Windings

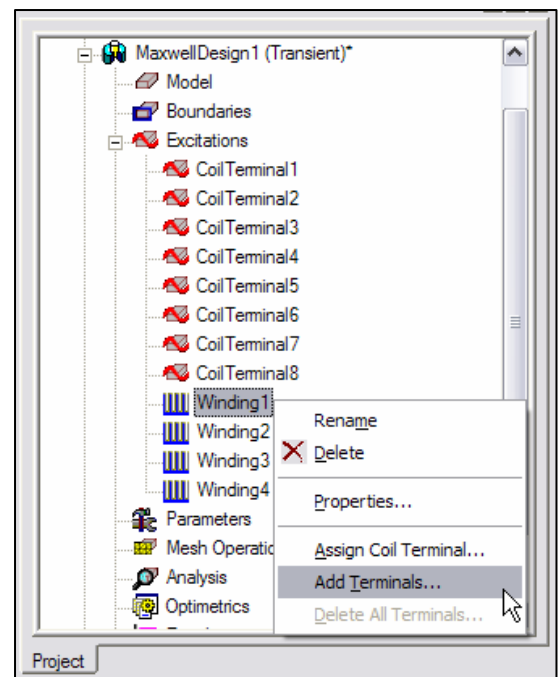
▲ To add windings

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**
 - ▲ **Winding4**
 - ▲ **CoilTerminal4**
 - ▲ **CoilTerminal8**



Example (Transient) - Stranded Conductors

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:

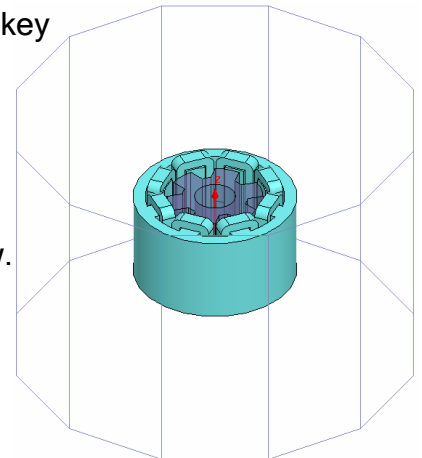
1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value of Name** type: **Region**
3. **Display Wireframe:** ☒ **Checked**
4. Click the **OK** button

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.**



Example (Transient) - Stranded Conductors

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 object)
 5. Click the **OK** button

Example (Transient) - Stranded Conductors

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

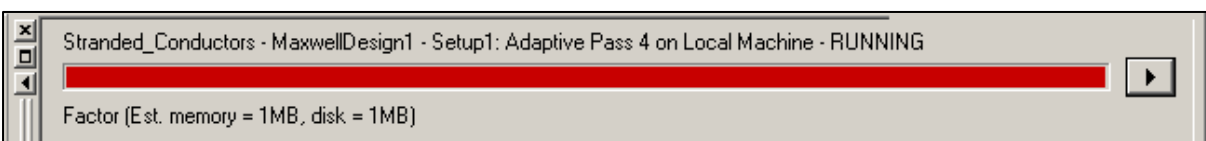
To validate the model:

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**

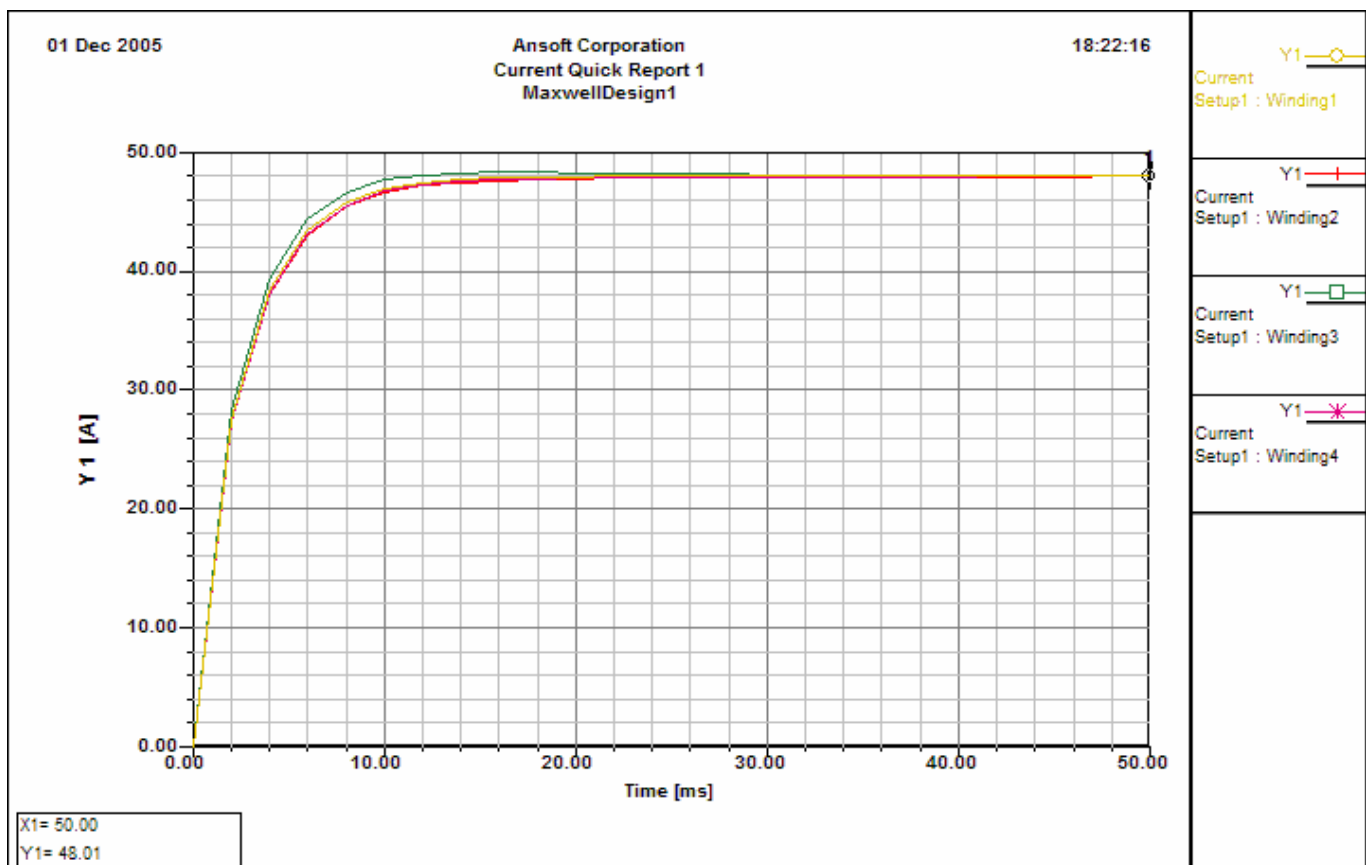


Example (Transient) - Stranded Conductors

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



Example (Transient) - Stranded Conductors

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 $\int$ (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>

Sc1 : ScalarZ[<Jx,Jy,Jz>]

Srf : Surface[Section1]
Sc1 : ScalarZ[<Jx,Jy,Jz>]

Sc1 : -48.0069242512792
Sc1 : /(Integrate(Surface[Section1], ScalarZ[<Jx,Jy,Jz>]), 150)

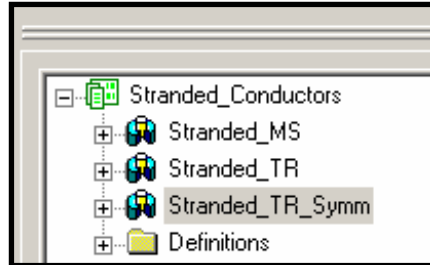
Example (Transient) - Stranded Conductors

Create Quarter Symmetry Geometry

- So far we have been working with full geometry. Often it is useful to use symmetry in order to reduce the problem size and thus decrease the solution time. In this section, we'll show how to create the symmetric model and its impact on stranded conductors.

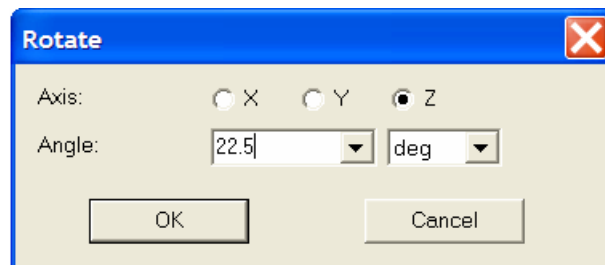
Copy the Existing Design

- Select the previous design, **Stranded_TR**, and right mouse click and select **Copy**.
- Click on project name, **Stranded_Conductors**, and right mouse click and select **Paste**; this will create a second design called **Stranded_TR1**. Change its name to **Stranded_TR_Symm** for Transient with symmetry. Double click on this design to make it active.



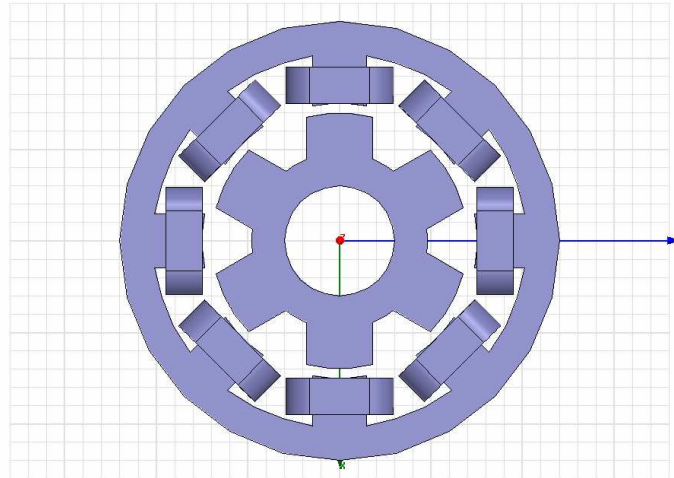
Rotate the Geometry to Create Quarter Symmetry

- Before splitting the model to create a 1/4 model, all of the objects need to be rotated. Make all of the objects visible and then select them. Next choose the menu item **Edit > Arrange > Rotate** and choose 22.5 degrees about Z.



- Next select only the rotor and choose the menu item **Edit > Arrange > Rotate** and choose 7.5 degrees about Z. Now the objects are rotated so that they can be properly cut on the symmetry planes.

Example (Transient) - Stranded Conductors

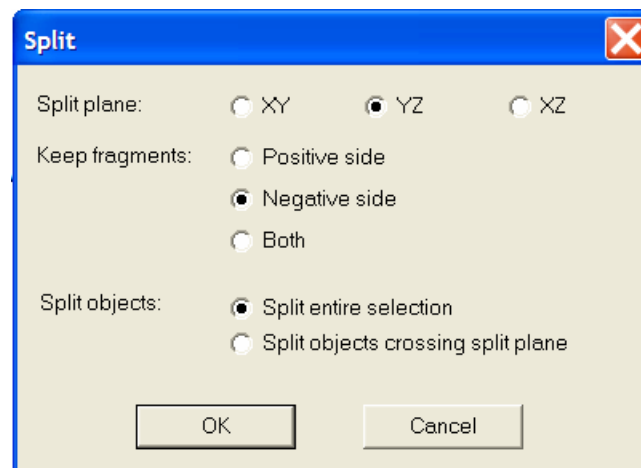


▲ Modify the Geometry to Create Quarter Symmetry

- ▲ Make all of the objects visible and then select them. Next select the menu item **3D Modeler > Boolean > Split** and choose

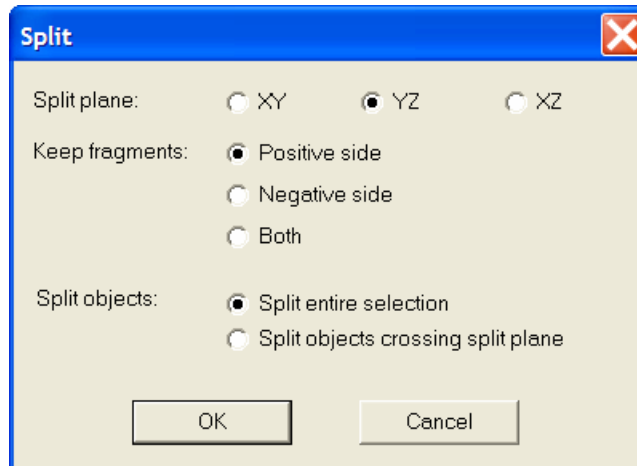
- ▲ Split Plane > YZ
- ▲ Keep Fragments > Positive Side
- ▲ Split Objects > Split Entire Selection

Note: By selecting Split Entire Selection instead of Split Objects Crossing Split Plane, all objects in negative X direction are automatically deleted, even if they are not cut.

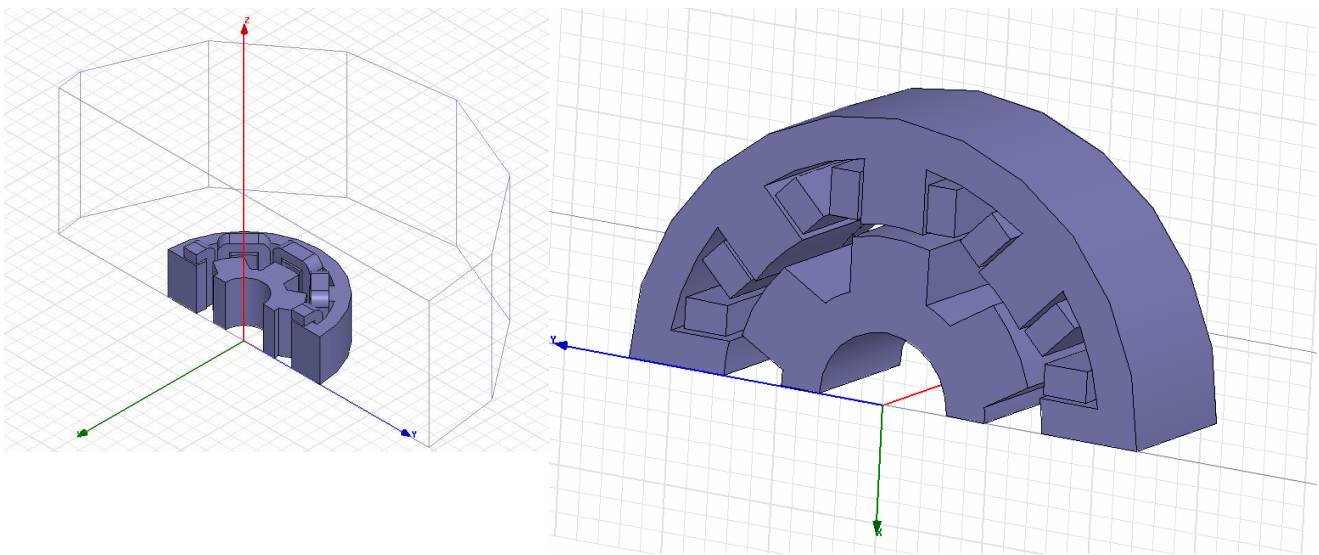


Example (Transient) - Stranded Conductors

- ▲ Select all of the objects and then select the menu item **3D Modeler > Boolean > Split** and choose:
 - ▲ Split Plane > XY
 - ▲ Keep Fragments > Positive Side
 - ▲ Split Objects > Split Entire Selection



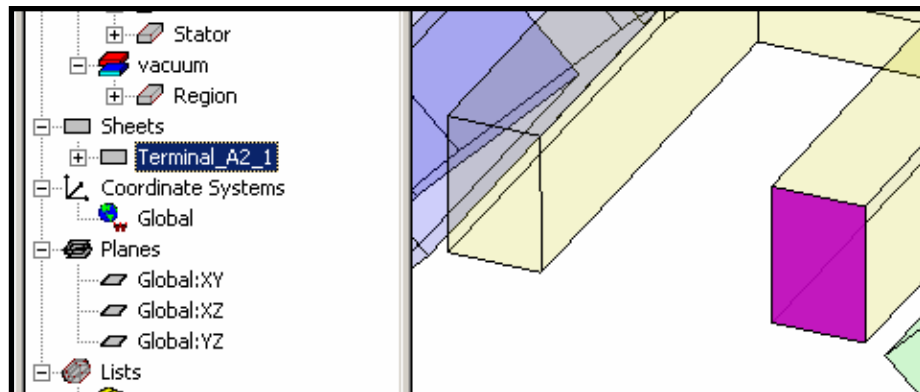
- ▲ The resulting geometry should look like this. Select View > Rotate to spin the view so that the ends of the coils are exposed and the faces can be selected. Select View > Zoom to enlarge the view.



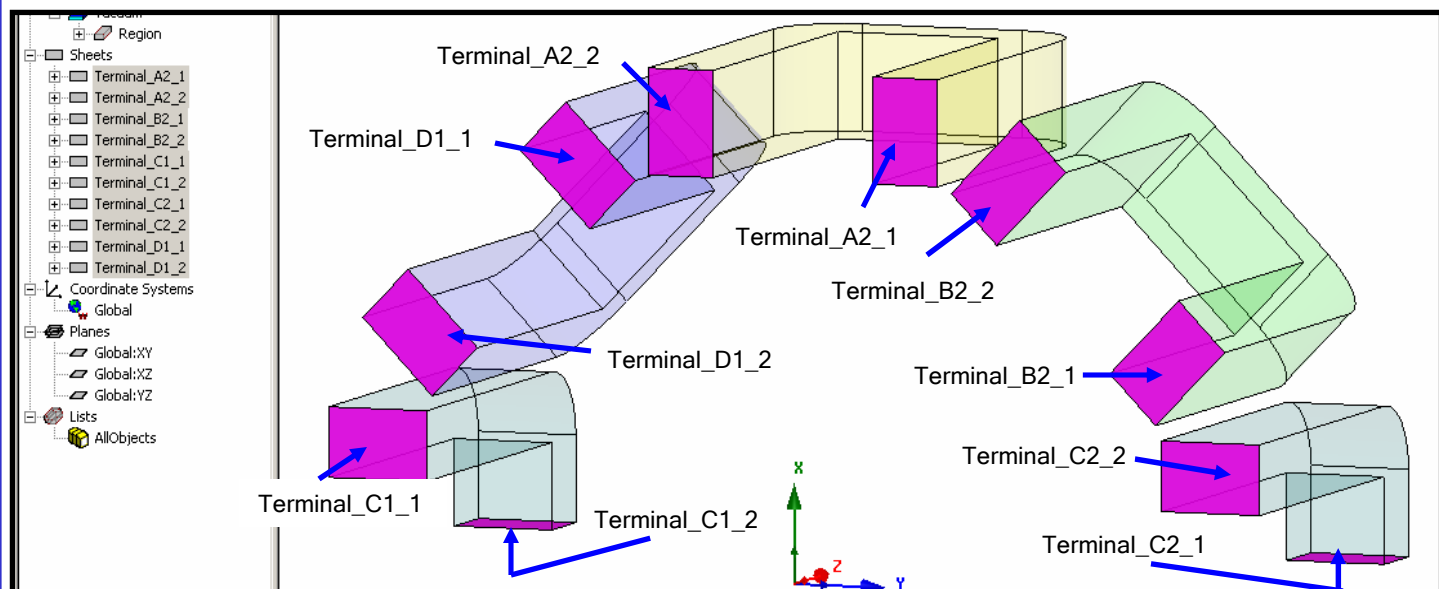
Example (Transient) - Stranded Conductors

Redefining the Terminals

- ▲ When the entire coil is not being modeled and the coil cuts the surface of the solution boundary (called Region in this example), terminals need to be defined that are coincident with the Coil and the Region.
 - ▲ Hide the region under **View > Active View > Visibility**
 - ▲ Select the menu item **Edit > Select > Faces** and click on the right face of **Coil_A2** that is coincident with the Region.
 - ▲ Select the menu item **3D Modeler > Surface > Create Object from Face**
 - ▲ Change the name from **ObjectFromFace1** to **Terminal_A2_1** to represent the number 1 Terminal on Coil A2



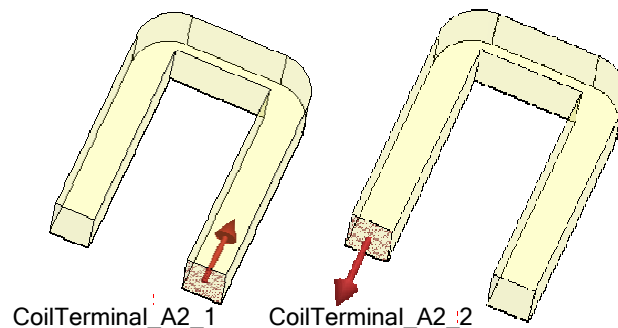
- ▲ Repeat this for the other coils, creating terminals from the faces the touch the boundary as shown below.



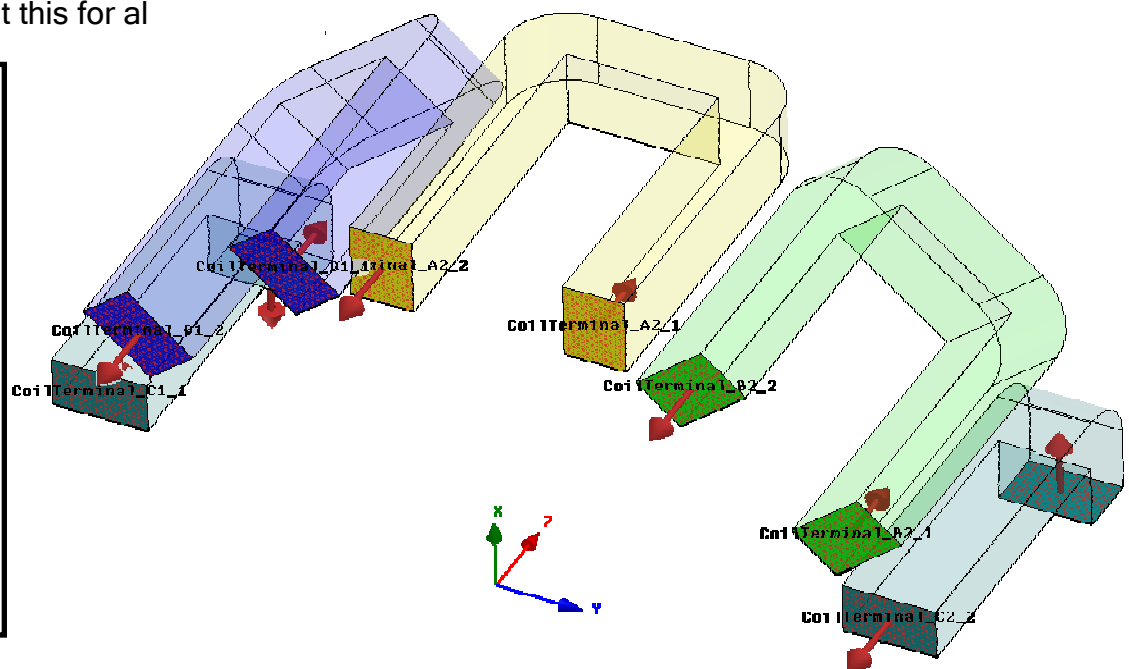
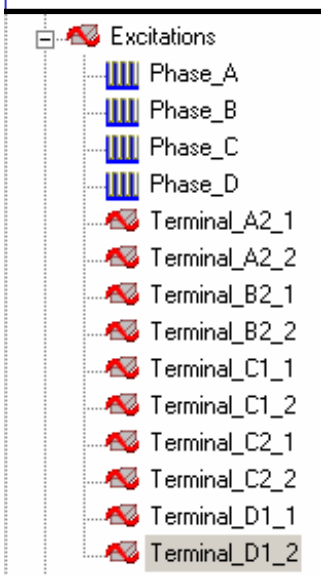
Example (Transient) - Stranded Conductors

Assign Terminals as Excitation Coil Terminals

- ▲ Select the Terminals just created.
- ▲ Select the menu item **Maxwell > Excitations > Assign > Coil Terminal**.
- ▲ Assign the **Number of Conductors** to 150
 - ▲ **Note:** The number of conductors for this fractional model is 150 which is the same as for the full model. The cross-section of the winding has not been reduced and so the objects modeled still have 150 conductors.
- ▲ Change the name of the Excitations to match the actual terminal. Select **CoilTerminal_1** and in Properties Window change the name to **CoilTerminal_A2_1**. Make sure that the terminal is going into the **Coil_A2** along the Z direction.
- ▲ Repeat this for all of the other Coil Terminals, making sure that each coil has one terminal that has the direction along +Z and the other terminal along -Z.
- ▲ The terminals for the A Coil should look like this:



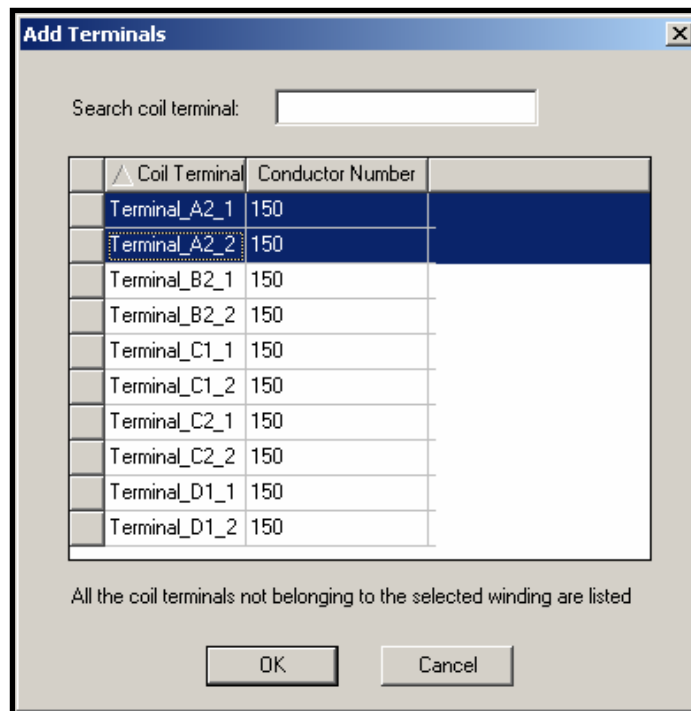
- ▲ Repeat this for all



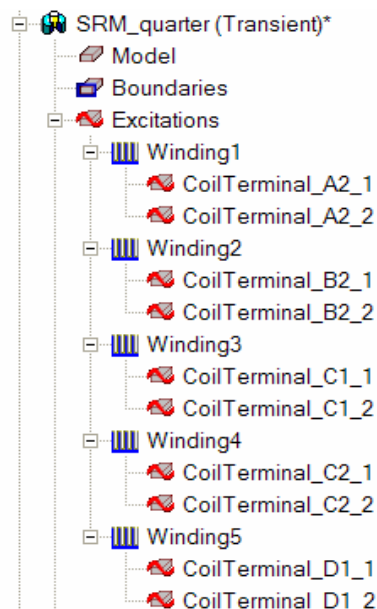
Example (Transient) - Stranded Conductors

Assign the Coil Terminals to the Windings

- ▲ The new Coil Terminals need to be assigned to the existing Windings
- ▲ Click on **Winding_1** Winding and then right mouse click and select **Add Terminals**. Choose **Terminal_A2_1** and **Terminal_A2_2**.



- ▲ Repeat this for the other four Windings



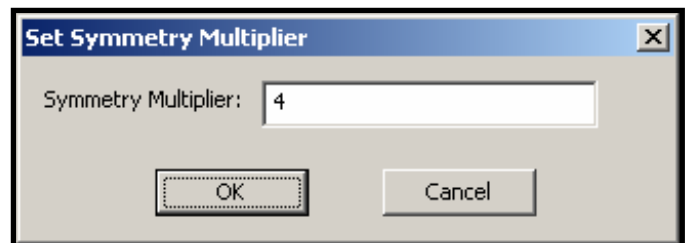
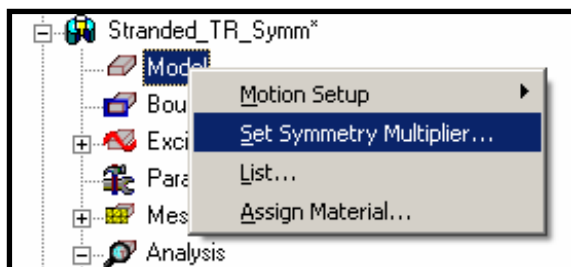
Example (Transient) - Stranded Conductors

Create a Mesh

- As before, we will need to create a manual mesh.
- First, delete any previous mesh operations under the Mesh Operations tree.
- Select the coils. Select the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 - Change the **Name** to Coils
 - Deselect **Restrict Length of Elements** check box
 - Select **Restrict the Number of Elements** check box and change the **Maximum Number of Elements** to be 4000. This is $\frac{1}{4}$ of the value that was used for the full model
- Select the Rotor and Stator. Right mouse click and select **Assign Mesh Operation > On Selection > Length Based**
 - Change the **Name** to Rotor_Stator
 - Deselect **Restrict Length of Elements** check box
 - Select **Restrict the Number of Elements** check box and change the **Maximum Number of Elements** to be 1000. Again, this is $\frac{1}{4}$ of the value that was used for the full model
- Select the menu item **Maxwell > Analysis Setup > Apply Mesh Operations**

Set Symmetry Factor

- Under Stranded_TR_Symm, select **Model** from the project tree. Right mouse click and select **Set Symmetry Multiplier** and use a value of 4.



Check the Validity of the Model

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



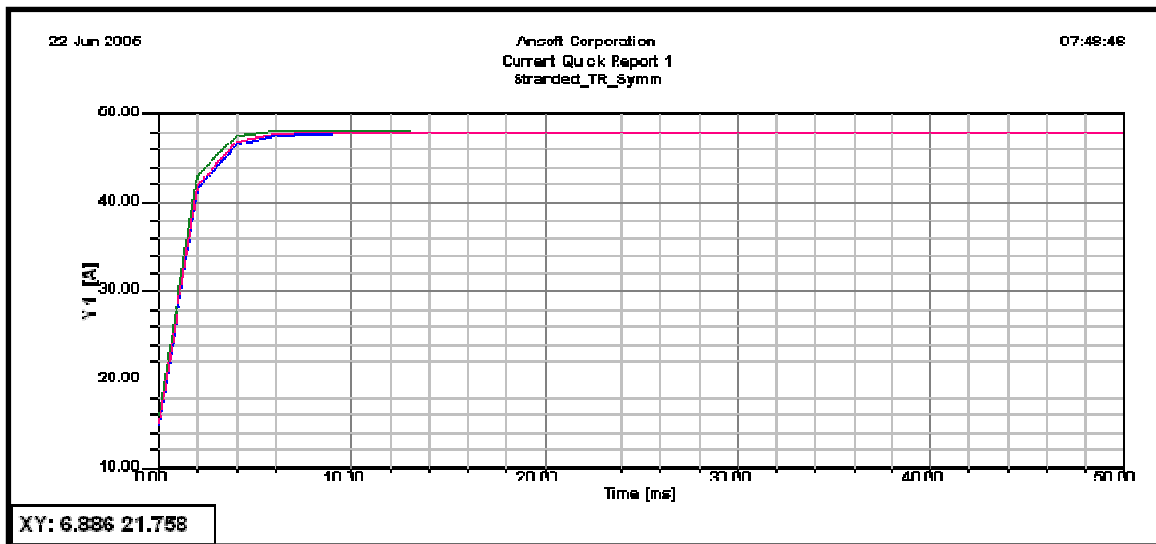
Example (Transient) - Stranded Conductors

Analyze

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

Results

- Click on **Results > Current Quick Report 1** to view the plot of the Winding Current

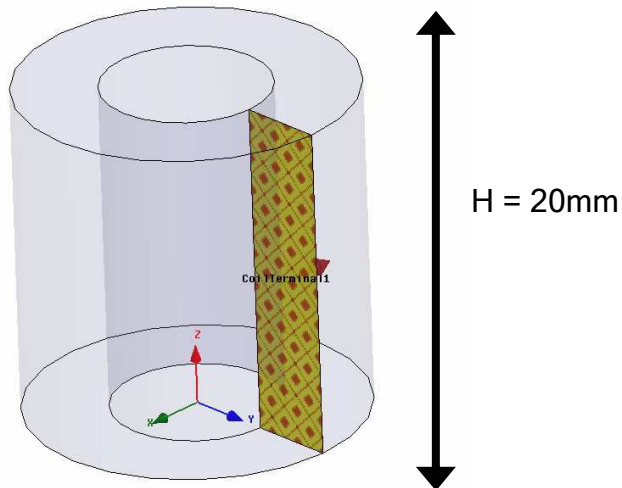


- As before, the four phase windings converge to the same DC value, 48 amps.
- Calculate the total current across the terminal
- Select the menu item **Maxwell > Fields > Calculator** and perform the following:
 - Time > 0.05 sec
 - Quantity > J
 - Scal? > ScalarZ
 - Geometry > Surface > Terminal_A1_1
 - , the integral symbol
 - Number > Scalar > 150, this is the conductor number
 - /, the divide symbol
 - Eval, which stands for evaluate
 - The value displayed is 47.98 amp.

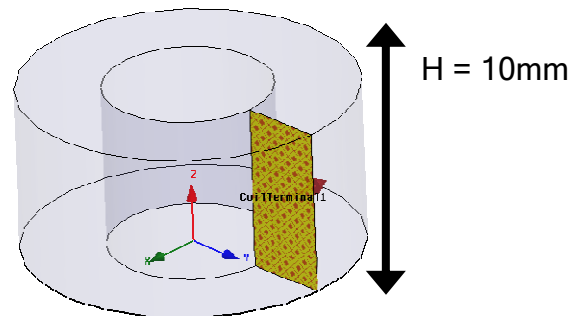
Example (Transient) - Stranded Conductors

Half Coils

Note: If the cross section of the coil is cut in half, then the number of conductors is half. An example for a simple coil is shown below:



Modeling the Full Coil:
Conductor Number = 150
Symmetry Multiplier = 1

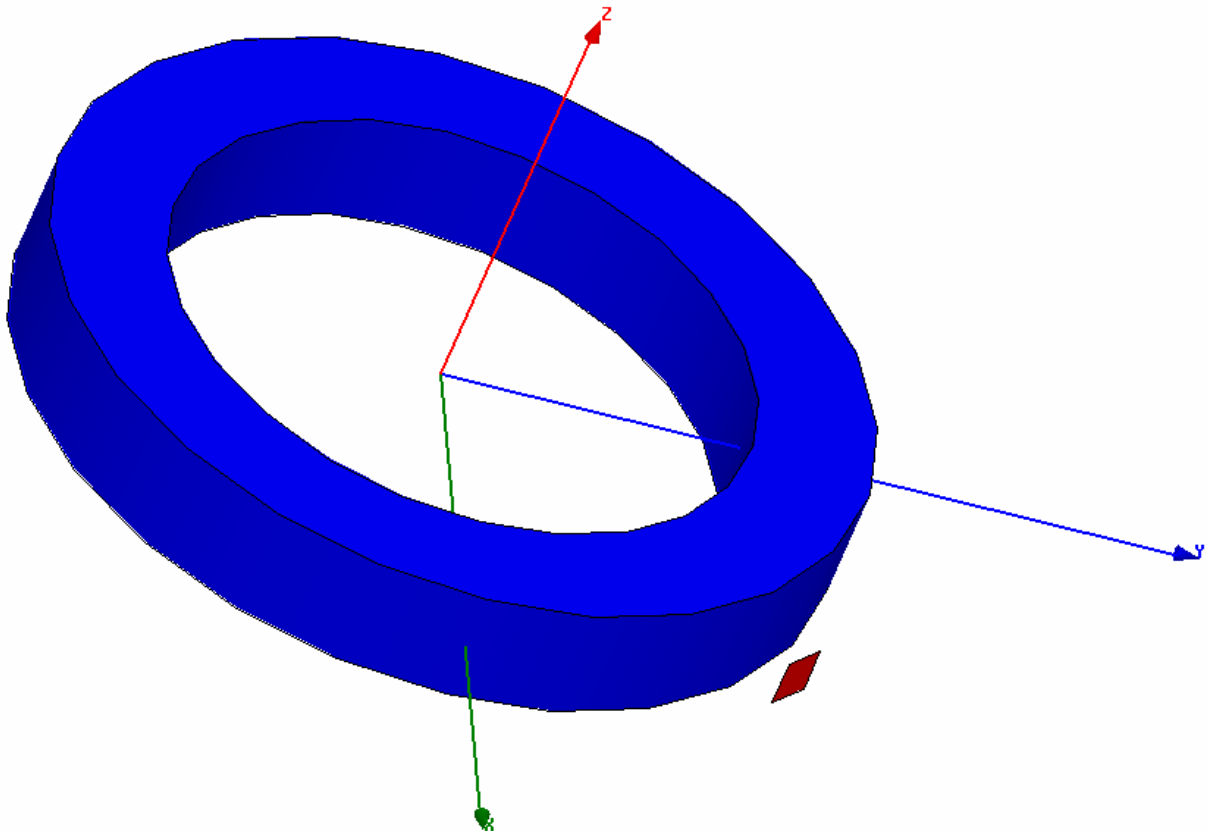


Cut the Full Coil in half and use an
Odd Symmetry Boundary:
Conductor Number = 75

Example (Transient) - Rotational Motion

Rotational Cylindrical Motion using the Transient Solver

- Transient Solver in Maxwell allows to be combined with motion. This motion can be translational, rotational cylindrical or rotational non-cylindrical (relay-type motion). This example shows on a simple geometry how to set-up rotational cylindrical motion project in a Transient Solver. It teaches how to set-up a band for rotational cylindrical motion, create and post process time-dependent field variables and create an animation. It shows how to utilize the periodicity of the configuration to reduce the size of the problem.
- The geometry consists of a cylindrical 4-pole radially magnetized NdFeB magnet with sinusoidal distribution of magnetization. The magnet rotates with a constant speed of 360 degrees per second around its axis. There is a Hall Sensor at certain distance from the magnet. The goal is to model a time-dependent magnetic field created by rotating magnet and follow the magnitude of this field by employing the Hall sensor. The role of the sensor is to calculate the average flux density passing through the sensor.



Example (Transient) - Rotational Motion

▀ 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: **Regular Polyhedron, Rectangle**
- ▀ Boolean Operations: **Split**
- ▀ Sweep Operations: **Around Axis**
- ▀ Model: **Band for rotational cylindrical motion**

- ▀ **Boundaries/Excitations**

- ▀ **Matching boundaries (Master/Slave)**
- ▀ **Natural outer boundary**

- ▀ **Material properties**

- ▀ **Edit material (4-pole NdFeB radially magnetized magnet with sinusoidal distribution of magnetization)**

- ▀ **Analysis**

- ▀ **Transient**

- ▀ **Results**

- ▀ **Fields Calculator**
- ▀ **Rectangular Plot**

- ▀ **Field Overlays:**

- ▀ **Flux Density Vector Plots**
- ▀ **Flux Density Vector Animation**

Example (Transient) - Rotational Motion

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:

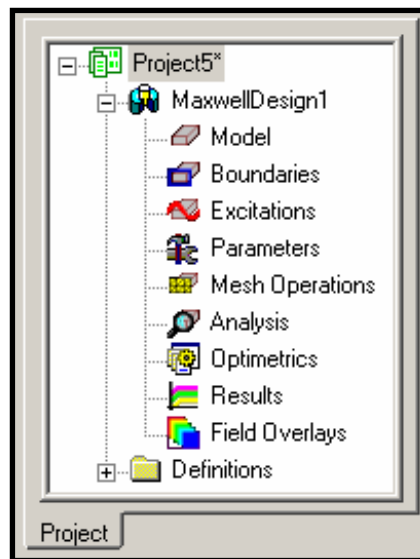
-  **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
 1. Select the menu item ***Tools > Options > Maxwell Options***
 2. Maxwell Options Window:
 1. Click the **General Options** tab
 -  Use Wizards for data entry when creating new boundaries: ☒ **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

Example (Transient) - Rotational Motion

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 > Transient** or right mouse click on MaxwellDesign1 and select **Solution Type ...**

Set Model Units

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

Example (Transient) - Rotational Motion

Create Magnet

- ▲ Outer Radius: 7 mm; Inner Radius: 5 mm; Height: 2 mm
- ▲ There are 2 possibilities how to create the magnet:
 1. Create 2 cylinders with respective outer and inner radii and subtract them
 2. Create a rectangle and sweep it around axis:
 - ▲ **3D Modeler > Drawing Plane > XZ**
 - ▲ **Draw > Rectangle**
 - ▲ Using the coordinate entry field, enter the first corner of the rectangle
 - ▲ **X: 5.0, Y: 0.0, Z: -1.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the opposite corner of the rectangle
 - ▲ **X: 7.0, Y: 0.0, Z: 1.0**, Press the **Enter** key
 - (Note here that the entries for the position are in ABSOLUTE values.)
 - ▲ Select **Rectangle 1**:
 - ▲ **Edit > Select > By Name - Rectangle 1** or click on the  icon and choose **Rectangle 1**.
 - ▲ **Draw > Sweep > Around Axis** - choose **z** as sweep axis and **360** degrees as Angle of sweep



- ▲ Click on the Attribute Tab and change the name from **Rectangle 1** to **Magnet**
- ▲ Change the Color to **Blue**

Example (Transient) - Rotational Motion

Create Hall Sensor

- ▲ A Hall Sensor is a square sheet (0.8x0.8 mm) located 1 mm from the outer radius of the magnet and rotated 45 degrees from the x-axis:
 - ▲ **3D Modeler > Drawing Plane > YZ**
 - ▲ **Draw > Rectangle**
 - ▲ Using the coordinate entry field, enter the first corner of the rectangle
 - ▲ **X: 8.0, Y: -0.4, Z: -0.4**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the opposite corner of the rectangle
 - ▲ **dX: 0.0, dY: 0.8, dZ: 0.8**, Press the **Enter** key
 - ▲ Click on the Attribute Tab and change the name from **Rectangle 1** to **Sensor**
 - ▲ Change the Color to **Red**
 - ▲ **Edit > Arrange > Rotate**
 - ▲ Axis: **z**; Angle: **45 deg**

Create Band

- ▲ In a Transient Solver, Band is an object that contains all rotating objects. In it is a way how to tell Maxwell which objects are steady and which are moving. In this example there is one moving (rotating) object: Magnet. For rotational cylindrical type of motion Band is usually a cylinder. It should be a solid cylinder, not a hollow cylinder. For translational type of motion Band cannot be a true surface object (cylinder). It must be a faceted object, such as Regular Polyhedron.
 - ▲ **3D Modeler > Drawing Plane > XY**
 - ▲ **Draw > Cylinder**
 - ▲ Using the coordinate entry field, enter the center of base:
 - ▲ **X: 0.0, Y: 0.0, Z: -6.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the radius:
 - ▲ **dX: 7.3, dY: 0.0, dZ: 0.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the height:
 - ▲ **dX: 0.0, dY: 0.0, dZ: 12.0**, Press the **Enter** key
 - ▲ Click on the Attribute Tab and change the name from **Cylinder 1** to **Band**
 - ▲ Transparent: **0.6**
 - ▲ Change the Color to **Brown**

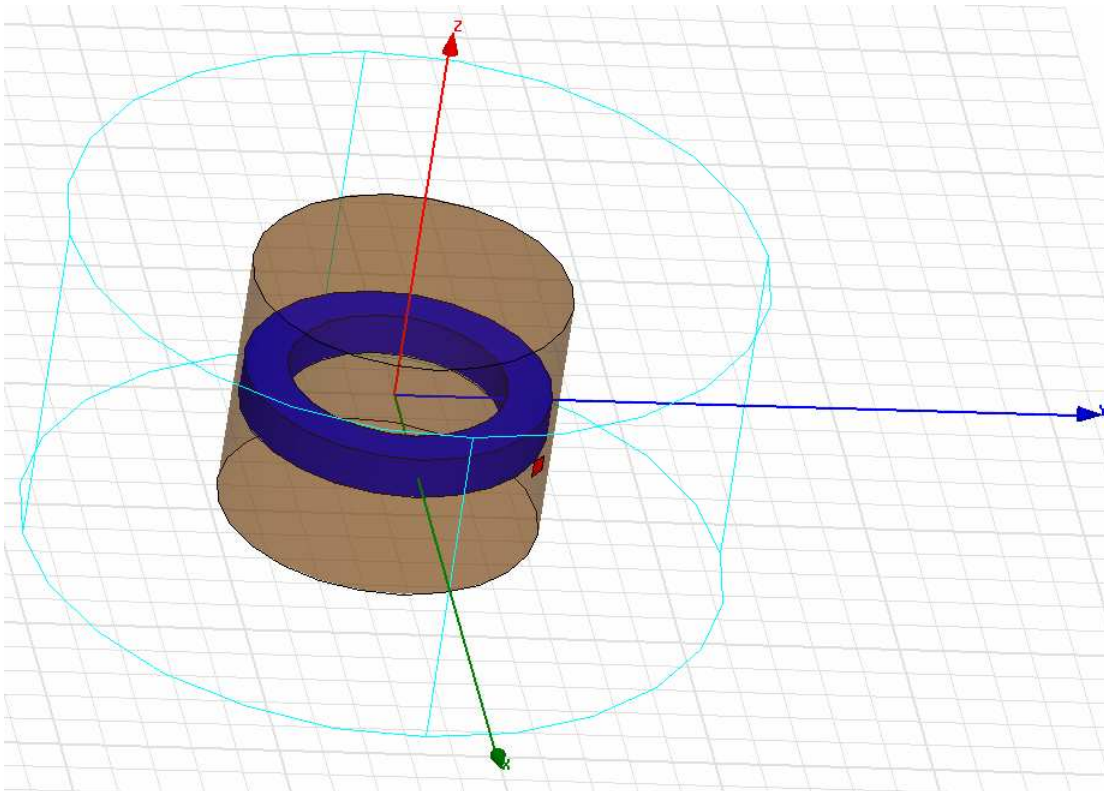
Example (Transient) - Rotational Motion

▲ Create Outer Boundary (Region)

- ▲ All objects in the model must be enclosed in one object the boundary of which serves as outer boundary. This object is called Region and can be created using **Draw > Region** command. Using this command a box with prescribed dimensions is created. In this example we wish that the Region is of cylindrical shape. Therefore, an additional cylinder far enough from all other objects will be created. This cylinder will serve as Region. The field is by default tangential to the outer boundary of such an object.

▲ **Draw > Cylinder**

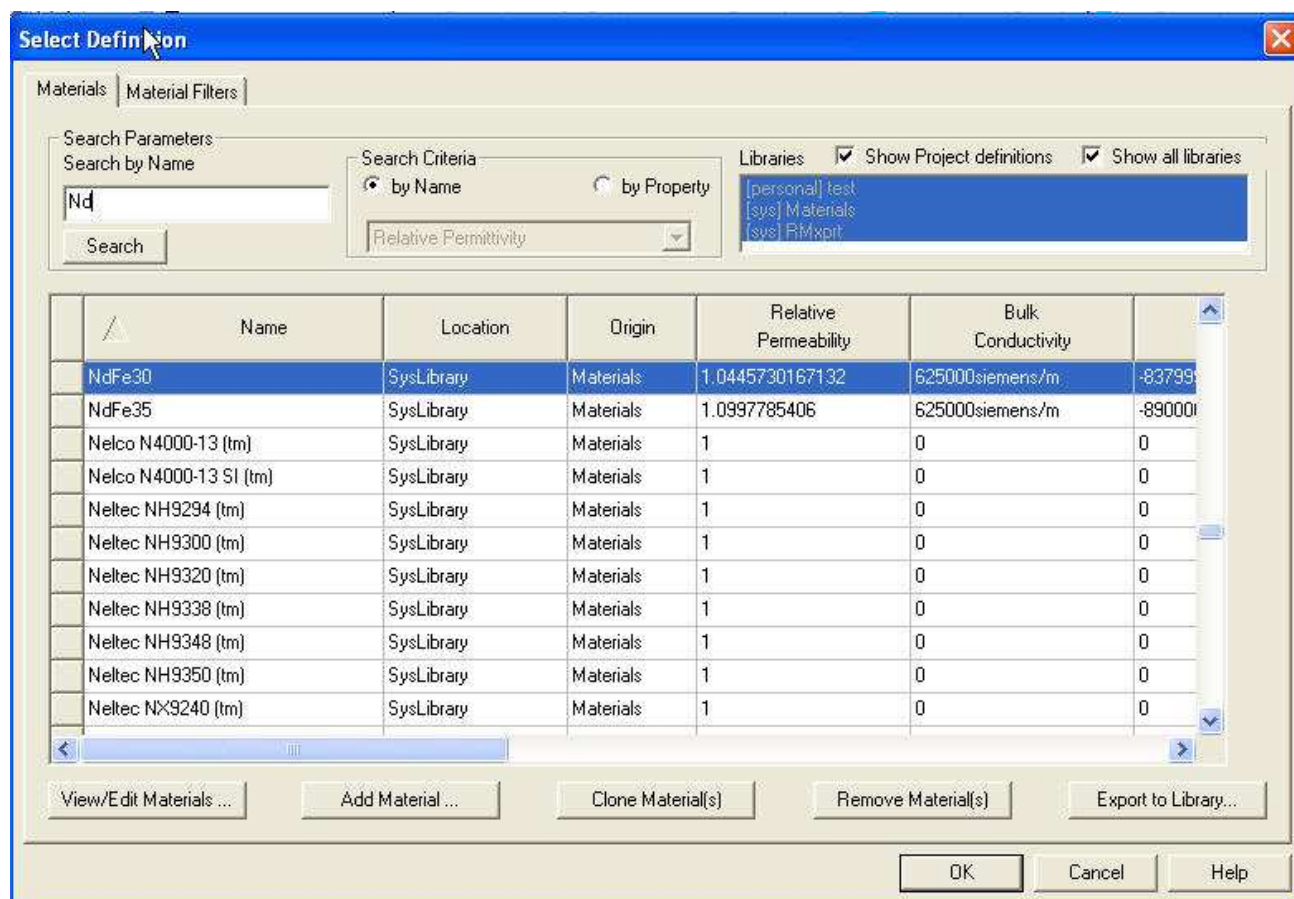
- ▲ Using the coordinate entry field, enter the center of base:
 - ▲ **X: 0.0, Y: 0.0, Z: -8.0**, Press the **Enter** key
- ▲ Using the coordinate entry field, enter the radius:
 - ▲ **dX: 16, dY: 0.0, dZ: 0.0**, Press the **Enter** key
- ▲ Using the coordinate entry field, enter the height:
 - ▲ **dX: 0.0, dY: 0.0, dZ: 16.0**, Press the **Enter** key
- ▲ Click on the Attribute Tab and change the name from **Cylinder 1** to **Region**
- ▲ Check the box **Display Wireframe**
- ▲ Change the Color to **Light Blue**



Example (Transient) - Rotational Motion

Define and assign material properties

- The only object that needs material definition and assignment is **Magnet**. All other objects are **vacuum** by default.
- To define a new NdFeB30 4-pole magnet material with sinusoidal distribution of magnetization:
 - Tools > Edit Configured Libraries > Materials**



- Select **NdFeB30** and click on **View/Edit Materials** (start typing NdFeB into the Search by Name window for faster selection)

Example (Transient) - Rotational Motion

Specify the following:

Material Name: ***magnet_material***

Material Coordinate System Type: ***Cylindrical***

(The magnet is radially magnetized so it is advantageous to use the cylindrical coordinate system. The direction of magnetization is determined by Unit Vector. For the cylindrical coordinate system the components of Unit Vector are ***R***, ***Phi*** and ***Z***, standing for ***radial***, ***circumferential*** and ***z***- directions. The values of ***R***, ***Phi*** and ***Z*** are 1, 0, 0 by default. For this example they do not have to be changed, because the magnet is radially magnetized.)

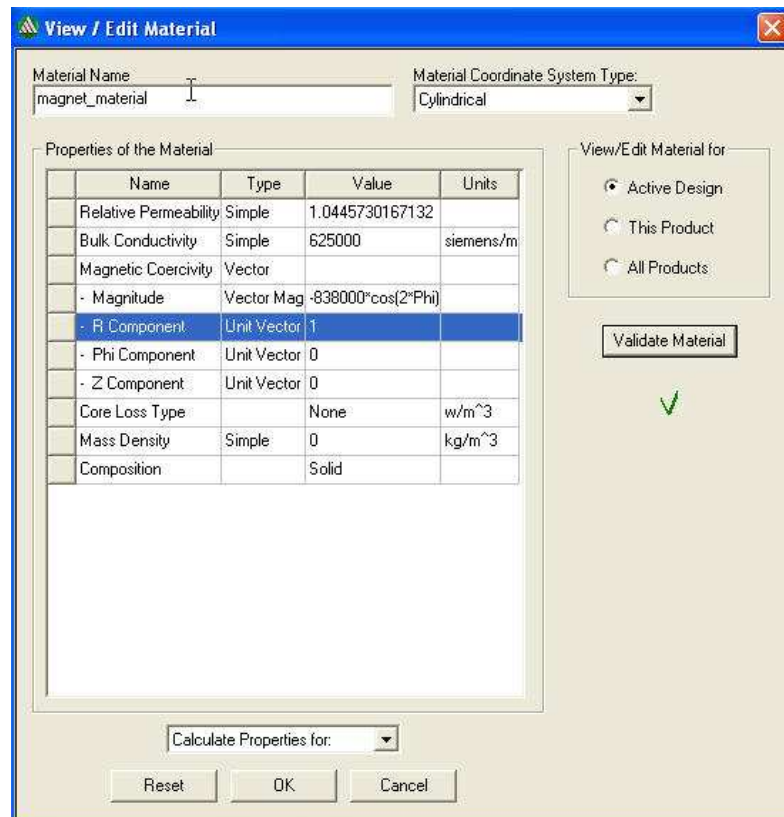
Magnetic Coercivity - Magnitude: ***-838000*cos(2*Phi)***

(Phi is a default symbol for an angle in the cylindrical coordinate system. In order to get 4-pole magnet the argument in cos is multiplied by two.)

Click on ***Validate Material*** to check the settings (green check mark indicates that everything is all right).

Using the coordinate entry field, enter the height:

Press the **OK** key



View / Edit Material

Material Name: Material Coordinate System Type:

Name	Type	Value	Units
Relative Permeability	Simple	1.0445730167132	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-838000*cos(2*Phi)	
- R Component	Unit Vector	1	
- Phi Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Core Loss Type		None	w/m^3
Mass Density	Simple	0	kg/m^3
Composition		Solid	

View/Edit Material for:

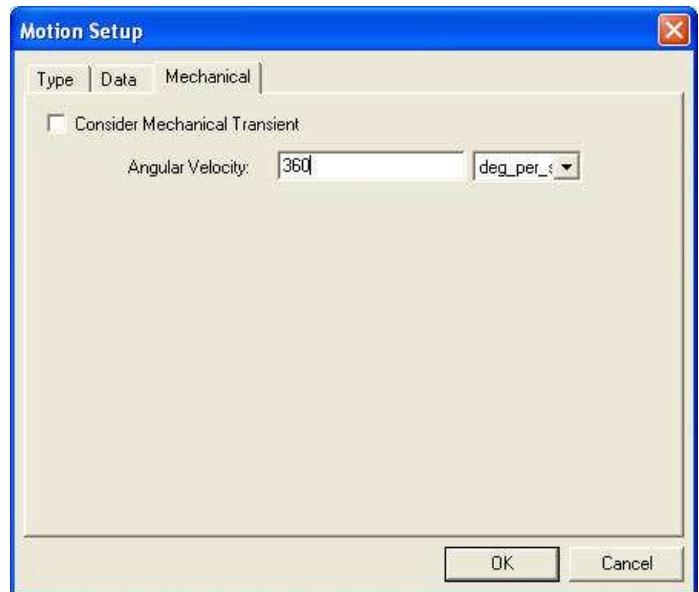
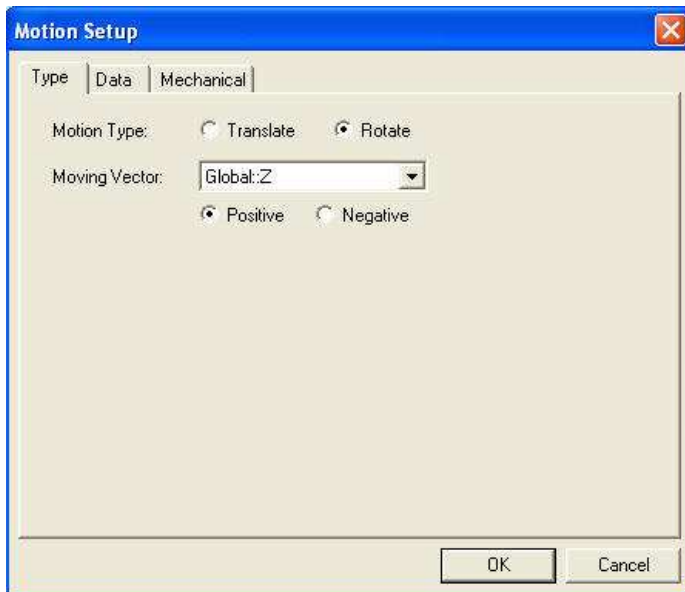
☒ Active Design
☐ This Product
☐ All Products

Example (Transient) - Rotational Motion

- ▲ To assign a newly created **magnet_material** to object **Magnet**.
 - ▲ Select **Magnet**.
 - ▲ **Edit > Select > By Name** - select **Magnet** and press OK key
 - ▲ Assign material:
 - ▲ **3D Modeler > Assign Material** - select **magnet_material** and press OK key

▲ Define Band and Motion Setup

- ▲ Select **Band**.
 - ▲ **Edit > Select > By Name** - select **Band** and press OK key
- ▲ Assign **Band**.
 - ▲ **Maxwell > Model > Motion Setup > Assign Band**



- ▲ Select **Type** Tab and specify **Rotate** radio button
- ▲ Select **Mechanical** Tab and specify **Angular Velocity: 360 deg_per_s**
(The magnet rotates at constant speed of 360 degrees per second - 1 revolution per second)

Example (Transient) - Rotational Motion

Create an Analysis Setup

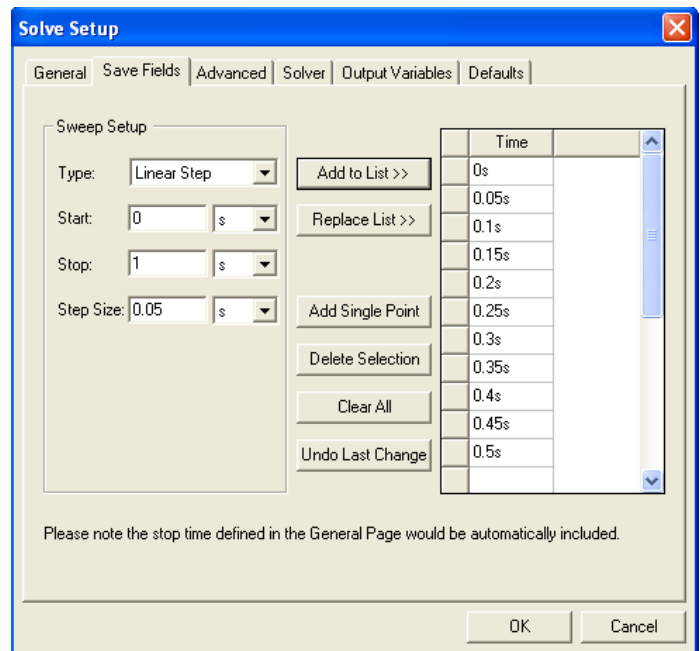
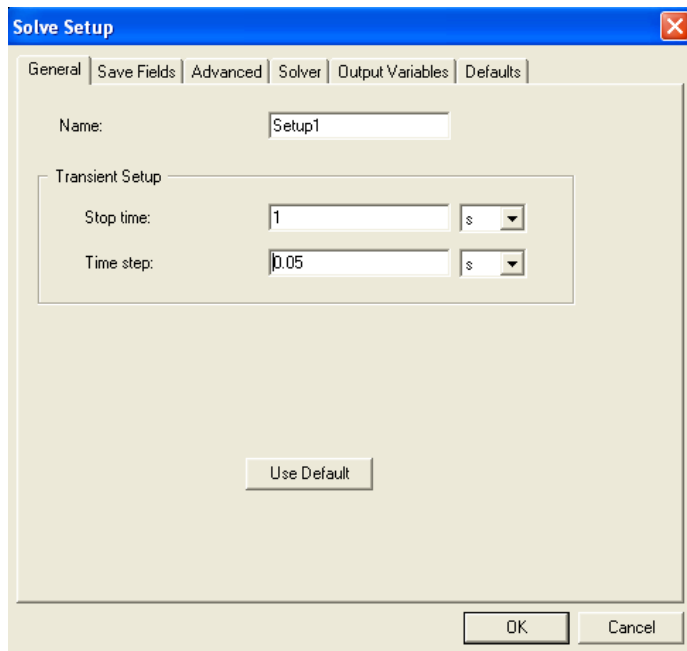
- Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**

- General Tab:** Stop time: **1 s**; Time step: **0.05 s**

(We will analyze one revolution. Since the speed is 360 degrees per second, to analyze one revolution, we have to solve 1 s in time. We will discretize the time domain into 20 time steps, meaning that the time step will be 0.05 s.)

- Save Fields Tab:** Type: **Linear Step**; Start: **0**; Stop: **1**; Step Size: **0.05** and click on **Add to List >>**. Finally press **OK** key.

(The field solution will be saved at the specified time instants.)



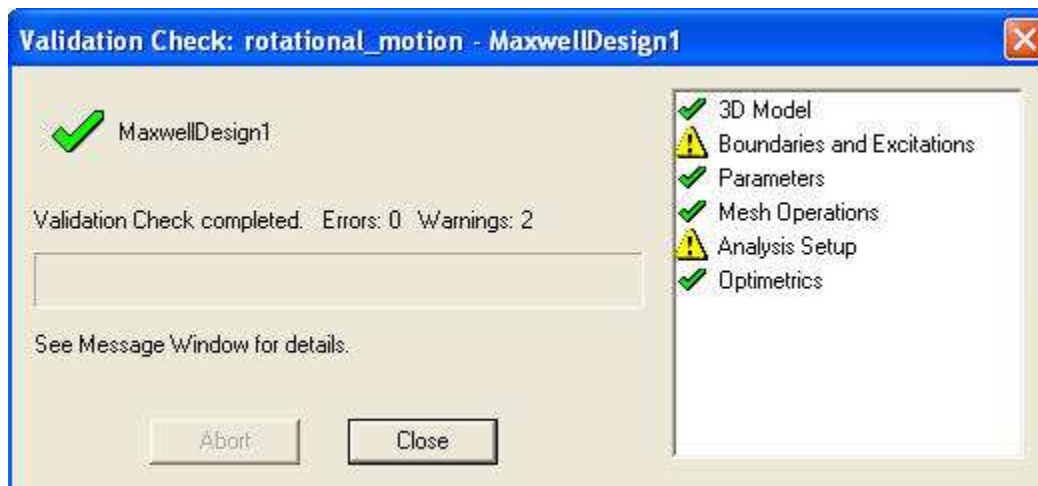
Example (Transient) - Rotational Motion

Save the Project

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

Check the Validity of the Model

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



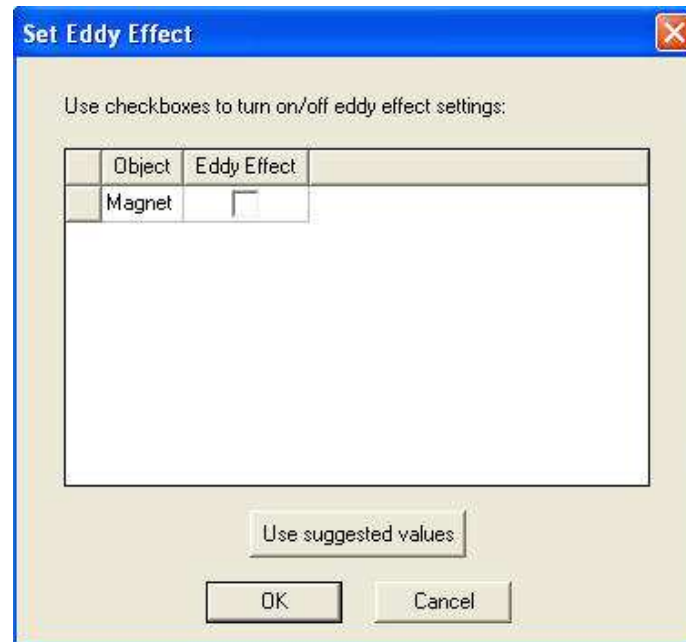
- ▲ The problem won't solve unless each object has a check mark. We can see that at the moment there are 2 warnings. From the **Message Window** we can read:

1. **Warning:** Eddy effect settings may need revisiting due to the recent changes in the design.

In This example we are not interested to compute eddy currents. We have to make sure that the objects that are made from conducting material (**Magnet**) are not assigned for eddy current computation.

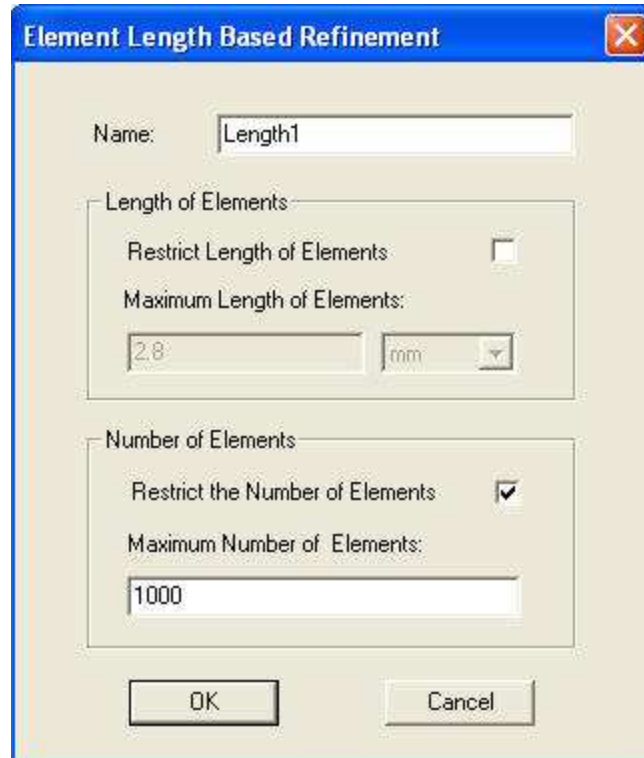
- ▲ **Maxwell > Excitations > Set Eddy Effects ...**
- ▲ Make sure that the box next to **Magnet** is unchecked

Example (Transient) - Rotational Motion



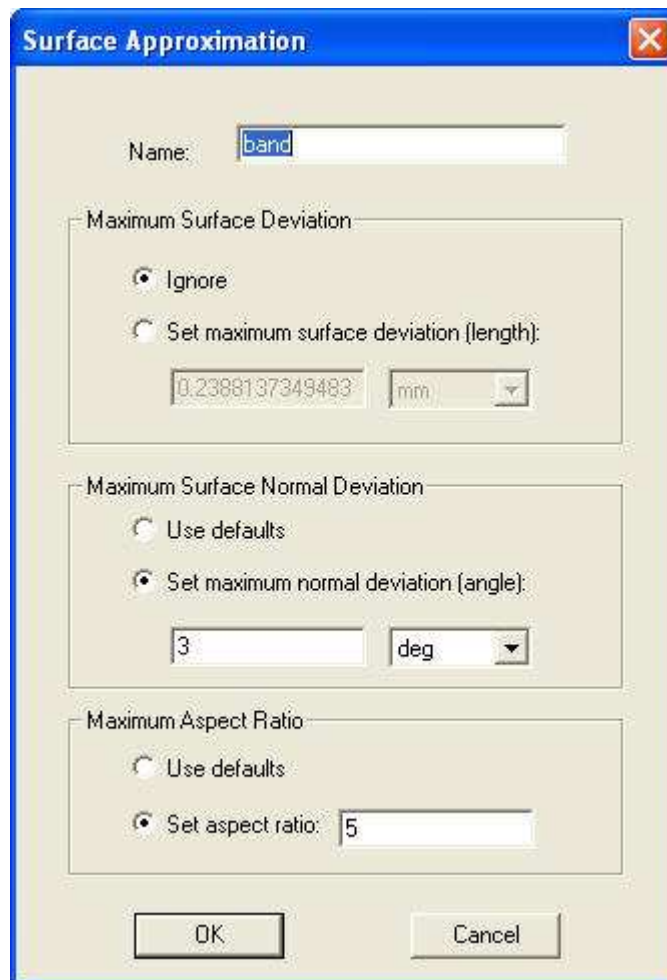
2. **Warning:** Currently no mesh operations or data link have been created for the setup(s): Setup1. The initial mesh will be used. The initial mesh is usually not fine enough to achieve an accurate solution.
This warning reminds us that we are using the Transient Solver in which there is not adaptive refinement of the mesh as in Magnetostatic or Eddy Current Solvers. This means that the mesh once created will be used throughout all the time steps. Sliding Band technology enables that the mesh is not recreated even if rotational cylindrical motion is involved. A fine enough mesh has to be manually created before solving any transient problem with motion.
- ▲ To refine the mesh by assigning the mesh operations:
 - ▲ Select **Magnet**.
 - ▲ **Edit > Select > By Name** - select **Magnet** and press OK key
 - ▲ Assign mesh operations:
 - ▲ **Maxwell> Mesh Operations > Assign > Inside Selection > Length Based...**

Example (Transient) - Rotational Motion



- ▲ Name: **magnet**
- ▲ Uncheck **Restrict Length of Elements**
- ▲ Check **Restrict the Number of Elements**
- ▲ Specify **1000** for **Maximum Number of Elements**
- ▲ Repeat the same procedure for **Region** (8000)
- ▲ Since a cylinder is a true (curved) surface object, we will use Surface Approximation in order to create a good mesh for this rotational motion problem:
 - ▲ Select **Band**:
 - ▲ **Edit > Select > By Name** - select **Magnet** and press OK key
 - ▲ Assign mesh operations:
 - ▲ **Maxwell> Mesh Operations > Assign > Surface Approximation**

Example (Transient) - Rotational Motion



- ▲ Name: **band**
- ▲ Maximum Surface Deviation: **Ignore**
- ▲ Maximum Surface Normal Deviation: **3 deg**
- ▲ Maximum Aspect Ratio: **5**

- ▲ Performing the Validity Check again gives the green check marks for all the objects. The project is ready to be analyzed.

▲ Analyze

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

Example (Transient) - Rotational Motion

Solution Data

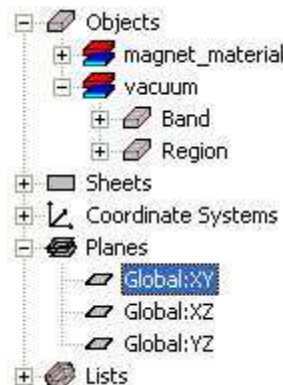
-  To view the Solution Data, select the menu item **Maxwell > Results > Solution Data**
-  Here you can view the **Solution** (if any, e.g. force) , **Profile** and the **Mesh Statistics**.

Plot Flux Density Vector on XY plane at time 0 s

-  To plot the flux density at specified time instant:
 -  **View > Set Solution Context**

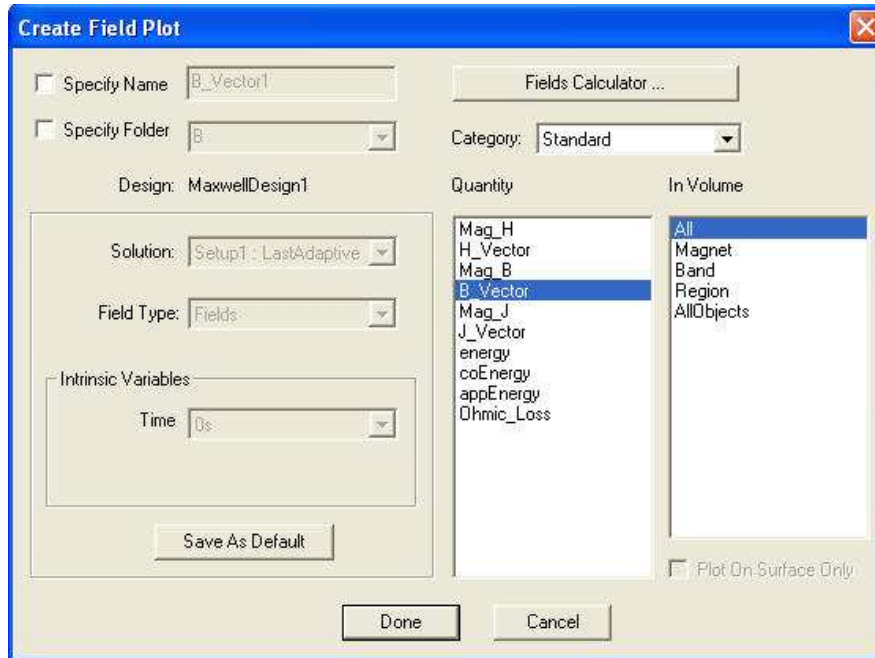


-  For **Time** specify **0 s**.
-  Select **XY** plane: From the history tree window expand **Planes** and select **Global: XY** plane

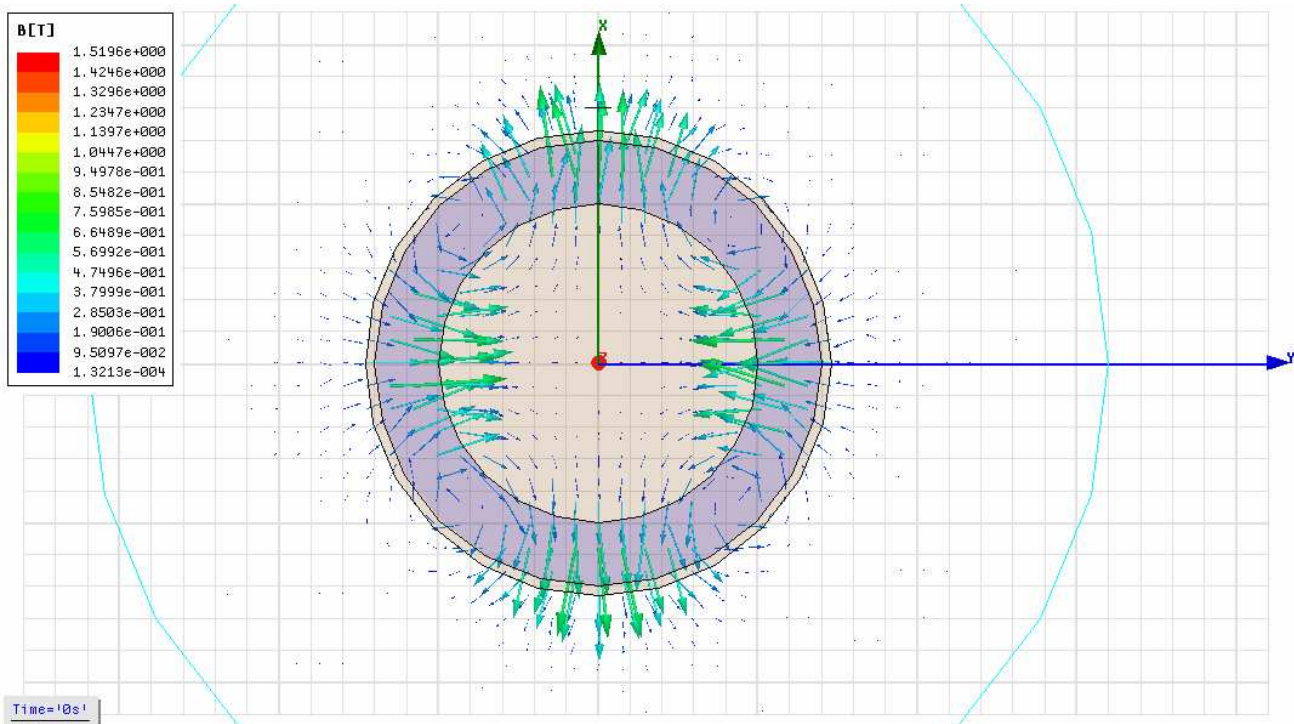


Example (Transient) - Rotational Motion

Maxwell > Fields > Fields > B > B_vector

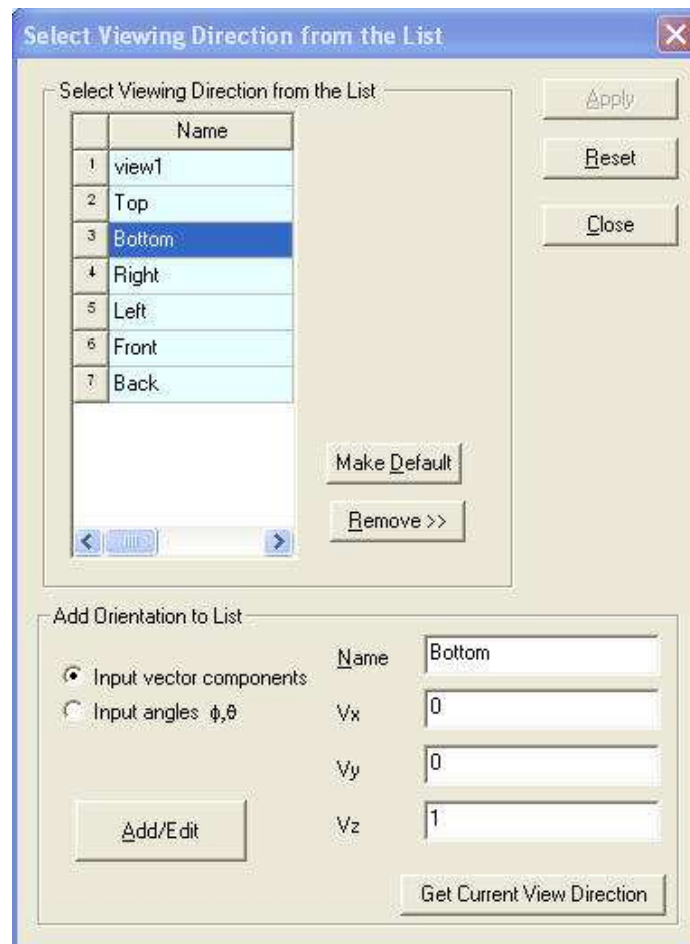


- Specify **B_vector** as Quantity and **All** as In Volume
- Press the **Done** key



Example (Transient) - Rotational Motion

- ▲ To set the viewing direction to the view of XY plane from the bottom:
 - ▲ **View > Modify Attributes > Orientation ...**
 - ▲ Specify **Bottom** as Viewing Direction
 - ▲ Click on **Apply** key and **Close** key



- ▲ Shortcut to Set the viewing direction to the view of XY plane from the bottom:
 - ▲ Press and hold **Alt** key
 - ▲ Double-click with the left mouse button on the central bottom region of 3D Modeler Window

Example (Transient) - Rotational Motion

- ▲ Modify the attributes of the vector plot:
 - ▲ **Maxwell > Fields > Modify Plot Attributes ...**
 - ▲ Select the specific plot (B)
- ▲ Or
 - ▲ Double-click on the Plot Legend

▲ Plot Hall Sensor Flux Density as a function of time

- ▲ To define a function **Bsensor** characterizing the average flux density through the surface of Sensor (use **Fields Calculator**):

- ▲ **Maxwell > Fields > Calculator ...**
 1. Input > Quantity > B
 2. Input > Geometry > Surface > Sensor
 3. Vector > Normal
 4. Undo
 5. Scalar > Integrate (the integral symbol)
 6. Input > Number > Scalar, Value 1
 7. Input > Geometry > Surface > Sensor
 8. Scalar > Integrate (the integral symbol)
 9. General > Divide (forward slash)

The final line in the Calculator register should look like this:

```
Sc1 : /(Integrate[Surface[Sensor], Dot(<Bx,By,Bz>, SurfaceNormal)], Integrate[Surface[Sensor], 1])
```

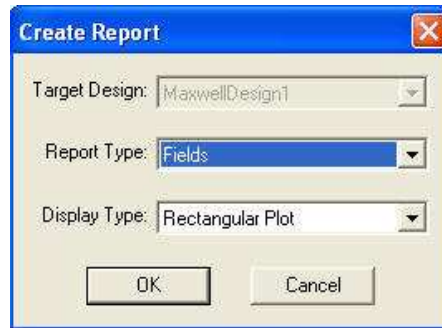
10. Add > Named Expression > Name: **Bsensor** > OK



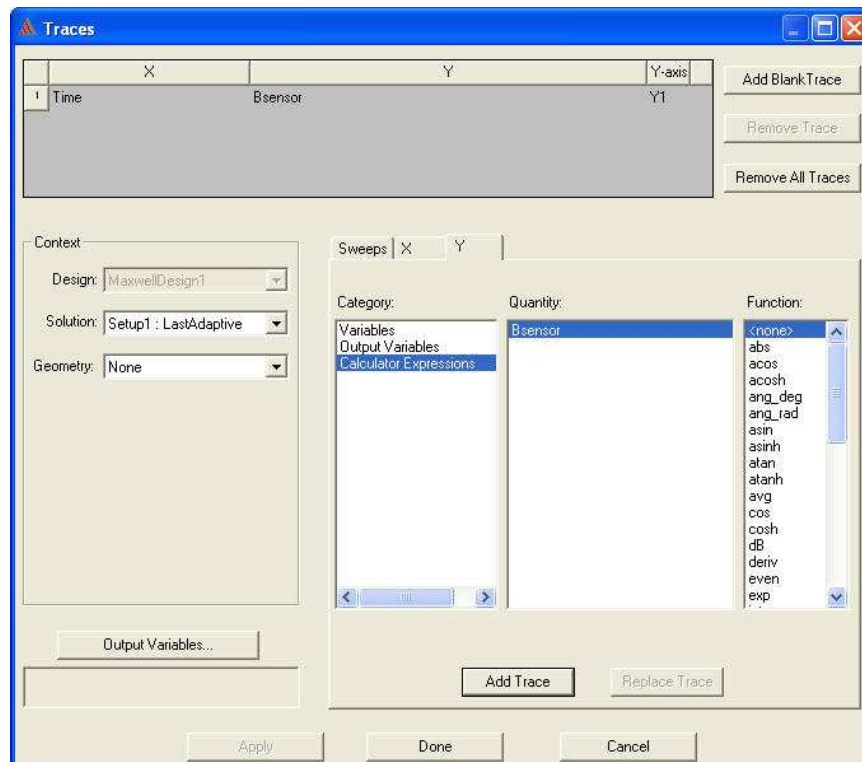
11. Click on **Done** key

Example (Transient) - Rotational Motion

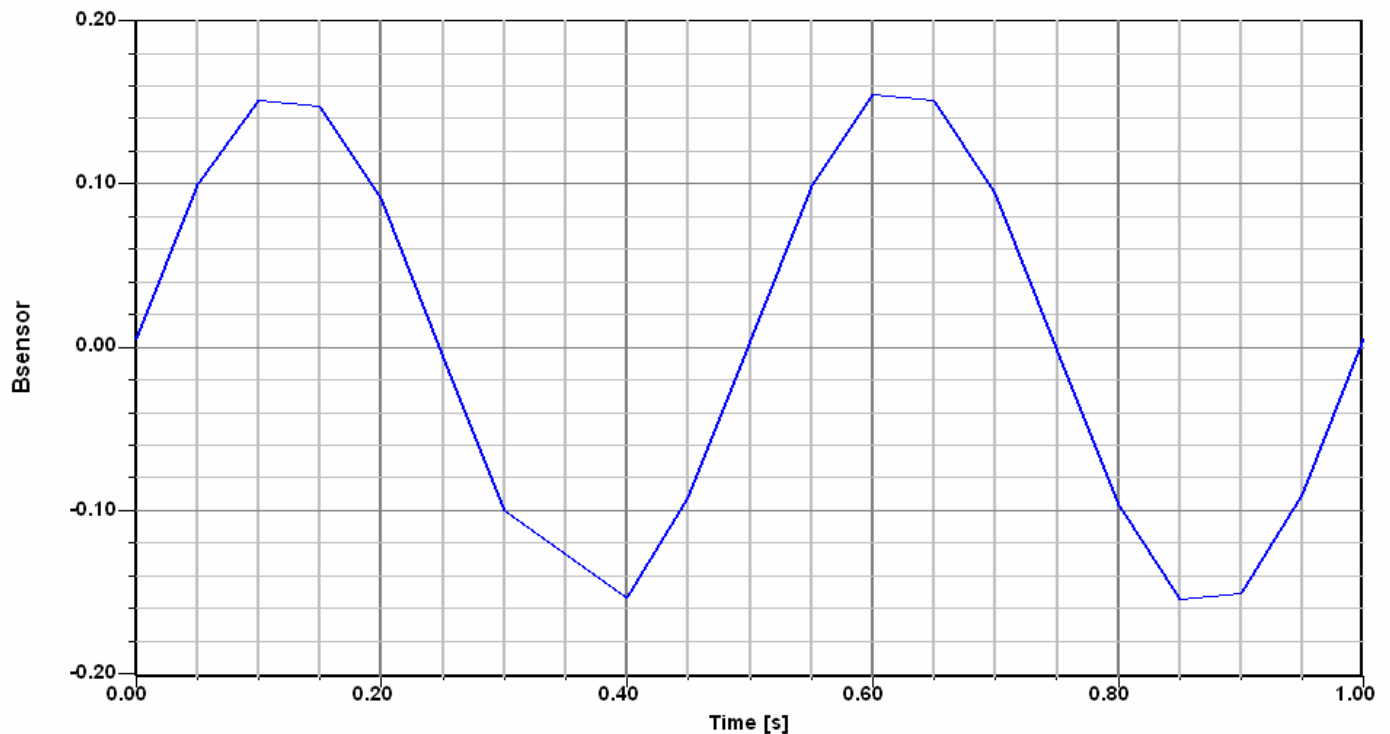
- ▲ To plot the defined function Bsensor as a function of time:
 - ▲ **Maxwell > Results > Create Report**
 - ▲ Specify: Report Type: **Fields**; Display Type: **Rectangular Plot**



- ▲ In the **Traces** Window specify from the Y Tab:
 - ▲ Category: **Calculator Expressions**
 - ▲ Quantity: **Bsensor**
 - ▲ Function: **none**
- ▲ Click on **Add Trace** key and **Done** key

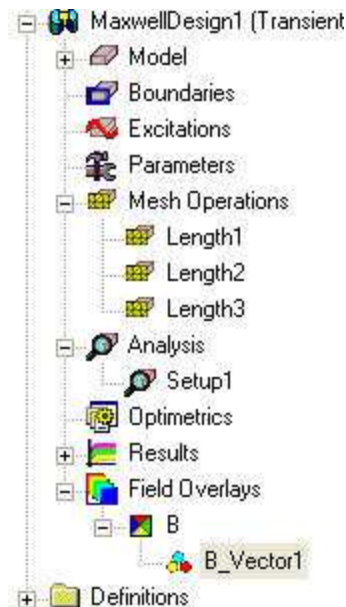


Example (Transient) - Rotational Motion

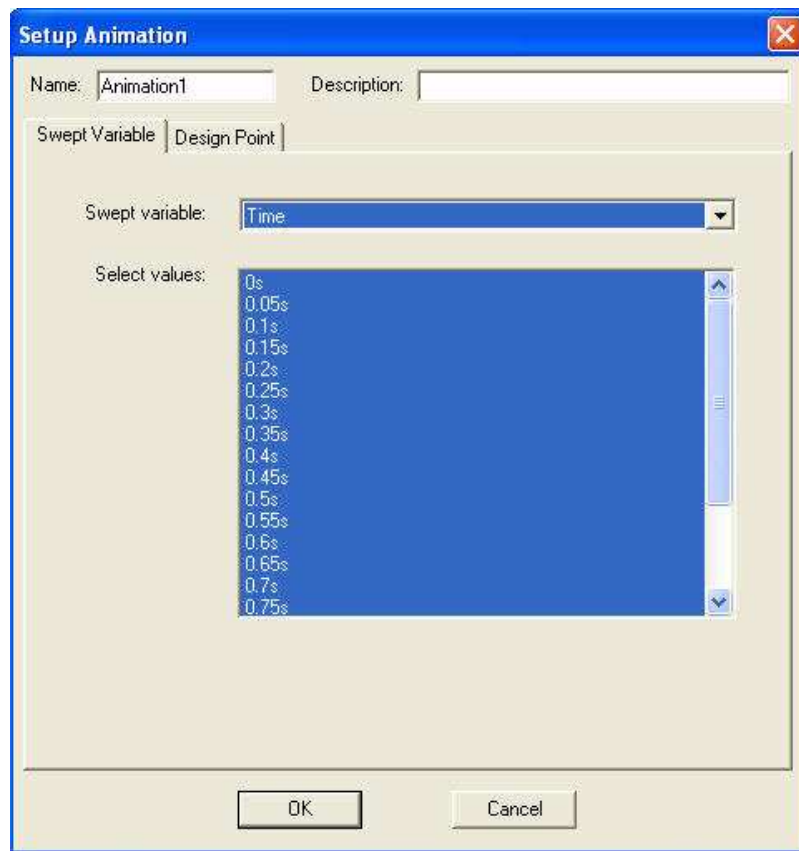


▲ Create an Animation of Flux Density Vector Plot

- ▲ Select already created plot of Flux Density Vector
 - ▲ At the Project Manager Window expand **Field Overlays** and **B**
 - ▲ Double-click on **B_vector1** to bring this vector plot back to view
 - ▲ Right-click on **B_vector1**
 - ▲ Select **Animate**

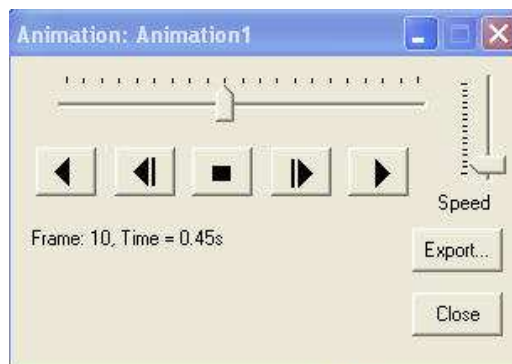


Example (Transient) - Rotational Motion



- ▲ In Setup Animation Window and Swept Variable Tab:
- ▲ Swept Variable: **Time**
- ▲ Select Values: Holding down the Shift key select all the values
- ▲ Press the **OK** key

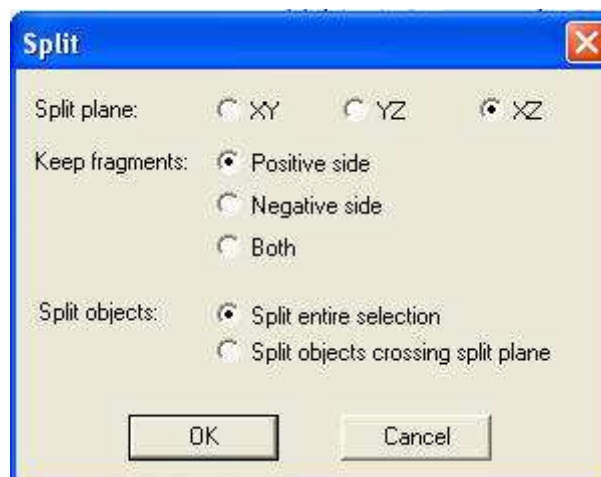
- ▲ To start and stop the animation, to control the speed of the animation and to export the animation use the buttons on the Animation Panel:



Example (Transient) - Rotational Motion

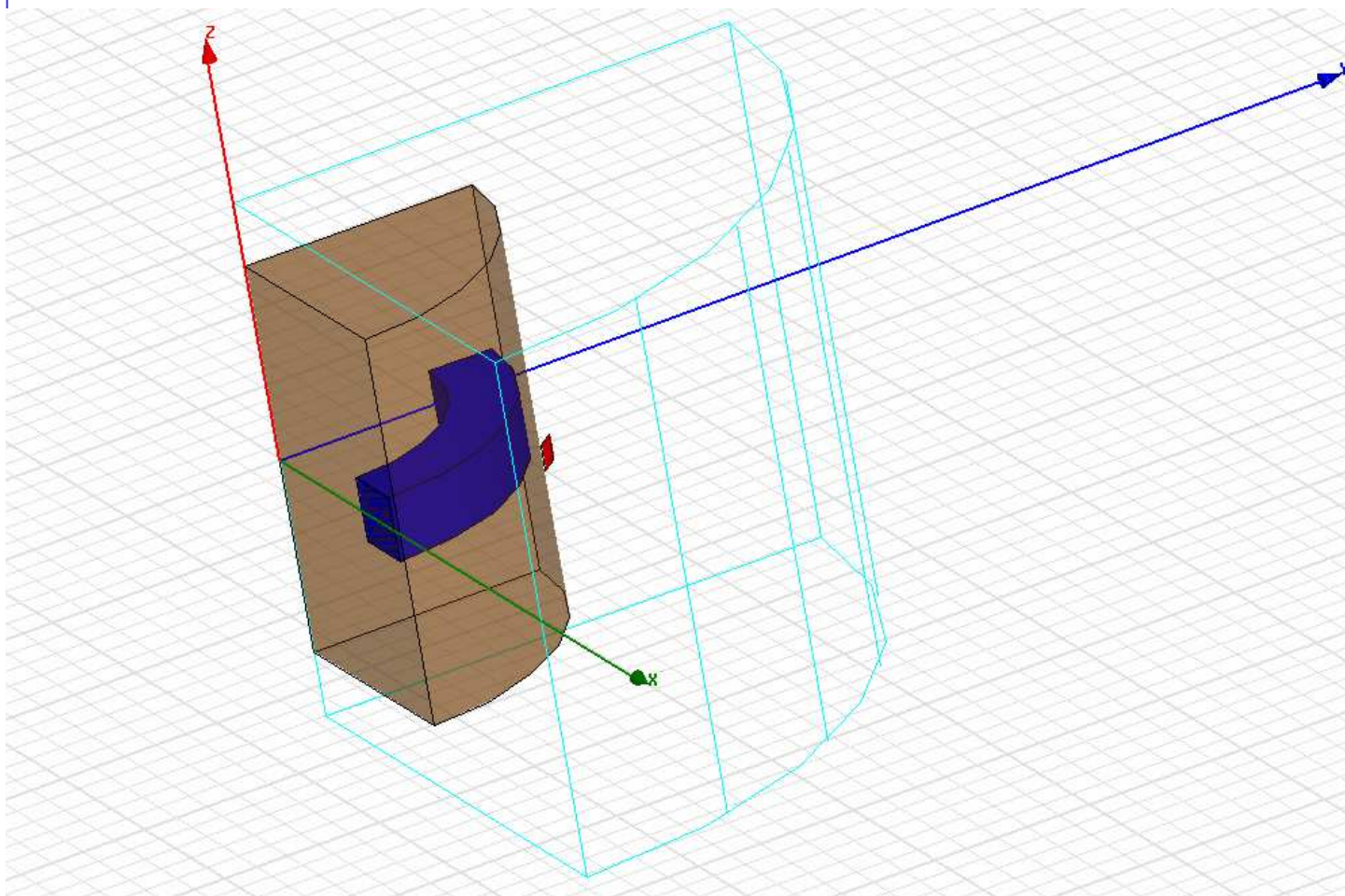
Utilize periodicity to reduce the size of the problem

- It is always advisable to utilize possible symmetries present in the model in order to reduce the size of the problem and in turn to reduce the computation time. The geometry of the problem studied in this section is obviously symmetrical. It is a 4-pole geometry where one pole pitch can be identified as the smallest symmetry sector. The field pattern repeats itself every pole pitch, which in this case is 90 degrees (see plot on page 5-1.16). To be precise there is so called negative periodicity present in the field pattern because the field along the x-axis is exactly the same as the field along the y-axis but with the opposite direction. The negative periodicity boundary conditions can be set-up in Maxwell using Master and Slave boundaries.
- In the Project Manager Window select the previous design, **full_geometry**, and right mouse click and select **Copy**.
- Click on project name, **rotational_motion**, and right mouse click and select **Paste**; this will create a second design called **rotational_motion1**. Change its name to **quarter_geometry**. Double click on this design to make it active.
- To create a 90 degrees geometry:
 - Select **Magnet, Band and Region**
 - Edit > Select > By Name** -select **Magnet, Band and Region**; OK
 - Split the selected object at **xz**-plane:
 - 3D Modeler > Boolean > Split**
 - Select **xz**-plane; Keep fragments: **Positive side**; **Split entire selection**



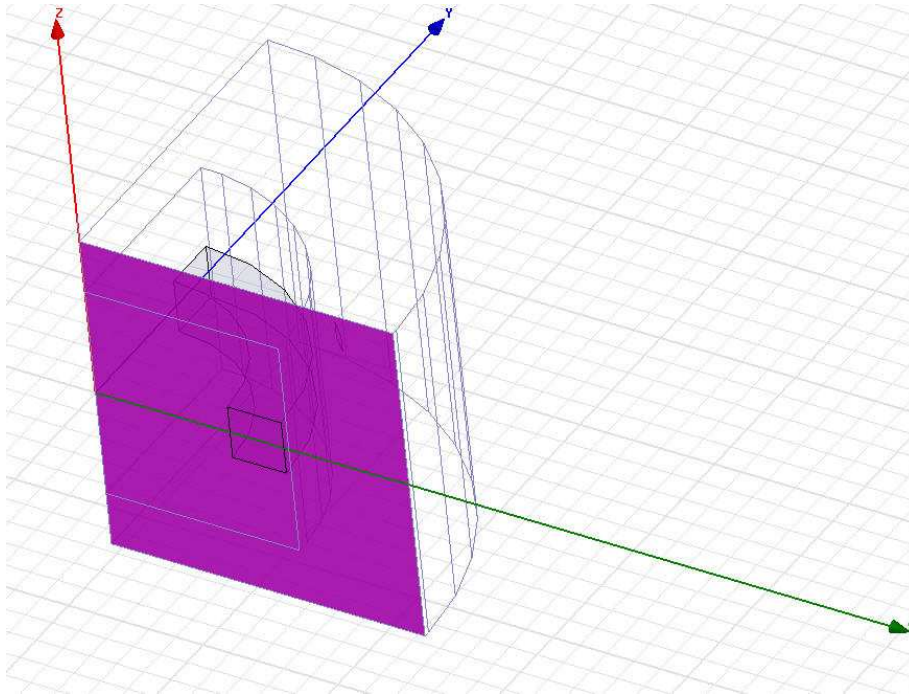
- OK
- Split the selected object at **yz**-plane:
 - 3D Modeler > Boolean > Split**
 - Select **yz**-plane; Keep fragments: **Positive side**; **Split entire selection**
 - OK

Example (Transient) - Rotational Motion

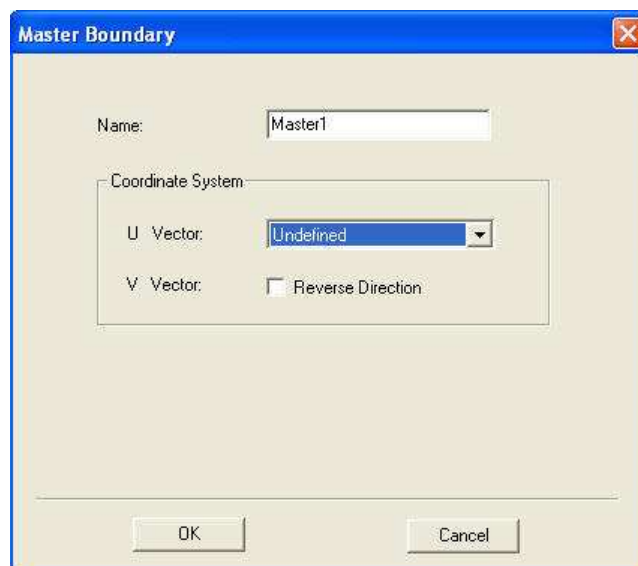


Example (Transient) - Rotational Motion

- ▴ To assign negative periodicity boundary conditions:
 - ▴ Press **F** on the keyboard in order to select Face Selection Mode
 - ▴ Using the mouse select the face of the **Region** at the **xz**-plane

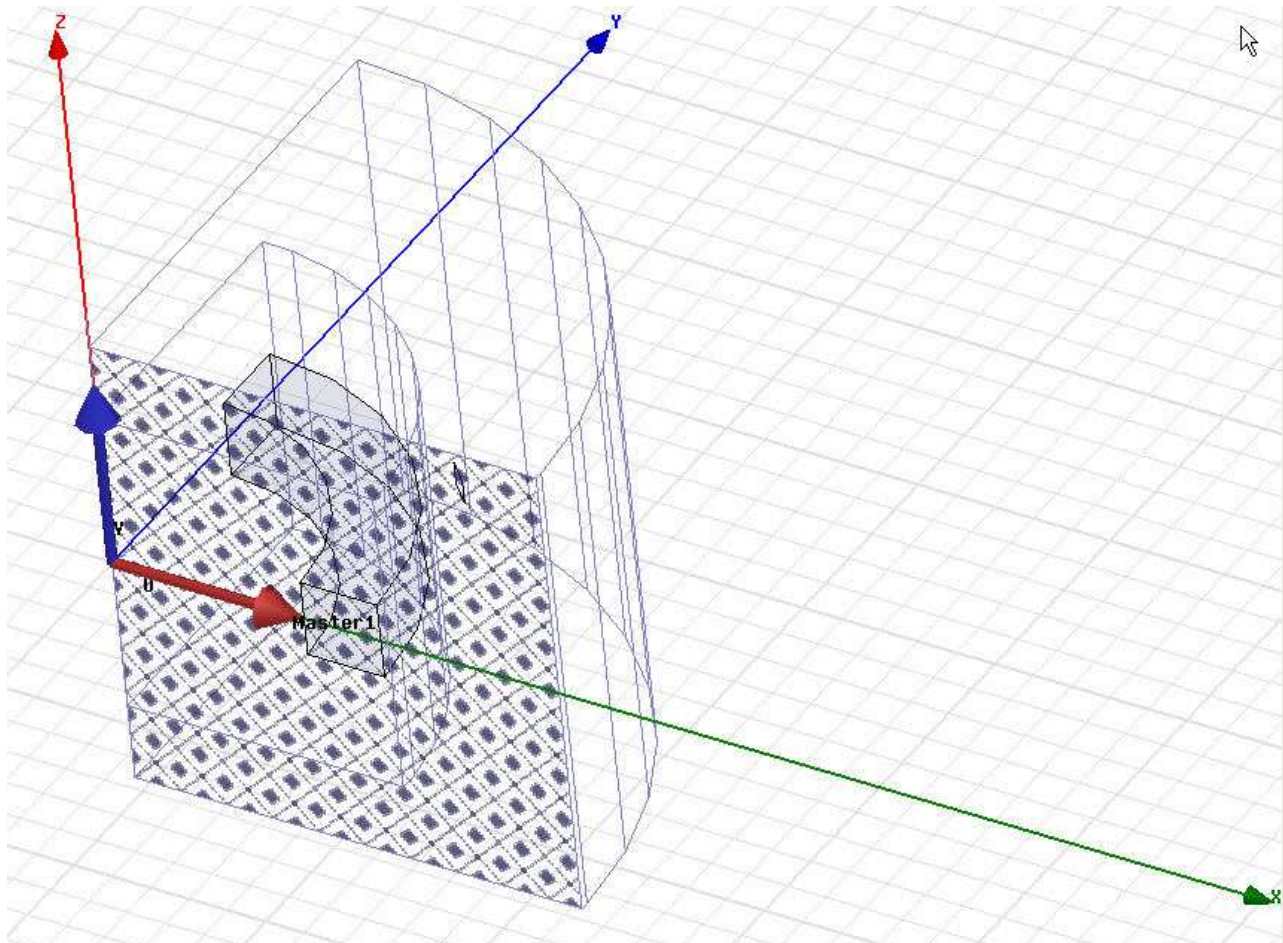


- ▴ To assign Master boundary:
 - Maxwell > Boundaries > Assign > Master**
 - U vector: **New Vector**



Example (Transient) - Rotational Motion

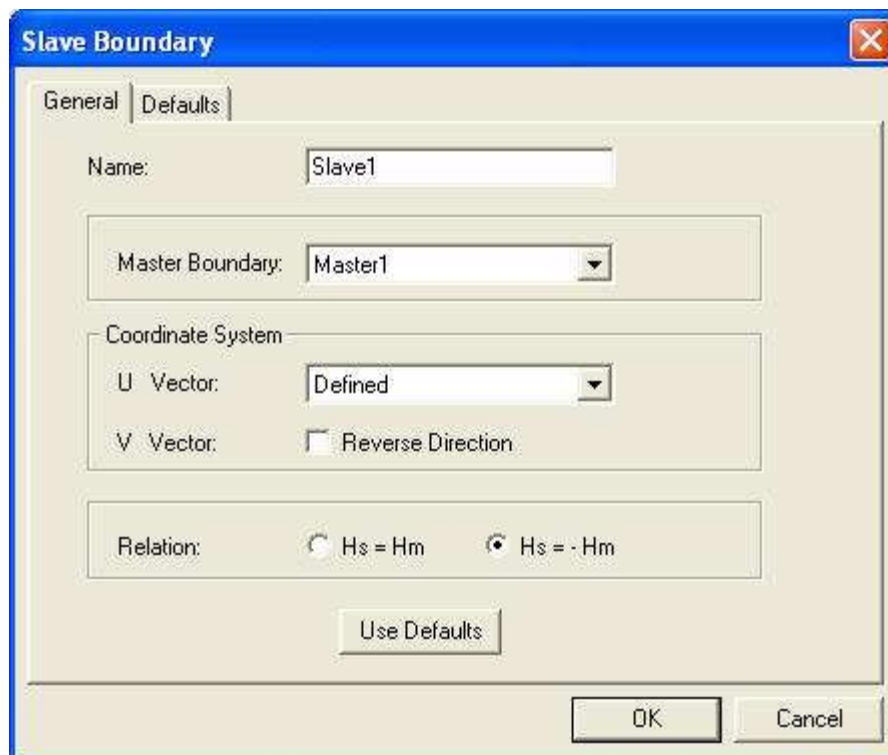
- ▲ Now define the beginning and the end of the U vector:
 - ▲ Using the coordinate entry field, enter the first corner of the rectangle
 - ▲ **X: 0.0, Y: 0.0, Z: 0.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the opposite corner of the rectangle
 - ▲ **dX: 12.0, dY: 0.0, dZ: 0.0**, Press the **Enter** key
- ▲ V vector: check **Reverse Direction**



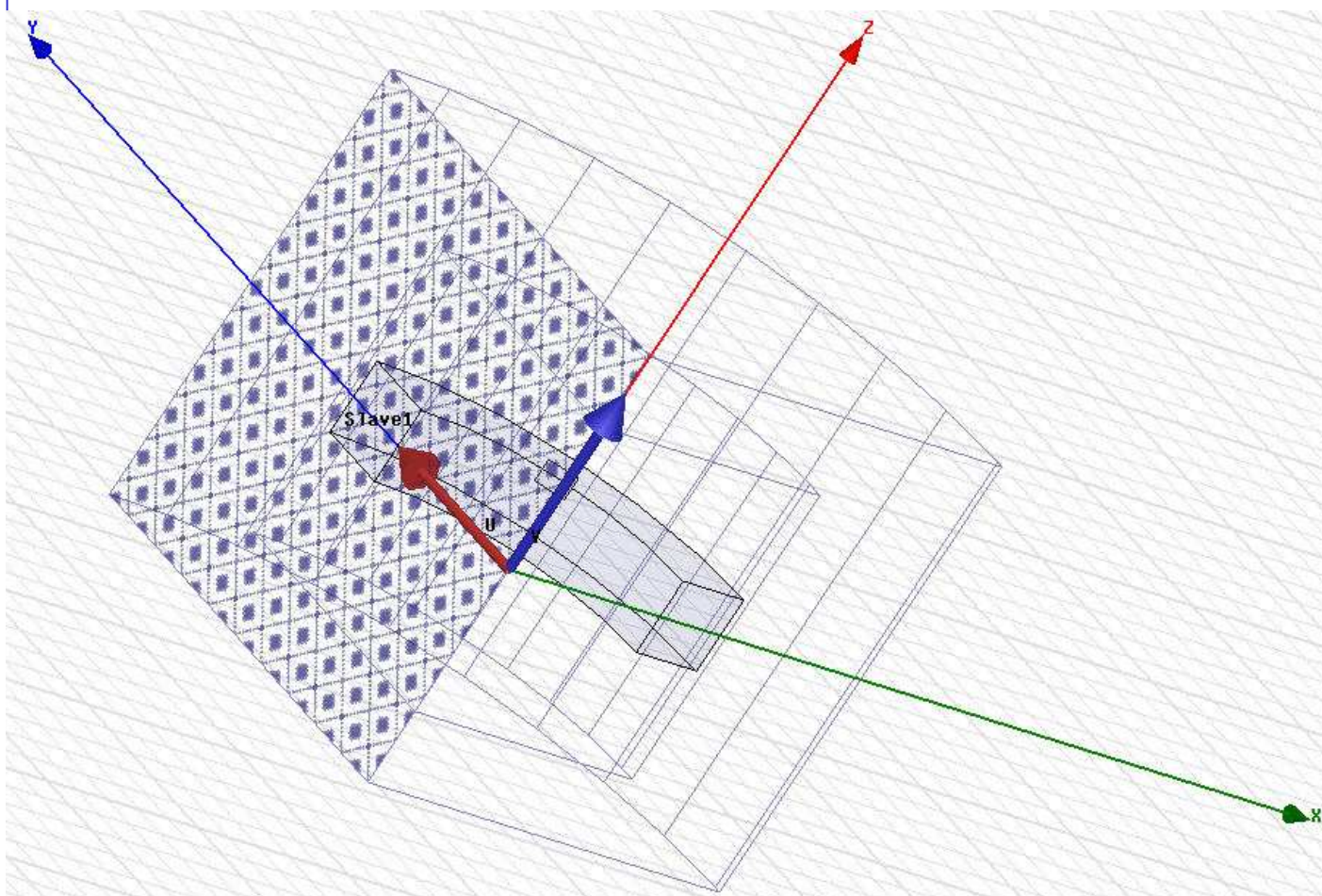
- ▲ Using the mouse select the face of the **Region** at the **yz**-plane
- ▲ To assign Slave boundary:
 - ▲ **Maxwell > Boundaries > Assign > Slave**
 - ▲ U vector: **New Vector**

Example (Transient) - Rotational Motion

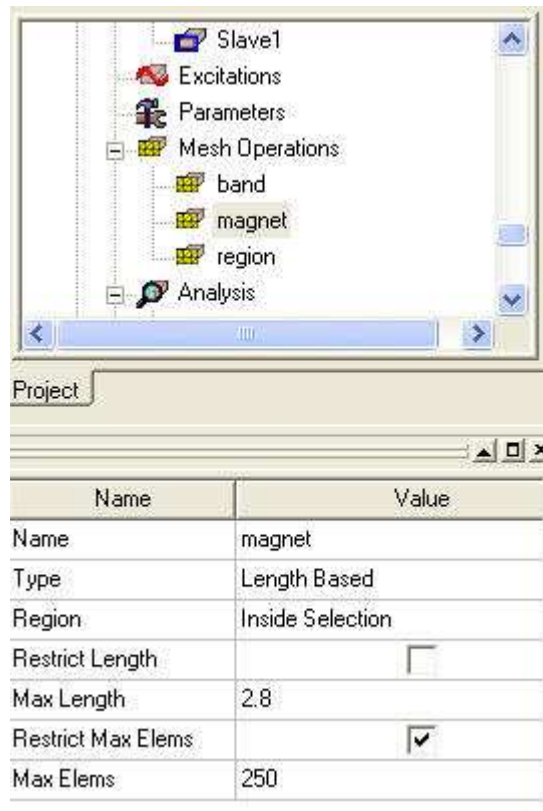
- ▲ Using the mouse select the face of the **Region** at the **yz**-plane
- ▲ To assign Slave boundary:
 - ▲ **Maxwell > Boundaries > Assign > Slave**
 - ▲ U vector: **New Vector**
 - ▲ Now define the beginning and the end of the U vector:
 - ▲ Using the coordinate entry field, enter the first corner of the rectangle
 - ▲ **X: 0.0, Y: 0.0, Z: 0.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the opposite corner of the rectangle
 - ▲ **dX: 0.0, dY: 12.0, dZ: 0.0**, Press the **Enter** key
 - ▲ V vector: check **Reverse Direction**
 - ▲ Relation: **Hs = - Hm** (negative periodicity)
 - ▲ **OK**



Example (Transient) - Rotational Motion



Example (Transient) - Rotational Motion



- ▲ We can now reassign the number of required elements in different objects (magnet, region):
 - ▲ In the Project Manager Window expand **Mesh Operations**
 - ▲ Left-click to select **magnet**
 - ▲ Reduce the number of maximum elements from 1000 to **250**
 - ▲ Repeat the same for region (8000 to **2000**)
- ▲ The surface approximation setting for the **band** remains unchanged.

Example (Transient) - Rotational Motion

- ▲ Since only one quarter of a geometry is considered, the energy, torque and force computed will be one quarter of the value computed with the full geometry. In order to change that, we have to set **Symmetry Multiplier** to 4:
- ▲ **Maxwell > Model > Set Symmetry Multiplier...**
- ▲ Type 4 and OK



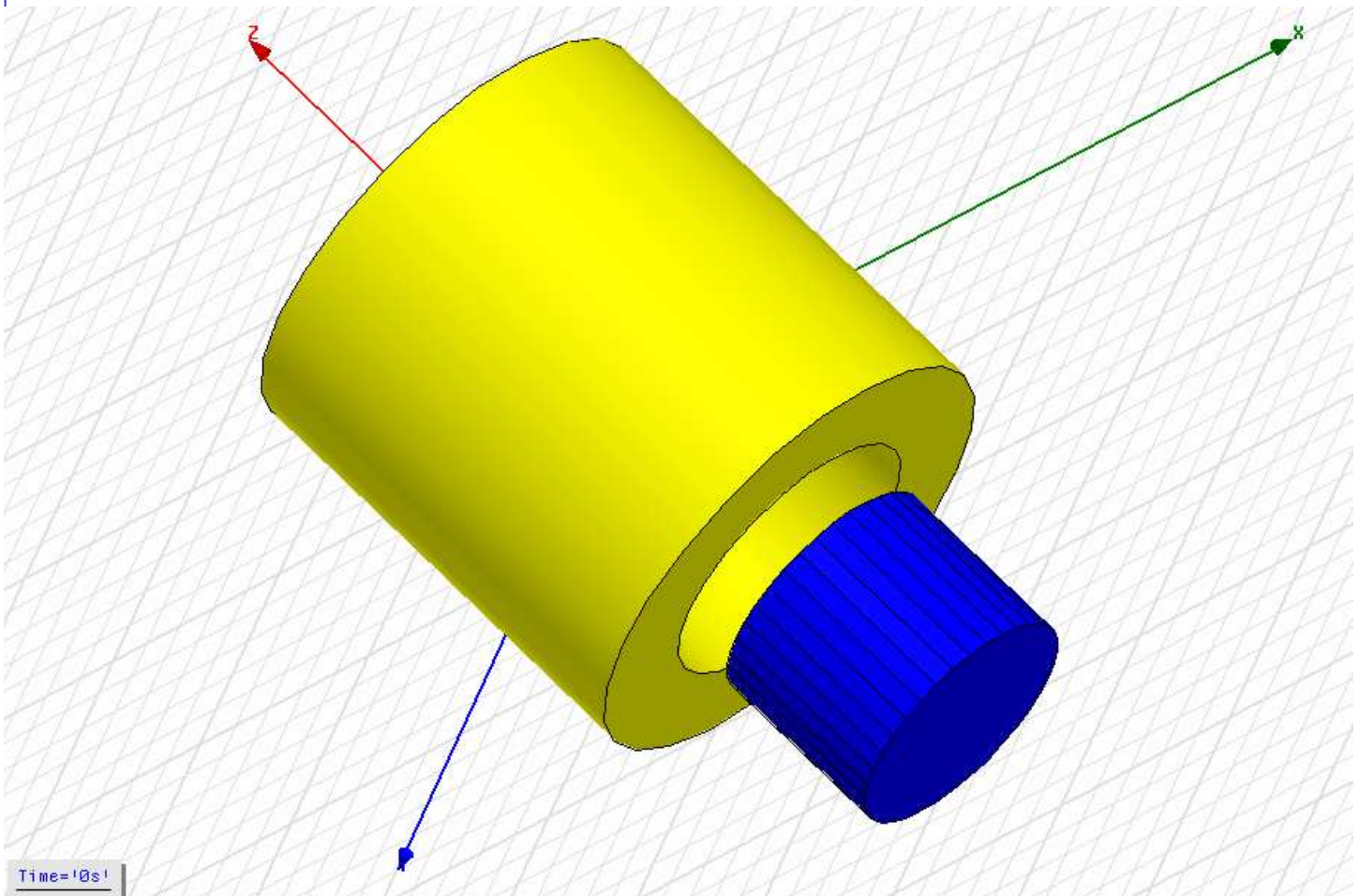
- ▲ **Analyze**
- ▲ Select the menu item **Maxwell > Analyze**.

- ▲ **Results**
- ▲ In the Project Manager Window expand Results and double-click on XY Plot 1 and view the results of Sensor Flux Density plot versus time. This plot should be very similar to the one already created for the full geometry.

Example (Transient) - Translational Motion

▲ Translational Motion using the Transient Solver

- ▲ Transient Solver in Maxwell allows to be combined with motion. This motion can be translational, rotational cylindrical or rotational non-cylindrical (relay-type motion). This example shows on a simple geometry how to set-up a translational motion project using the Transient Solver of Maxwell.
- ▲ The geometry consists of a cylindrical NdFeB magnet with uniform distribution of magnetization in the axial direction (z-direction). The magnet moves through a copper hollow cylindrical structure (coil) with a constant speed of 1 mm per second. As the magnet moves the eddy currents are induced and flow in the copper structure. The goal is to model a time-dependent magnetic field created by moving magnet and determine the eddy currents in the copper structure.



Example (Transient) - Translational Motion

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: **Regular Polyhedron, Rectangle**
-  Boolean Operations: **Split**
-  Sweep Operations: **Around Axis**
-  Model: **Band for translational motion**

- **Boundaries/Excitations**

-  **Natural outer boundary**

- **Material properties**

-  **Edit material (NdFeB axially magnetized magnet with uniform distribution of magnetization)**

- **Analysis**

-  **Transient**

- **Results**

-  **Fields Calculator**
-  **Rectangular Plot**

- **Field Overlays:**

-  **Current Density Vector Plots**
-  **Current Density Vector Animation**

Example (Transient) - Translational Motion

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**

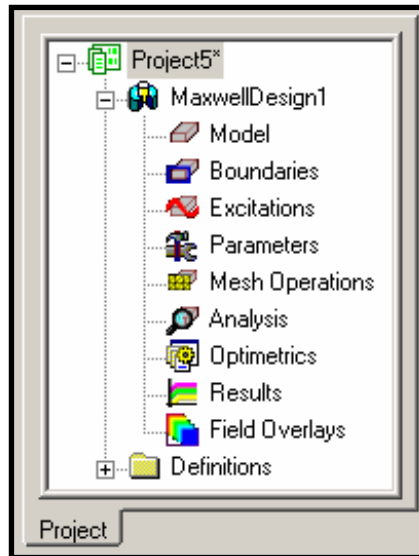
3. Click the **OK** button

Example (Transient) - Translational Motion

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



To rename the design: Right-click on **MaxwellDesign1** and select **Rename**; change the name from **MaxwellDesign1** to **moving_magnet**

Set Solution Type

Select the menu item: **Maxwell > Solution Type** and specify **Transient** or right mouse click on **MaxwellDesign1** and select **Solution Type** and specify **Transient**

Set Model Units

Select the menu item **3D Modeler > Units > Select Units: cm (centimeters)**

Example (Transient) - Translational Motion

Create Magnet

- ▲ Regular Polyhedron with Outer Radius: 1.3 cm and Height: 2 cm positioned 1 cm below **XY** plane:
 - ▲ **3D Modeler > Drawing Plane > XY**
 - ▲ **Draw > Regular Polyhedron**
 - ▲ Using the coordinate entry field, enter the center of the base of the regular polyhedron
 - ▲ **X: 0.0, Y: 0.0, Z: -1.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the radius
 - ▲ **dX: 0.0, dY: 1.3, dZ: -1.0**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the height
 - ▲ **dX: 0.0, dY: 0.0, dZ: 2.0**, Press the **Enter** key
 - ▲ Number of Segments: **36**
 - ▲ Click on the Attribute Tab and change the name from **Regular Polyhedron 1** to **Magnet**
 - ▲ Change the Color to **Blue**

Create Coil and Coil Terminal

- ▲ Outer Radius: 2.5 cm; Inner Radius: 1.5 cm; Height: 5 cm
- ▲ Create a rectangle in **XZ** plane and sweep it around z-axis
 - ▲ **3D Modeler > Drawing Plane > XZ**
 - ▲ **Draw > Rectangle**
 - ▲ Using the coordinate entry field, enter the first corner of the rectangle
 - ▲ **X: 0.0, Y: 1.5, Z: -2.5**, Press the **Enter** key
 - ▲ Using the coordinate entry field, enter the opposite corner of the rectangle
 - ▲ **dX: 0.0, dY: 1.0, dZ: 5.0**, Press the **Enter** key
 - ▲ Click on the Attribute Tab and change the name from **Rectangle 1** to **CoilTerminal**
 - ▲ Select **CoilTerminal** and **Edit > Copy Edit > Paste** (a copy CoilTerminal1 will be created)
 - ▲ **Edit > Select > By Name - CoilTerminal 1**
 - ▲ **Draw > Sweep > Around Axis** - choose **z** as sweep axis and **360** degrees as Angle of sweep
 - ▲ Click on the Attribute Tab and change the name from **CoilTerminal 1** to **Coil**
 - ▲ Change the Color to **Yellow**
 - ▲ Transparent: **0.7**

Example (Transient) - Translational Motion

Create Band

- ▲ In Transient Solver, Band is an object that contains all moving objects. In other words it is a way how to tell Maxwell which objects are steady and which are moving. In this example there is one moving (translating) object: **Magnet**. For translational type of motion Band cannot be a true surface object (cylinder). It must be a faceted object, such as Regular Polyhedron:

- ▲ **3D Modeler > Drawing Plane > XY**

- ▲ **Draw > Regular Polyhedron**

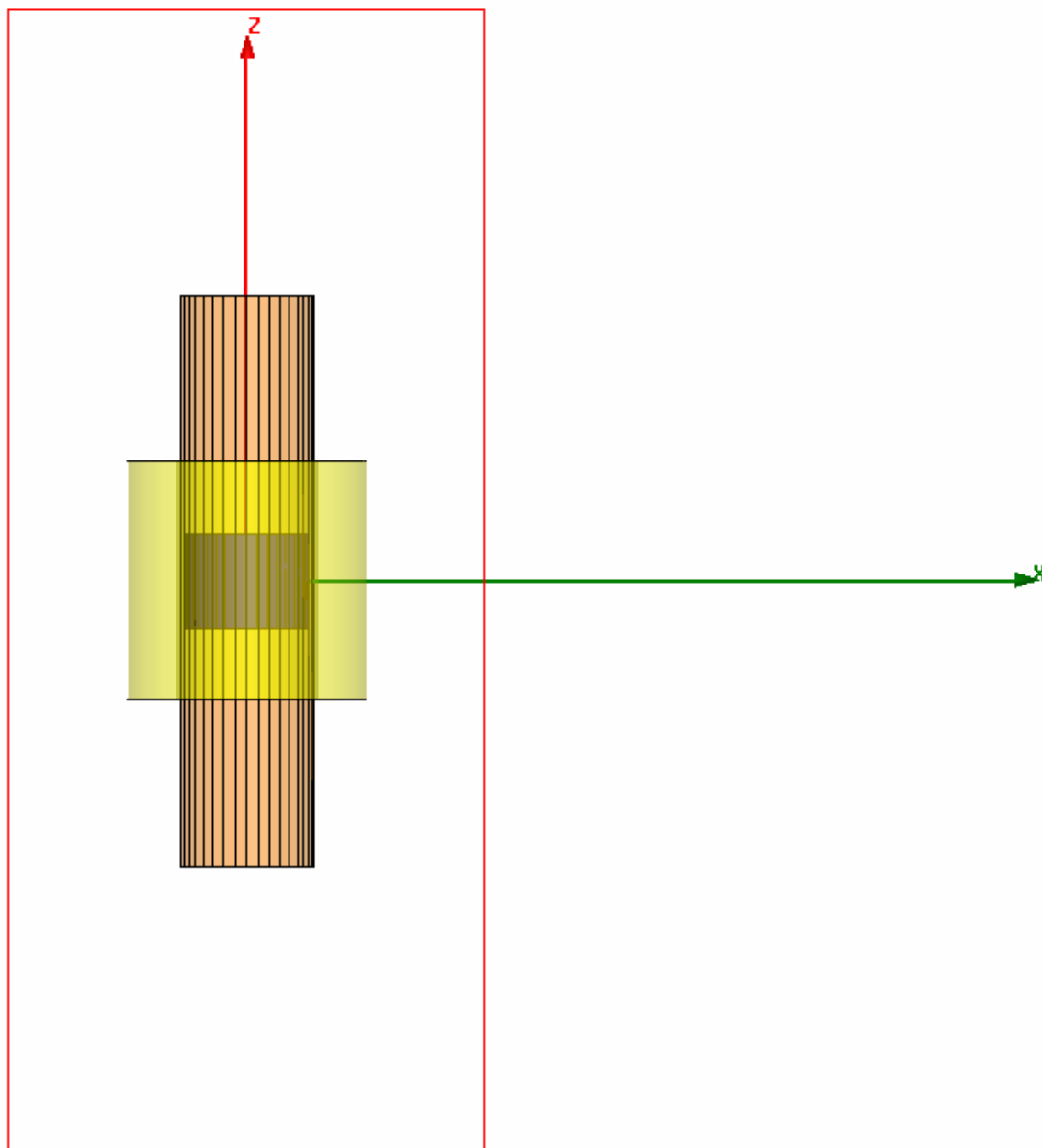
- ▲ Using the coordinate entry field, enter the center of base:
▲ **X: 0.0, Y: 0.0, Z: -6.0**, Press the **Enter** key
- ▲ Using the coordinate entry field, enter the radius:
▲ **dX: 0.0, dY: 1.4, dZ: -6.0**, Press the **Enter** key
- ▲ Using the coordinate entry field, enter the height:
▲ **dX: 0.0, dY: 0.0, dZ: 12.0**, Press the **Enter** key
- ▲ Number of Segments: **36**
- ▲ Click on the Attribute Tab and change the name from **Cylinder 1** to **Band**
- ▲ Transparent: **0.7**
- ▲ Change the Color to **Brown**

Create Outer Boundary (Region)

- ▲ **Draw > Region**. Padding Data: **One**; Padding Percentage: **50**; **OK**



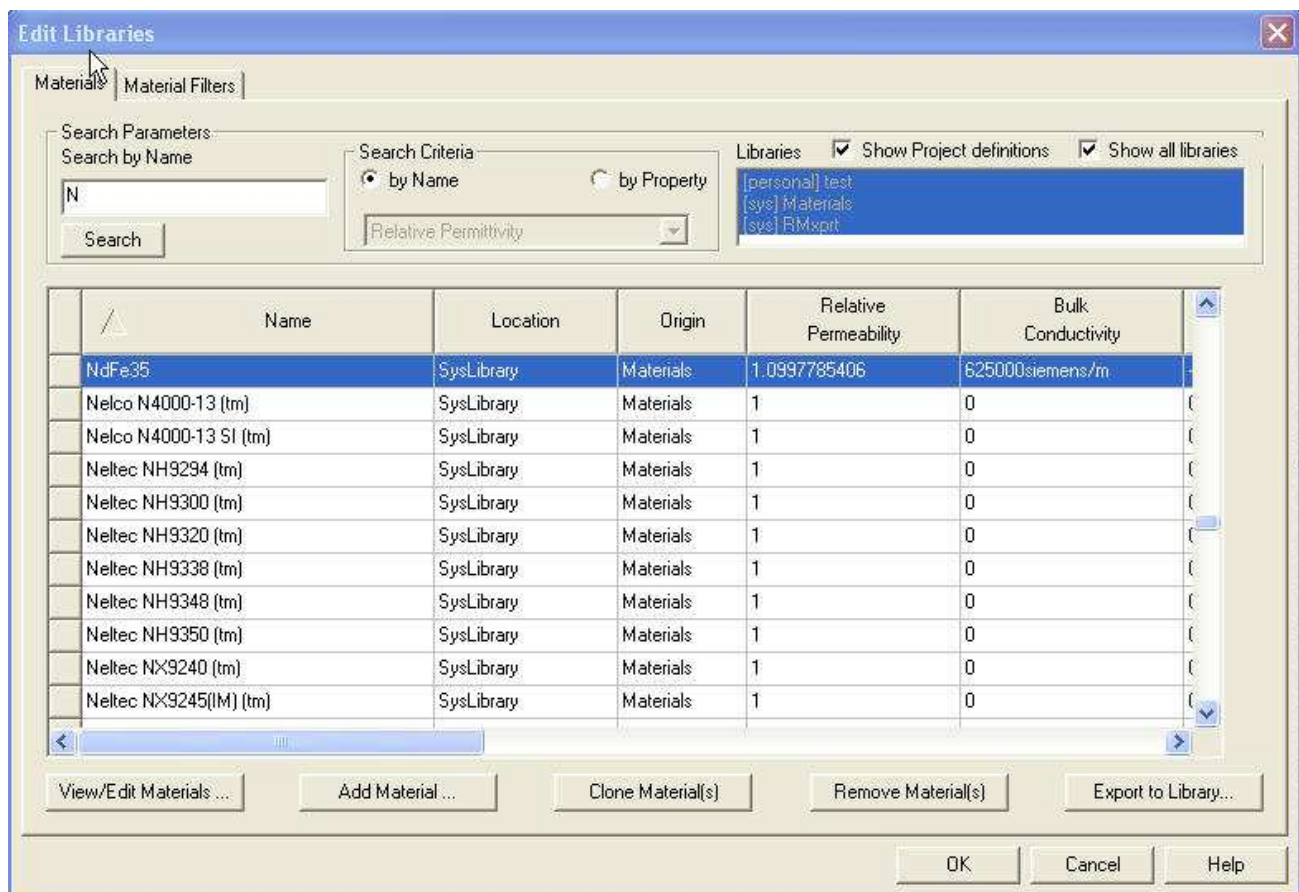
Example (Transient) - Translational Motion



Example (Transient) - Translational Motion

Define and assign material properties

- ▲ **Magnet** and **Coil** need material assignment. All other objects are vacuum by default.
- ▲ To define a new NdFeB35 magnet material with uniform distribution of magnetization in the axial (positive z) direction:
 - ▲ **Tools > Edit Configured Libraries > Materials**



- ▲ Select **NdFeB35** and click on **View/Edit Materials** (start typing NdFeB into the Search by Name window for faster selection)

Example (Transient) - Translational Motion

Specify the following:

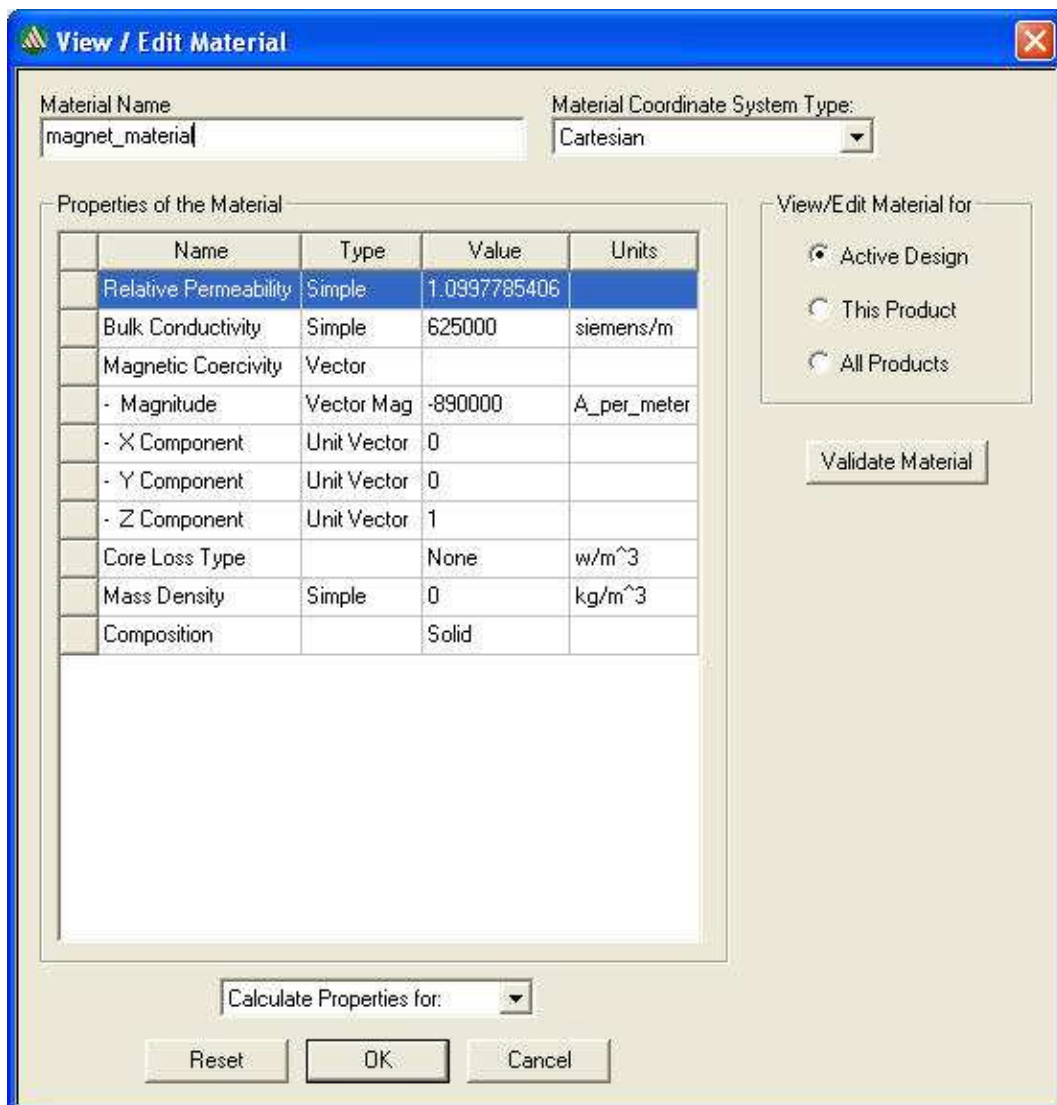
Material Name: **magnet_material**

Material Coordinate System Type: **Cartesian**

The direction of magnetization is determined by Unit Vector. For the Cartesian coordinate system the components of Unit Vector are **X**, **Y** and **Z**. The values of **X**, **Y** and **Z** are 1, 0, 0 by default. For this example they have to be changed, because the magnet is magnetized in positive **z**-direction:

X: 0; Y: 0; Z: 1

Click on **Validate Material** to check the settings (green check mark indicates that everything is all right); **OK**



View / Edit Material

Material Name: Material Coordinate System Type:

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Simple	1.0997785406	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-890000	A_per_meter
- X Component	Unit Vector	0	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	1	
Core Loss Type		None	w/m^3
Mass Density	Simple	0	kg/m^3
Composition		Solid	

View/Edit Material for:

☒ Active Design
☐ This Product
☐ All Products

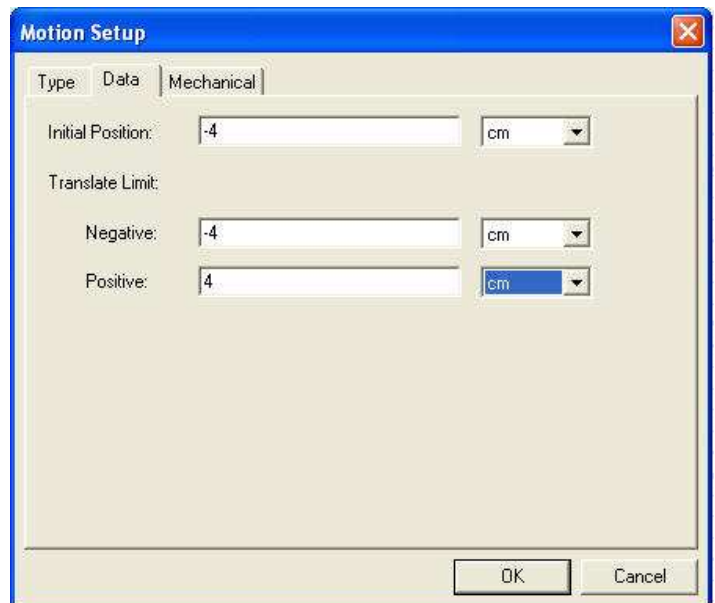
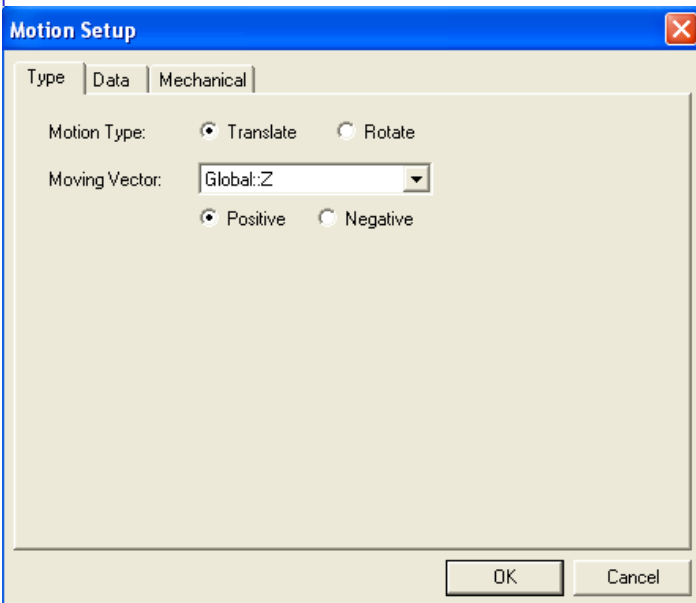
Calculate Properties for:

Example (Transient) - Translational Motion

- ▲ To assign a newly created **magnet_material** to object **Magnet**:
 - ▲ Select **Magnet**:
 - ▲ **Edit > Select > By Name** - select **Magnet** and press **OK** key
 - ▲ Assign material:
 - ▲ **3D Modeler > Assign Material** - select **magnet_material** and press **OK** key
- ▲ To assign material **copper** to object **Coil**:
 - ▲ Select **Coil**:
 - ▲ **Edit > Select > By Name** - select **Coil** and press **OK** key
 - ▲ Assign material:
 - ▲ **3D Modeler > Assign Material** - select **copper** and press **OK** key

Define Band and Motion Setup

- ▲ Select **Band**:
 - ▲ **Edit > Select > By Name** - select **Band** and press **OK** key
- ▲ Assign **Band**:
 - ▲ **Maxwell > Model > Motion Setup > Assign Band**

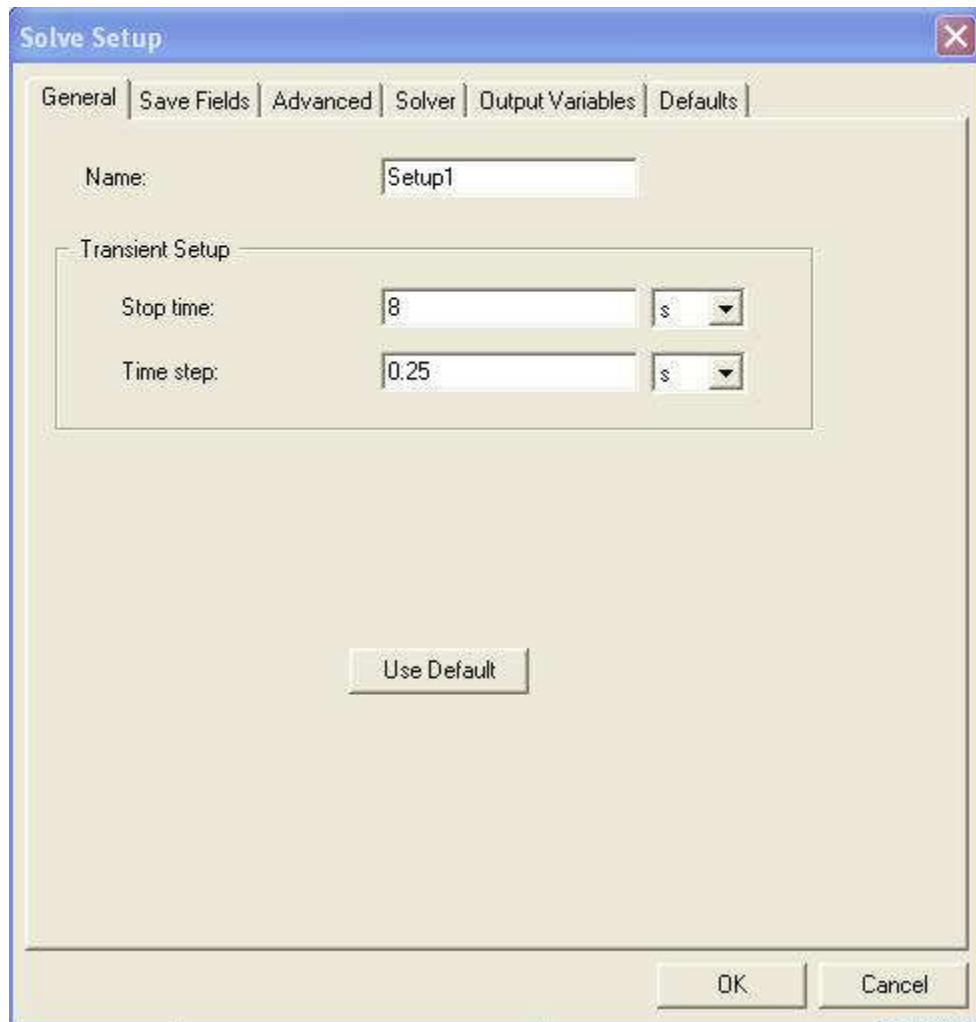


- ▲ Select **Type** Tab and specify **Translate** radio button.
- ▲ Select **Data** Tab and specify: Initial Position: **-4 cm**; Translate Limit: Negative: **-4 cm**; Positive: **4 cm**
- ▲ Select **Mechanical** Tab and specify **Angular Velocity: 1 cm_per_s**
(The magnet moves at constant speed of 1 cm per second in the positive z direction.)

Example (Transient) - Translational Motion

▲ Create an Analysis Setup

- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**
- ▲ **General** Tab: Stop time: **8 s**; Time step: **0.25 s**
- ▲ **Save Fields** Tab: Type: **Linear Step**; Start: **0**; Stop: **8**; Step Size: **0.5** and click on **Add to List >>**. Finally press **OK** key.
(The field solution will be saved at the specified time instants.)



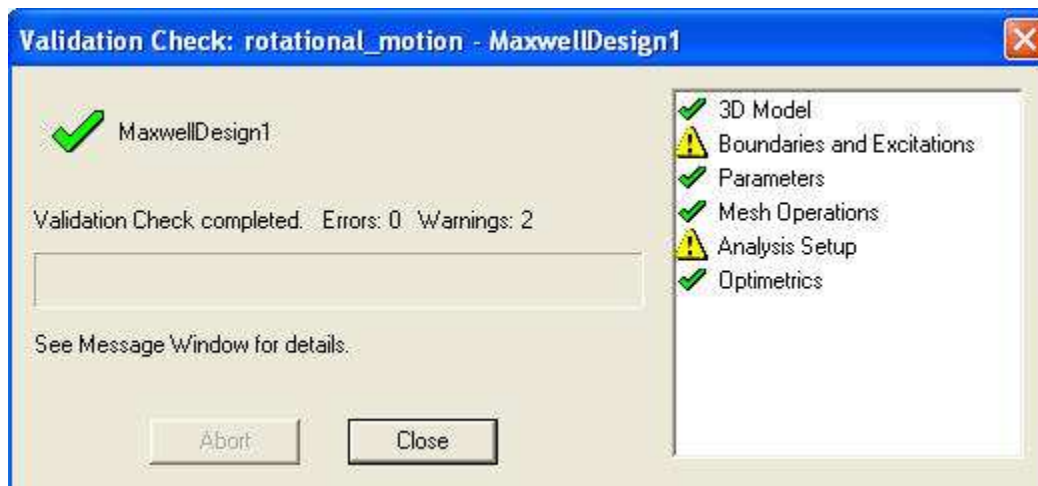
Example (Transient) - Translational Motion

Save the Project

- ▲ Select the menu item **File > Save As**
- ▲ From the Save As window, type in **translational_motion**
- ▲ 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 will not solve unless each item has a check mark. We can see that at the moment there are 2 warnings. From the **Message Window** we can read:

1. **Warning:** Eddy effect settings may need revisiting due to the recent changes in the design.

In this example we are particularly interested to compute eddy currents induced in the solid copper coil. We have to make sure that the objects **Coil** is assigned for eddy current computation:

- ▲ **Maxwell > Excitations > Set Eddy Effects**
- ▲ Make sure that the box next to **Coil** is checked and all other boxes are unchecked (**Magnet** is unchecked)

Example (Transient) - Translational Motion

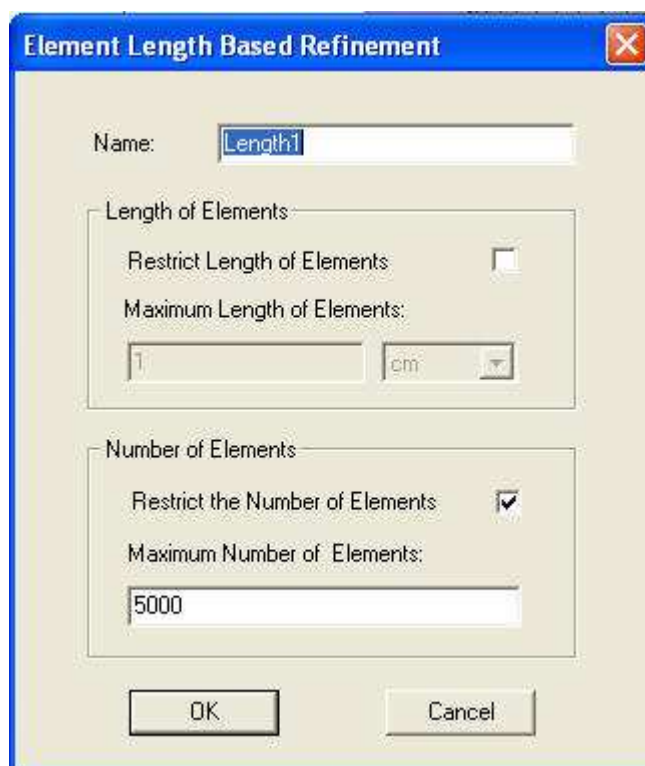


2. **Warning:** Currently no mesh operations or data link have been created for the setup(s): Setup1. The initial mesh will be used. The initial mesh is usually not fine enough to achieve an accurate solution.

For demonstration purposes in this particular example we are interested to see the induced eddy currents at the cross section of the coil. Let us therefore refine the cross section by assigning appropriate mesh operations:

- To refine the mesh by assigning the mesh operations:
 - Select **CoilTerminal**:
 - **Edit > Select > By Name** - select **CoilTerminal** and press OK
 - Assign mesh operations:
 - **Maxwell> Mesh Operations > Assign > Inside Selection > Length Based...**

Example (Transient) - Translational Motion



- ▲ Name: **Length1**
- ▲ Uncheck **Restrict Length of Elements**
- ▲ Check **Restrict the Number of Elements**
- ▲ Specify **5000** for **Maximum Number of Elements**
- ▲ Performing the Validity Check again gives the green check marks for all the objects. The project is ready to be analyzed.

▲ Analyze

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

Example (Transient) - Translational Motion

▲ Solution Data

- ▲ To view the Solution Data, select the menu item **Maxwell > Results > Solution Data**
- ▲ Here you can view the **Solution** (if any, e.g. force) , **Profile** and the **Mesh Statistics**.

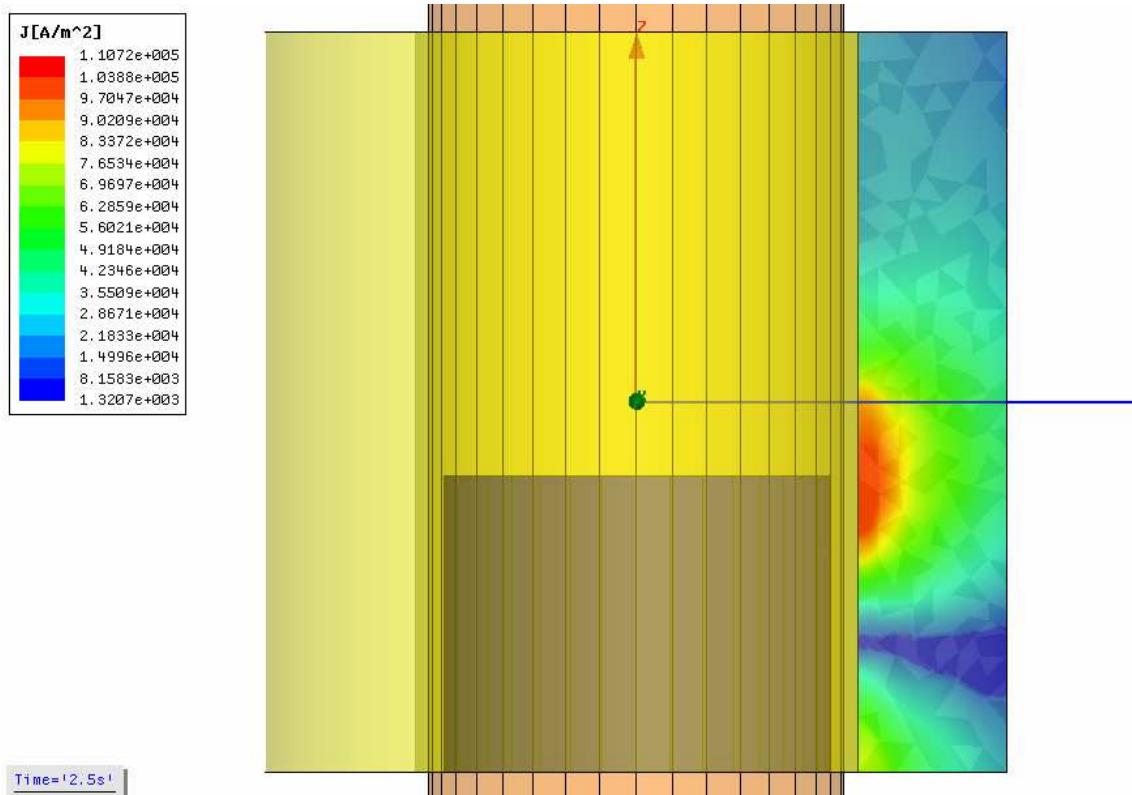
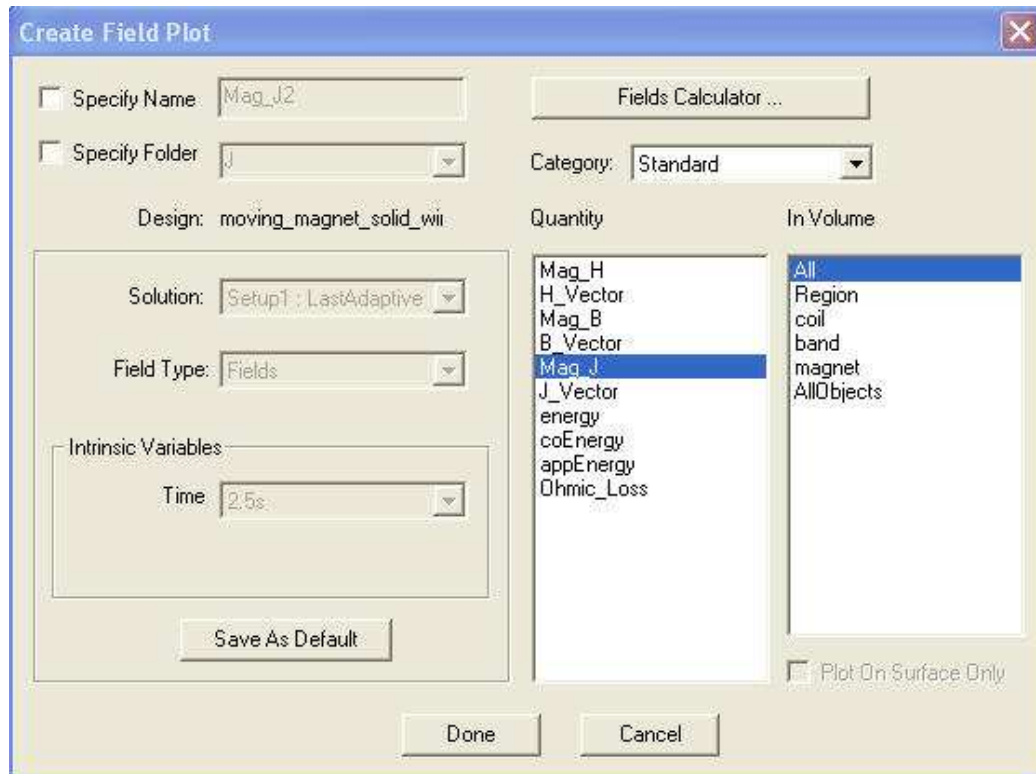
▲ Plot Current Density Magnitude across the Coil Terminal at time 2.5 s

- ▲ To plot the flux density at specified time instant:
 - ▲ **View > Set Solution Context**



- ▲ For **Time** specify **2.5 s**.
- ▲ Select **CoilTerminal**
- ▲ **Maxwell > Fields > Fields > J > mag_J**
- ▲ Press the **Done** key

Example (Transient) - Translational Motion



Example (Transient) - Translational Motion

Plot Induced Current through the coil as a function of time

- ▲ To define a function **It** characterizing the current through the surface of CoilTerminal (use **Fields Calculator**):

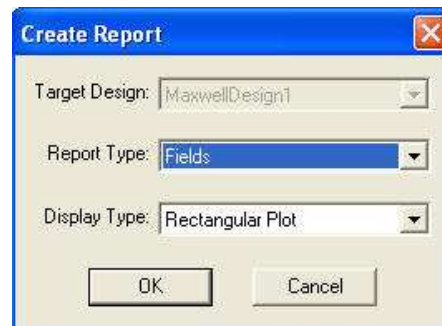
- ▲ **Maxwell > Fields > Calculator ...**

1. Input > Quantity > J
2. Input > Geometry > Surface > CoilTerminal
3. Vector > Normal
4. Scalar > Integrate (the integral symbol)
5. Add > Named Expression > Name: **It** > OK
6. Click on **Done** key

- ▲ To plot the defined function **It** as a function of time:

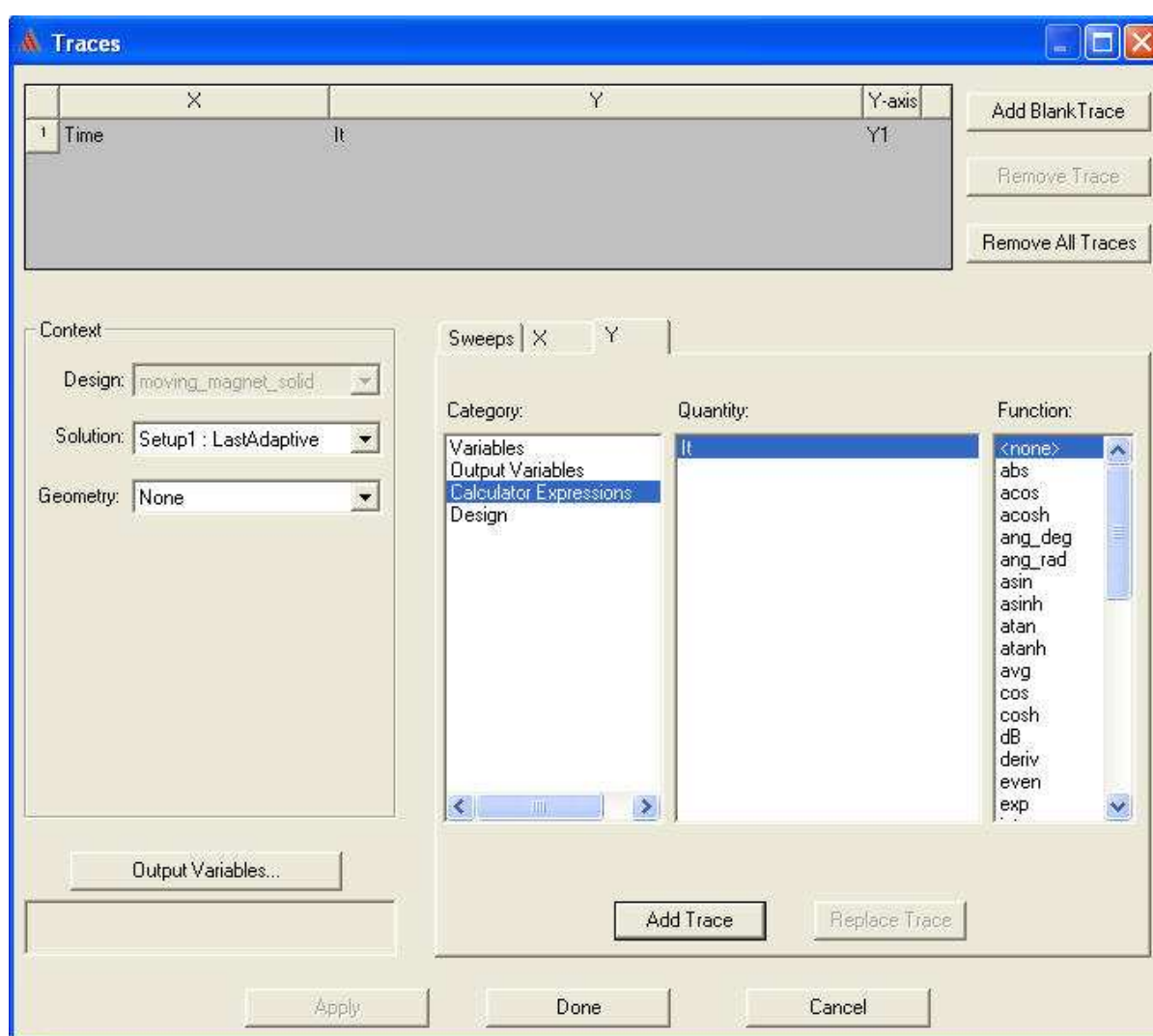
- ▲ **Maxwell > Results > Create Report**

- ▲ Specify: Report Type: **Fields**; Display Type: **Rectangular Plot**

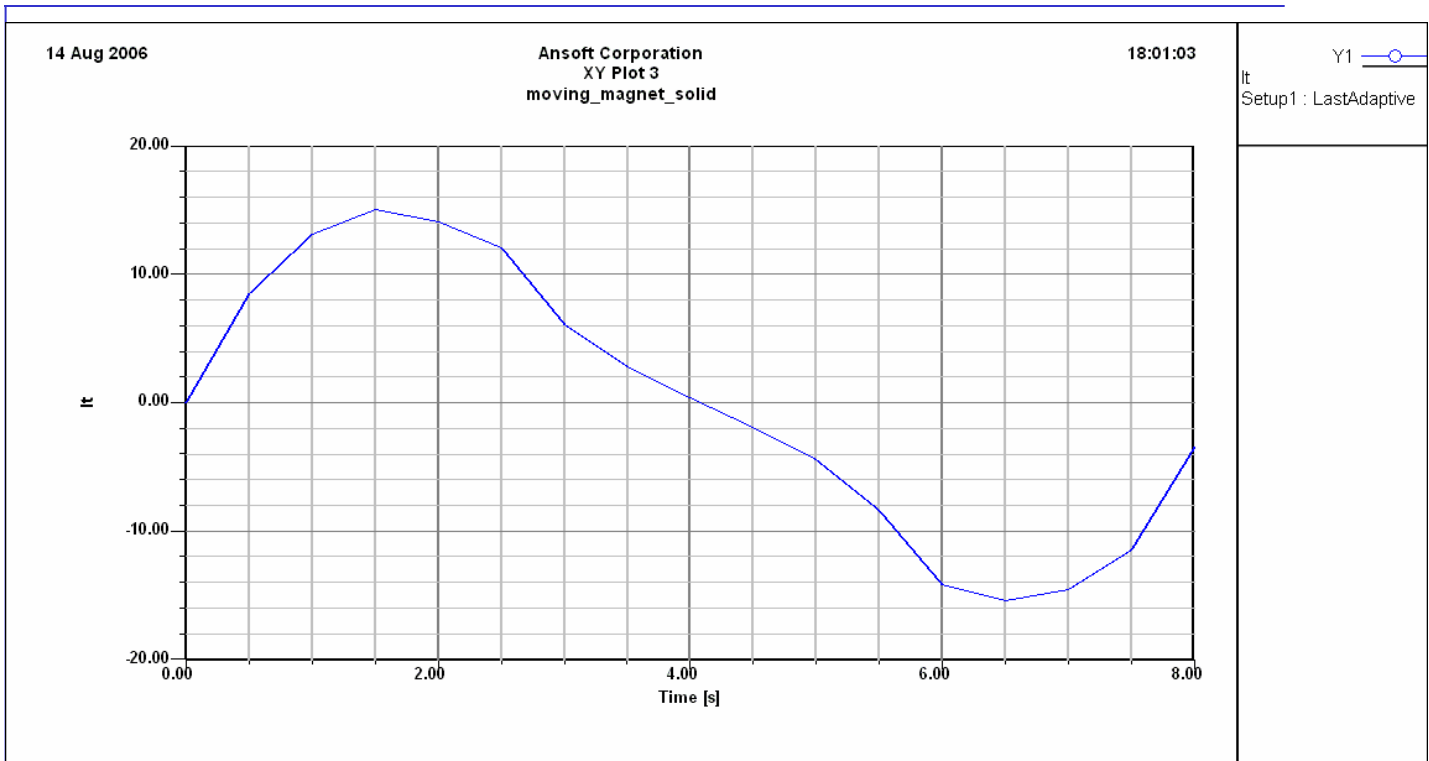


Example (Transient) - Translational Motion

- ▲ In the **Traces** Window specify from the Y Tab:
 - ▲ Category: **Calculator Expressions**
 - ▲ Quantity: **It**
 - ▲ Function: **none**
- ▲ Click on **Add Trace** key and **Done** key



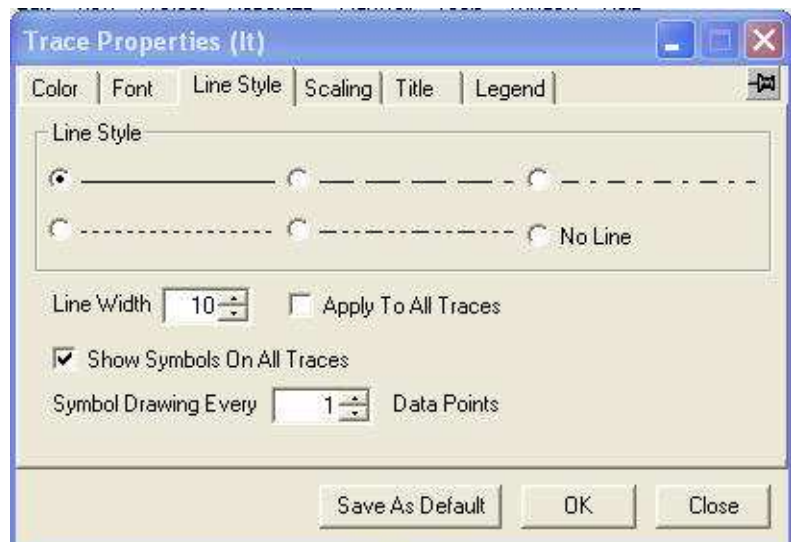
Example (Transient) - Translational Motion



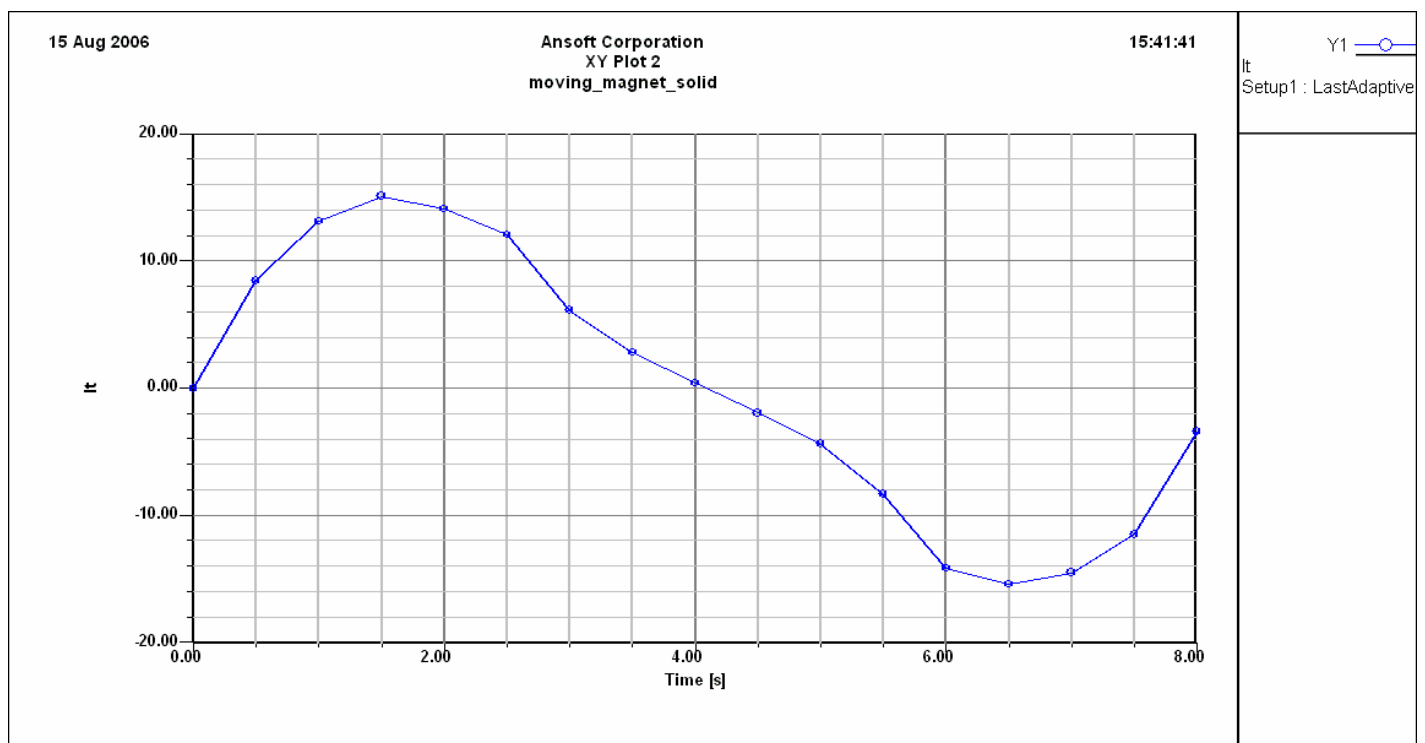
- ▲ The generated trace is continuous without any discrete symbols showing where the calculated points are. In order to show the trace with the discrete calculated pints, the current 2D report has to be modified:
- ▲ Select the trace you want to modify:
 - ▲ Point the mouse pointer on the trace and left-click or left-click on the area in the rectangle in the upper right corner specifying the trace you want to modify. The selected trace changes its color.

Report2D > Properties

- ▲ Select a **Line Style** Tab
- ▲ Check Show Symbol on all Traces
- ▲ Symbol Drawing Every 1 Data Points
- ▲ **OK**



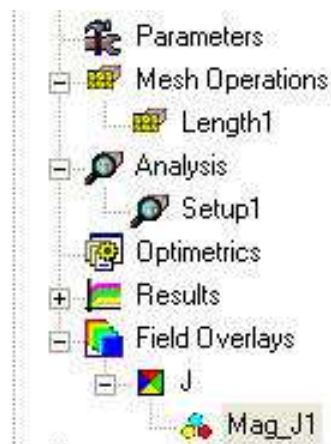
Example (Transient) - Translational Motion



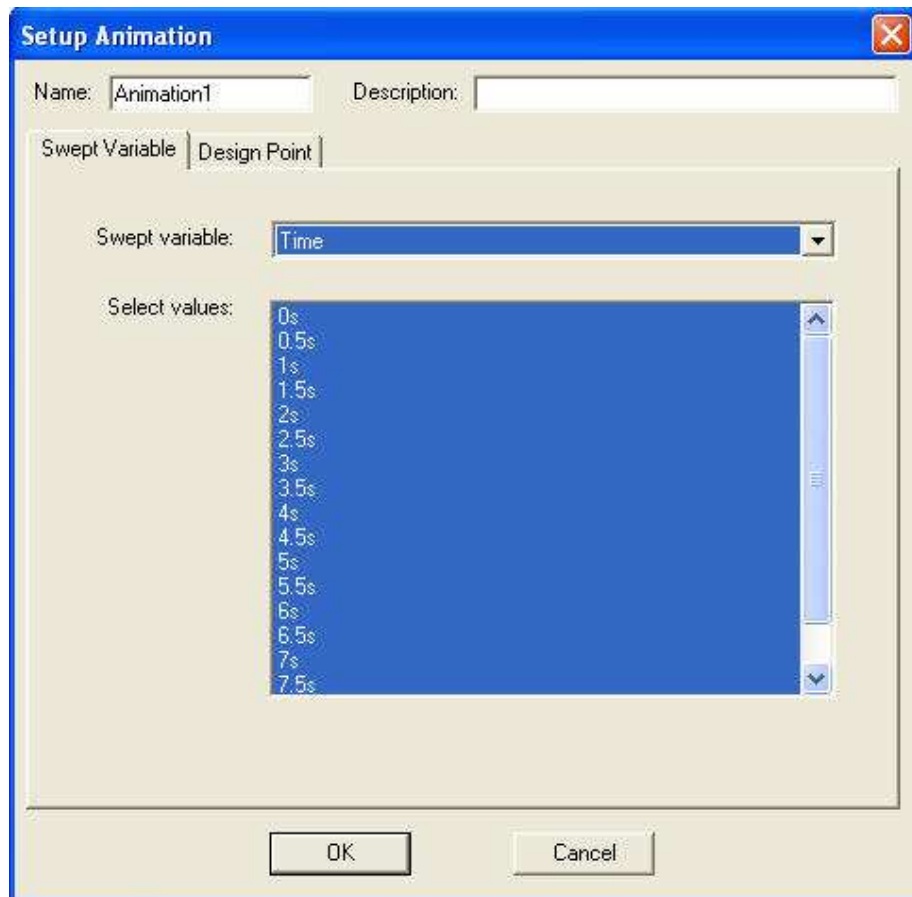
- ▲ Note that the points appear every 0.5s, even though the specified time step was 0.25s. The reason for this is that the quantity I_t is a field associated quantity. The field associated quantities are computed as post-processing quantities from the saved field solution, which in our case is every 0.5s.

▲ Create an Animation of Current Density Magnitude Plot

- ▲ Select already created plot of Current Density
 - ▲ At the Project Manager Window expand **Field Overlays** and **J**
 - ▲ Double-click on **Mag_J1** to bring this plot back to view
 - ▲ Right-click on **Mag_J1**
 - ▲ Select **Animate** and Click on **New...**

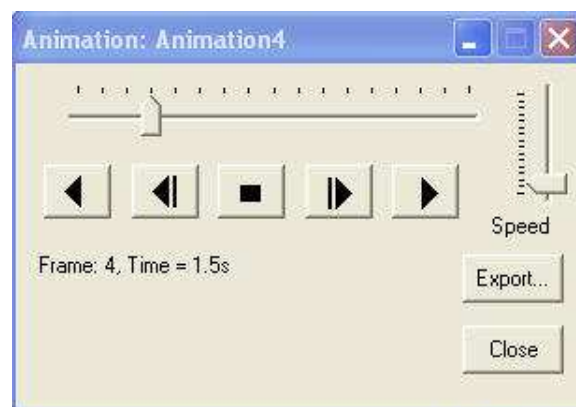


Example (Transient) - Translational Motion



- ▲ In Setup Animation Window and Swept Variable Tab:
- ▲ Swept Variable: **Time**
- ▲ Select Values: Holding down the Shift key select all the values
- ▲ Press the **OK** key

- ▲ To start and stop the animation, to control the speed of the animation and to export the animation use the buttons on the Animation Panel:



THIS PAGE INTENTIONALLY LEFT BLANK



Chapter 8.0 - Electric Examples



8.1 -

THIS PAGE INTENTIONALLY LEFT BLANK

Chapter 9.0 - Basic Exercises

-  9.1 - Electrostatic
-  9.2 - DC Conduction
-  9.3 - Magnetostatic
-  9.4 - Parametric
-  9.5 - Transient
-  9.6 - Transient with Circuit Editor

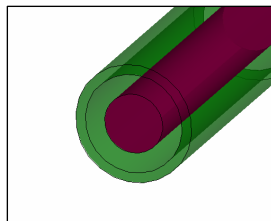
THIS PAGE INTENTIONALLY LEFT BLANK

Introduction on the Electrostatic Solver

-  This note introduces the Electro Static solver based on some simple examples. This solver is meant to solve the static electric field without current flowing in conductors (conductors are in electrostatic equilibrium). The conductors are considered perfect such that there is no electric field inside conductors.

Capacitance of a Cylindrical Capacitor

-  Suppose we have a long coaxial line. We want to know what is the electric field distribution based on the potential (or the charges) that are applied on each conductor. We also want to determine the capacitance.



Draw the Model

-  Click on the menu item **Project > Insert MaxwellDesign**
-  Click on the menu item **Maxwell > Solution Type**
 -  Select **Electric**
 -  Select the radio button **Electrostatic**
-  Click on the menu item **Draw > Cylinder** or click on the  icon
 -  For center position, enter **0;0;- 4 mm**
 -  For radius, enter **0.6 mm**
 -  For height, enter **25 mm**
 -  Change the name to **Inner**
 -  Change the material to **copper**
 -  Change the color and transparency level at your convenience.

Basic Exercises - Electrostatic Solver

- ▲ Create a cylinder
 - ▲ For center position, enter **0; 0 ; - 4 mm**
 - ▲ For radius, enter **1.2 mm**
 - ▲ For height, enter **25 mm**
 - ▲ Change the name to **Outer**
 - ▲ Change the material to **copper**
 - ▲ Change the color and transparency level at your convenience.
- ▲ Create a third cylinder
 - ▲ For center position, enter **0; 0; - 4mm**
 - ▲ For radius, enter **1. mm**
 - ▲ For height, enter **25 mm**
 - ▲ Change the name to **Air**
 - ▲ Assign material to **vacuum**
 - ▲ Change the color and transparency level at your convenience.
- ▲ Subtract **Air** from **Outer**
 - ▲ Select **Air** and **Outer**
 - ▲ **3D Modeler > Boolean > Subtract**
 - ▲ Blank Parts: **Outer**; Tool Parts: **Air**
- ▲ Select the menu item **Draw > Region**
 - ▲ For the padding data, choose **6**
 - ▲ For the Padding Percentage, enter +/- **300** for X and Y, **0** for +/- Z

▲ Assign Excitation

- ▲ Based on the assumptions that the conductors are in electrostatic equilibrium, we assign voltage potential on the object itself (and not on the surface of the object line in the other solvers).
- ▲ Select the object **Inner**, select the menu item **Maxwell > Excitations > Assign > Voltage**. As an alternative, once the object is selected, you can right click and select **Assign Excitations > Voltage**.
 - ▲ For the voltage, enter **-1kV**
- ▲ Select the object **Outer**, select the menu item **Maxwell > Excitations > Assign > Voltage**.
 - ▲ For the voltage, enter **1kV**

Assign Executive Parameter

- ▲ in addition to the field, we are interested by the Capacitance value as well as the force applied to the inner armature.
- ▲ Select the menu item **Maxwell > Parameters > Assign > Matrix**
 - ▲ Include Voltage1 and Voltage2 in the capacitance computation
- ▲ Select the object Inner
- ▲ Select the menu item **Maxwell > Parameters > Assign > Force**
 - ▲ Choose the virtual force

Create Analysis Setup

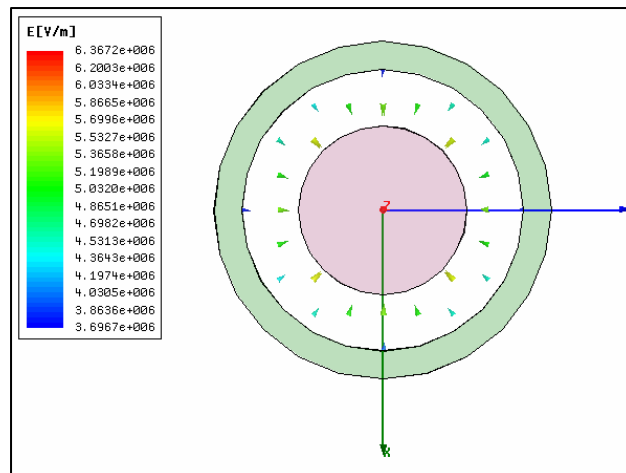
- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**
 - ▲ For the **Percent Error**, enter 5%
 - ▲ For the **Refinement per Pass** (Convergence tab), put 50%

Solve the Problem

- ▲ Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

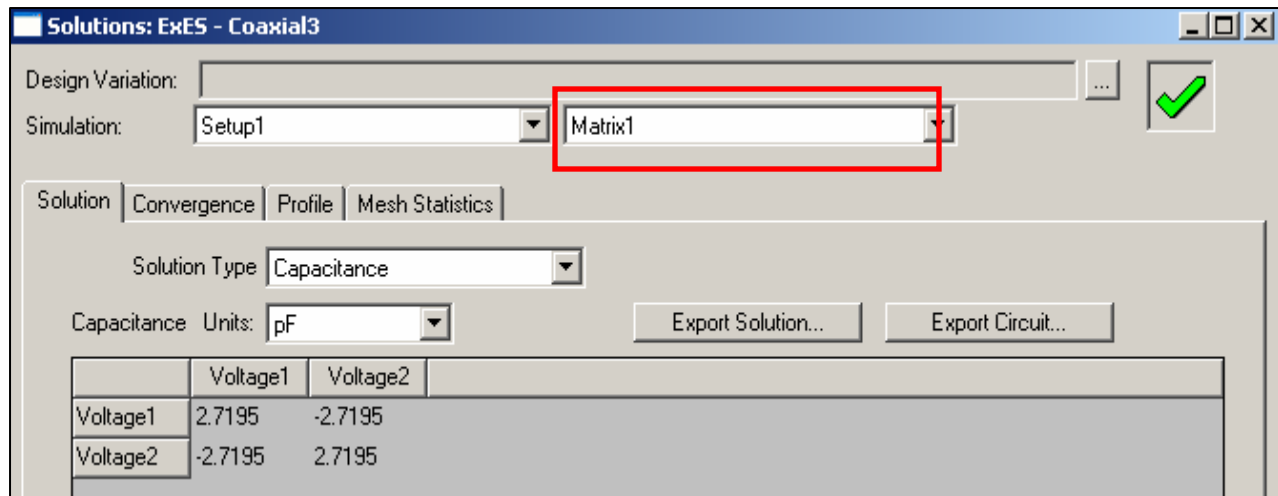
Plot the electric field

- ▲ From the modeler history tree, select the plane **Global:XY**.
- ▲ On the 3D modeler window, right click and select **Fields > E_Vector**



▲ Get the capacitance value

- ▲ From the Project window, right click on Setup1. From the context menu, select the entry Solutions
- ▲ From the pull down menu, select **Matrix1**



- ▲ In our problem, we only have two conductors, therefore the capacitance values are symmetrical.
- ▲ From the pull down menu, select **Force1**. It gives you the force applied to the inner object.
- ▲ The analytical value of the capacitance per meter for an infinite long coaxial wire is given by the following formula:

$$C = 2\pi\epsilon_0 / \ln(b/a) \text{ (a and b being the inside and outside diameters)}$$

- ▲ The analytical value would be therefore **1.089e-10 F/m** (a = 0.6mm, b = 1mm)
- ▲ In our project, then length of the conductor is 25 mm, therefore the total capacitance is **2.722 pF**. We obtain a good agreement with the obtained result.

Capacitance of a planar capacitor

-  In this example we illustrate how to simulate a simple planar capacitor made of two parallel plates

-  **Draw the model**
-  Click on the menu item **Project > Insert MaxwellDesign**
-  Name the design **Plate**
-  Click on the menu item **Maxwell > Solution Type**
 -  Select **Electric**
 -  Select the radio button **Electrostatic**
-  Select the menu item **Draw > Box** to create a plate
 -  For the first position corner, enter **0;0;0 mm**
 -  For the Xsize, enter **25 mm**
 -  For the Ysize, enter **25mm**
 -  For the Zsize, enter **2mm**
 -  For the material property, enter **pec** (perfect conductor)
 -  Name the first box **DownPlate**
-  It is not needed to draw the plate; it is possible in this case just to apply a voltage to the surface of the region.
-  Select the menu item **Draw > Box** to create a plate
 -  For the first position corner, enter **0;0;3 mm**
 -  For the Xsize, enter **25 mm**
 -  For the Ysize, enter **25mm**
 -  For the Zsize, enter **2mm**
 -  Name the box **UpPlate**
-  Create **Region** with 0% Padding Percentage. By doing this all the fringing is neglected.

-  **Assign Excitation**
-  Select the object **UpPlate**, select the menu item **Maxwell > Excitations > Assign > Voltage**. As an alternative, once the object is selected, you can right click and select **Assign Excitations > Voltage**.
 -  For the voltage, enter **0V**
-  Select the upper face of the box **Region**, select the menu item **Maxwell > Excitations > Assign > Voltage**.
 -  For the voltage, enter **1V**

Assign Executive Parameter

- ▲ Select the menu item **Maxwell > Parameters > Assign > Matrix**
 - ▲ Include Voltage1 and Voltage2 in the capacitance computation

Create Analysis Setup

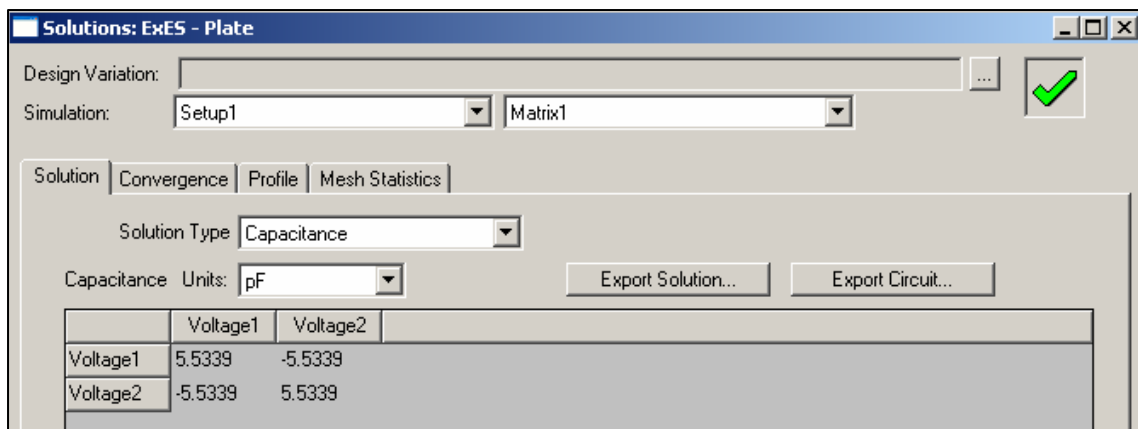
- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**
 - ▲ For the **Percent Error**, enter 1%
 - ▲ For the **Refinement per Pass** (Convergence tab), put 50%

Solve the Problem

- ▲ Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**. The problem is really easy, therefore the solution is obtained almost immediately.

Get the capacitance value

- ▲ From the Project window, right click on Setup1. From the pull down menu, select **Matrix1**



 Get the capacitance value (Continued)

- 
- The analytical value of the capacitance for two parallel plates is given by:

$C = \epsilon_0 A / d$ (A is the area of the plate and d is the thickness of the dielectrics)

$$A = 625 \times 10^{-6} \text{ m}^2$$

$$d = 1 \times 10^{-3} \text{ m}$$

$$\epsilon_0 = 8.854 \times 10^{-12} \text{ F/m}$$

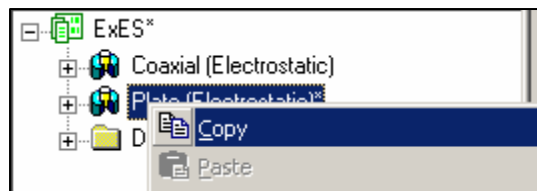
- 
- Using the above formula, we obtain
- 5.53 pF**
- . This value matches the value obtained using the finite elements

Capacitance of a planar capacitor with perturbation

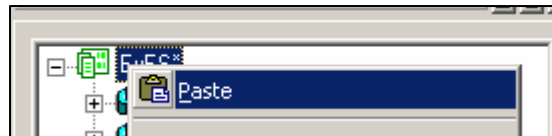
- Here we look at the consequence for the capacitance value when a conductive object is placed in the dielectrics region. We will use the previous design geometry to start with

Draw the model

- Click on the previous design **Plate**. Right Mouse click and select **Copy**



- Click on the project name, Right Mouse click and select **Paste**



- Based on this design, we draw a metal copper object. Select the menu item **Draw> Box**

- For the base point, enter **10;10;2.4 mm**
- For the Xsize, enter **5 mm**
- For the Ysize, enter **5mm**
- For the Zsize, enter **0.2mm**
- For the material property, enter **pec** (perfect conductor)
- Name this object **SmallPlate**

Modify Analysis Setup

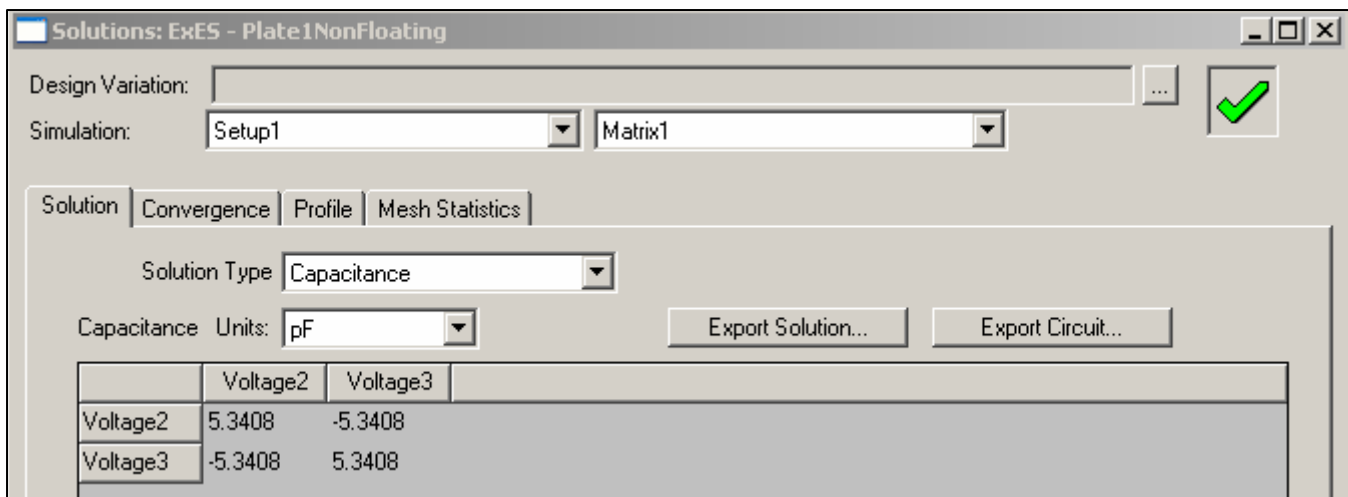
- ▲ We want to lower the error percentage since we have included the small piece of metal.
- ▲ From the project manager window, double click on setup1 (under the Analysis folder name).
 - ▲ For the **Percent Error**, enter 0.1%
 - ▲ For the **Refinement per Pass** (Convergence tab), put 50%

Solve the Problem

- ▲ Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**.

Get the capacitance value

- ▲ From the Project window, right click on Setup1. From the pull down menu, select **Matrix1**



Get the capacitance value (Continued)

- We obtain quite a similar capacitance than without the **SmallPlate** object. Actually, when a conductor is surrounding by insulator, Maxwell does not compute any voltage on it. In other words, the **SmallPlate** object will not have any charge on it, the capacitance value is not affected.
- This illustrates the fact that in this case, a dedicated excitation boundary needs to be used, the **floating** boundary.

Assign Excitation (2nd round)

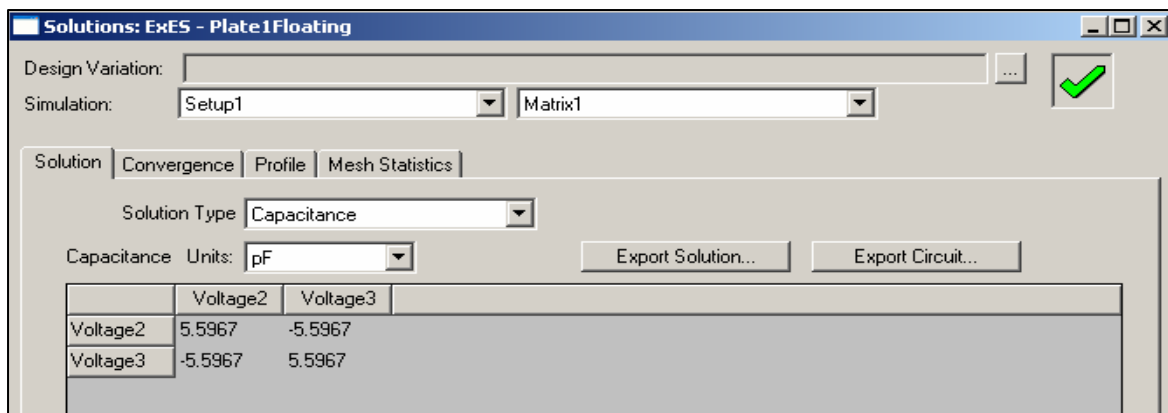
- Select the object **SmallPlate**, right click and select **Assign Excitations > Floating**.
- There is not charge on this plate, so enter **0** Coulomb.

Solve the Problem

- Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**.

Get the capacitance value

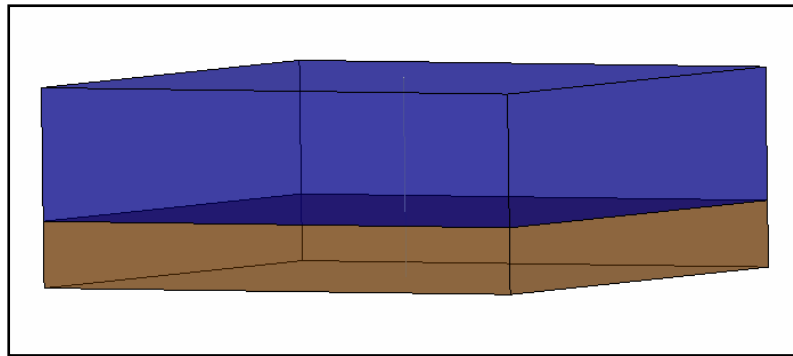
- From the Project window, right click on Setup1. From the pull down menu, select **Matrix1**. We obtain now the good capacitance value.



Introduction on the DCconduction Solver

- This note introduces the DC conduction solver. Only conductors are considered in the process.

Parallel Plates with Non-uniform media



- In this example, we want to determine the DC resistance between two plates with a non-uniform media made of ferrite and sea water. We do not need to draw the plates; we just need to draw the two media regions.

Draw the Model

- Click on the menu item **Project > Insert MaxwellDesign**
- Name the design **Plates**
- Click on the menu item **Maxwell > Solution Type**
 - Select **Electric**
 - Select the radio button **DCconduction**
- Select the menu item **Draw > Box** to create the first region
 - For the first position corner, enter **0;0;0 mm**
 - For the Xsize, enter **4 mm**
 - For the Ysize, enter **4mm**
 - For the Zsize, enter **0.5mm**
 - For the material property, enter **ferrite** Name the first box **Solid**

Draw the Model (Continued)

-  Select the menu item **Draw > Box** to create the second region
 -  For the first position corner, enter **0;0;0.5 mm**
 -  For the Xsize, enter **4 mm**
 -  For the Ysize, enter **4mm**
 -  For the Zsize, enter **1mm**
 -  For the material property, enter **water_sea**
 -  Name the first box **Liquid**

Assign Excitation

-  We apply the voltage to the plates. We did not draw the plates, because we can apply a voltage to the top and bottom part of the regions.
-  Select the top face of the **liquid** object and select the menu item **Maxwell > Excitations > Assign > Voltage**
 -  For the voltage, enter **10V**
-  Select the bottom face of the **solid** object and select the menu item **Maxwell > Excitations > Assign > Voltage**
 -  For the voltage, enter **-10V**

Create Analysis Setup

-  Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**. Leave the default parameters

Solve the Problem

-  Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

Compute the DC Resistance

-  The DC resistance is not directly computed by Maxwell3D. However, we already know the applied voltage. We need the DC current that flows through the media. The current is obtained by taking the integral of Z component of **J** on a cross-section of the region. To define any cross section, we define a local coordinate system, and use the planes defined by this local CS
-  Let's define a local coordinate system. Select the menu item **3D Modeler > Coordinate System > Create > Relative CS > Offset**.
 -  Enter **0 mm, 0mm, 1mm** for the position of the origin. The CS is called by default **RelativeCS1**.
-  Select the menu item **Maxwell > Fields > Calculator**. The field calculator is a tool that can compute any quantities using geometric entities and fields. The total current going through the media is the integral of the current density **J** on the plane **XY** of the local CS.
 -  Select **Quantity > J**
 -  Select **Scalar > ScalarZ**
 -  Select **Geometry > Surface > RelativeCS1:XY**, then click **OK**
 -  Select the **Integral** symbol
 -  Click on **Eval**
-  The total current appears, closed to **-6.37e-3 A**. The value is very low; it makes sense because the conductivity of medium is very low. The negative sign is just a matter of sign convention due to the CS orientation.
-  The DC resistance is given by $R = \text{Voltage} / \text{Current}$. The difference of potential between the two plates is 20 V. We obtain **R = 3140 Ohm**
-  The analytical value of the resistance is given by the following formula:
 -  **$R = \sigma_2 h_1 + \sigma_1 h_2 / (\sigma_1 \sigma_2 A)$** where σ_1, σ_2 are the conductivity of the two medium, h_1, h_2 the thickness of the two medium and **A** the surface of the plates.

▲ Compute the DC Resistance (Continued)

▲ Here, $\sigma_1 = 1\text{e-}2$ Siemens/m, $h_1 = 0.5\text{e-}3$ m (ferrite); $\sigma_2 = 4$ Siemens/m, $h_2 = 1\text{e-}3$ mm (sea water) ; $A = 16\text{e-}6$ m². This gives

▲ $R = 3140.62$ Ohm

▲ The two values are very similar.

▲ DC current flow in conductors

▲ In this example, we illustrate the capability of the DC current solver to reconstruct the current paths flowing in different conductors

▲ Draw the Model

▲ Click on the menu item **Project > Insert MaxwellDesign**

▲ Name the design **Conductors**

▲ Click on the menu item **Maxwell > Solution Type**

▲ Select **Electric**

▲ Select the radio button **DCconduction**

▲ Select the menu item **Draw > Box** to create the first conductor

▲ For the first position corner, enter **1 ;-0.6 ;0 mm**

▲ For the Xsize, enter **1 mm**

▲ For the Ysize, enter **0.2 mm**

▲ For the Zsize, enter **0.2 mm**

▲ For the material property, enter **copper** Name the first box **Conductor**

▲ Select the object **conductor**, then select the menu item **Edit > Duplicate > Along Line**

▲ Enter **0;0;0 mm** for the first point of the vector

▲ Enter **0 ;0.4;0 mm** for the second point of the vector

▲ For the total number, enter **4**

Draw the Model (Continued)

Select the menu item **Draw > Box**

-  For the first position corner, enter **0.8 ;-1 ;0 mm**
-  For the Xsize, enter **0.2 mm**
-  For the Ysize, enter **2.2 mm**
-  For the Zsize, enter **0.2 mm**
-  For the material property, enter **copper** Name the box **Conductor_4**

Select the menu item **Draw > Box**

-  For the first position corner, enter **0.8 ;-0.4 ;0 mm**
-  For the Xsize, enter **-1.2 mm**
-  For the Ysize, enter **-0.2 mm**
-  For the Zsize, enter **0.2 mm**
-  For the material property, enter **copper** Name the box **Conductor_5**

Select the object **Conductor_5** and select the menu item **Edit > Duplicate > Mirror**

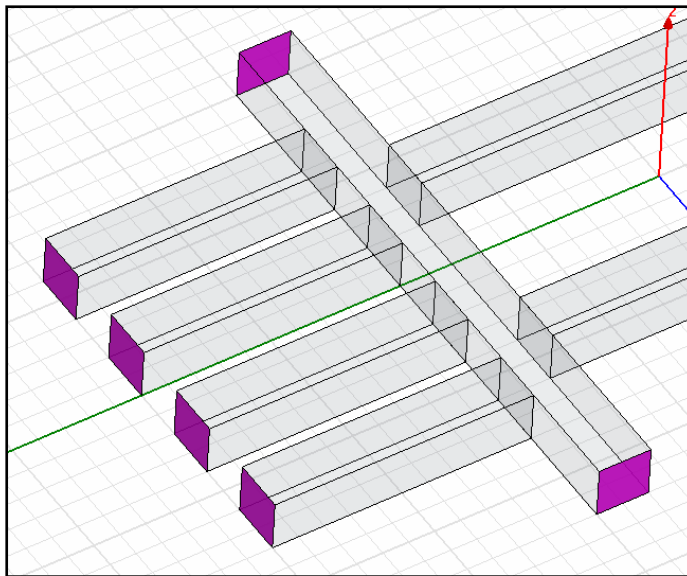
-  For the anchor point on the mirror plane, enter **0;0;0 mm**
-  For the target point, enter **0;1;0 mm**

Select the menu item **Draw > Box**

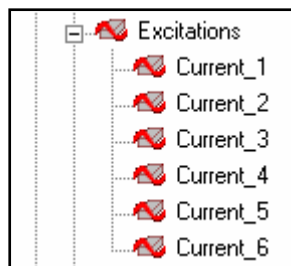
-  For the first position corner, enter **-0.4 ,0.6 ,0 mm**
-  For the Xsize, enter **-0.4 mm**
-  For the Ysize, enter **-1.2 mm**
-  For the Zsize, enter **0.2 mm**
-  For the material property, enter **copper** Name the last box **Conductor_sink**

Assign Excitation

- Switch the Selection mode to face by clicking on **f**
- Select the out faces of the objects **Conductor**, **Conductor_1**, **Conductor_2**, **Conductor_3**, **Conductor_4** as shown on the picture:

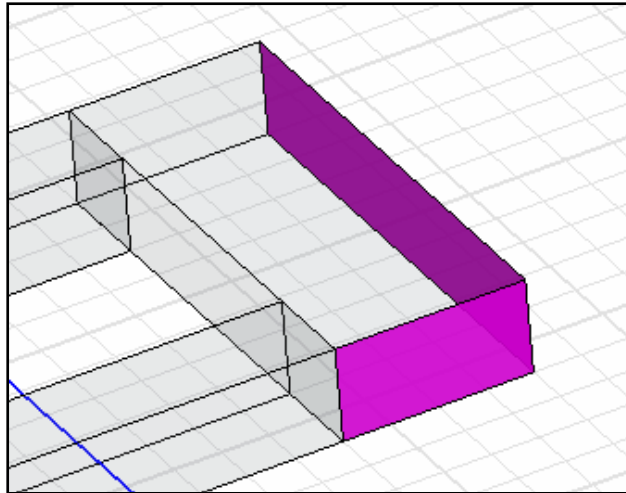


- Right mouse click and from the context menu select **Assign Excitation > Current**
 - Enter **1 A** as the current value
- Maxwell creates 6 current excitations of **1A**, each for every faces selected. The default current orientation is the current going inside the objects, so we do not need to swap any current direction.



Assign Excitation (Continued)

- ▲ We need to define a current sink. It ensures that the total current flowing through the outside surface of a conduction path is exactly zero. A current sink can be any collection of surfaces.
- ▲ Select the two faces of the **Conductor_sink** object as shown below



- ▲ Right mouse click and from the context menu select **Assign Excitation > Sink**

Assign Mesh Operations

- ▲ Select all the objects, right mouse click and select **Assign Mesh Operations > Inside Selection > Length Based**
 - ▲ Unselect the radio button Restrict length of elements
 - ▲ Select the radio button Restrict the number of elements
 - ▲ Enter **10,000** for the maximum number of elements

▲ Create Analysis Setup

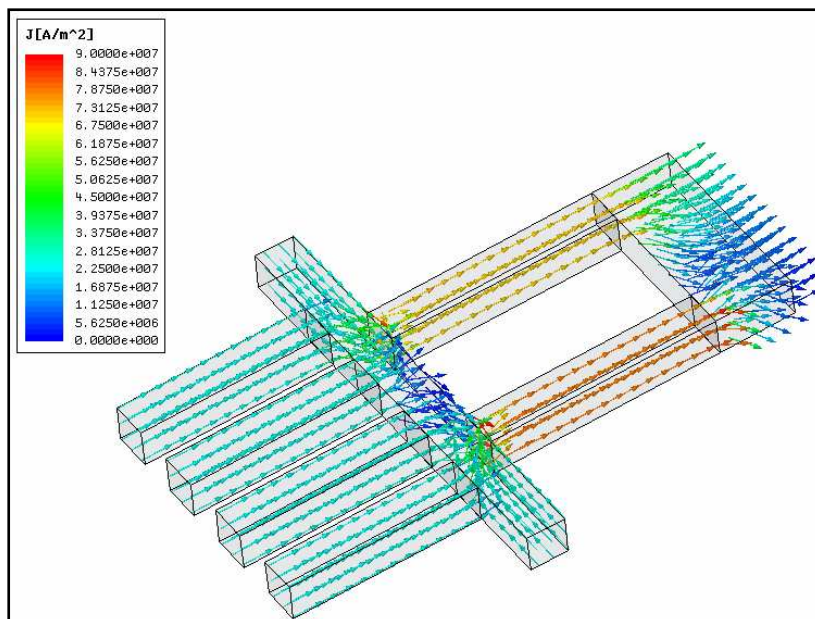
- ▲ Select the menu item **Maxwell > Analysis Setup > Add Solution Setup**.
Leave the default parameters

▲ Solve the Problem

- ▲ Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

▲ Plot the Current

- ▲ Select all the objects, right mouse click and select **Fields > J_Vector**. The 3D plot of the current vectors appears. It is possible to change the parameters of the plots:
 - ▲ On the **Scale** tab, check the radio button **Use Limits** and enter **0** and **90000000** as Min and Max
 - ▲ On the Marker/Arrow, uncheck the **Map size** radio button, then adjust the Size of the arrows
- ▲ You should get a similar plot.



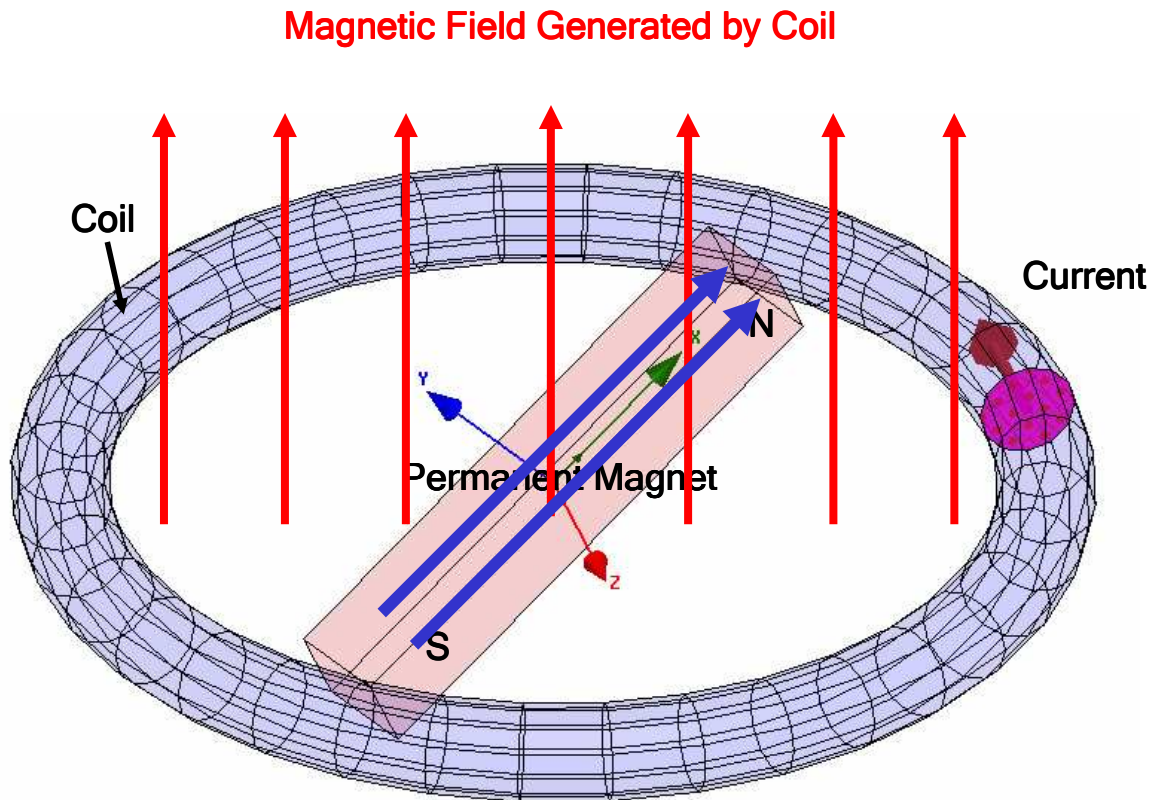
Basic Exercise - Magnetostatic Torque Calculation

Torque calculation in Magnetostatic Solver

- This exercise will discuss how to set up a torque calculation in the Magnetostatic Solver.

Problem Description

- As shown in the following graph, the current in the coil generates a magnetic field pointing upward. The permanent magnet in the middle is magnetized along X-axis, hence there is a torque generated along Z-axis.



Basic Exercise - Magnetostatic Torque Calculation

Draw the Coil

- ▲ Open up **Maxwell 3D V11**
- ▲ Click on the menu item **Project > Insert Maxwell Design**
- ▲ Click on the menu item **Maxwell > Solution Type > Magnetostatic**
- ▲ Click on the menu item **Draw > Regular Polygon**
 - ▲ X,Y, Z: **0,5,0**, Enter
 - ▲ DX, DY, DZ: **0.5,0,0**, Enter
 - ▲ Number of Segments: **12**
 - ▲ Change its name from **Polygon1** to **Coil**
- ▲ Select the **Coil** and click on the menu item **Draw > Sweep > Around Axis**
 - ▲ Change its material to: **copper**
 - ▲ Change its color and transparency if desired

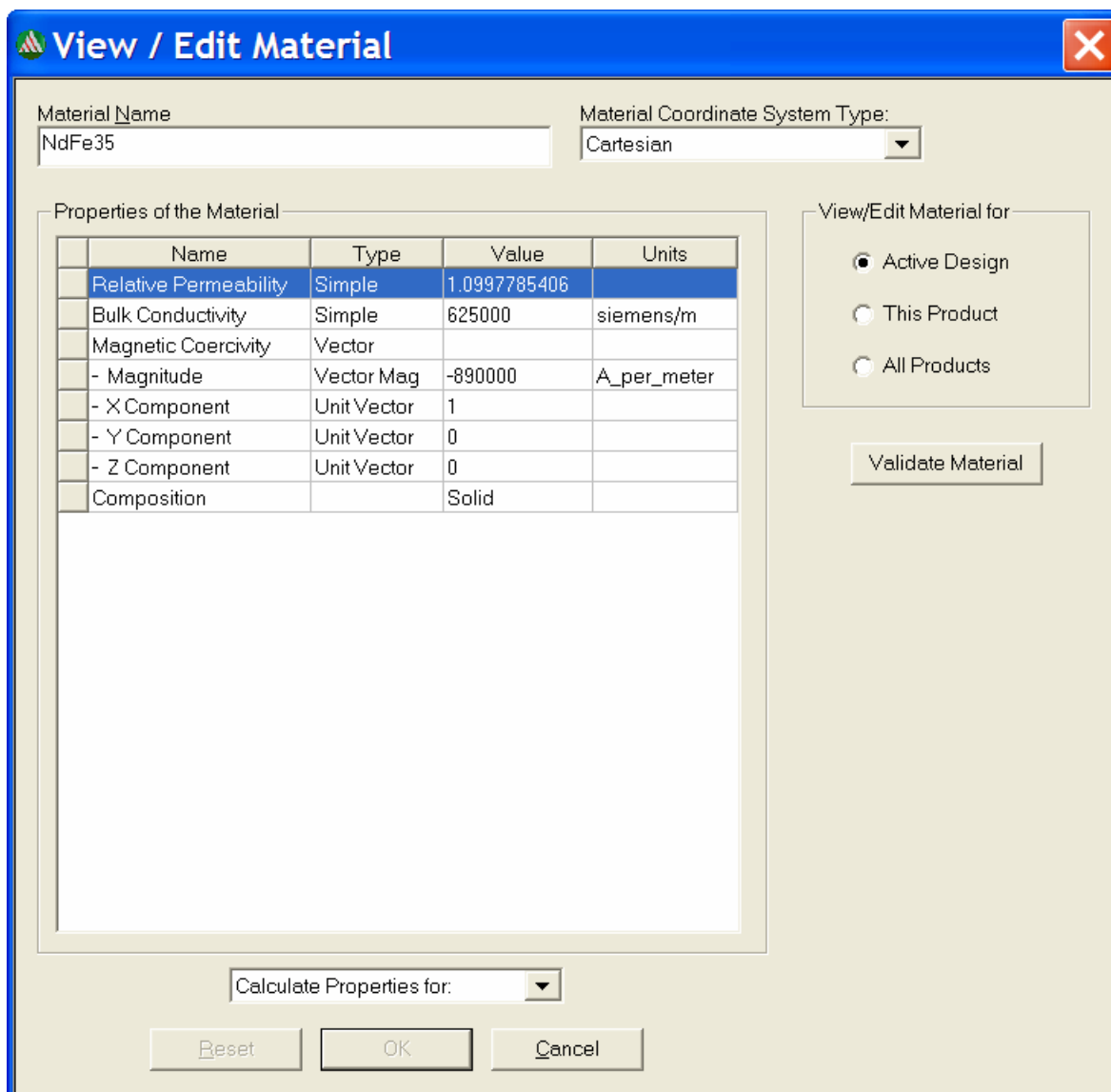


Draw the Permanent Magnet

- ▲ Click on the menu item **Draw > Box**
 - ▲ X, Y, Z: **-3,-0.5,-0.5**, Enter
 - ▲ DX, DY, DZ: **6,1,1**, Enter
 - ▲ Change its name to: **Magnet**
 - ▲ Change its material to: **NdFe35**
 - ▲ Change its color and transparency if desired

Basic Exercise - Magnetostatic Torque Calculation

- Make sure the magnetization of the magnet is assigned along X-axis



View / Edit Material

Material Name: NdFe35

Material Coordinate System Type: Cartesian

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Simple	1.0997785406	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-890000	A_per_meter
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Composition		Solid	

View/Edit Material for:

☒ Active Design

☐ This Product

☐ All Products

Validate Material

Calculate Properties for:

Reset OK Cancel

Create Section

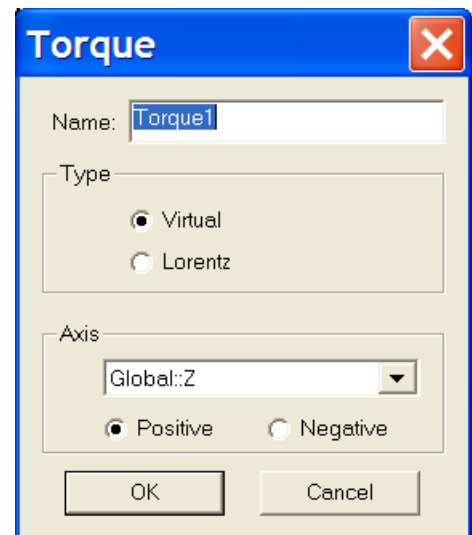
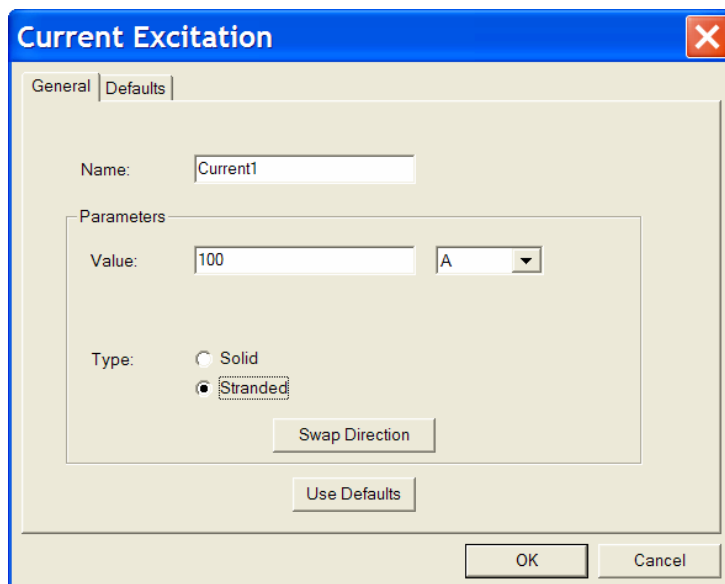
- Select **Coil** and then click on the menu item **3D Modeler > Surface > Section:**
 - Section Plane: **XY**
- Two sheets are create and to separate them click on **3D Modeler > Boolean > Separate Bodies**
- Select **Section1_1** and click on the menu item **Edit > Delete**

Basic Exercise - Magnetostatic Torque Calculation

- ▲ Select **Section1** and change its color if necessary
- ▲ Click on **File > Save**:
 - ▲ **Magnetostatic_Force.mxwl** for Basic Exercise Magnetostatic Force calculation
- ▲ Click on the menu item **Draw > Region**:
 - ▲ Padding Percentage: **100**
- ▲ Select **Region** and click on the menu item **View > Hide Selection > All views**.

Assign Excitation

- ▲ Select the **Section1** and click on the menu item **Maxwell > Excitations > Assign > Current**:
 - ▲ Name: **Current1**
 - ▲ Value: **100**
 - ▲ Type: **Stranded**



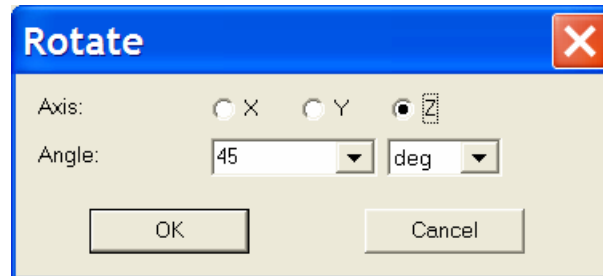
Assign Torque Calculation

- ▲ Select the object **Magnet** and right click on the project tree item **Parameters > Assign > Torque**:
 - ▲ Name: **Torque1**
 - ▲ Type: **Virtual**
 - ▲ Axis: **Global::Z, Positive**

Basic Exercise - Magnetostatic Torque Calculation

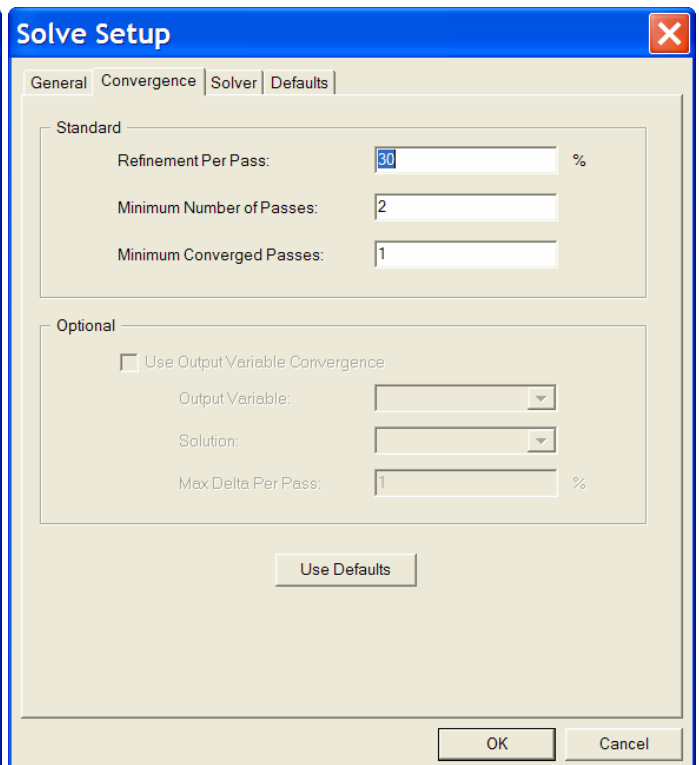
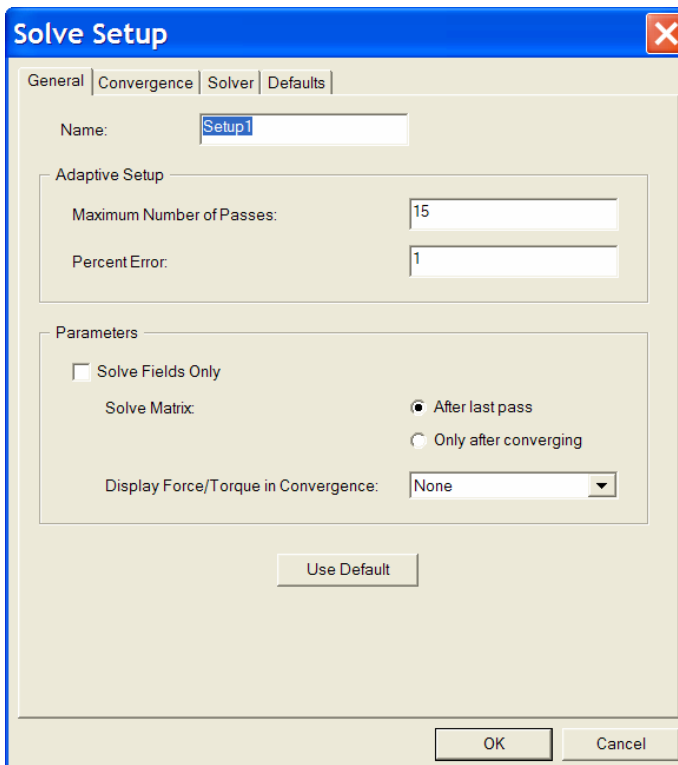
Rotate the Coil

- Select object **Coil** and sheet **Section1**, right click, then **Edit > Arrange > Rotate**



Create Analysis Setup

- Click on **Maxwell > Analysis Setup > Add Solution Setup**
 - Maximum Number of Passes: **15**
 - Refinement per Pass: **30**
 - Click on **OK**



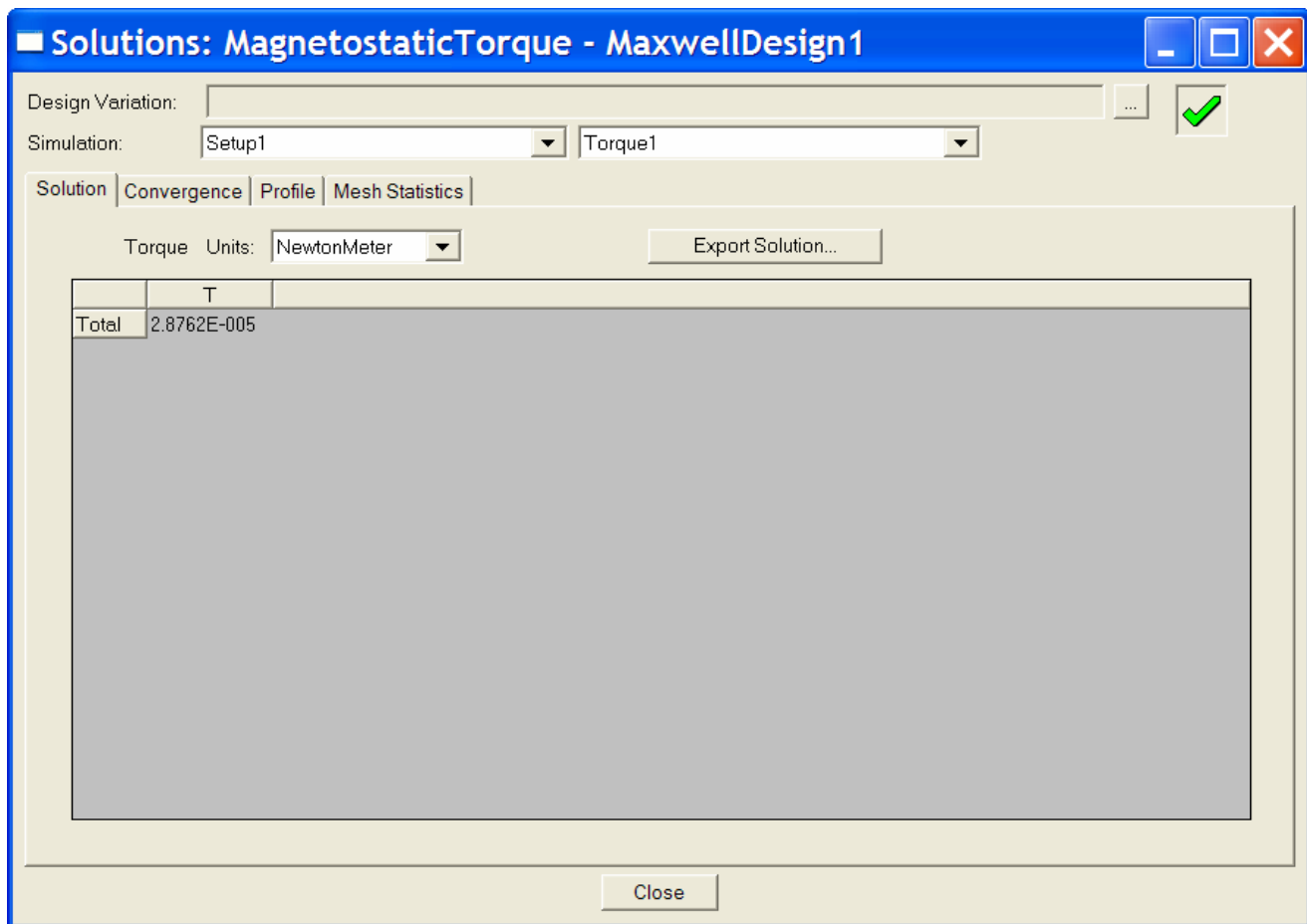
Solve the Problem

- Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

Basic Exercise - Magnetostatic Torque Calculation

View Calculated Torque Result

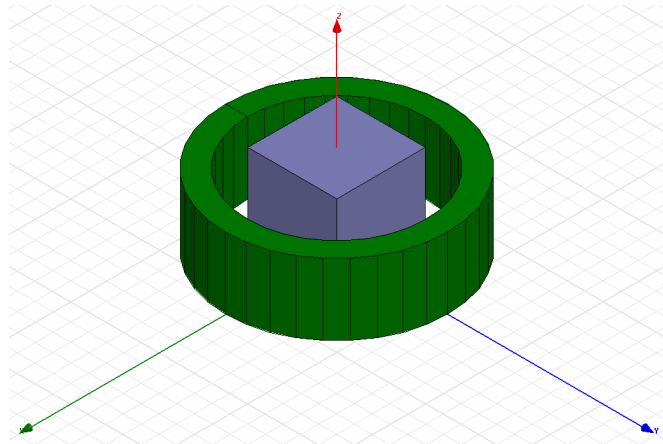
- Click on menu item **Maxwell > Results > Solution Data**.
- Click on **Solution** tab, you will see the calculated torque, it is updated automatically after each pass.



- This concludes the Basic Exercise for Magnetostatic Torque Calculation.

Parametric study using a coil and iron slug.

- A Magnetostatic problem will be used to demonstrate the setup of a parametric solution using Optimetrics. The coil current and the dimensional length of an iron slug will be varied and the force on the slug will be observed.



- Click on the menu item **Project > Insert Maxwell Design**
- Click on the menu item **Maxwell > Solution Type > Magnetostatic**

Draw the Coil

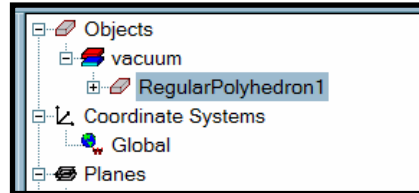
- The user will draw two concentric cylinders and perform a “Boolean subtract” to create the coil.
- Draw the first cylinder. Click on the menu item **Draw > Regular Polyhedron**
 - Arbitrarily select a center of base (origin), a radius, and a height using the mouse. Then, assign a number of segments. When finished, the **Properties** window will appear.
 - Edit the values as shown in the panel below to create the first cylinder.

Properties: TrainingExercise_Parametric - MaxwellDesign3 - 3D Modeler

Command		Attribute	
Name	Value	Unit	Evaluated Value
Command	CreateRegularPolyhedron		
Coordinate System	Global		
Center Position	0,0,0	mm	0mm, 0mm, 0mm
Start Position	1.25,0,0	mm	1.25mm, 0mm, 0mm
Axis	Z		
Height	0.8	mm	0.8mm
Number of Segments	36		36

Basic Exercise - Parametric

- ▲ The user will create a second cylinder using copy and paste.
 - ▲ Select the first cylinder, “RegularPolyhedron1” from the Design Tree.



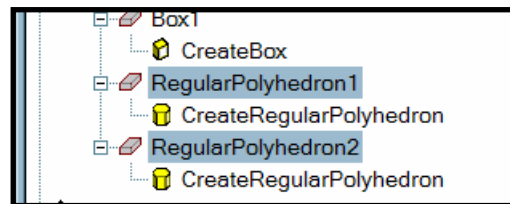
- ▲ Copy and Paste the Polyhedron by using the **Edit > Copy** and **Edit > Paste**.
- ▲ In the Design Tree, double click on “RegularPolyhedron2” and edit the values given in the **Properties** window below.

Properties: TrainingExercise_Parametric - MaxwellDesign3 - 3D Modeler

Command

Name	Value	Unit	Evaluated Value
Command	CreateRegularPolyhedron		
Coordinate System	Global		
Center Position	0,0,0	mm	0mm, 0mm, 0mm
Start Position	1,0,0	mm	1mm, 0mm, 0mm
Axis	Z		
Height	0.8	mm	0.8mm
Number of Segments	36		36

- ▲ The user will finish the creation of the coil by performing a Boolean subtract of the two polyhedron.
 - ▲ Select the both polyhedrons from the Design Tree by holding the Ctrl key and clicking on both objects.

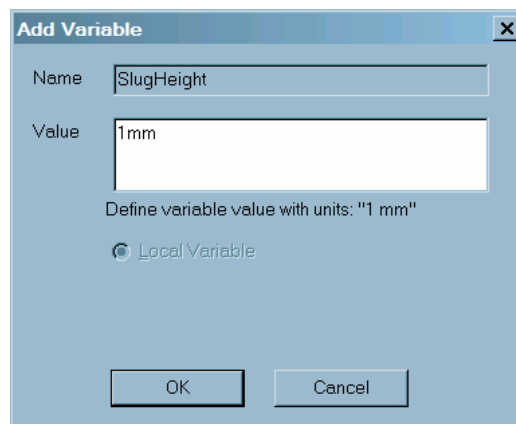
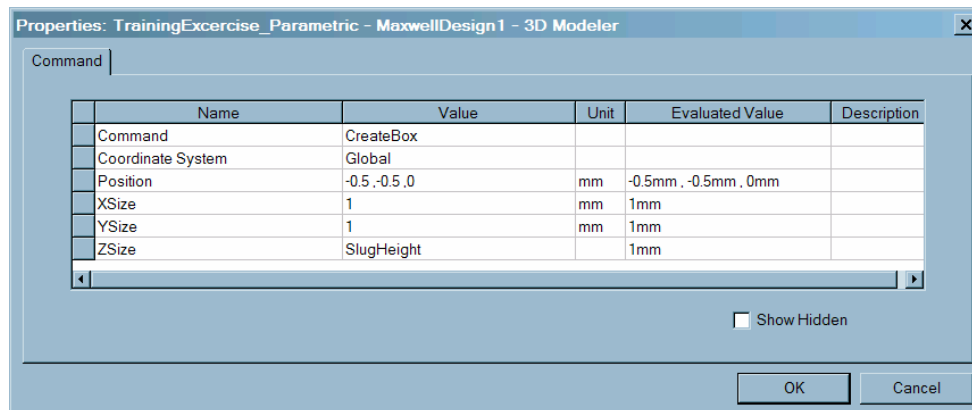


- ▲ Perform the Boolean subtract.
 - ▲ **3D Modeler > Boolean > Subtract**
 - ▲ Blank Park: **Polyhedron1**
 - ▲ Tool Park: **Polyhedron2**
- ▲ Select the polyhedron object in the Design Tree and double click it to edit it's properties.
 - ▲ Name: **Coil**
 - ▲ Material: Select **copper** from the material database.

Basic Exercise - Parametric

Draw the iron Slug.

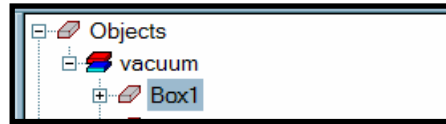
- ▲ The user will draw the iron slug using a Box.
 - ▲ Click on the Menu item: **Draw > Box**
 - ▲ Arbitrarily select a Box position (origin), the opposite corner of the base rectangle, and a height using the mouse. When finished, the Properties window will appear.
 - ▲ Edit the values as shown in the Property window below to create the box. Note that **SlugHeight** is entered as a variable for the **ZSize** of the box. When prompted, define the value of **SlugHeight** as "1mm."



- ▲ **Note:** By defining a variable name (SlugHeight) it becomes a design variable. The design variables are accessible in the Property window by clicking on the design name in the Project Manager. Or, by clicking:
 - ▲ **Maxwell > Design Properties . . .**

Basic Exercise - Parametric

- Double click on the newly created **Box1** to bring up the property window.



- Edit the name of the Slug and assign the Material as iron.

- Name: **Slug**

- Material: Select **iron** from the material database.

- Note:** The material properties for iron has a linear permeability. This means that no non-linear BH curve is being used in this example.

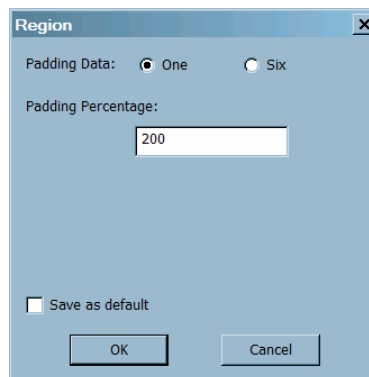
Create a Problem Region

- Create a problem region that surrounds the coil and slug.

- 3D Modeler > Region**

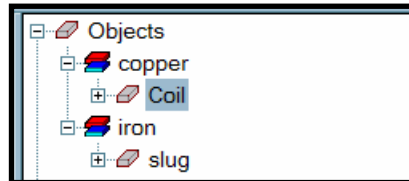
- Assign a padding percentage as 200, as shown below and select **OK**.

- The user may edit the padding percentages in the Properties window as desired.

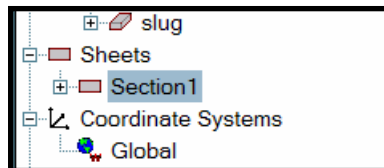


Assign the Excitation.

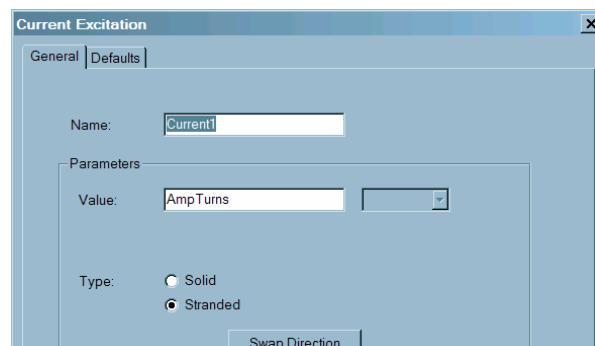
- Select the Coil from the Design Tree.



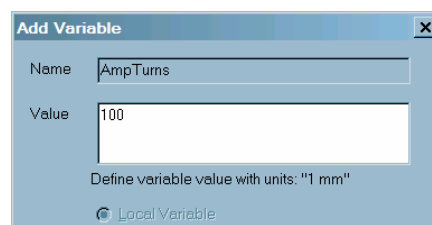
- Section the coil to create a sheet cross section.
 - 3D Modeler > Surface > Section.**
- Select the **YZ radial button**.
- Separate the section into two objects.
 - 3D Modeler > Boolean > Separate Bodies**
- Delete the unnecessary section, "Section1_1" by selecting it in the Design Tree and pressing the Delete key.
- Select **Section1** and assign an excitation in the project manager window.



- Click on the Menu item **Maxwell > Excitations > Assign > Current**
- Assign a variable called **AmpTurns** as shown below.
- Select a **Stranded Winding**.

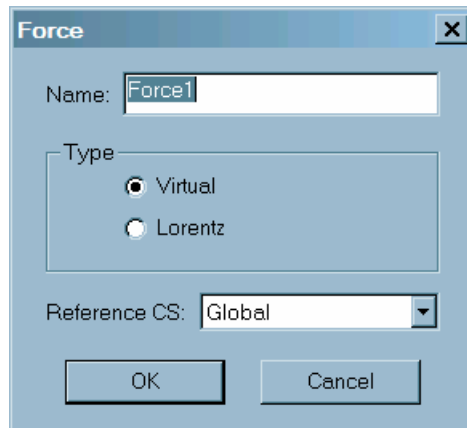


- Define the value of AmpTurns as **100** in the Add Variable window.



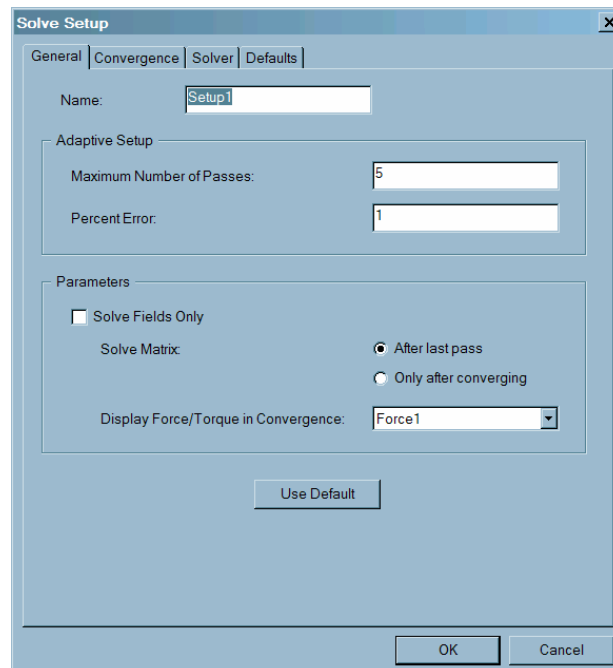
Assign a Force Parameter Calculation

- ▲ Select the Slug from the Design Tree.
- ▲ Assign a force in the project manager window.
 - ▲ Click on the Menu item **Maxwell > Parameters > Assign > Force**.
- ▲ Select the Virtual Force as shown below.



Create and solve a nominal problem

- ▲ Click on the Menu item **Maxwell > Analysis > Add Solution Setup . . .**
- ▲ Change the default **Minimum Number of Passes** to 5 to speed up the solution.
- ▲ Select **Force1** to be displayed in the Convergence, as shown below. Click **OK**.



Basic Exercise - Parametric

- ▲ Run the nominal solution.
 - ▲ In the Project Manager window, right click on **Setup1** and select **Analyze**.
 - ▲ During or after the solution is complete, inspect the Convergence and Force results by viewing the appropriate Tab in the Solution window.
 - ▲ In the Project Manager Window, right click on **Setup1** and select **Convergence**.
 - ▲ Your results should look similar to those shown below.

Solutions: TrainingExercise_Parametric - MaxwellDesign4

Design Variation: AmpTurns='100' SlugHeight='1mm'

Simulation: Setup1

Solution | Convergence | Profile | Mesh Statistics

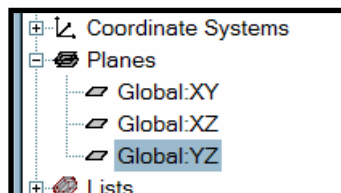
Pass	Tets	Total Energy (J)	Delta Energy (%)	Energy Error (%)	Mag(F)/T	Output Var.	Output Var. Delta (%)
1	854	4.1618e-005	N/A	286.49	0.055857	N/A	N/A
2	1219	2.0988e-005	98.295	47.378	0.0059399	N/A	N/A
3	1636	1.9166e-005	9.5035	21.318	0.0044213	N/A	N/A
4	2151	1.8133e-005	5.7005	15.416	0.0034147	N/A	N/A
5	2840	1.7765e-005	2.0709	4.431	0.0035897	N/A	N/A

Number of Passes:
Completed 5
Maximum 5
Minimum 2

Energy Error/Delta Energy (%)
Target (1, 1)
Current (4.431, 2.0709)

View: ☒ Table ☐ Plot

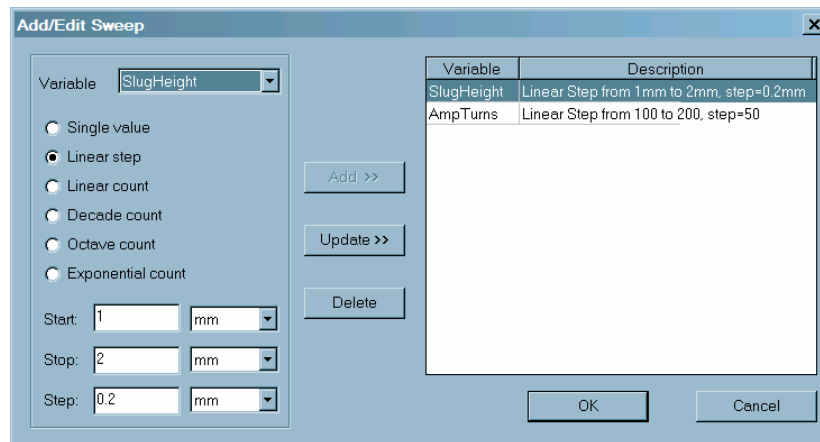
- ▲ Plot a field result.
 - ▲ Select the **Global: YZ** plane from in the Design Tree under **Planes**.



- ▲ Plot the B-Field on this plane.
 - ▲ Click on the Menu item **Maxwell > Fields > Fields > B > Mag_B**.
 - ▲ Click Done in the next window.
 - ▲ To view the fields inside the slug, expand **Field Overlays** in the Project Manager and click on **Mag_B1**.
 - ▲ Your result should show about 0.2 Tesla in the Center of the Slug which is well within the linear region of the BH curve for most steels.

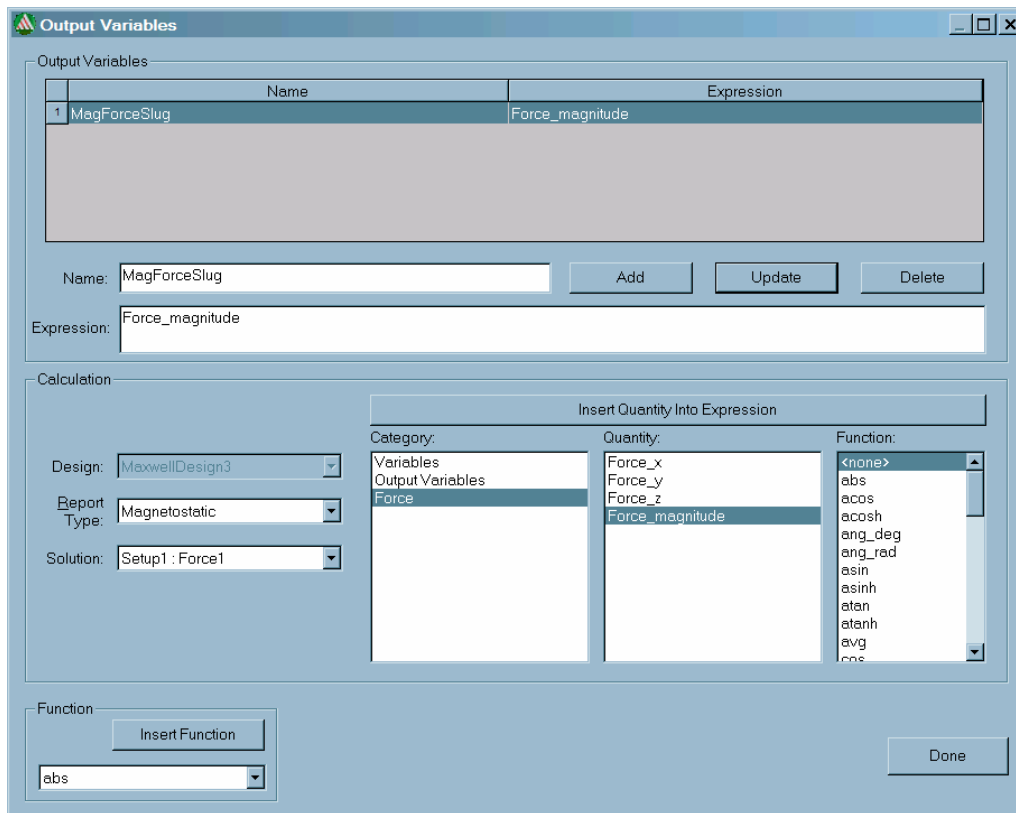
Create a Parametric solution

- Click on the Menu item **Maxwell > Optimetrics > Add Parametric . . .**
- Click **Add. . .** in the Setup Sweep Analysis window to define the parameters to be swept in the analysis.
- Select the **SlugHeight** variable for definition, and assign **Start =1, Stop =2, and Step = 0.2**, and click the **Add >>** button.
- Similarly, select the **AmpTurns** variable for definition and assign a **Start =100, Stop =200, and Step = 50**, and click the **Add >>** button.
- Click **OK..**

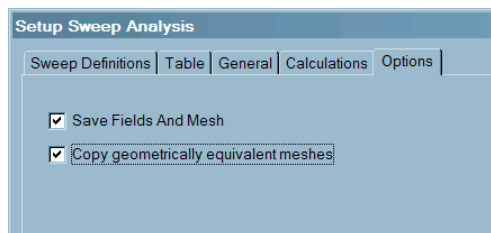


- Click on the **Table** tab to inspect the combination of solutions that have been created. There should be 18 solutions since we defined 6 variations of SlugHeight and 3 variations of AmpTurns.
- Next, select the **Calculations** tab to define which outputs will be calculated for each parametric solution.
- Click the **Add** button to add a calculation.
- Click **Edit Calculation . . .** to select the output calculation.
 - Select : **Category: Force,**
Quantity: Force_magnitude
Insert Quantity Into Expression
Name: type MagForceSlug
 click **Add**
 - The **Edit Calculation** window should appear as follows:

Basic Exercise - Parametric

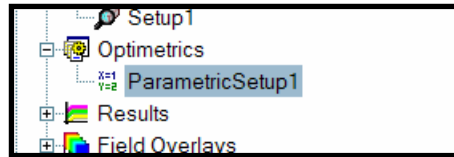


- ▲ Click **Done**
- ▲ The Calculation **MagForceSlug** should appear in in the **Calculation** tab
- ▲ In the Options Tab, click both boxes for **Save Fields And Mesh**, and **Copy Geometrically Equivalent Meshes**.



Solve the Parametric problem

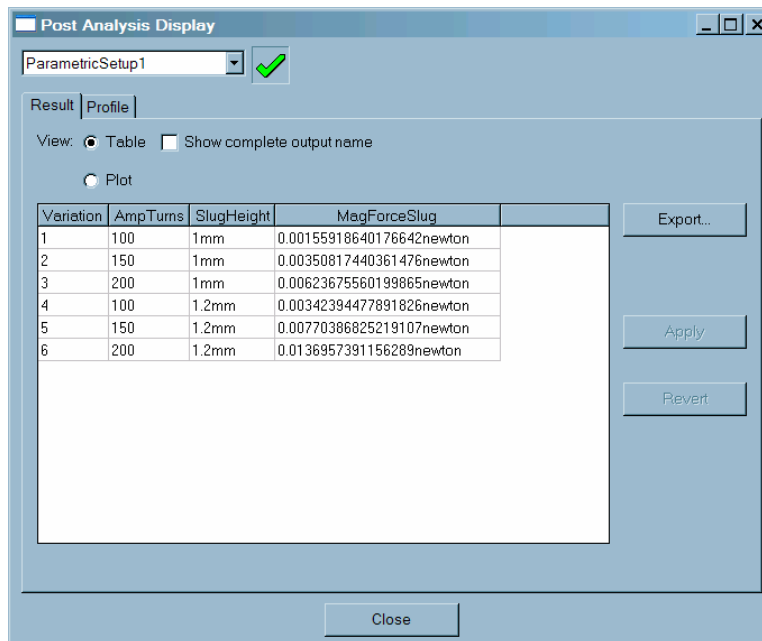
- In the Project Manager window, under **Optimetrics**, right click on **ParametricSetup1**, and select **Analyze**.



- Note:** the solving criteria is taken from the nominal problem, **Setup1**. Each parametric solution will re-mesh if the geometry has changed or the energy error criteria is not met as defined in **Setup1**.

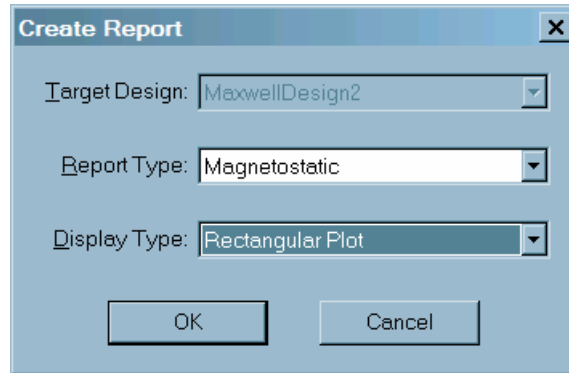
- View the solution progress:

- In the Project Manager window, right click on **ParametricSetup1**, and select **View Analysis Result . . .**
- Click the Table button to view all the results in tabulation form.
- The full parametric solution should take about 5 minutes depending on the speed of the machine.

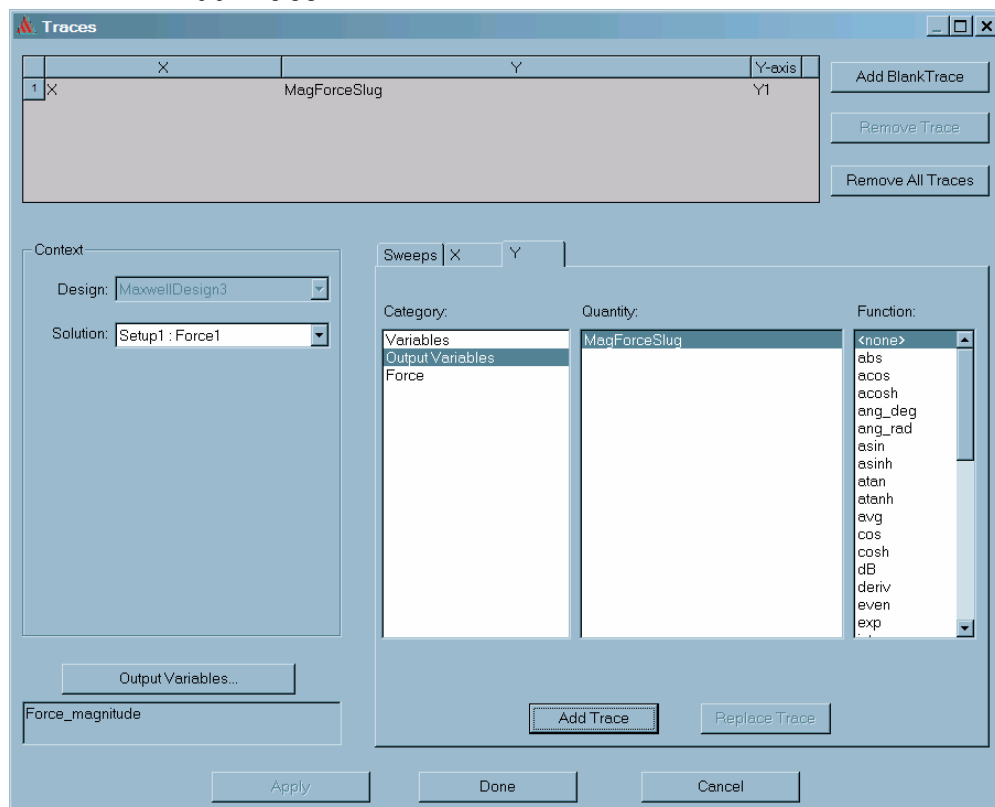


Graph the Force vs. AmpTurns vs. SlugHeight

- Right Click on **Results** in the Project Manager, and select **Create Report**.
- Choose the Report Type and Display Type as shown below.
- Click **OK**.

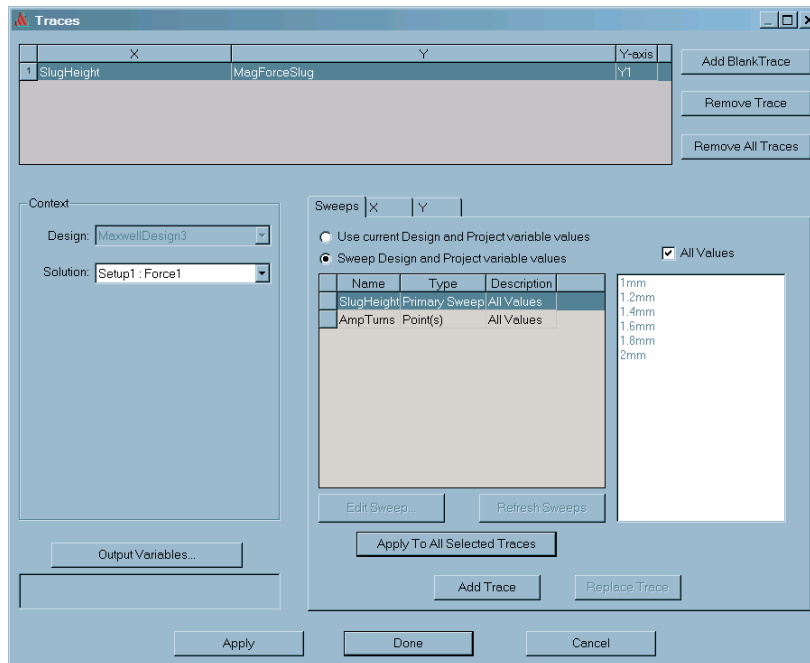


- Define the values to be plotted on the y-axis. Select the **Y** tab,
- Select **Output Variables** in the **Category** window.
- Select **MagForceSlug** in the **Quantity** Window.
- Click **Add Trace**.

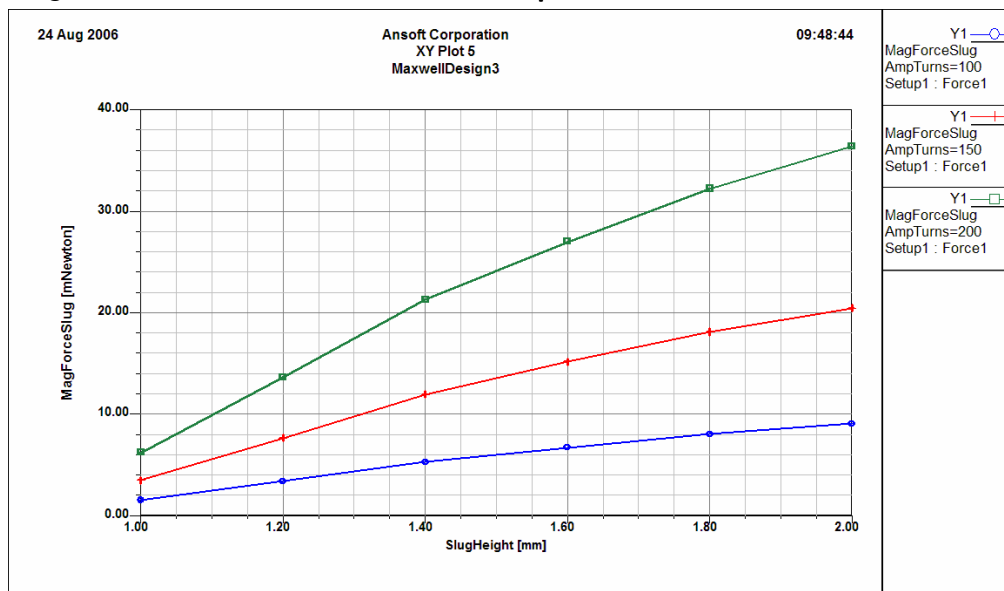


Basic Exercise - Parametric

- ▲ Click on the **Sweeps** tab.
- ▲ Select the first trace in the top of the window by clicking on **"1"**.
 - ▲ Check the **Sweep Design and Project variable value** button.
 - ▲ Click on **Apply to All Selected Traces**.
- ▲ Click **Done**.



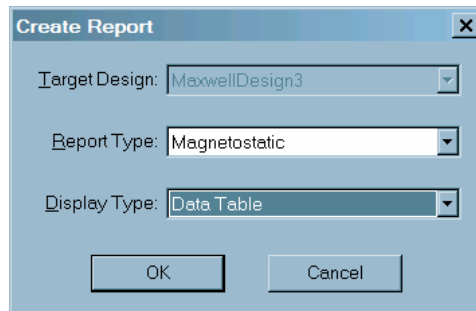
- ▲ The graph should appear as shown below.
- ▲ The graph may be edited by double clicking on the legend, axis, or trace.
- ▲ Right click on the table and select **Export to File** to save the data as a text file



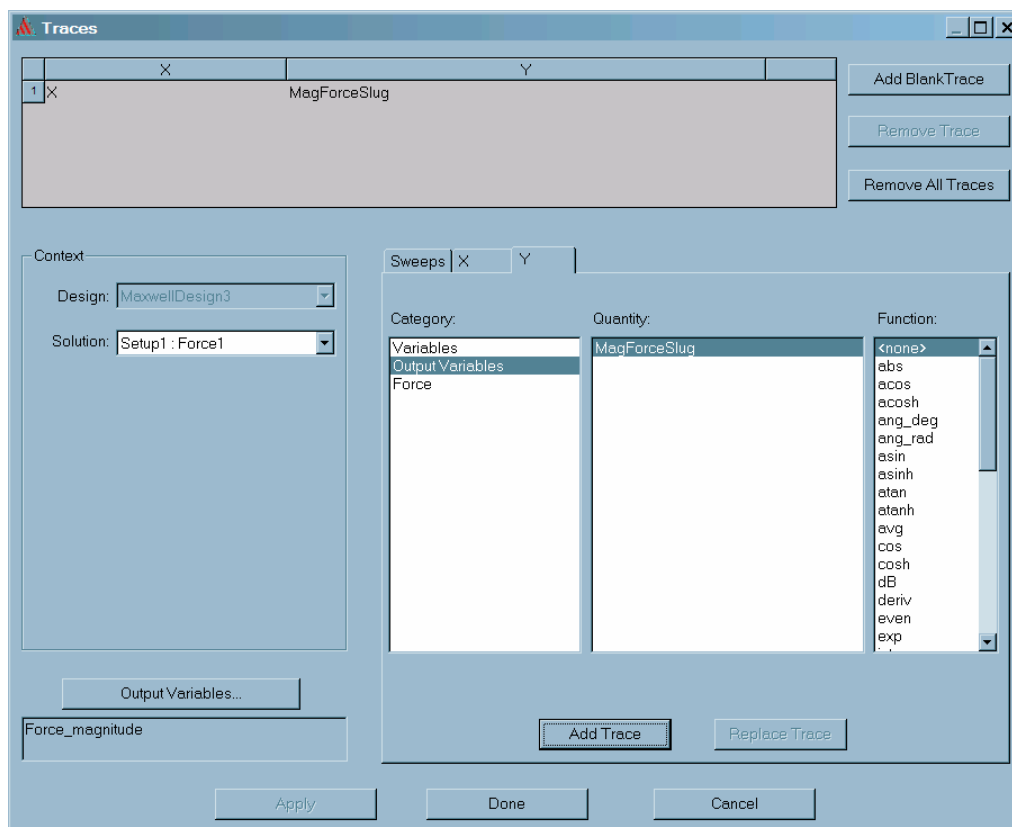
Basic Exercise - Parametric

Create a Table of SlugHeight, Force, and AmpTurns

- Right Click on Results in the Project Manager, and select **Create Report**.
- Choose the Report Type and Display Type as shown below.
- Click **OK**.

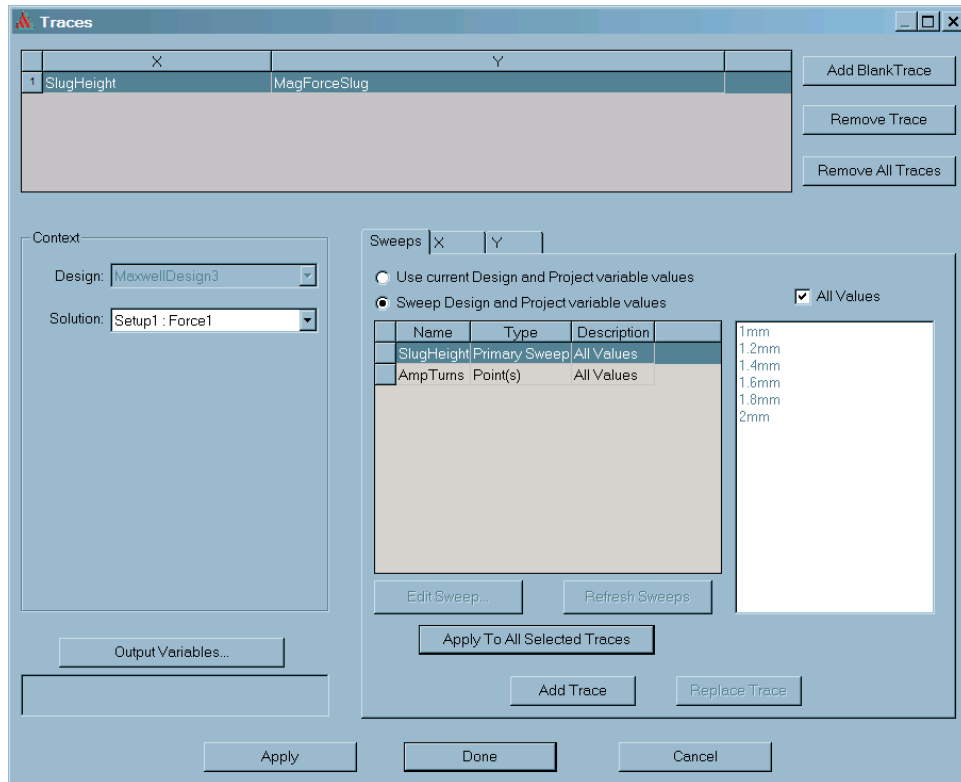


- Define the values to be listed in the Table. Select the **Y** tab,
 - Select **OutputVariables** in the **Category** window
 - Select **MagForceSlug** in the **Quantity** window.
 - Click on **Add Trace**



Basic Exercise - Parametric

- ▲ Click on the **Sweeps** tab.
- ▲ Select the first trace in the top of the window by clicking on **"1"**.
 - ▲ Check the **Sweep Design and Project variable value** button.
 - ▲ Click on **Apply to All Selected Traces**.
- ▲ Click **Done**.



- ▲ The table should appear as shown below.
- ▲ Right click on the table and select **Transpose** to transpose the table matrix, or right click on the table and select **Export to File** to save the data as a text file.

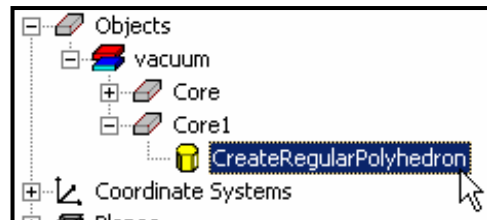
	SlugHeight [mm]	MagForceSlug [mNewton] Amp Turns=100 Setup1 : Force1	MagForceSlug [mNewton] Amp Turns=150 Setup1 : Force1	MagForceSlug [mNewton] Amp Turns=200 Setup1 : Force1
1	1.000000	1.553562	3.495508	6.214239
2	1.200000	3.409108	7.670489	13.636390
3	1.400000	5.313148	11.954611	21.252554
4	1.600000	6.746904	15.180570	26.987658
5	1.800000	8.049570	18.111523	32.198341
6	2.000000	9.096648	20.467430	36.386532

Inductor using transient source

- This exercise will discuss how to use transient sources as the excitation for an inductor coil.

Draw the Inductor

- Click on the menu item **Project > Insert Maxwell Design**
- Click on the menu item **Maxwell > Solution Type > Transient**
- Click on the menu item **Draw > Regular Polyhedron**
 - Center: **0,0,0**
 - Start Position: **2.0, 2.0, 0**
 - Height: **20.0**
 - Number of Segments: **24**
 - Change its name to: **Core**
 - Change its material to: **ferrite**
 - Change its color if desired
- Select the **Core** and click on the menu item **Edit > Copy**
- Click on the menu item **Edit > Paste**, the new objects name is **Core1**
- In the object tree click on **Core1** and then click on **CreateRegularPolyhedron**

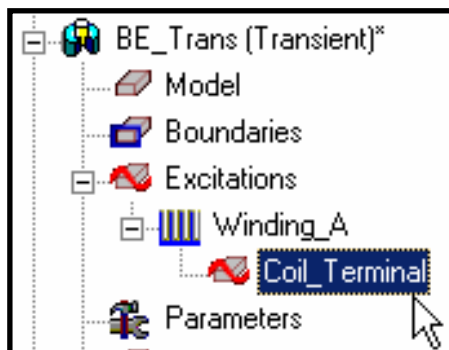


- In the **Properties** window change the following:
 - Name: **Coil**
 - Material: **Copper**
 - Start Position: **3.0, 3.0, 0**
 - Change its color
- Select **Core** and **Coil** and then click on **3D Modeler > Boolean > Subtract**:
 - Blank Part: **Coil**
 - Tool Part: **Core**
 - Clone objects before subtracting: ☒ **checked**
- Select the **Coil** and then click on the menu item **3D Modeler > Surface > Section**:
 - Section Plane: **YZ**

- ▲ Two sheets are create and to separate them click on **3D Modeler > Boolean > Separate Bodies**
- ▲ Select **Section1_1** and click on the menu item **Edit > Delete**
- ▲ Select **Section1** and in the properties window change the following:
 - ▲ Name: **Terminal**
 - ▲ Change its color
- ▲ Click on the menu item **Draw > Region:**
 - ▲ Padding Percentage: **500**
- ▲ Change the name of the design to:
 - ▲ **BE_Trans.mxwl** for Basic Exercise Transient

▲ Assign Excitation

- ▲ Select the **Terminal** and click on the menu item **Maxwell > Excitations > Assign > Coil Terminal:**
 - ▲ Name: **Coil_Terminal**
 - ▲ Number of Conductors: **150**
- ▲ Click on the menu item **Maxwell > Excitations > Add Winding**
 - ▲ Name: **Winding_A**
 - ▲ Type: **Voltage**
 - ▲ Stranded: ☒ **Checked**
 - ▲ Initial Current: **0.0 amps**
 - ▲ Resistance: **25 ohm**
 - ▲ Inductance: **0 H**
 - ▲ Voltage: **0 V** ... This will be changed later
- ▲ Select **Winding_A** from the project tree under **Excitation** and right mouse click and select **Add Terminals:**
 - ▲ Select: **Coil_Terminal**
- ▲ The project tree should look like this:



Create the Excitation

- ▲ The excitation for this problem will be a voltage source with a 1KHz triangular wave superimposed on a 50 Hz sine wave that has a 50 volt DC offset.
- ▲ Click on the menu item **Maxwell > Design Datasets** and then **Add** a new dataset
 - ▲ Name: **DSet_A**
 - ▲ Coordinates:
 - ▲ X1 = 0 Y1 = 0
 - ▲ X2 = 250e-6 Y2 = 1
 - ▲ X3 = 750e-6 Y3 = -1
 - ▲ X4 = 1e-3 Y4 = 0
- ▲ Select **Winding_A** from the Project Tree and right mouse click and select **Properties** and type in the following:
 - ▲ Voltage: $50 + 25 \cdot \sin(2 \cdot \pi \cdot 50 \cdot \text{Time}) + 5 \cdot \text{pwl\_periodic}(\text{DSet\_A}, \text{Time})$
 - ▲ The first term is a 50 V DC offset
 - ▲ The second term is a 25 Vp-p, 50 Hz sine wave
 - ▲ The third term is a 5 Vp-p, 1 KHz triangular wave

Apply Mesh Operations

- ▲ The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created
- ▲ Select the **Core** and click on the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 - ▲ Restrict Length Of Elements: **Uncheck**
 - ▲ Restrict Number of Elements: ☒ **Check**
 - ▲ Maximum Number of Elements: **1000**
- ▲ Select the **Coil** and click on the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 - ▲ Restrict Length Of Elements: **Uncheck**
 - ▲ Restrict Number of Elements: ☒ **Check**
 - ▲ Maximum Number of Elements: **1000**

Create Analysis Setup

- ▲ Click on **Maxwell > Analysis Setup > Add Solution Setup**
 - ▲ Stop Time: **20 ms**
 - ▲ Time Step: **50 us**

Basic Exercise - Transient

Solve the Problem

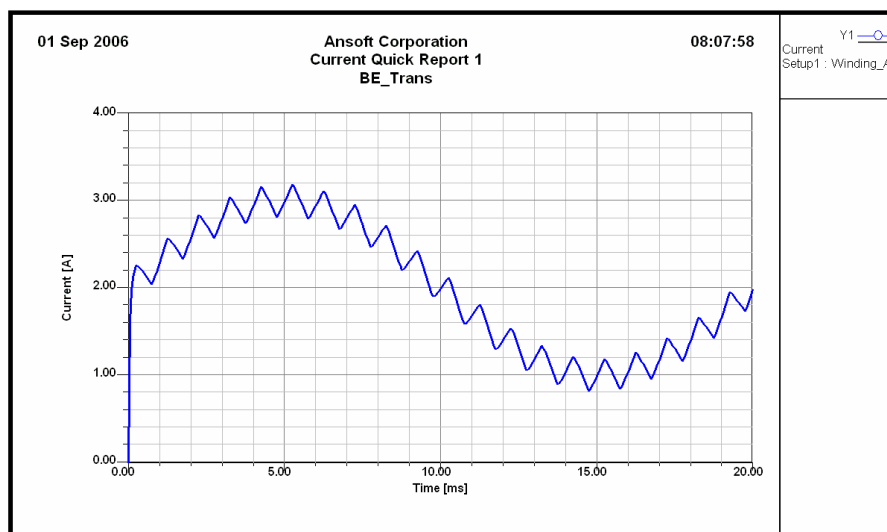
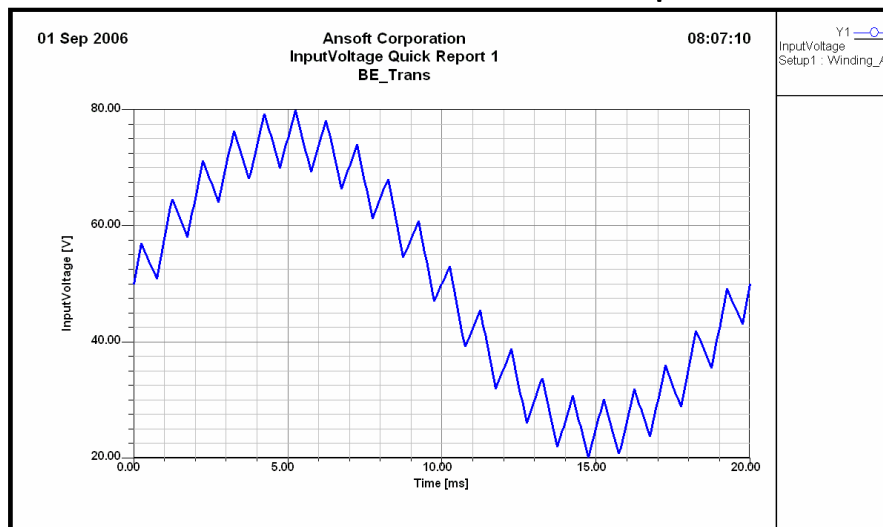
-  Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

Save the Design

-  Click on **File > Save** to save the design and results

Plot the Voltage and Current

-  Click on **Maxwell > Results > Create Quick Report** and select **Input Voltage**.
-  Click on **Maxwell > Results > Create Quick Report** and select **Current**



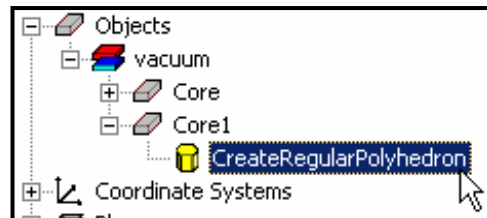
This concludes the Basic Example for Transient Sources

Inductor using transient source with external circuits

- This exercise will discuss how to link a transient source generated by an external circuit to an inductor coil.

Draw the Inductor

- Click on the menu item **Project > Insert Maxwell Design**
- Click on the menu item **Maxwell > Solution Type > Transient**
- Click on the menu item **Draw > Regular Polyhedron**
 - Center: **0,0,0**
 - Start Position: **2.0, 2.0, 0**
 - Height: **20.0**
 - Number of Segments: **24**
 - Change its name to: **Core**
 - Change its material to: **ferrite**
 - Change its color if desired
- Select the **Core** and click on the menu item **Edit > Copy**
- Click on the menu item **Edit > Paste**, the new objects name is **Core1**
- In the object tree click on **Core1** and then click on **CreateRegularPolyhedron**



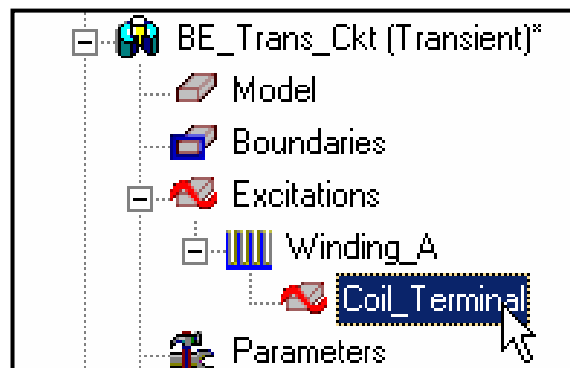
- In the **Properties** window change the following:
 - Name: **Coil**
 - Material: **Copper**
 - Start Position: **3.0, 3.0, 0**
 - Change its color
- Select **Core** and **Coil** and then click on **3D Modeler > Boolean > Subtract**:
 - Blank Part: **Coil**
 - Tool Part: **Core**
 - Clone objects before subtracting: ☒ **checked**
- Select the **Coil** and then click on the menu item **3D Modeler > Surface > Section**:
 - Section Plane: **YZ**

Basic Exercise - Transient with Circuits

- ▲ Two sheets are create and to separate them click on **3D Modeler > Boolean > Separate Bodies**
- ▲ Select **Section1_1** and click on the menu item **Edit > Delete**
- ▲ Select **Section1** and in the properties window change the following:
 - ▲ Name: **Terminal**
 - ▲ Change its color
- ▲ Click on the menu item **Draw > Region:**
 - ▲ Padding Percentage: **500**
- ▲ Change the name of the design to:
 - ▲ **BE_Trans_Ckt.mxl** for Basic Exercise Transient with Circuits

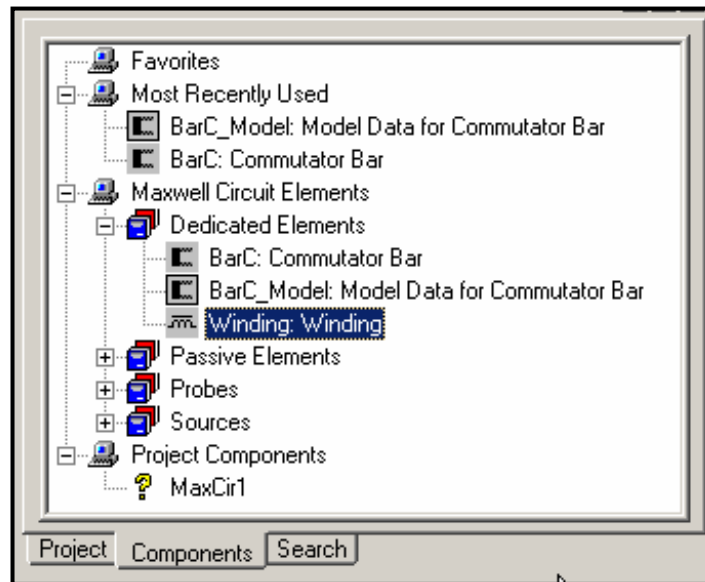
Assign Excitation

- ▲ Select the **Terminal** and click on the menu item **Maxwell > Excitations > Assign > Coil Terminal:**
 - ▲ Name: **Coil_Terminal**
 - ▲ Number of Conductors: **150**
- ▲ Click on the menu item **Maxwell > Excitations > Add Winding**
 - ▲ Name: **Winding_A**
 - ▲ Type: **External**
 - ▲ Stranded: ☒ **Checked**
 - ▲ Initial Current: **0.0 amps**
- ▲ Select **Winding_A** from the project tree under **Excitation** and right mouse click and select **Add Terminals:**
 - ▲ Select: **Coil_Terminal**
- ▲ The project tree should look like this:



Create an External Circuit

- ▲ To access Maxwell Circuit Editor, click the Microsoft **Start** button, select **Programs > Ansoft > Maxwell 11 > Maxwell Circuit Editor**
- ▲ Select the **Components** tab and choose **Maxwell Circuit Elements > Dedicated Elements > Winding** and drag this component onto the sheet

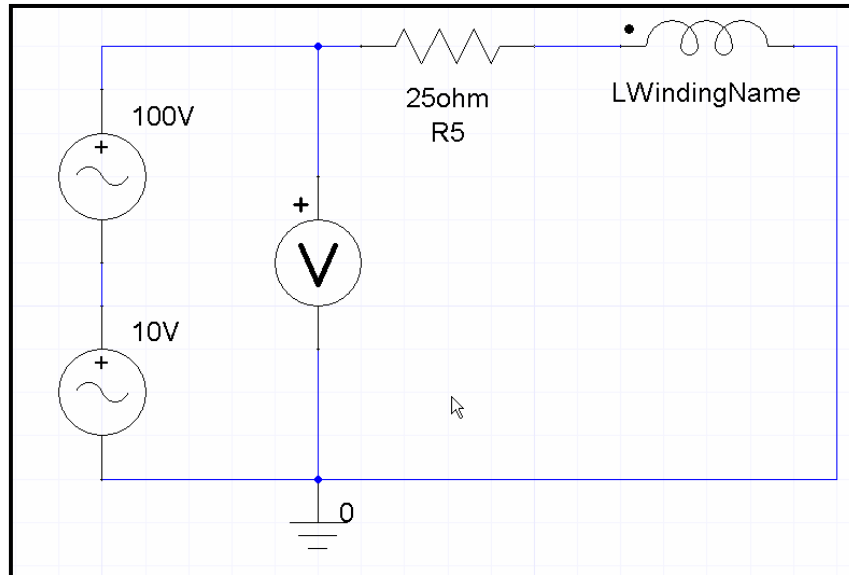


- ▲ In the properties window change the following:
 - ▲ Name: **Winding_A**

Note: This name has to be **exactly** the same name as using the Winding definition described in the previous section “**Assign Excitation**”
- ▲ Select **Sources > VSin** drag this component onto the sheet:
 - ▲ VA: **100 volts**
 - ▲ VFreq: **60 Hz**
- ▲ Select **Source > VSin** drag this component onto the sheet:
 - ▲ VA: **10 volts**
 - ▲ VFreq: **1000 Hz**
- ▲ Select **Passive Elements > Res** and drag this component onto the sheet:
 - ▲ R: **25 ohms**
- ▲ Connect all of the elements together with wires and add a ground
- ▲ Select **Probes > Voltmeter** and place it between the two voltage sources and ground

Basic Exercise - Transient with Circuits

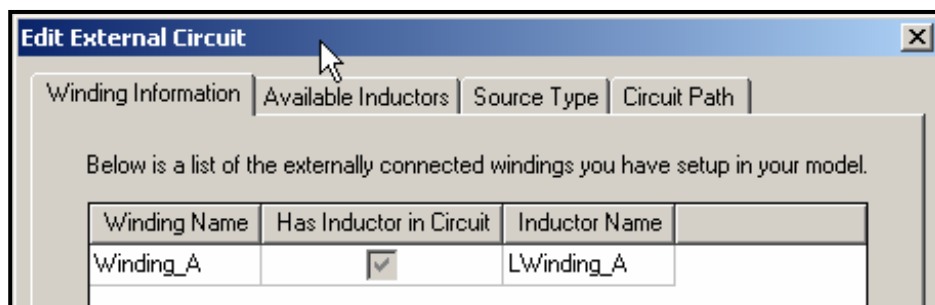
- ▲ The circuit should look like this:



- ▲ Click on **Maxwell Circuit > Export Netlist:**
 - ▲ File Name: **Inductor_Circuit.sph**
- ▲ Click on **Edit > Save As:**
 - ▲ **BE_Circuit.amcp** for Basic Exercise Circuit

▲ Link the circuit file to the Maxwell project

- ▲ In the Maxwell BE_Trans_Ckt.mxl project click on the menu item **Maxwell > Excitations > External Circuits > Edit External Circuit.**
 - ▲ Import Circuit: **Inductor_Circuit.sph**
- ▲ The Edit External Circuit Panel should look like this:



Basic Exercise - Transient with Circuits

Apply Mesh Operations

-  The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created
-  Select the **Core** and click on the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 -  Restrict Length Of Elements: **Uncheck**
 -  Restrict Number of Elements: ☒ **Check**
 -  Maximum Number of Elements: **1000**
-  Select the **Coil** and click on the menu item **Maxwell > Mesh Operations > Assign > On Selection > Length Based**.
 -  Restrict Length Of Elements: **Uncheck**
 -  Restrict Number of Elements: ☒ **Check**
 -  Maximum Number of Elements: **1000**

Create Analysis Setup

-  Click on **Maxwell > Analysis Setup > Add Solution Setup**
 -  Stop Time: **20 ms**
 -  Time Step: **50 us**
 -  Click on **OK**

Solve the Problem

-  Select **Setup1** from under **Analysis** in the project tree, right mouse click and select **Analyze**

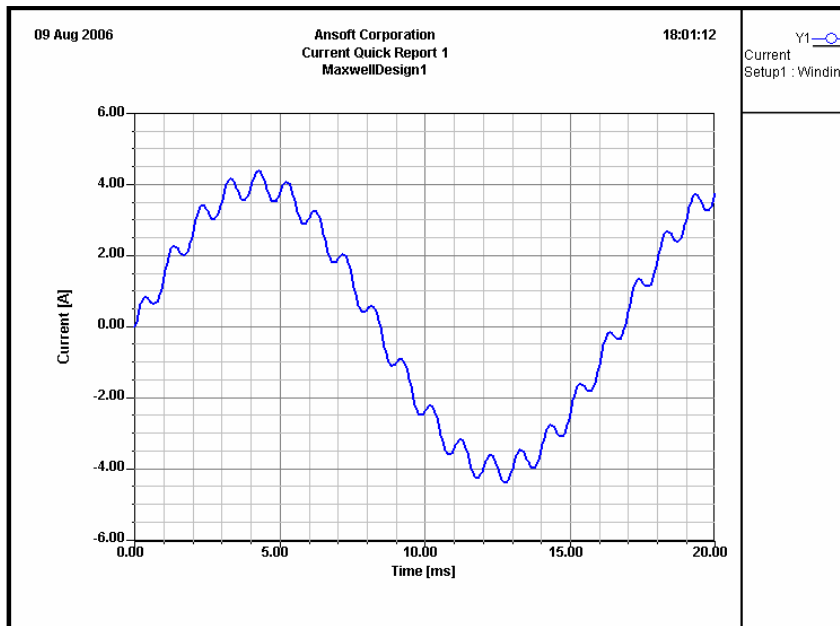
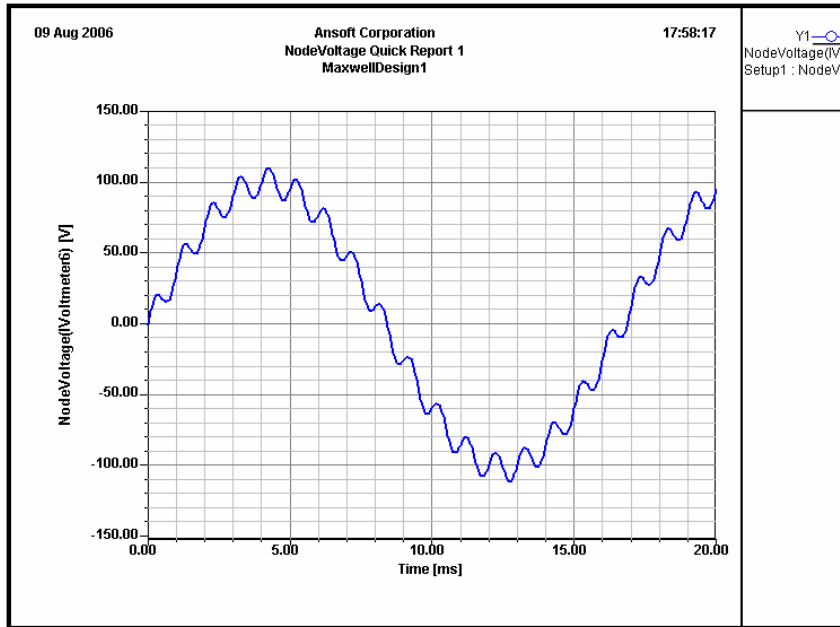
Save the Design

-  Click on **File > Save** to save the design and results

Basic Exercise - Transient with Circuits

Plot the Voltage and Current

- Click on **Maxwell > Results > Create Quick Report** and select **Node Voltage**.
- Click on **Maxwell > Results > Create Quick Report** and select **Current**



This concludes the Basic Example for Transient Sources Using External Circuits



- ❖ 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.

Maxwell[®] v.11

- ▶ Maxwell solves the electromagnetic field problems by solving Maxwell's equations in a finite region of space with appropriate boundary conditions and — when necessary — with user-specified initial conditions in order to obtain a solution with guaranteed uniqueness.

- ▶ **Electric 3D fields:**

- ▶ Electrostatic 3D fields in dielectrics
- ▶ Electric 3D fields in conductors
- ▶ A combination of the first two with conduction solutions being used as boundary conditions for an electrostatic problem.

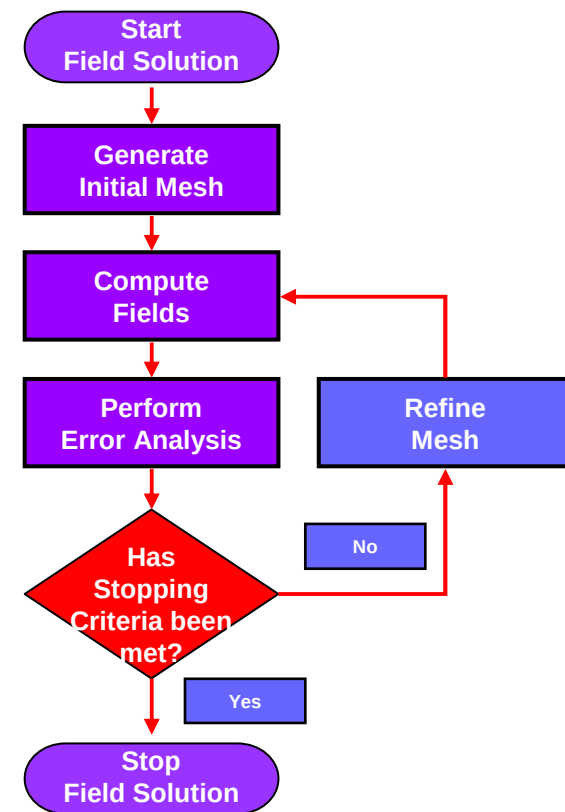
- ▶ **Magnetostatic 3D fields**

- ▶ **Eddy current 3D fields**

- ▶ **Transient 3D fields**

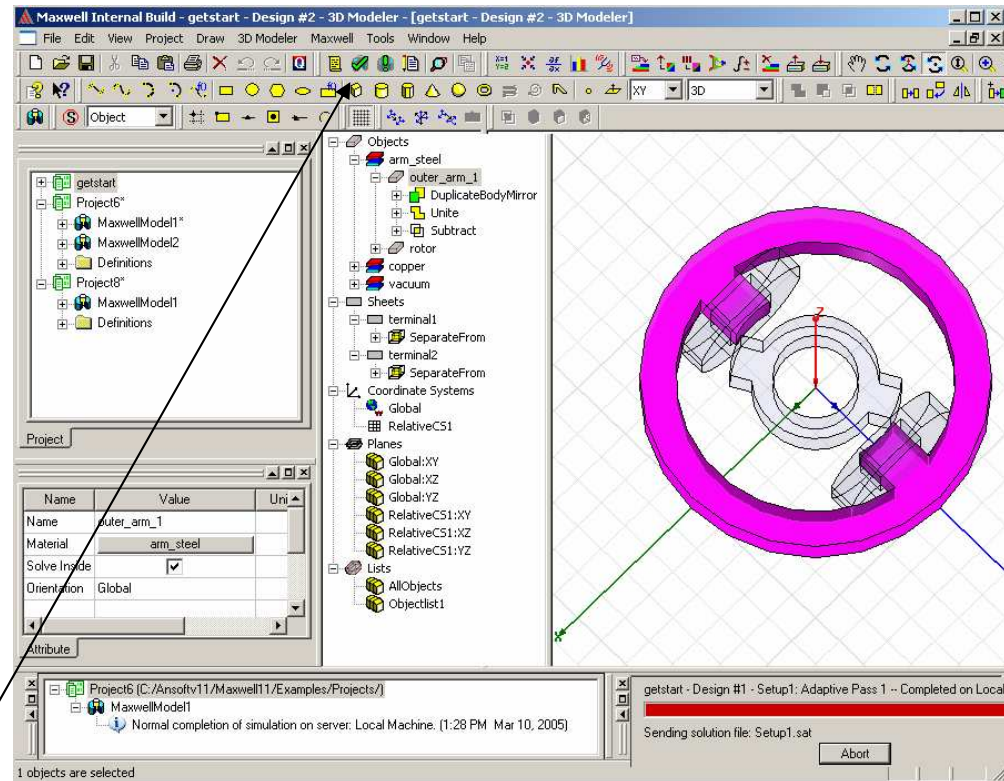
FEM and adaptive meshing

- ❖ In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into tetrahedral elements.
- ❖ All the model solids are meshed automatically by the mesher.
- ❖ The assembly of all tetrahedra is referred to as the finite element mesh of the model or simply the mesh.



GUI - Desktop

- ❖ The complex functionality built into the Maxwell 3D solvers is accessed through the main user interface (called the desktop).
- ❖ Problem can be setup in a fairly arbitrary order (rather than following the steps in a precise order as was required in previous versions of Maxwell).
- ❖ A new “validation check” has been added to insure that all required steps are completed



ACIS solid modeling kernel

- ❖ The underlying solid modeling technology used by Ansoft products is provided by ACIS geometric modeler. ACIS version 14 is presently used.
- ❖ Users can create directly models using primitives and operations on primitives.
- ❖ In addition, users can import models saved in a variety of formats (.sm2 .gds .sm3 .sat .step .iges .dwg .sld .geo .stl)
- ❖ When users import models into Ansoft products, translators are invoked that convert the models to an ACIS native format (sat format).
- ❖ Exports directly .sat, .dxf, .sm3, .sm2, .step, .iges

Supported platforms

- ▲ Windows (2000 Professional, XP Professional, XP Professional x64 Edition, Server 2003)
- ▲ Windows Sun Solaris
- ▲ Red Hat Enterprise Linux v3

System requirements (Windows)

▀ 32-Bit System Requirements

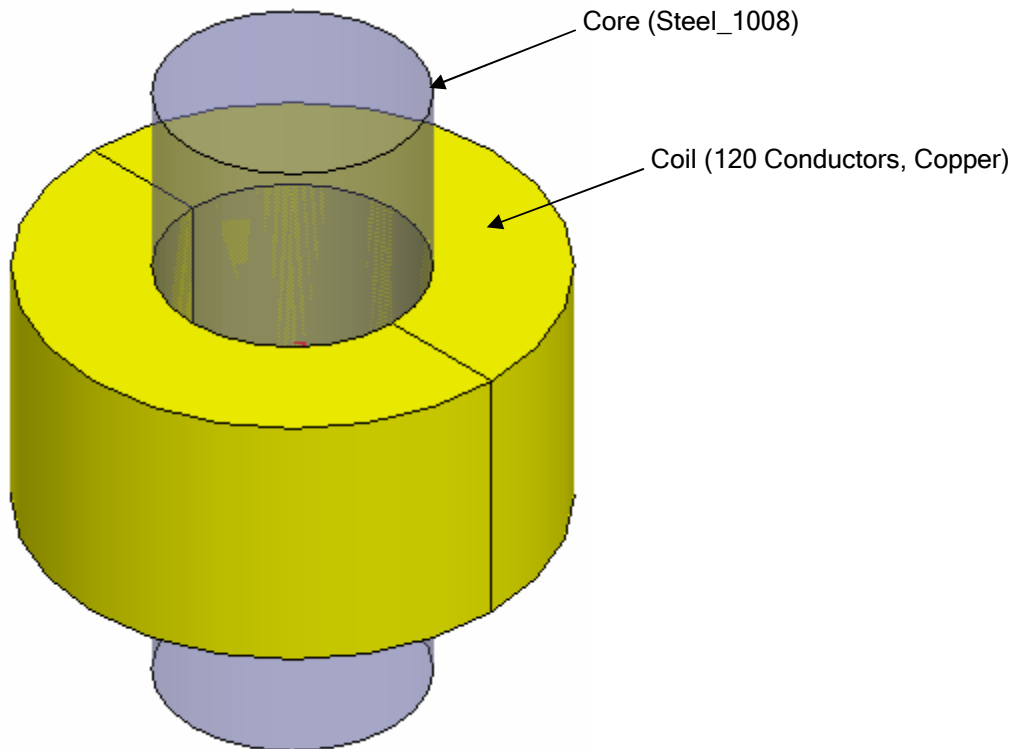
▀ Minimum System Requirements:

- ▀ Pentium-based processor: 500 MHz
- ▀ Hard Drive Space (for Maxwell software): 200 MB
- ▀ RAM: 512 MB

▀ Recommended Minimum Configuration (for Optimal Performance):

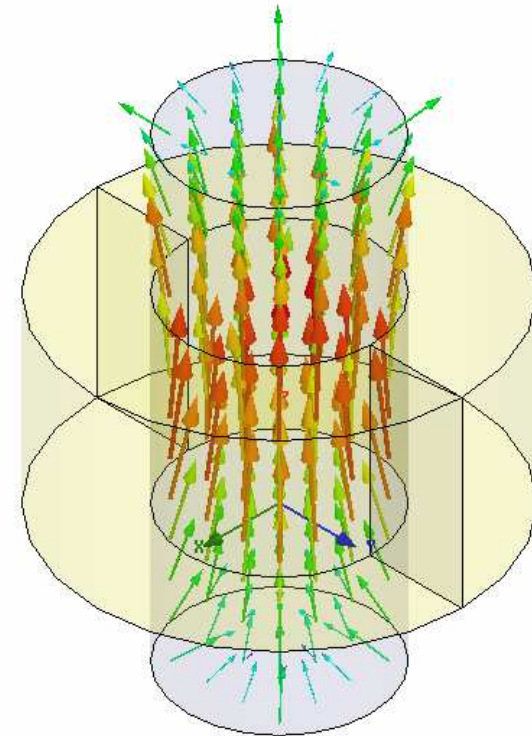
- ▀ Pentium-based processor: 2 GHz
- ▀ Hard Drive Space (for Maxwell software and temporary files): 500 MB
- ▀ RAM: 2 GB

Quick Example - Coil



Maxwell 3D

- 3D field solver
- Solves for the fields in an arbitrary volume



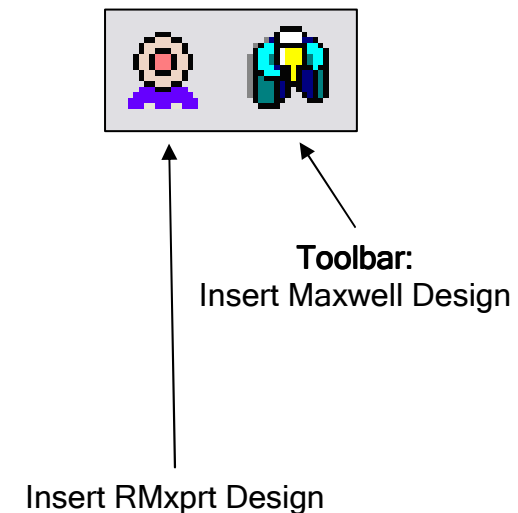
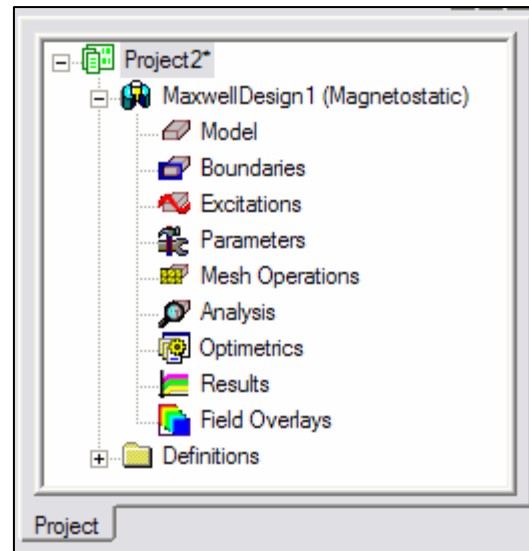
Starting Maxwell

- Click the Microsoft **Start** button, select **Programs**, and select the *Ansoft > Maxwell 11 > Maxwell 11*
- Or Double click on the Maxwell 11 icon on the Windows Desktop



Adding a Design

- When you first start Maxwell a new project will be automatically added to the Project Tree.
- To insert a Maxwell Design to the project, select the menu item *Project > Insert Maxwell Design*



Ansoft Desktop

Menu
bar

Project
Manager
with project
tree

Message
Manager

Status
bar

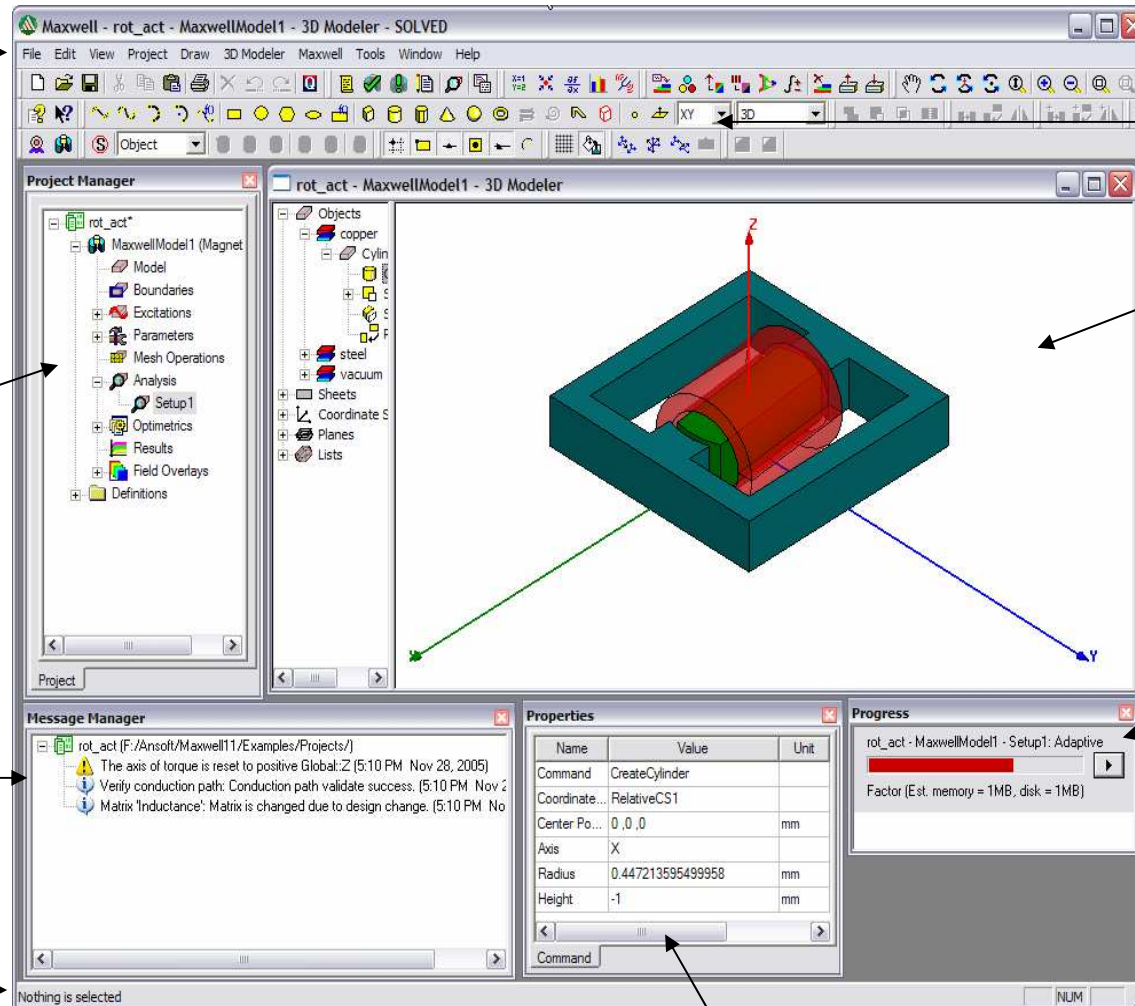
Toolbars

3D Modeler
Window

Progress
Window

Coordinate Entry Fields

Property Window

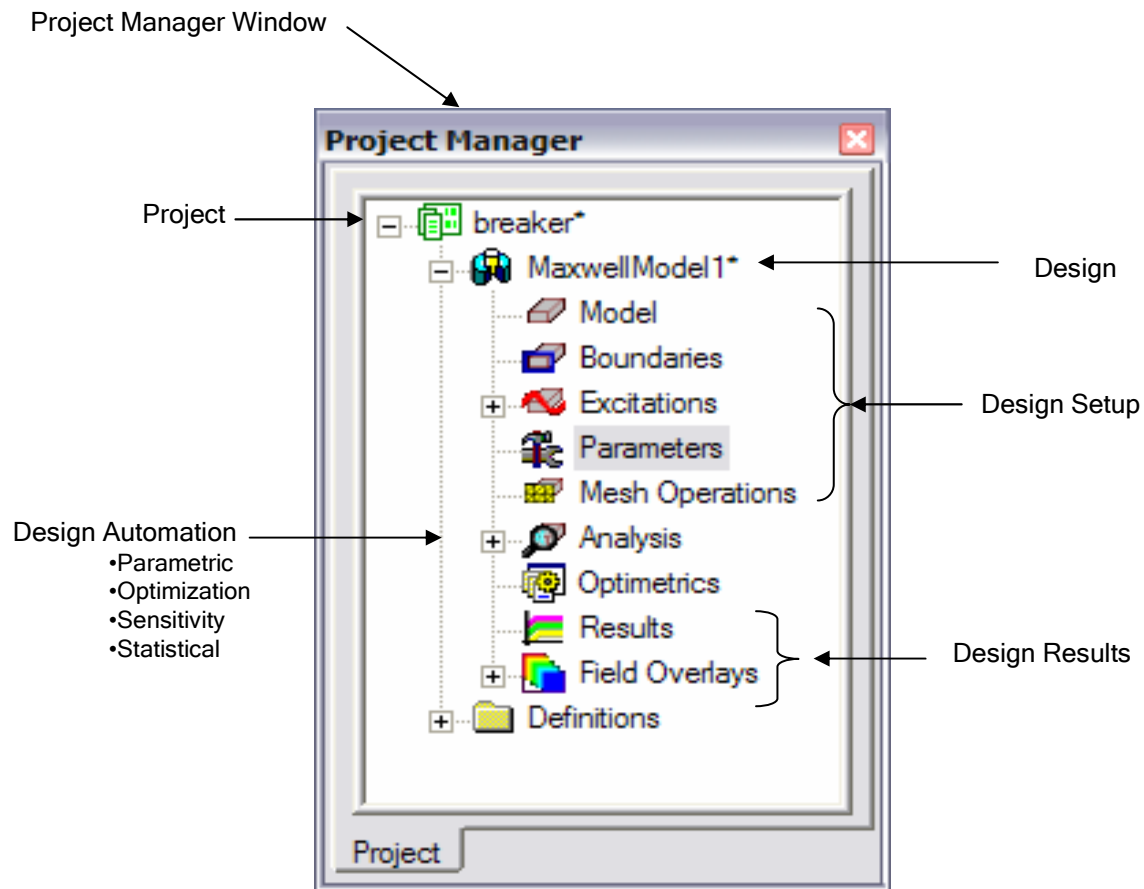


Enter the box position.

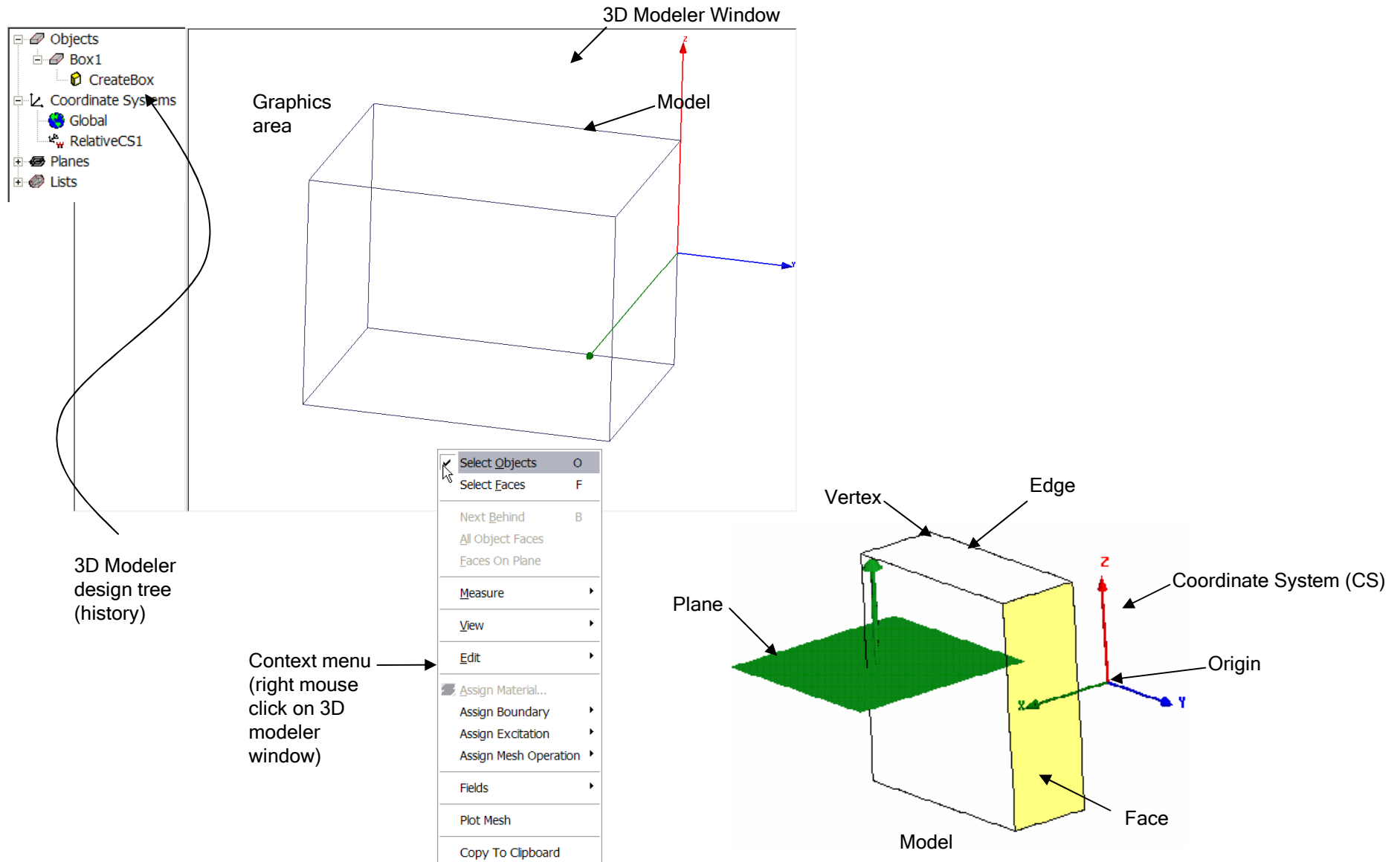
X: 350.00 Y: -100.00 Z: 0.0000 Absolu Cartesia mil

▲ Ansoft Desktop - Project Manager

- ▲ Multiple Designs per Project
- ▲ Multiple Projects per Desktop
- ▲ Integrated Optimetrics Setup (requires license for analysis)



Ansoft Desktop - 3D Modeler



Set Solution Type

To set the solution type:

1. Select the menu item *Maxwell > Solution Type*
2. Solution Type Window:
 1. Choose **Magnetostatic**
 2. Click the OK button

To edit notes: *Maxwell > Edit Notes*

Maxwell - Solution Types

Magnetostatic

- Static magnetic fields, forces, torques, and inductances caused by DC currents, static external magnetic fields, and permanent magnets. Linear or nonlinear materials.

Eddy Current

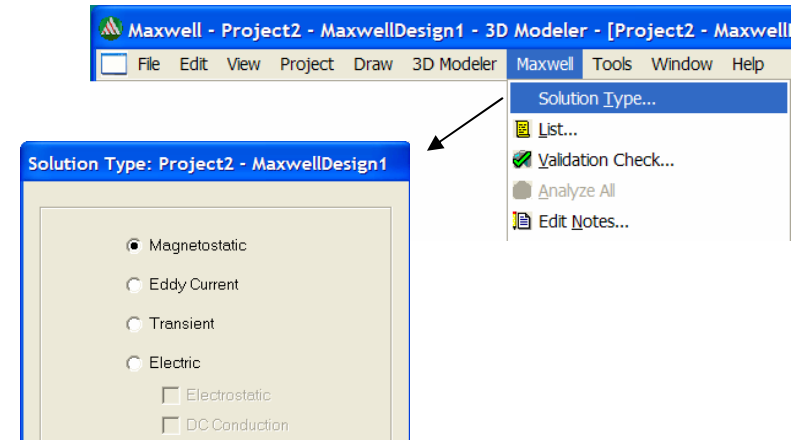
- Sinusoidally-varying magnetic fields, forces, torques, and impedances caused by AC currents and oscillating external magnetic fields. Linear materials only. Full wave solver considers displacement currents. Induced fields such as skin and proximity effects considered.

Transient

- Transient magnetic fields caused by time-varying or moving electrical sources and permanent magnets. Linear or nonlinear materials. Induced fields such as skin and proximity effects considered. Sources can be DC, sinusoidal, or transient voltages or currents. Can use external schematic circuit.

Electrostatic

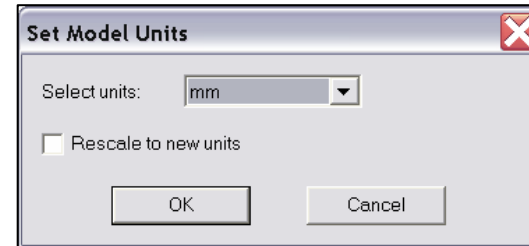
- Static electric fields, forces, torques, and capacitances caused by voltage distributions and charges. Linear materials only.



Set Model Units

To set the units:

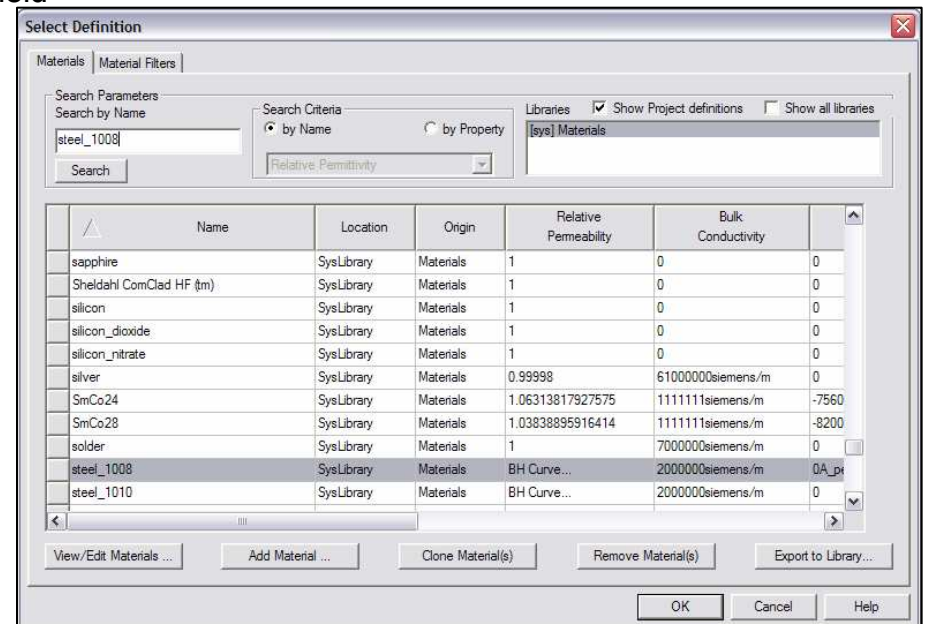
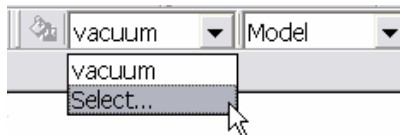
1. Select the menu item **3D Modeler > Units**
2. Set Model Units:
 1. Select Units: **mm**
 2. Click the **OK** button



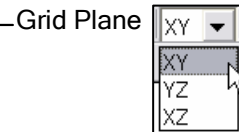
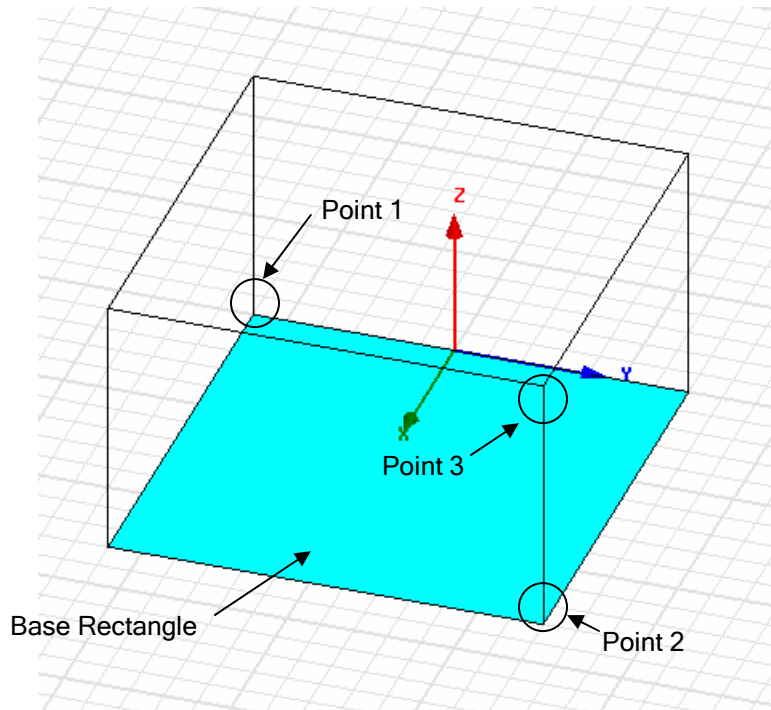
Set Default Material

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



3D Modeler - Create a Primitive



Note: Point 1 and Point 2 are drawn in the grid plane, while Point 3 is drawn in the 3rd direction

X: 0 Y: 0 Z: -0.35

Coordinate Entry Fields

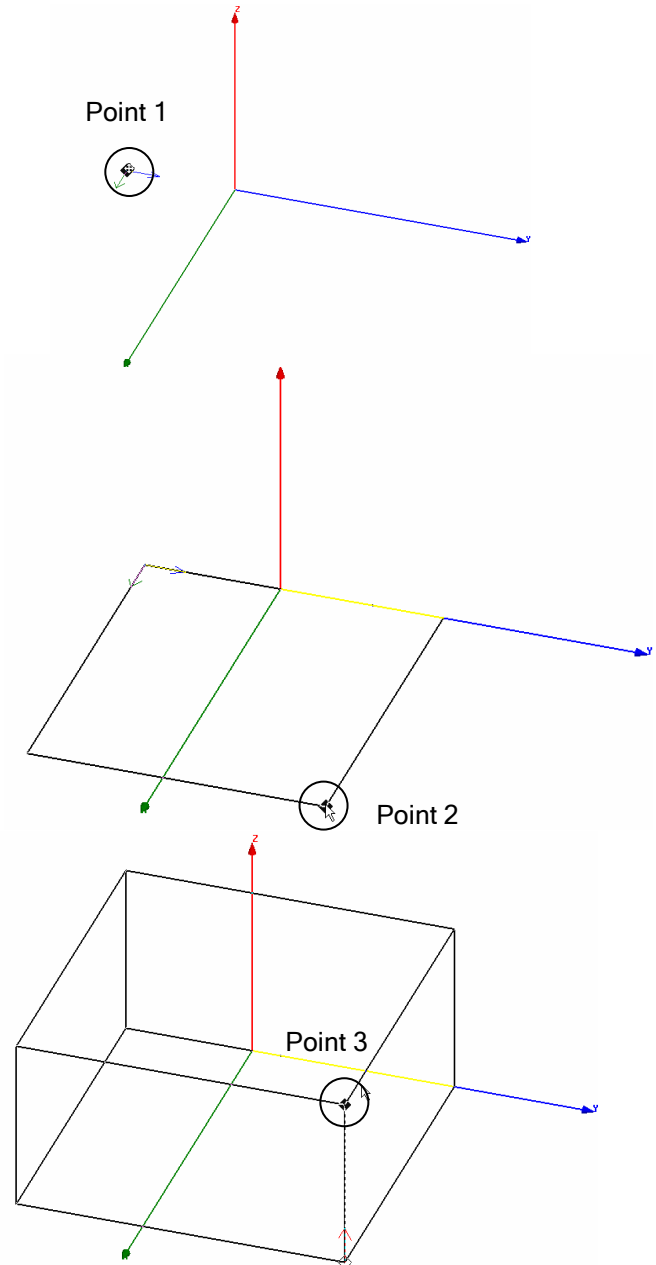
dX: 0.0000 dY: 0.0000 dZ: 0.0000 Relativ Cartesia mm

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



3D Modeler - Object Properties

Commands
(dimensions
and history)

Properties: Project4 - MaxwellDesign1 - 3D Modeler

Command | Attribute

Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate System	Global		
Center Position	0,0,0	mm	0mm, 0mm, 0mm
Axis	Z		
Radius	0.447213595499958	mm	0.447213595499958mm
Height	0.6	mm	0.6mm

☐ Show Hidden

OK Cancel

Attributes
(properties
of the object)

Properties: Project4 - MaxwellDesign1 - 3D Modeler

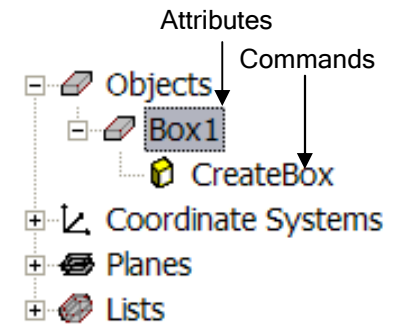
Command | Attribute

Name	Value	Unit	Evaluated Value	Description	Read-only
Name	Cylinder1				☐
Material	vacuum				☐
Solve Inside	☒				☐
Orientation	Global				☐
Model	☒				☐
Display Wireframe	☐				☐
Color	Edit				☐
Transparent	0				☐

☐ Show Hidden

OK Cancel

In History Tree:



3D Modeler - Attributes

Name	Value	Unit
Name	Box1	
Material	vacuum	
Solve Inside	☒	
Orientation	Global	
Model	☒	
Display Wireframe	☐	
Color	Edit	
Transparent	0	

Attribute

Solve Inside - if unchecked meshes but no solution inside (like the old exclude feature in material manager)

Model - if unchecked, the object is totally ignored outside of modeler with no mesh and no solution

Select Definition

Materials | Material Filters

Search Parameters
Search by Name
Nd
Search

Search Criteria
by Name by Property
Relative Permeability

Libraries ☒ Show Project definitions ☒ Show all libraries
[personal] test
[sys] Materials
[sys] RMxpri

Name	Location	Origin	Relative Permeability	Bulk Conductivity	
NdFe30	SysLibrary	Materials	1.0445730167132	625000siemens/m	-83799
NdFe35	SysLibrary	Materials	1.0997785406	625000siemens/m	-89000
Nelco N4000-13 (tm)	SysLibrary	Materials	1	0	0
Nelco N4000-13 SI (tm)	SysLibrary	Materials	1	0	0
Neltec NH9294 (tm)	SysLibrary	Materials	1	0	0
Neltec NH9300 (tm)	SysLibrary	Materials	1	0	0
Neltec NH9320 (tm)	SysLibrary	Materials	1	0	0
Neltec NH9338 (tm)	SysLibrary	Materials	1	0	0
Neltec NH9348 (tm)	SysLibrary	Materials	1	0	0
Neltec NH9350 (tm)	SysLibrary	Materials	1	0	0
Neltec NX9240 (tm)	SysLibrary	Materials	1	0	0

View/Edit Materials ... Add Material ... Clone Material(s) Remove Material(s) Export to Library...

OK Cancel Help

View / Edit Material

Material Name
vacuum

Material Coordinate System Type:
Cartesian

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Simple	1	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
Composition		Solid	

View/Edit Material for
☒ Active Design
☐ This Product
☐ All Products

Validate Material

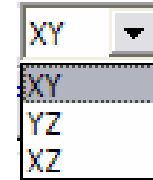
Calculate Properties for:

Reset OK Cancel

Set Grid Plane

To set the Grid Plane:

- Select the menu item *3D Modeler > Grid Plane > XY*



Create Core

To create the coax pin:

- Select the menu item *Draw > Cylinder*
- Using the coordinate entry fields, enter the center position
 - X: 0.0, Y: 0.0, Z: -3.0, Press the **Enter** key

X:	0	Y:	0	Z:	-3	Absolute	Cartesian	mm

- Using the coordinate entry fields, enter the radius of the cylinder
 - dX: 0.0, dY: 2.0, dZ: 0.0, Press the **Enter** key

dX:	0	dY:	2	dZ:	0	Relative	Cartesian	mm

- Using the coordinate entry fields, enter the height of the cylinder
 - dX: 0.0, dY: 0.0 dZ: 10.0, Press the **Enter** key

dX:	0	dY:	0	dZ:	10	Relative	Cartesian	mm

Continued on Next Page

Create Core (Continued)

To Parameterize the Height

1. Select the **Command** tab from the **Properties** window
2. Height: **H**
3. Press the **Tab** key
4. Add Variable Window
 1. Value: **10mm**
 2. Click the **OK** button

To set the name:

1. Select the **Attribute** tab from the **Properties** window.
2. For the **Value** of **Name** type: **Core**

To set the material:

1. Select the **Attribute** tab from the **Properties** window
2. Click on the button in Material value: set to steel_1008

To set the color:

1. Select the **Attribute** tab from the **Properties** window.
2. Click the **Edit** button

To set the transparency:

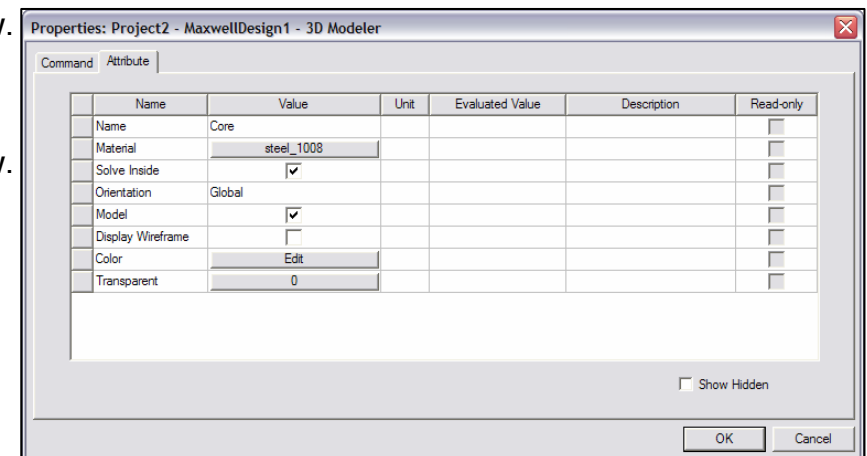
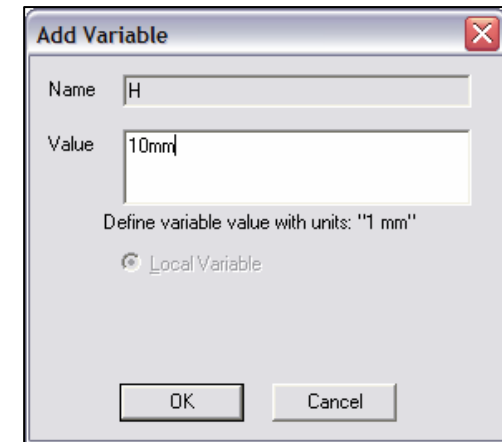
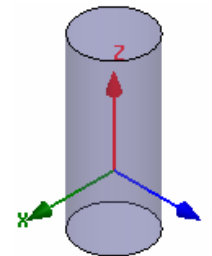
1. Select the **Attribute** tab from the **Properties** window.
2. Click the **OK** button

To finish editing the object properties

1. Click the **OK** button

To fit the view:

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

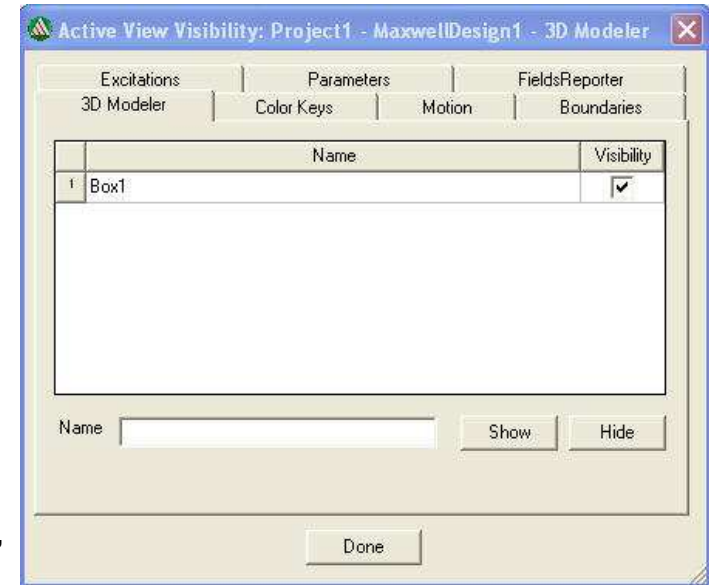


3D Modeler - Views

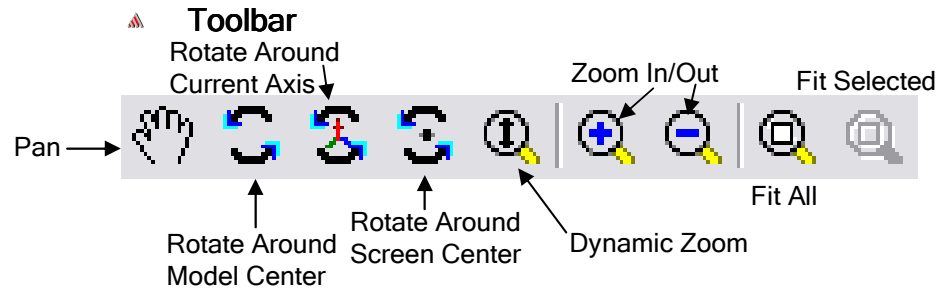
- ▲ **View > Modify Attributes >**
 - ▲ **Orientation** - Predefined/Custom View Angles
 - ▲ **Lighting** - Control angle, intensity, and color of light
 - ▲ **Projection** - Control camera and perspective
 - ▲ **Background Color** - Control color of 3D Modeler background
- ▲ **View > Active View Visibility** - Controls the display of: 3D Modeler Objects, Color Keys, Boundaries, Excitations, Field Plots
- ▲ **View > Options** - Stereo Mode, Drag Optimization, Color Key Defaults, Default Rotation
- ▲ **View > Render > Wire Frame** or **Smooth Shaded** (Default)
- ▲ **View > Coordinate System > Hide** or **Small (Large)**
- ▲ **View > Grid Setting** - Controls the grid display



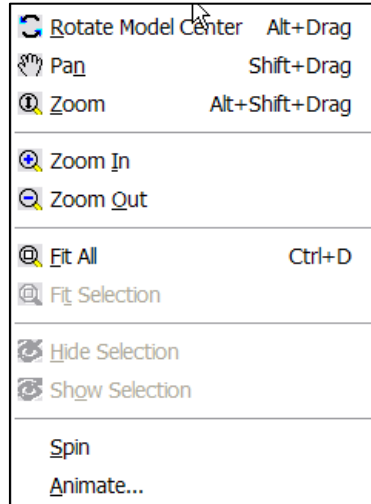
Toolbar: Toggle Grid Visibility



Changing the View



Context Menu



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:

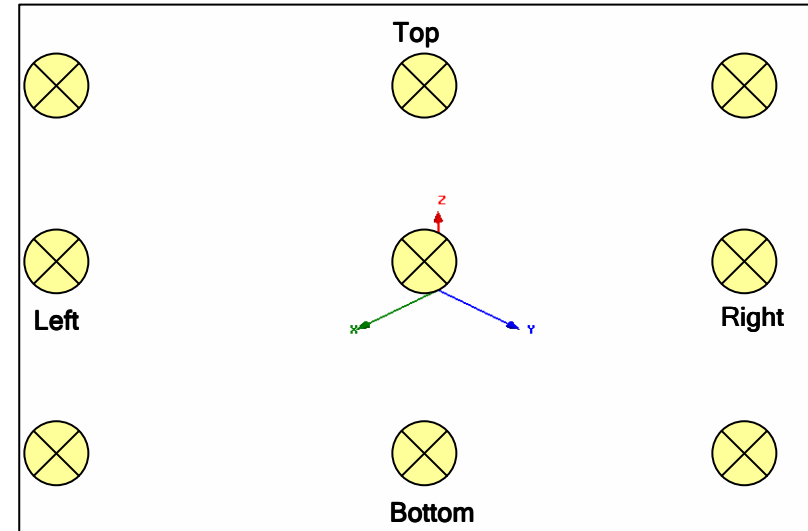
- ALT + Drag** - Rotate

- 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

Predefined View Angles



Set Default Material

To set the 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

To create the coil for the current to flow:

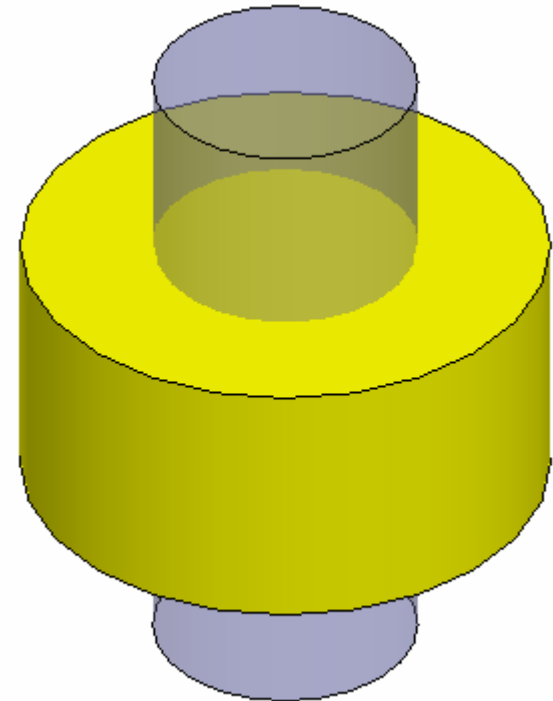
1. Select the menu item **Draw > Cylinder**
2. Using the coordinate entry fields, enter the center position
 - ▲ X: 0.0, Y: 0.0, Z: 0.0, Press the **Enter** key
4. Using the coordinate entry fields, enter the radius of the cylinder
 - ▲ dX: 0.0, dY: 4.0, dZ: 0.0, Press the **Enter** key
5. Using the coordinate entry fields, enter the height of the cylinder
 - ▲ dX: 0.0, dY: 0.0 dZ: 4.0, 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 fit the view:

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



Overlapping Objects

About Overlapping Objects

- When the volume of a 3D object occupies the same space as two or more objects you will receive an overlap error during the validation process. This occurs because the solver can not determine which material properties to apply in the area of overlap. To correct this problem, Boolean operations can be used to subtract one object from the other or the overlapping object can be split into smaller pieces that are completely enclosed within the volume of another object. When an object is completely enclosed there will be no overlap errors. In this case, the material of the interior object is used in the area of overlap.



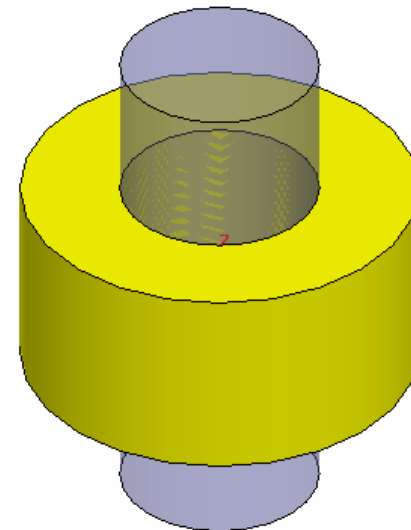
Complete the Coil

To select the objects Core and Coil:

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

To complete the Coil:

1. Select the menu item *3D Modeler > Boolean > Subtract*
2. Subtract Window
 - Blank Parts: **Coil**
 - Tool Parts: **Core**
 - Clone tool objects before subtract: ☒ **Checked**
 - Click the **OK** button



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

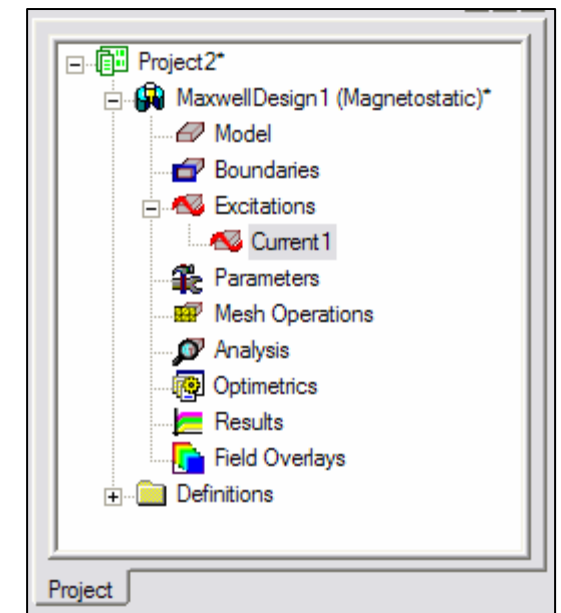
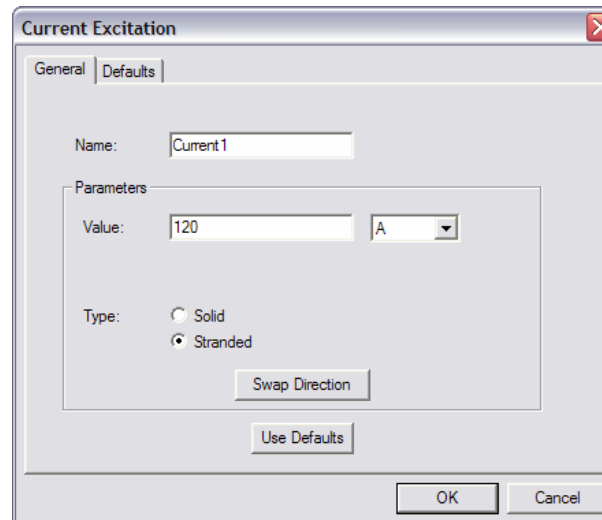
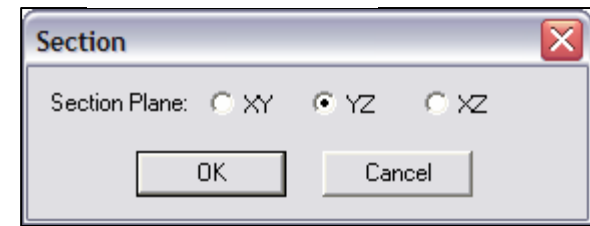
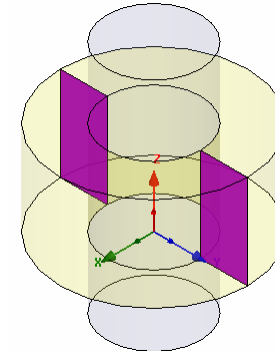
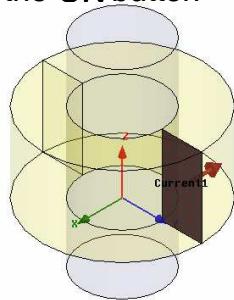
1. Select the menu item **3D Modeler > Surface > Section**
 1. Section Plane: **YZ**
 2. Click the **OK** button

Separate Bodies

1. Select the menu item **3D Modeler > Boolean > Separate Bodies**
2. Delete the extra sheet which is not needed

Assign Excitation

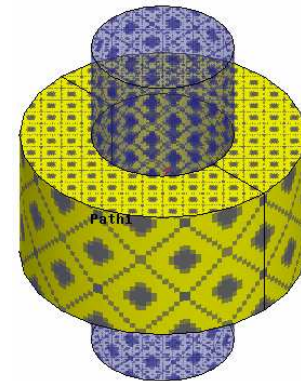
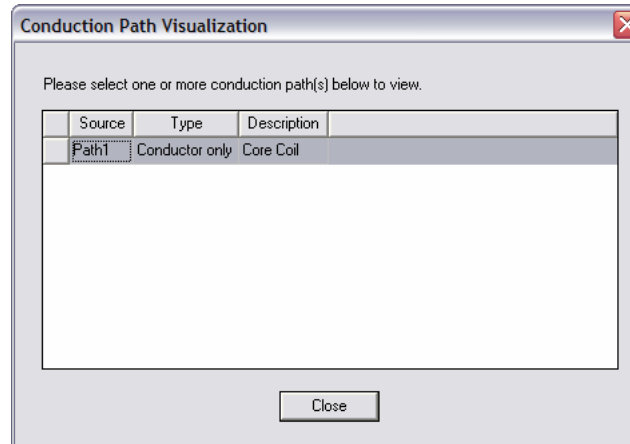
1. Select the menu item **Maxwell > Excitations > Assign > Current**
2. Current Excitation : General
 1. Name: **Current1**
 2. Value: **120 A**
 3. Type: **Stranded**
3. Click the **OK** button



Show Conduction Path

Show Conduction Path

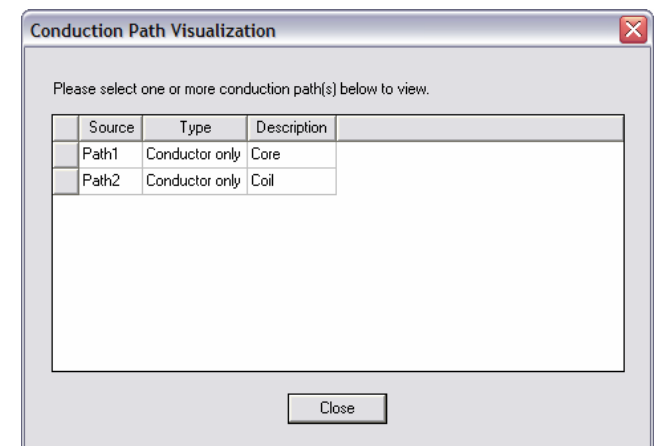
1. Select the menu item **Maxwell > Excitations > Conduction Path > Show Conduction Path**
2. From the Conduction Path Visualization dialog, select the row1 to visualize the conduction path in on the 3D Model.
3. Click the **Close** button



Fix Conduction Path

- To solve this isolation problem an insulating boundary condition will be used.

1. Select the outer face of the core by typing **f** on the keyboard and select the outer face of the core.
2. Select the menu item **Maxwell > Boundaries > Assign > Insulating**
3. Insulating Boundary
 1. Name: **Insulating1**
 2. Click the **OK** button
4. Follow the instructions from above to redisplay the Conduction Path



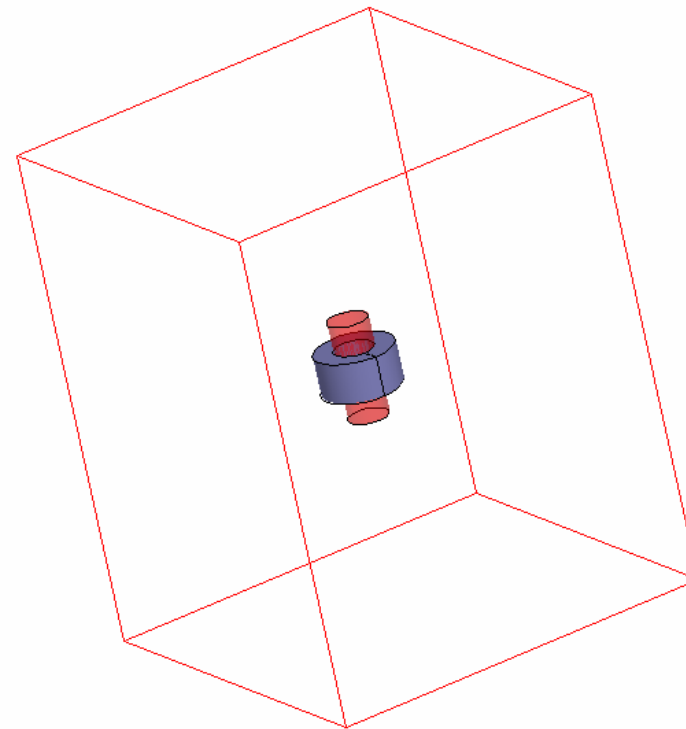
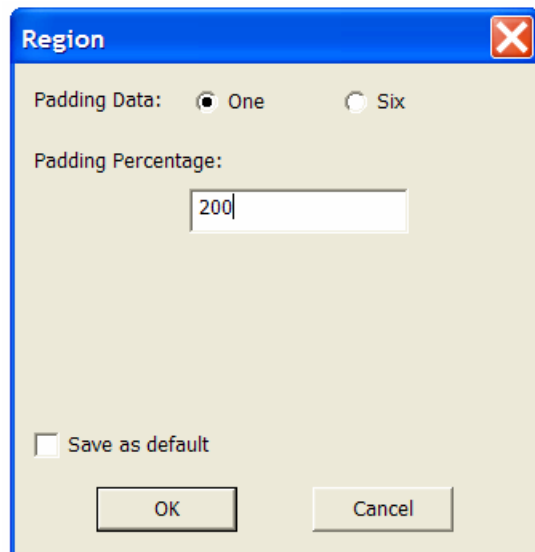
Define a Region

- Before solving a project a region has to be defined. A region is basically an outermost object that contains all other objects. The region can be defined by a special object in **Draw->Region**. This special region object will be resized automatically if your model changes size.

- A ratio in percents has to be entered that specifies how much distance should be left from the model.

To define a Region:

1. Select the menu item **Draw > Region**
 1. Padding Data: **One**
 2. Padding Percentage: **200**
 3. Click the **OK** button



Note: Since there will be considerable fringing in this device, a padding percentage of at least 2 times, or 200% is recommended

Maxwell - Solution Setup

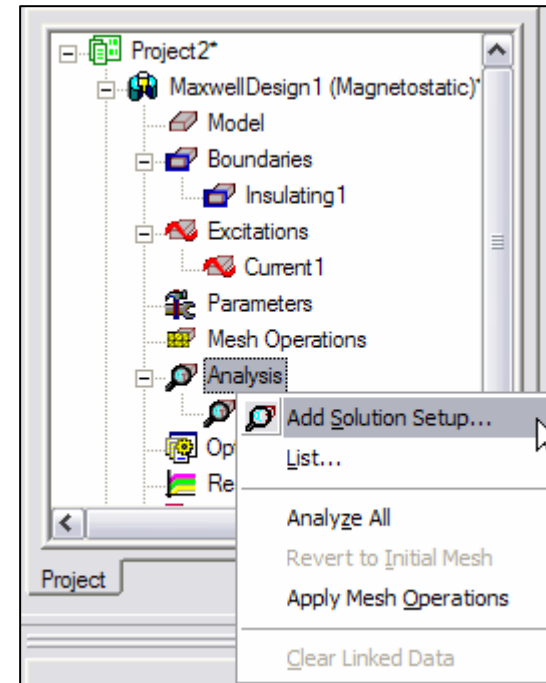
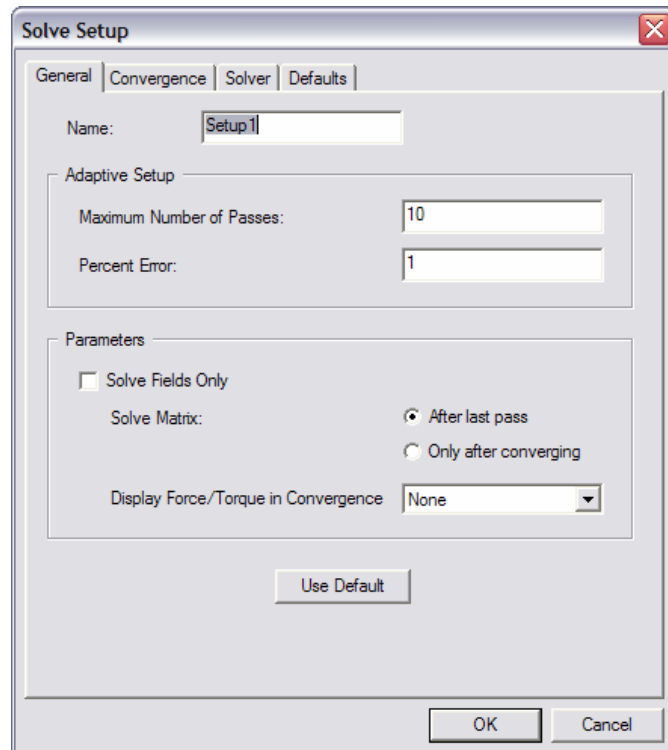
Creating an Analysis Setup

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:
 - Maximum Number of Passes: **10**
 - Percent Error: **1**
 2. Click the **OK** button



Add Solution Setup



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_coil**
3. Click the **Save** button

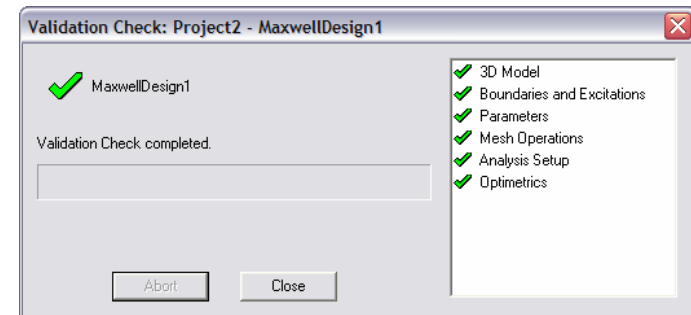
Analyze

Model Validation

To validate the model:

1. Select the menu item **Maxwell > Validation Check**
2. Click the **Close** button

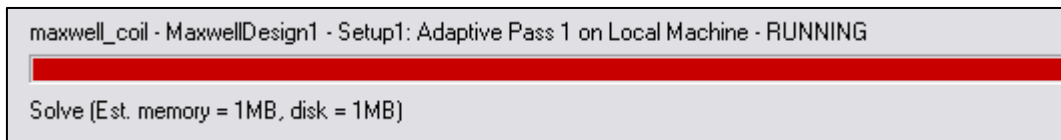
Note: To view any errors or warning messages, use the Message Manager.



Analyze

To start the solution process:

1. Select the menu item **Maxwell > Analyze All**



Validate Analyze All

View detailed information about the progress

In the Project Tree click on **Analysis->Setup1** with the right mouse button and select **Profile**

The image shows the Maxwell 3D Project Tree on the left. The 'Analysis' folder is expanded, and 'Setup1' is selected. A right-click context menu is open over 'Setup1', with the 'Profile...' option highlighted. Two arrows originate from this menu: one points to the 'Solutions: getting - MaxwellDesign1' window, and the other points to the 'Solutions: Ex_overview_example - MaxwellDesign1' window.

Solutions: getting - MaxwellDesign1

Design Variation: [dropdown]
Simulation: Setup1

Tabs: Solution | Convergence | **Profile** | Mesh Statistics

Task	Real Time	CPU Time	Memory	Information
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:00	00:00:00	322 K	815 matrix, 92KB disk
Solver DRS2	00:00:01	00:00:01	24.8 M	9793 matrix, 2MB disk
Solver DRS2	00:00:01	00:00:01	24.8 M	9793 matrix, 2MB disk
Solver DRS2	00:00:00	00:00:01	24.8 M	9793 matrix, 2MB disk
adapt	00:00:05	00:00:05	39.2 M	7674 tetrahedra
Adaptive Passes converged				
Solution Process				
Total	00:00:46	00:00:28	Elapsed time : 00:03:17, Peak Memory Usage : 93 M Time: 09/21/2005 13:13:03, Status: Normal Completion	

Buttons: Export..., Close

Solutions: Ex_overview_example - MaxwellDesign1

Design Variation: H*10mm'
Simulation: Setup1

Tabs: Solution | Convergence | **Profile** | Mesh Statistics

Number of Passes

Completed 9
Maximum 10
Minimum 2

Energy Error/Delta Energy (%)

Target (1, 1)
Current (0.98685, 0.17046)

View: ☒ Table ☐ Plot

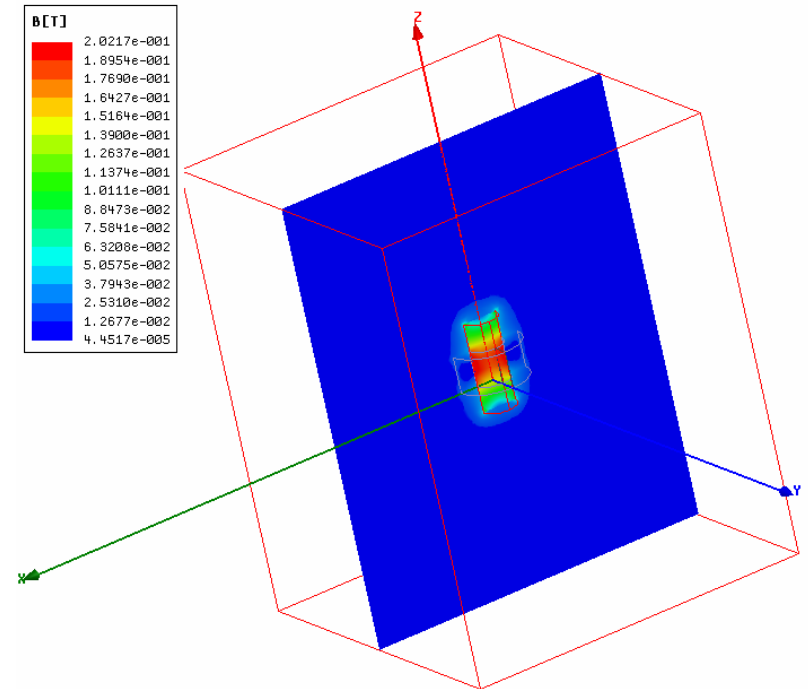
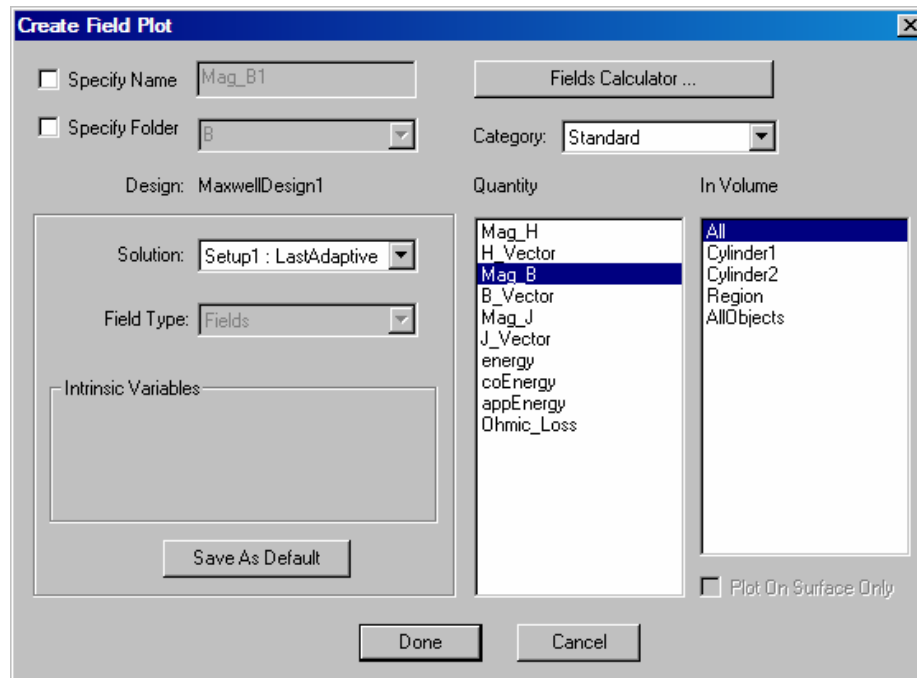
Pass	Tets	Total Energy (J)	Delta Energy (%)	Energy Error (%)	Mag(F/T)	Output Var
1	1199	0.00033687	N/A	48.82	N/A	N/A
2	1554	0.00021194	58.95	44.965	N/A	N/A
3	1984	0.00015116	40.209	18.892	N/A	N/A
4	2642	0.0001338	12.972	10.059	N/A	N/A
5	3444	0.00012913	3.6171	4.4529	N/A	N/A
6	4490	0.00012764	1.1674	2.6971	N/A	N/A
7	5870	0.00012698	0.5165	1.7476	N/A	N/A
8	7681	0.00012672	0.2096	1.4612	N/A	N/A
9	9960	0.0001265	0.17046	0.98685	N/A	N/A

Buttons: Close

Field Overlays

To create a field plot:

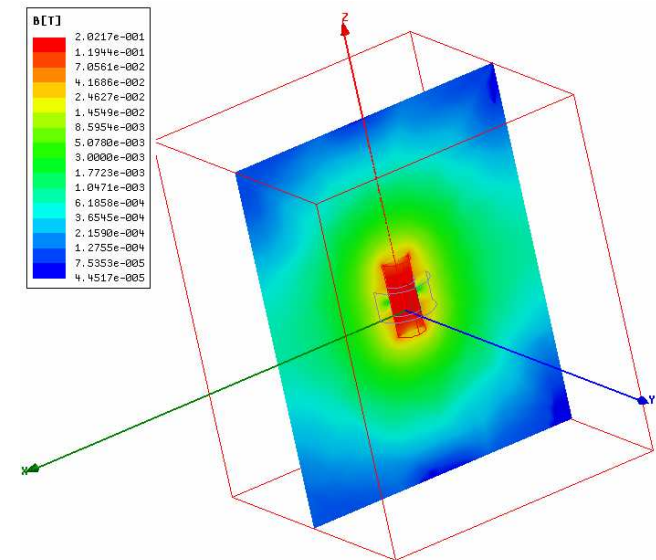
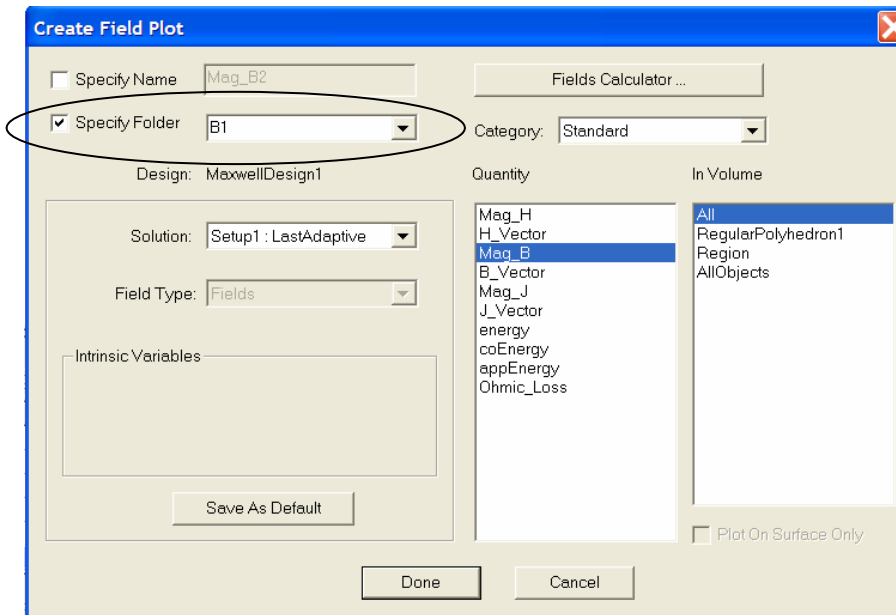
1. Select the Global XZ Plane
 1. Using the Model Tree, expand **Planes**
 2. Select **Global:XZ**
2. Select the menu item **Maxwell > Fields > Fields > B > Mag_B**
3. Create Field Plot Window
 1. Solution: **Setup1 : LastAdaptive**
 2. Quantity: **Mag_B**
 3. In Volume: **All**
 4. Click the **Done** button



Field Overlays

To create a second field plot of same quantity, but different scale:

1. Select the Global XZ Plane
 1. Using the Model Tree, expand **Planes**
 2. Select **Global:XZ**
2. Select the menu item **Maxwell > Fields > Fields > B > Mag_B**
3. Create Field Plot Window
 1. Specify Folder: **B1**
 2. Solution: **Setup1 : LastAdaptive**
 3. Quantity: **Mag_B**
 4. In Volume: **All**
 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: **B**
 2. Click the **OK** button
3. B-Field Window:
 1. Click the **Scale** tab
 1. Scale: **Log**
 2. Click the **Close** button
4. Hide the plots by clicking in the project tree and unchecking Plot Visibility

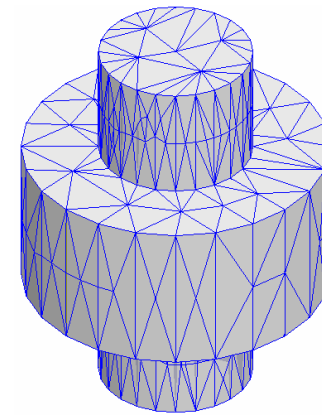
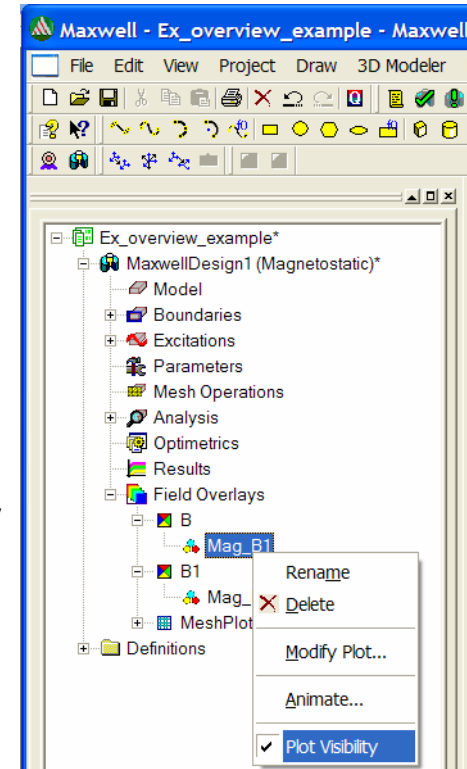
▲ **Mesh Overlay**

▲ To select the objects Core and Coil:

1. Select the menu item **Edit > Select > By Name**
2. Press and hold the CTRL key and select Core and Coil from the list
3. Click the **OK** button

▲ To create a mesh plot:

1. Select the menu item **Maxwell > Fields > Plot Mesh**
2. Create Mesh Window:
 1. Click the **Done** button

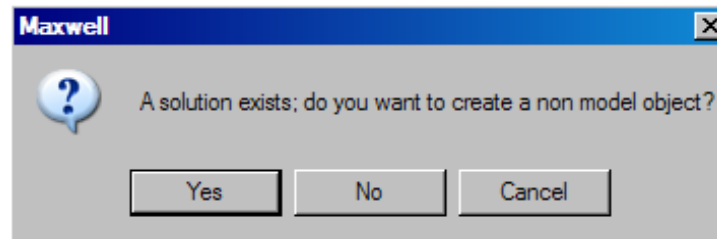


Plot the z-component of B Field along the line

1. Create a line
2. Calculate the z-component of B using the **Calculator tool**
3. Create report - plot the desired graph

To create a line

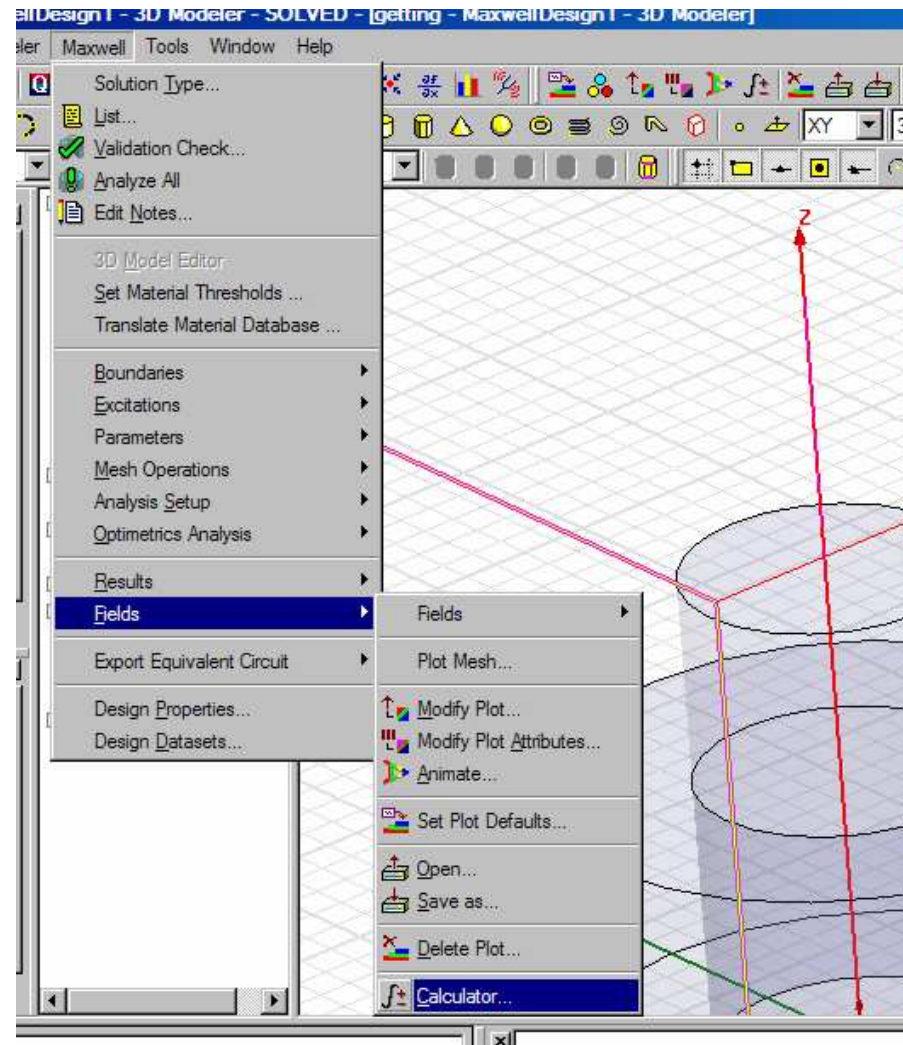
1. Select the menu item **Draw > Line**. The following window opens:



2. Answer **YES** to create a non model object and not to destroy the solution.
3. Place the cursor at the top face of the core and let the cursor snap to the center.
4. Click the left mouse button and hold the z button on the keyboard while moving the cursor up. Left click at some distance from the core and double click left to end the line.
5. Leave the values in the upcoming dialog box and close the dialog by pressing OK.

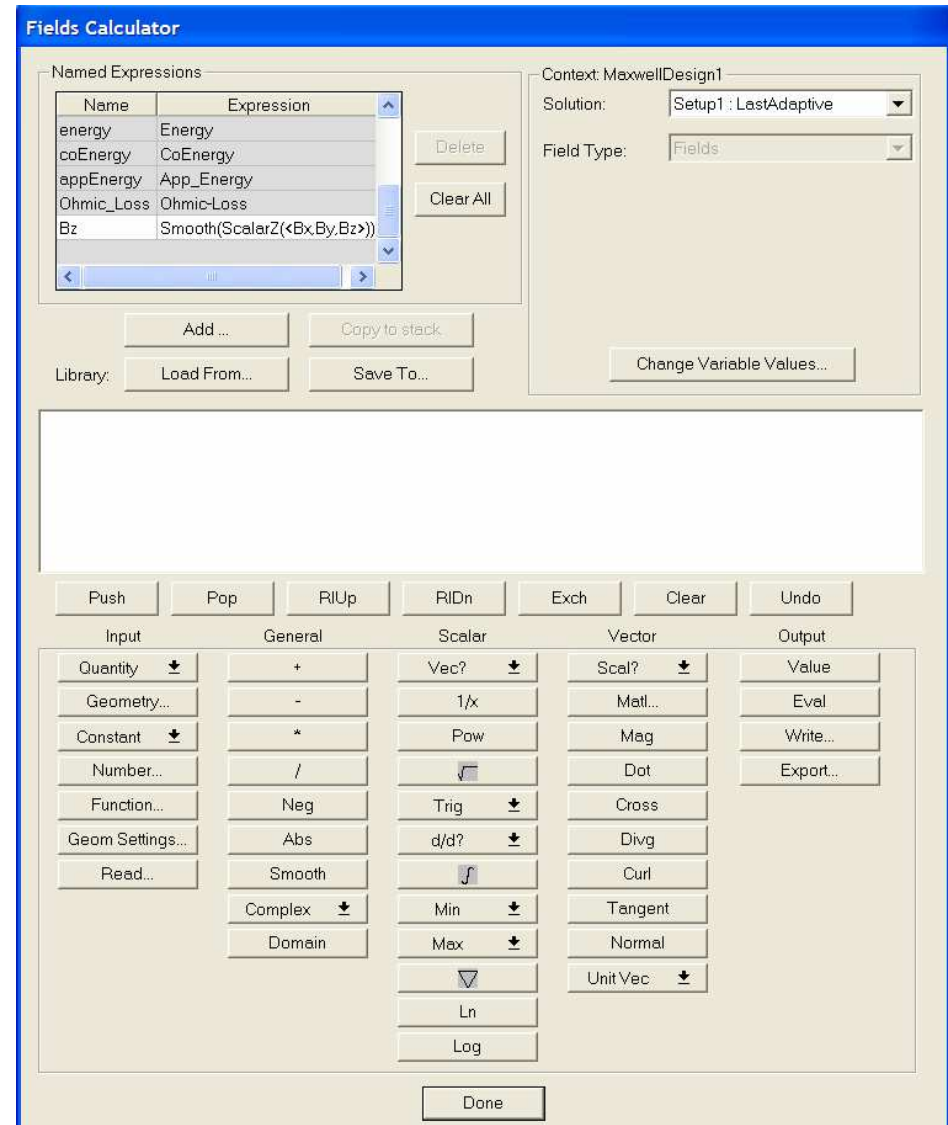
▲ To bring up the Calculator tool

1. Select the menu item **Maxwell->Fields->Calculator**



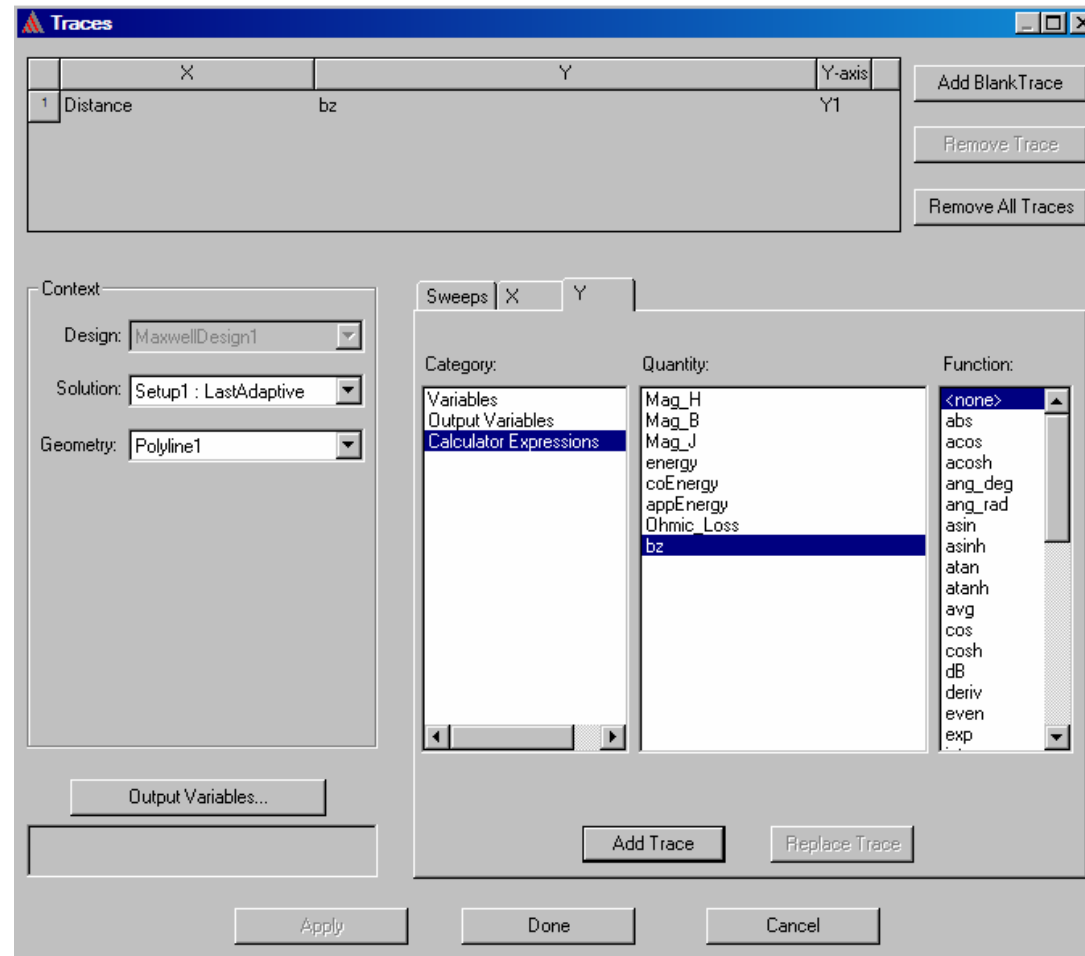
▲ To calculate the z-component of B field

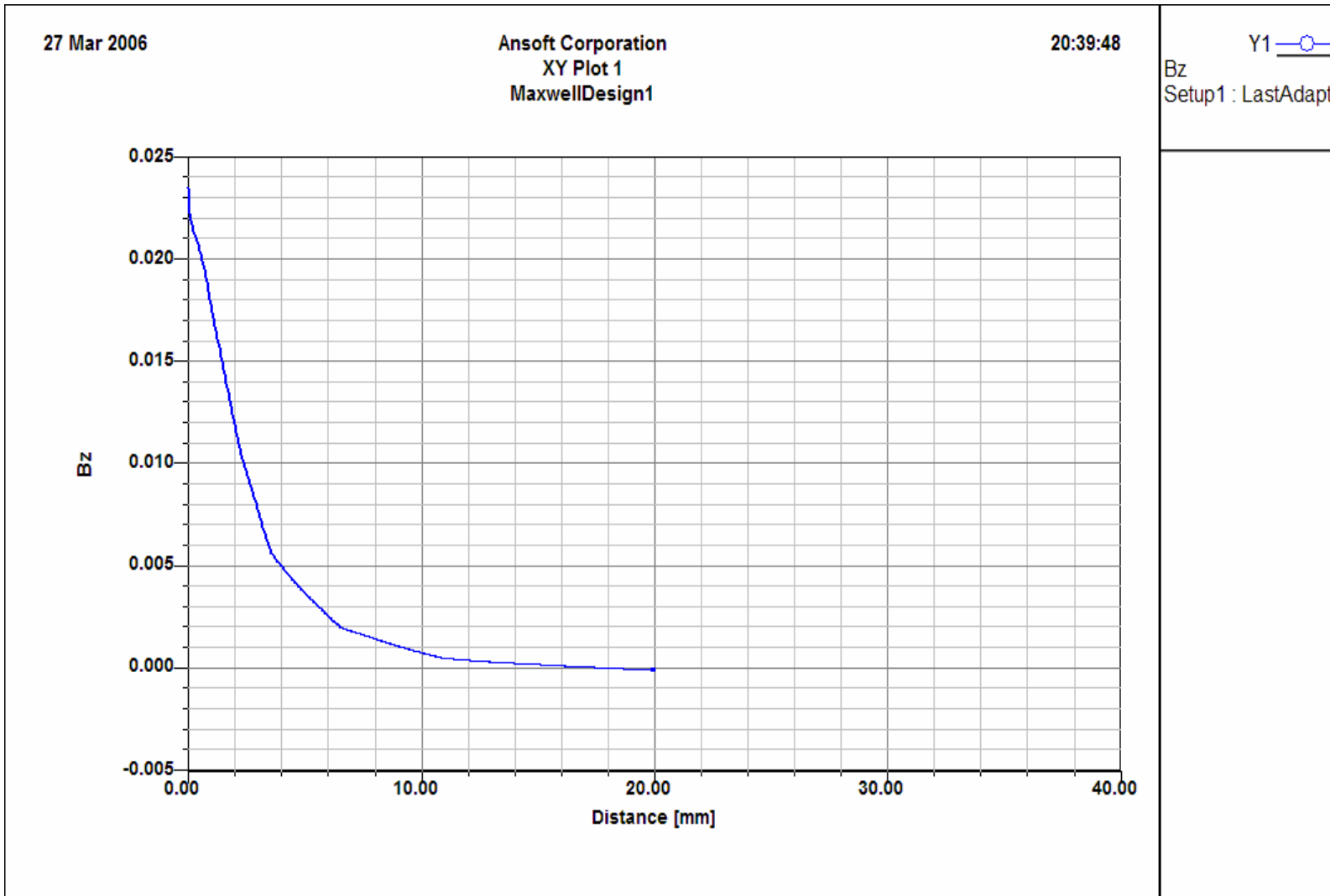
1. In the Input Column select the **Quantity B**
2. In the Vector Column select the **ScalarZ** component
3. In the General Column select **Smooth**
4. Press the **Add** button in the upper part of the calculator and enter a name that describes the calculation
- enter **Bz** for B field in z-direction
5. Click **Done** to close the calculator



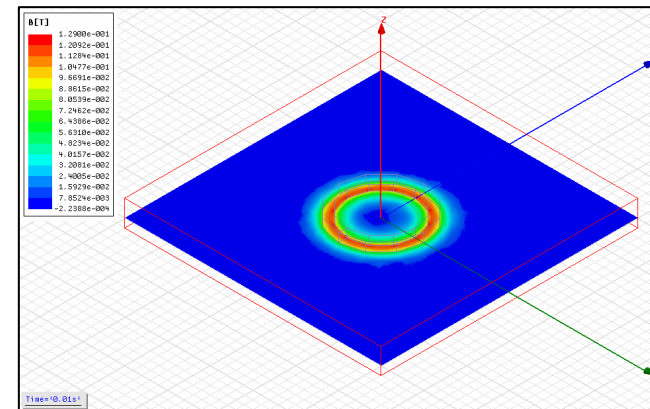
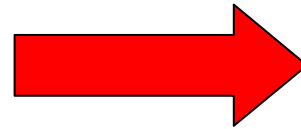
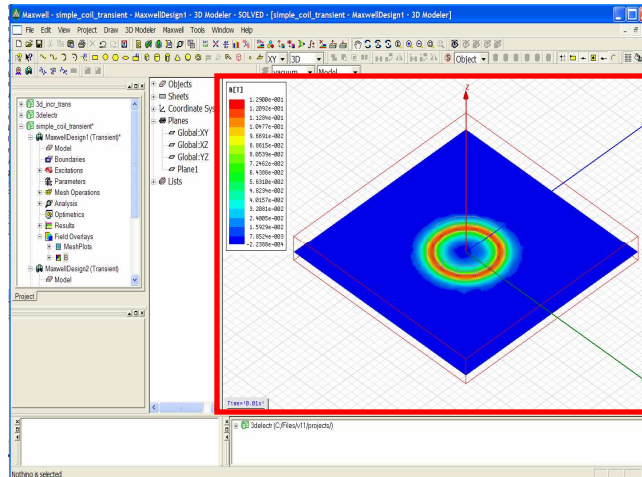
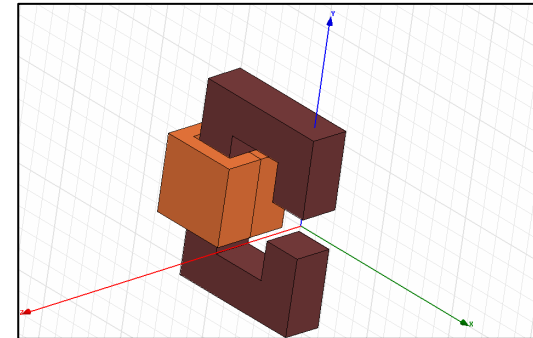
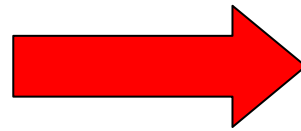
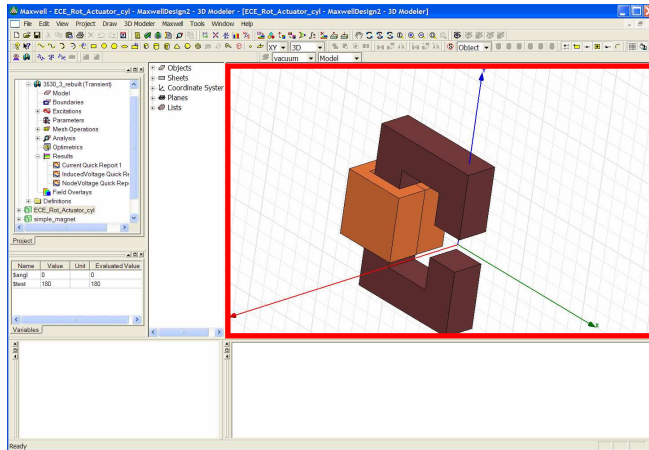
▲ To create report (plot the desired graph)

1. Select the menu item **Maxwell->Result->Create Report**
2. Enter Fields and Rectangular Plot and click the **OK** button
3. Select the following: Geometry - Polyline1; Category - Calculator Expressions; Quantity - Bz
4. Press the **Add Trace** to copy your selection to the upper part of the window and click on the **Done** button



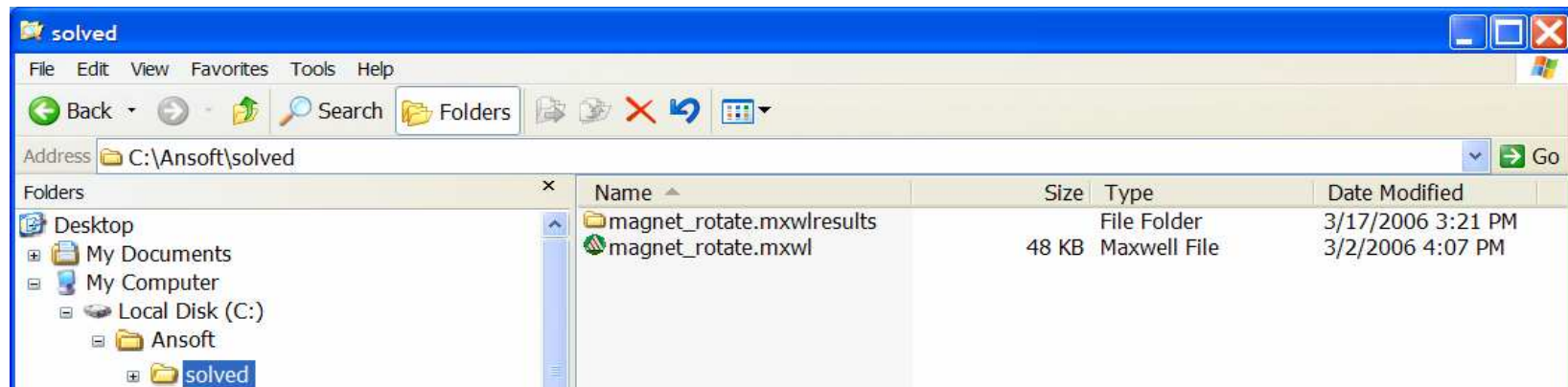
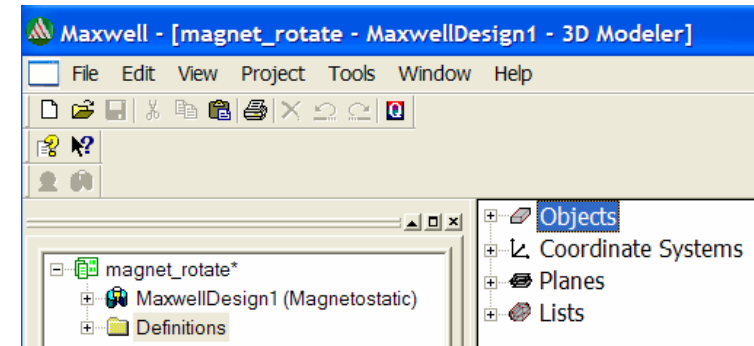


- To save drawing Window or a plot to clipboard
1. Select the menu item *Edit->Copy to Clipboard as Image*



File Structure

- Everything regarding the project is stored in an ascii file
 - File: **<project_name>.mxwl**
 - Double click from Windows Explorer will open and launch Maxwell v11
- Results and Mesh are stored in a folder named **<project_name>.mxwresults**
- Lock file: **<project_name>.lock.mxwl**
 - Created when a project is opened
- Auto Save File: **<project_name>.mxwl.auto**
 - When recovering, software only checks date
 - If an error occurred when saving the auto file, the date will be newer then the original
 - Look at file size (provided in recover dialog)

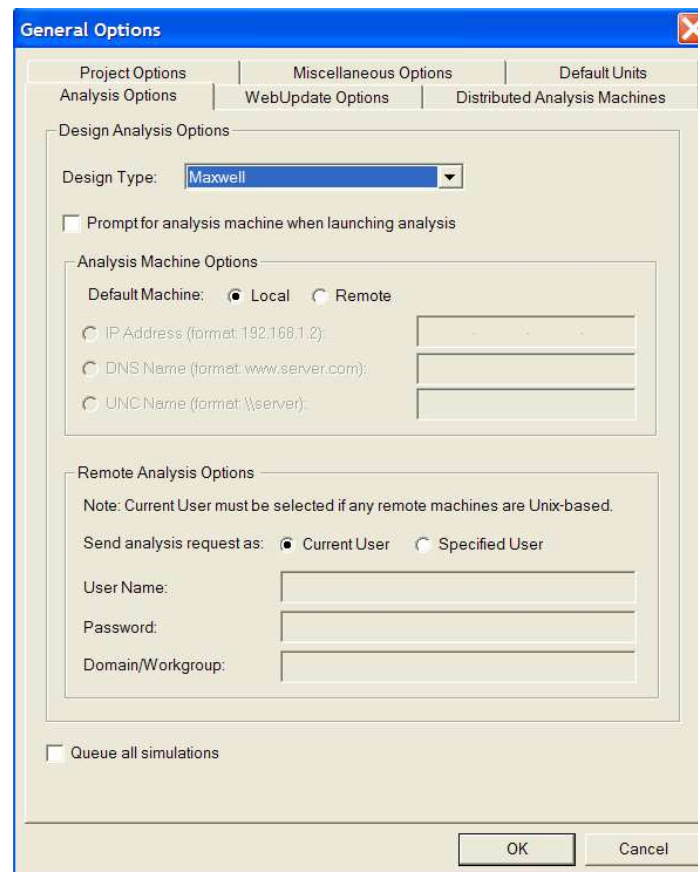
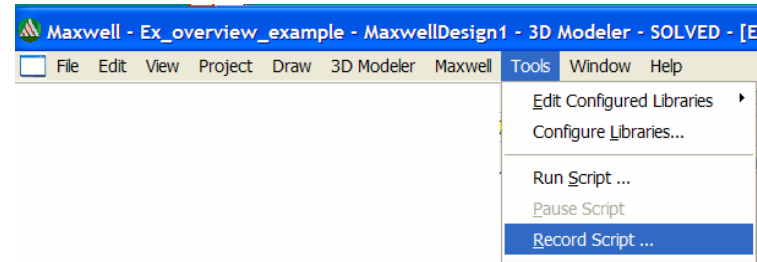


Scripts

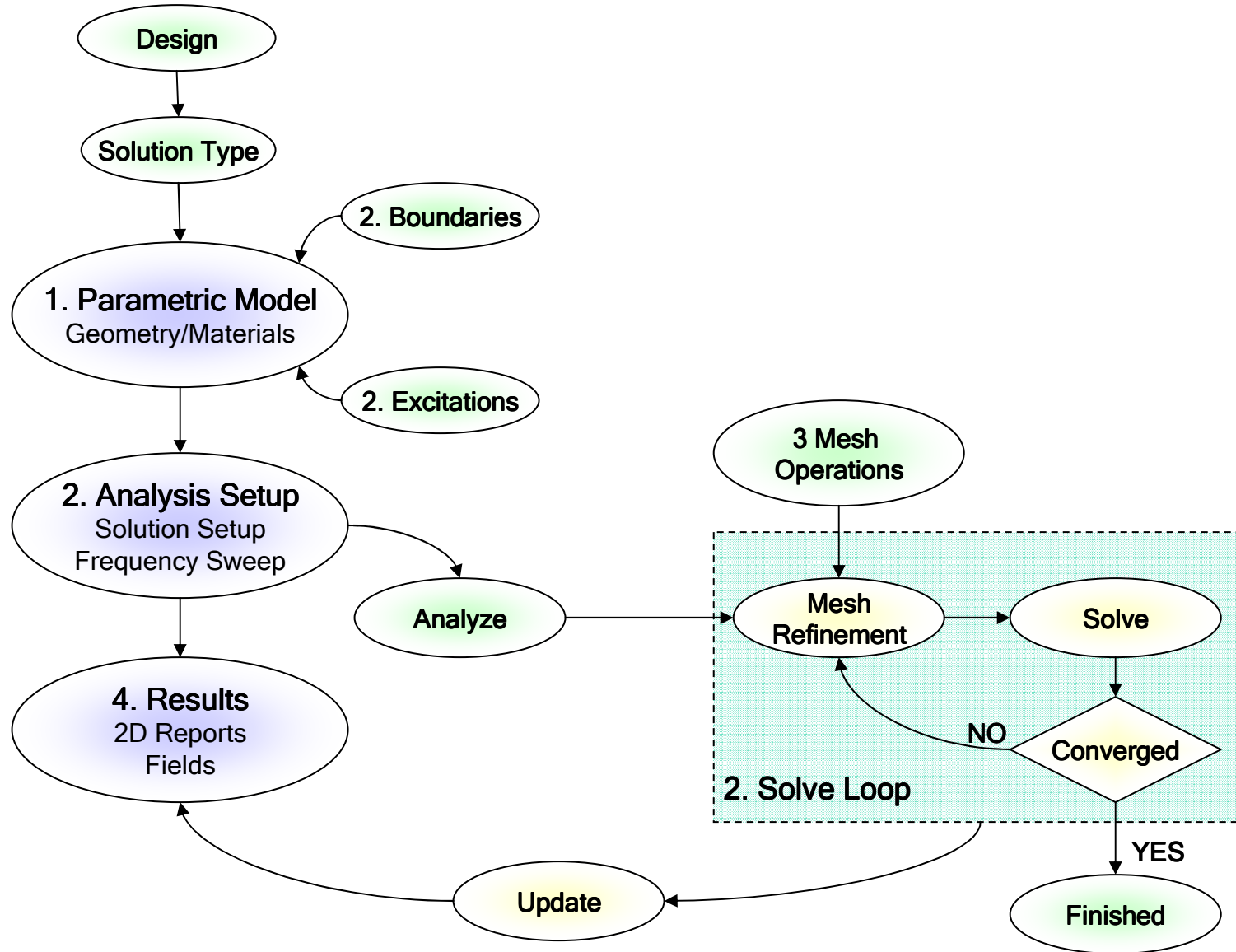
- Default Script recorded in v11
- Visual Basic Script

Remote Solve (Windows Only)

- Tools > Options > General Options > Analysis Options*



The Process

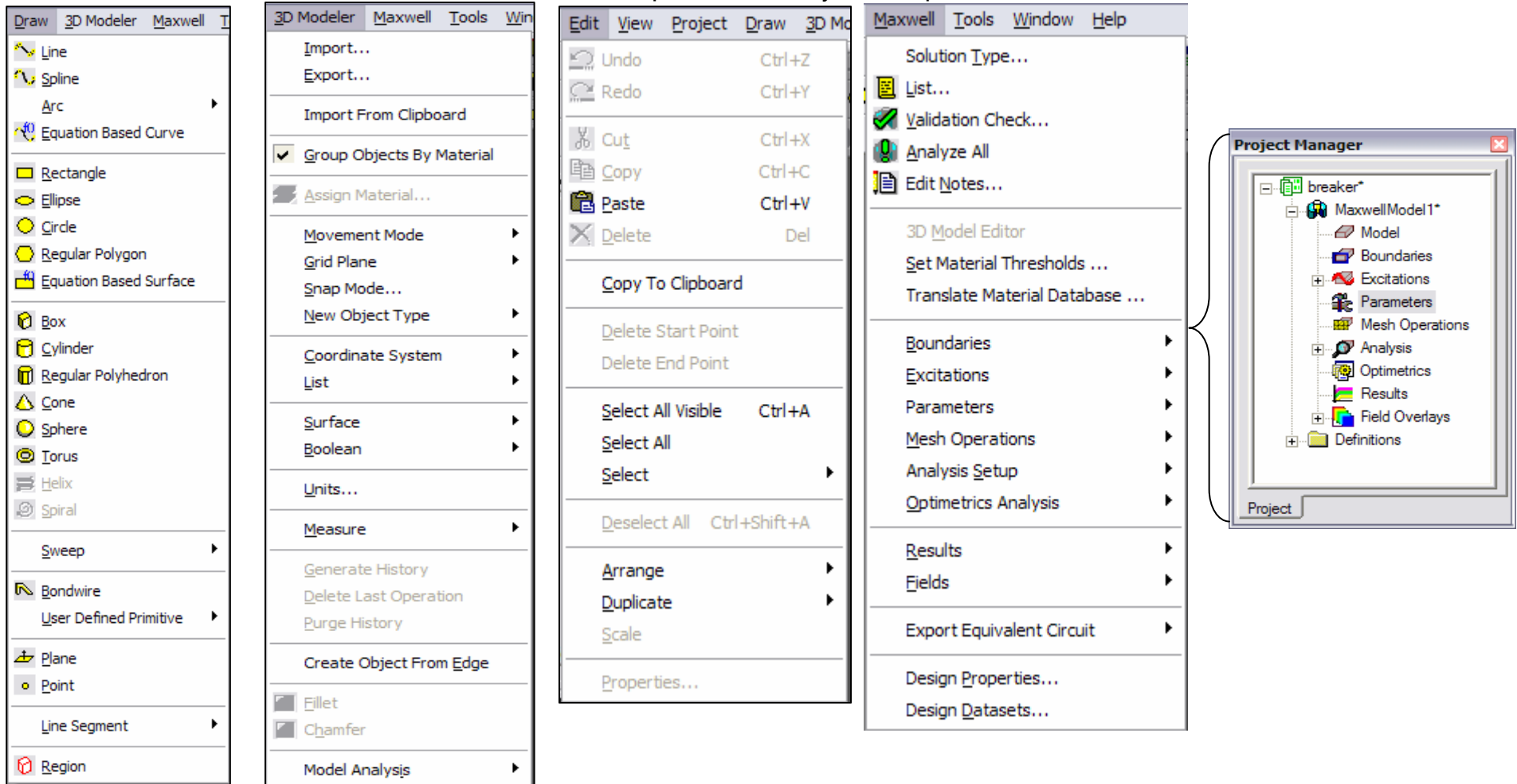


Menu Structure

- ▲ **Draw** - Primitives
- ▲ **3D Modeler** - Settings and Boolean Operations
- ▲ **Edit** - Copy/Paste, Arrange, Duplicate
- ▲ **Maxwell** - Boundaries, Excitations, Mesh Operations, Analysis Setup, Results

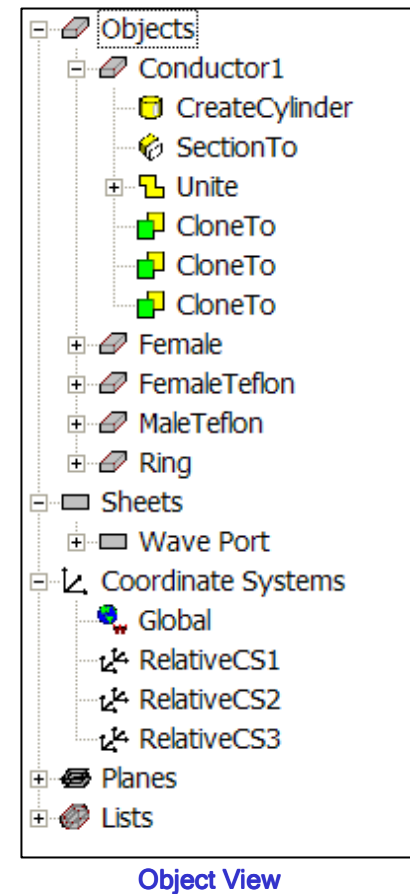
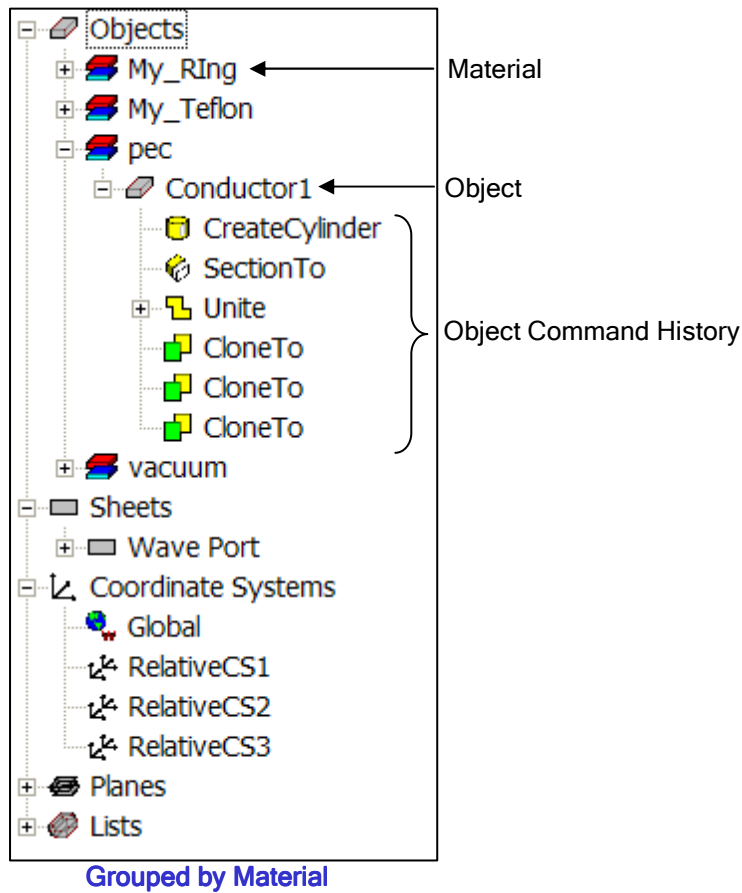
Note:

- Edit Copy/Paste is a dumb copy for only dimensions and materials, but not the history
- Edit > Duplicate clones objects include dimensions, materials and the creation history with all variables



3D Modeler - Model Tree

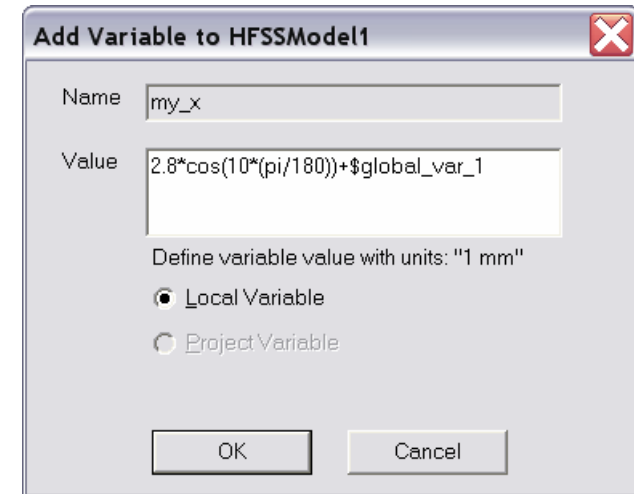
Select menu item *3D Modeler > Group by Material*



3D Modeler - Commands

Parametric Technology

- ▀ Dynamic Edits - Change Dimensions
- ▀ Add Variables
 - ▀ Project Variables (Global) or Design Variables (Local)
 - ▀ Animate Geometry
 - ▀ Include Units - Default Unit is meters
- ▀ Supports mixed Units



Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	-1, -1.6, 0	mm
XSize	2.6	mm
YSize	2.8	mm
ZSize	1	mm

Name	Value	Unit
Comma...	CreateBox	
Coordin...	Global	
Position	-1, -1.6, 0	mm
XSize	my_x	
YSize	2.8	mm
ZSize	1	mm

3D Modeler - Primitives

2D Draw Objects

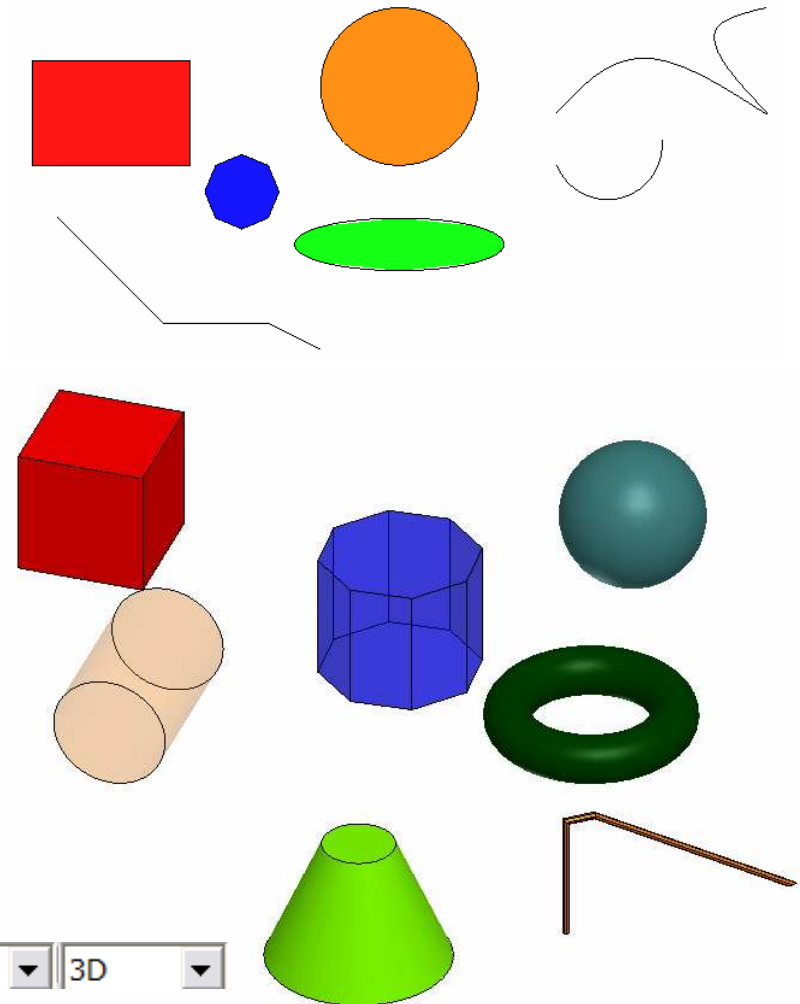
- The following 2D Draw objects are available:
 - Line, Spline, Arc, Equation Based Curve, Rectangle, Ellipse, Circle, Regular Polygon, Equation Based Surface

3D Draw Objects

- The following 3D Draw objects are available:
 - Box, Cylinder, Regular Polyhedron, Cone, Sphere, Torus, Helix, Spiral, Bond Wire

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, select the Regular Polyhedron or Regular Polygon.



Toolbar: 2D Objects

Toolbar: 3D Objects

3D Modeler - Boolean Operations/Transformations

3D Modeler > Boolean >

-  **Unite** - combine multiple primitives
 -  Unite disjoint objects (**Separate Bodies** to separate)
-  **Subtract** - remove part of a primitive from another
-  **Intersect** - keep only the parts of primitives that overlap
-  **Split** - break primitives into multiple parts along a plane (XY, YZ, XZ)
-  **Split Crossing Objects** - splits objects along a plane (XY, YZ, XZ) only where they intersect
-  **Separate Bodies** - separates objects which are united but not physically connected into individual objects



← **Toolbar: Boolean**

-  **3D Modeler > Surfaces > Move Faces** - Resize or Reposition an objects face along a normal or vector.

Edit > Arrange >

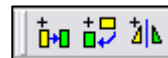
-  **Move** - Translates the structure along a vector
-  **Rotate** - Rotates the shape around a coordinate axis by an angle
-  **Mirror** - Mirrors the shape around a specified plane
-  **Offset** - Performs a uniform scale in x, y, and z.



← **Toolbar: Arrange**

Edit > Duplicate >

-  **Along Line** - 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



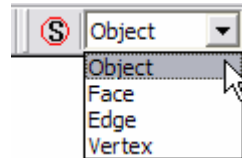
← **Toolbar: Duplicate**

-  **Edit > Scale** - Allows non-uniform scaling in the x, y, or z direction

3D Modeler - Selection

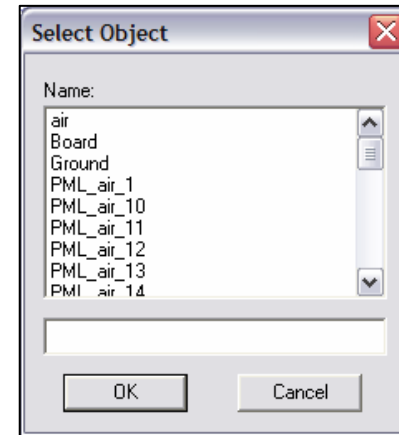
Selection Types

- ▀ Object (Default)
- ▀ Face
- ▀ Edge
- ▀ Vertex



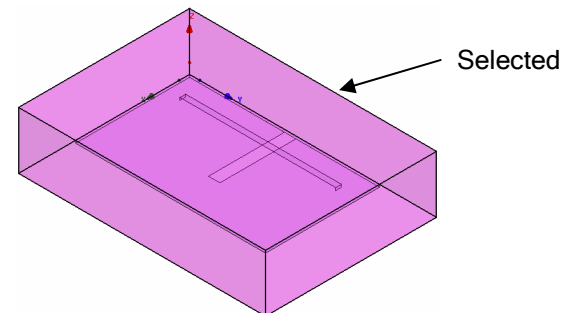
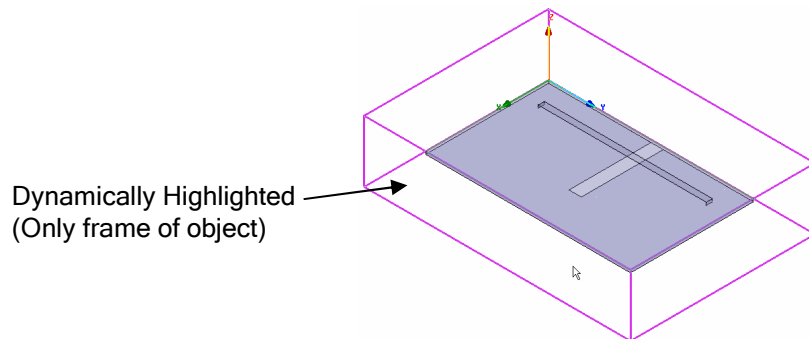
Selection Modes

- ▀ All Objects
- ▀ All Visible Object
- ▀ By Name

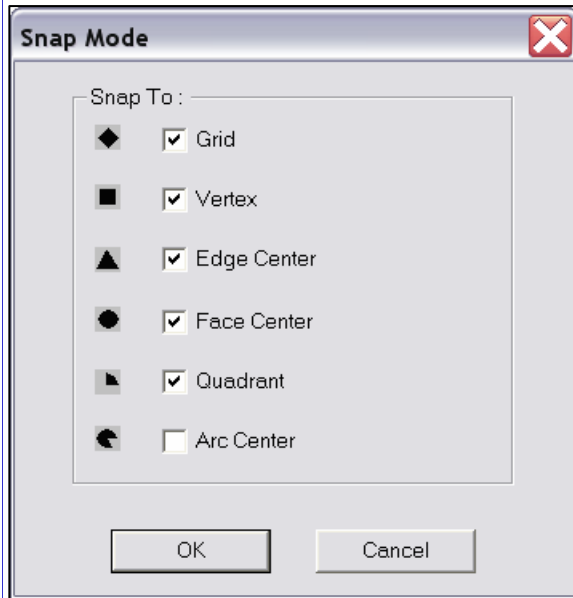


- ▀ **Highlight Selection Dynamically** - By default, moving the mouse pointer over an object will dynamically highlight the object for selection. To select the object simply click the left mouse button.

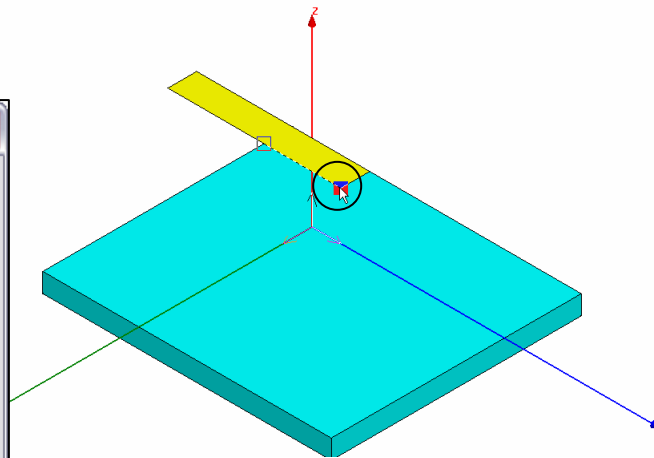
- ▀ **Multiple Object Selection** - Hold the **CTRL** key down to graphically select multiple objects
- ▀ **Next Behind** - To select an object located behind another object, select the front object, press the **b** key to get the next behind. Note: The mouse pointer must be located such that the next behind object is under the mouse pointer.
- ▀ **To Disable:** Select the menu item *Tools > Options > 3D Modeler Options*
 - ▀ From the **Display Tab**, uncheck **Highlight selection dynamically**



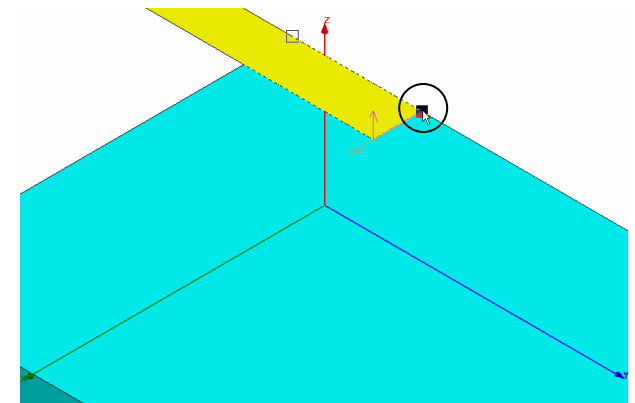
3D Modeler - Moving Around



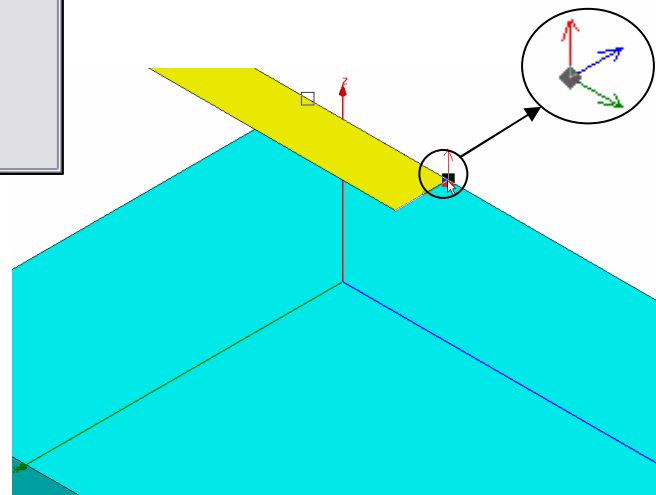
Toolbar: Snap Mode



Step 1: Start Point

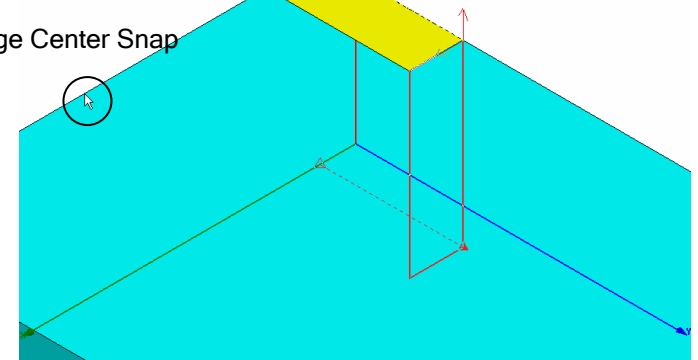


Step 2: Hold X key (to only allow motion in X-direction) and select vertex point



Step 3: CTRL+Enter Keys (sets a temporary local reference)

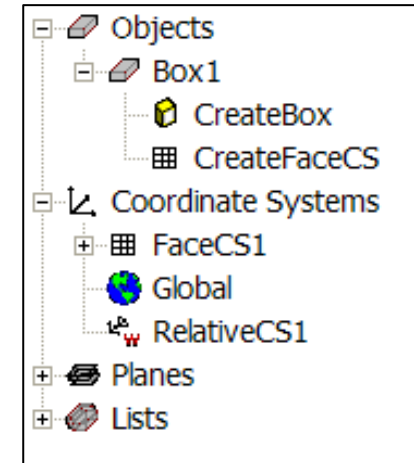
Edge Center Snap



Step 4: Hold Z key and set height

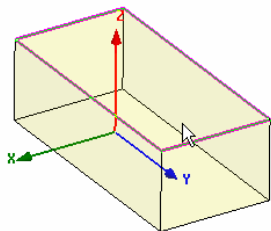
3D Modeler - Coordinate Systems

- ▲ Can be Parameterized
- ▲ **Working Coordinate System**
 - ▲ 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

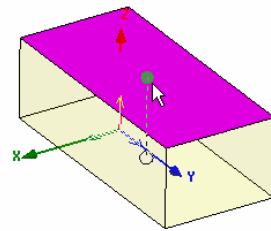


Toolbar: Coordinate System

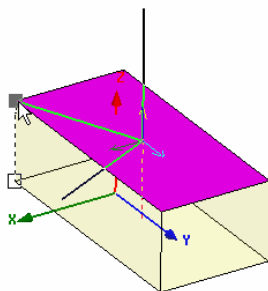
- ▲ **Face CS** (setting available to automatically switch to face coordinate system in the 3D Modeler Options)



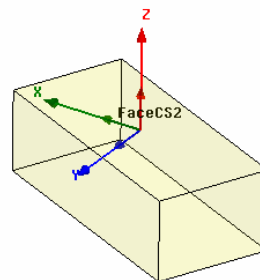
Step 1: Select Face



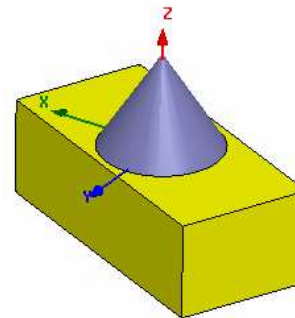
Step 2: Select Origin



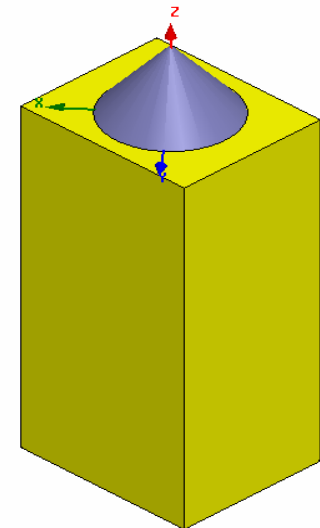
Step 3: Set X-Axis



New Working CS



Cone created with Face CS

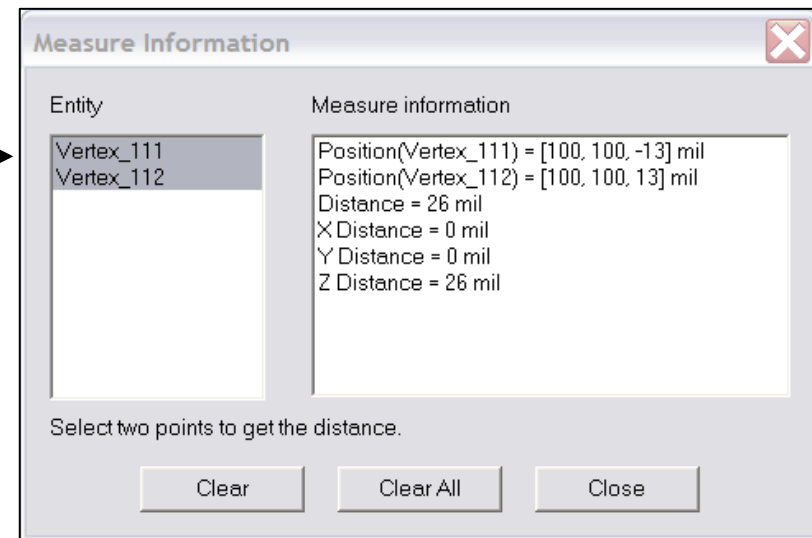


Change Box Size and Cone is automatically positioned with the top face of the box

Measure

3D Modeler > Measure >

- Position - Points and Distance
- Length - Edge Length
- Area - Surface Area
- Volume - Object Volume

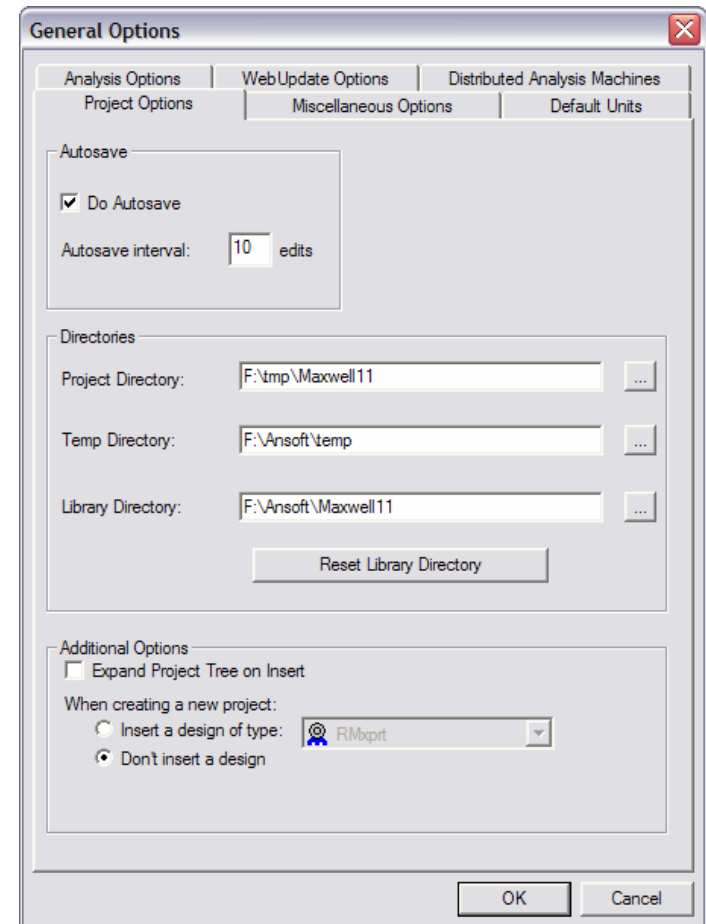
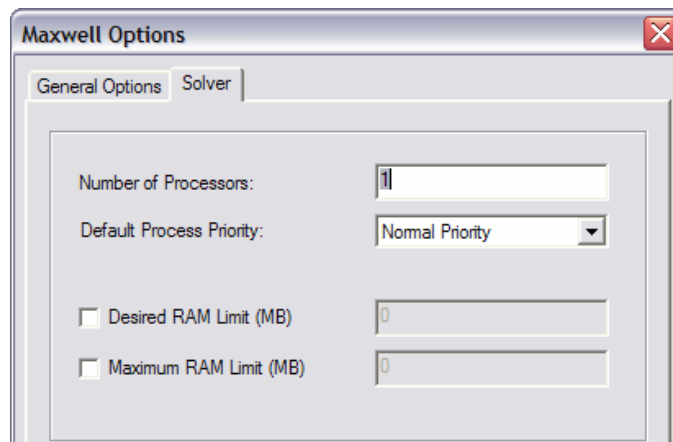


Options - General

- Tools > Options > General Options > Project Options
 - Temp Directory - Location used during solution process
 - Make sure it is at least 512MB free disk.

Options - Maxwell

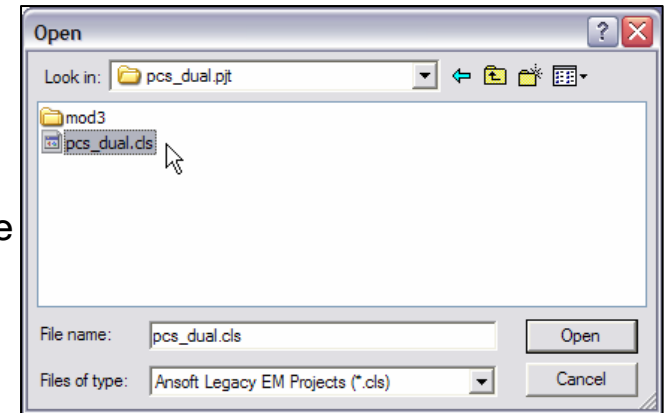
- Tools > Options > Maxwell Options > Solver
 - Set Number of Processors = 2 for 1 dual-core processor or two single-core processors. Requires additional license
 - Default Process Priority - set the simulation priority from Critical (highest) to Idle (lowest)
 - Desired RAM Limit - determines when solver will use hard disk off core (leave it unchecked for auto-detect)
 - Maximum RAM Limit - determines when swapping will occur with hard drive (leave it unchecked for auto-detect)



❖ Converting Older Maxwell Projects (pre-Maxwell v11) to Maxwell v11

❖ From Maxwell v10.0,

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



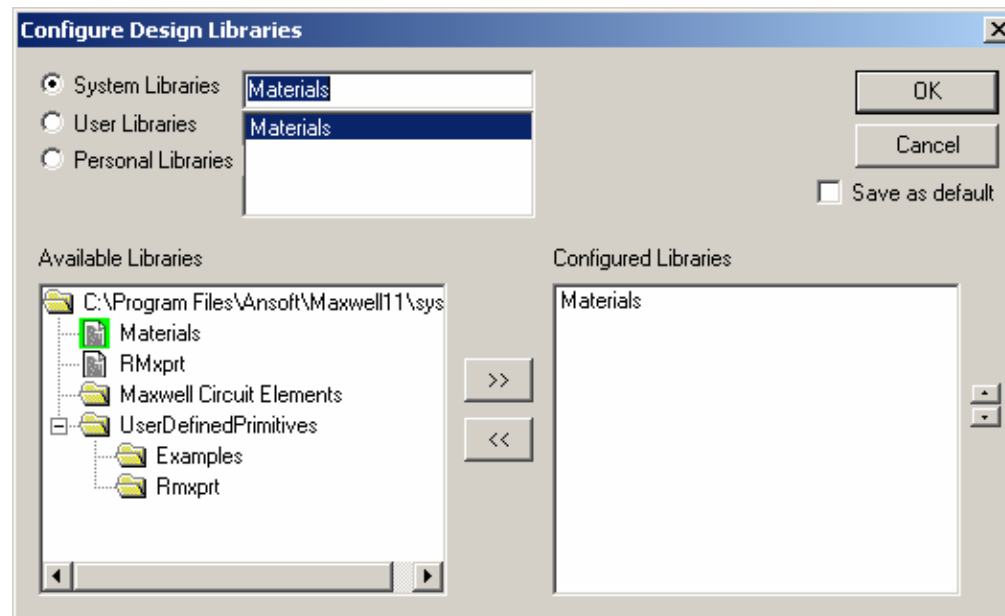
❖ What is Converted?

- ❖ Converts Entire Model: Geometry, Materials, Boundaries, Sources and Setup
- ❖ *Solutions, Optimetrics projects and Macros are not converted*



Break

- Click “Add Material ...”. The Material is only available in Project
- To add a material in the user or personal library: click on “Export Library” and save it in the desire library.
- In the main project window, click on *Tools > Configured Libraries*. Locate the library to have the material available for all the projects.



Materials Setup - Editing

View / Edit Material

Material Name: Material Coordinate System Type:

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Nonlinear	BH Curve...	
Bulk Conductivity	Simple	2000000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
Composition		Solid	

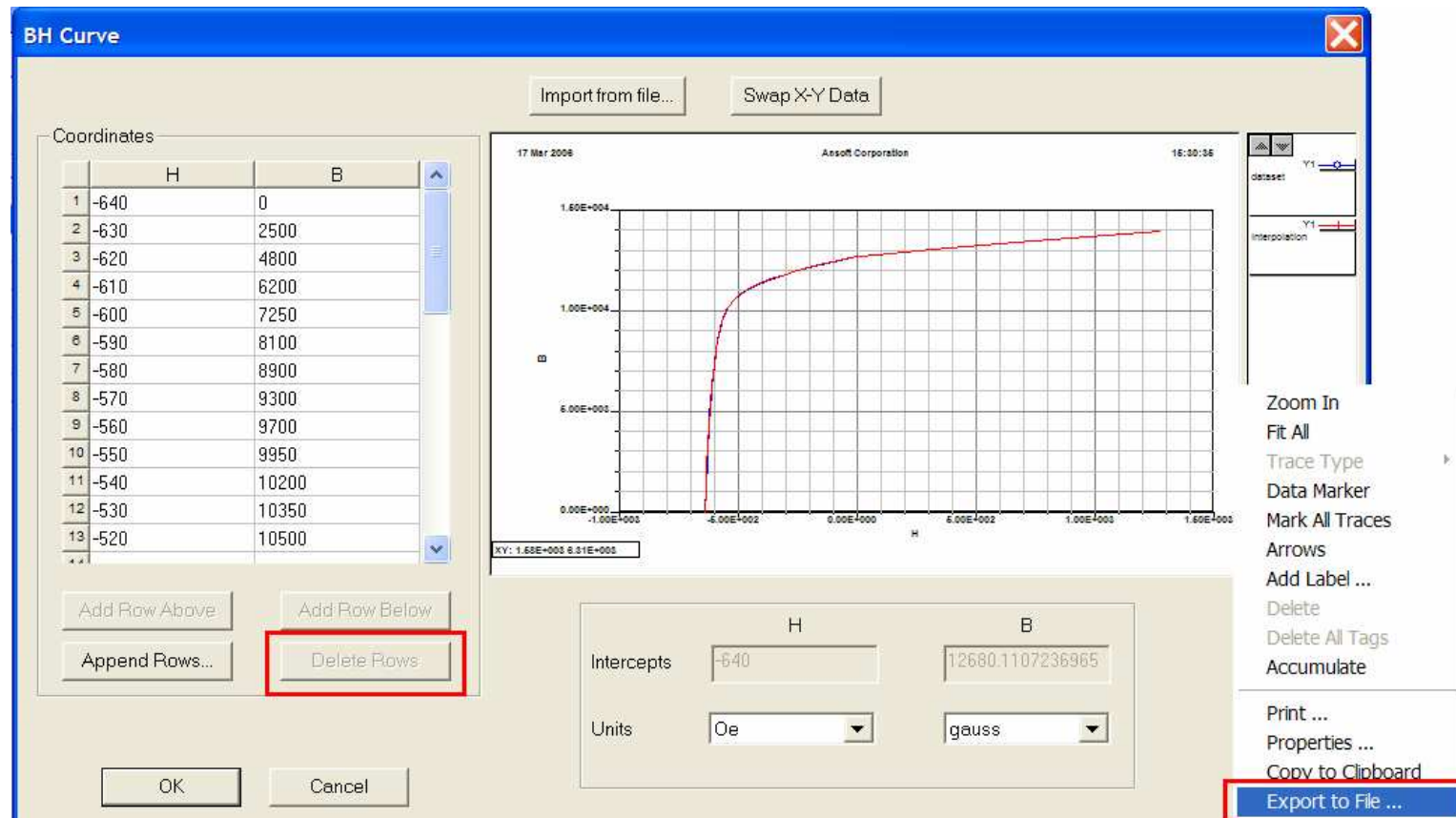
View/Edit Material for

☒ Active Design
☐ This Product
☐ All Products

Calculate Properties for:

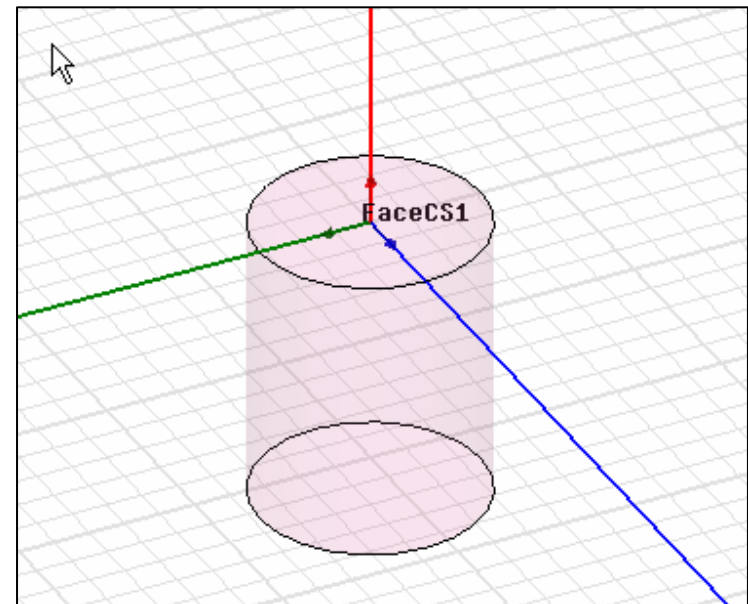
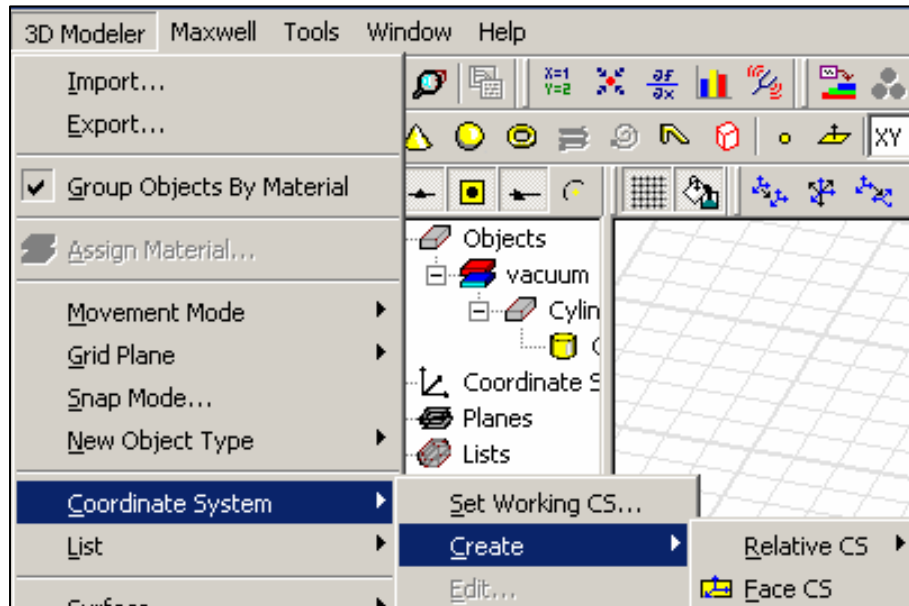
Material Setup - BH curve

- Lamination model to account for stacking factor (modifies permeability, not conductivity)
- Robust BH curve entry - can delete points Create the Object
- To export BH curve for use in future, right-mouse-click on curve and select Export to File...

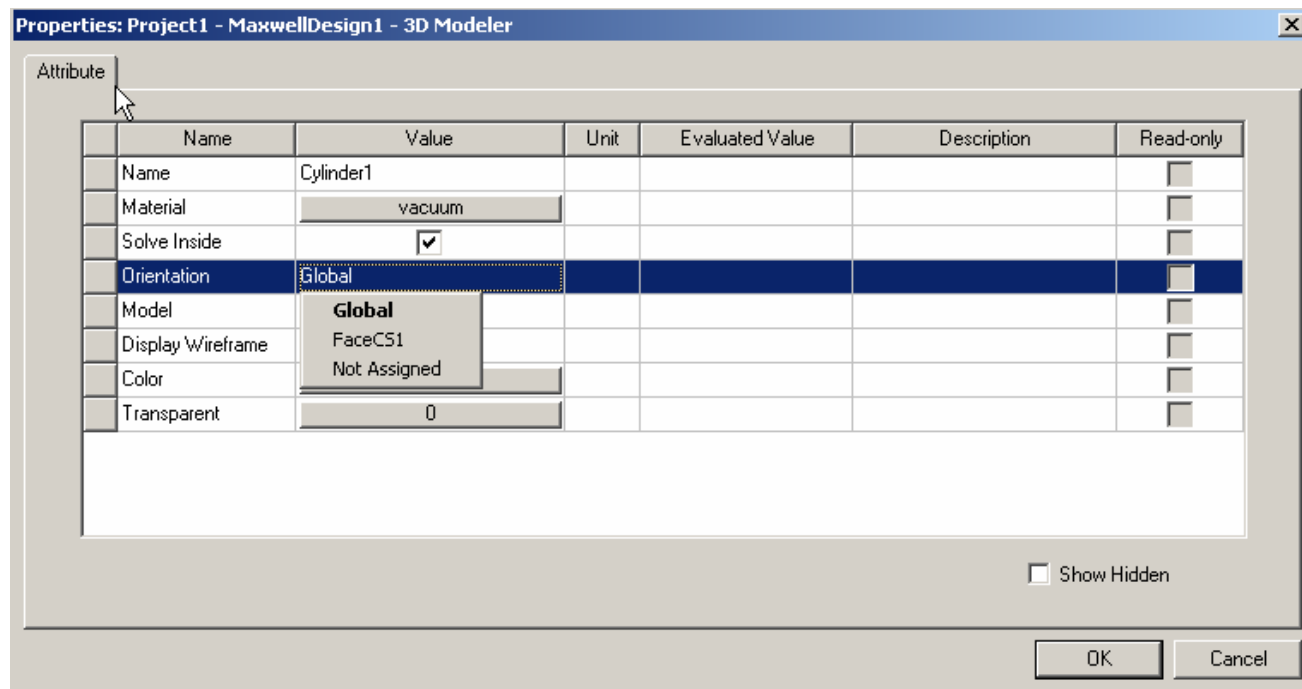


Material Setup - Permanent Magnets:

- ▲ Create the Object
- ▲ Create a Face Coordinate System (Face CS)



- Enter the Materials Library and choose the Desired Material
- Permanent Magnet Orientation refers by default to the global CS. To change the reference CS for orientation, open the property window and change the CS reference



- ▲ In the Material Window, you can assign orientation
- ▲ The Orientation can be described in Cartesian, Cylindrical, Spherical

View / Edit Material

Material Name: NdFe35 Material Coordinate System Type: Cartesian

Properties of the Material

Name	Type	Value	Units
Relative Permeability	Simple	1.0997785406	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-890000	A_per_meter
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Composition		Solid	

View/Edit Material for:

☒ Active Design
☐ This Product
☐ All Products

Validate Material

Calculate Properties for: [Dropdown]

Reset OK Cancel

Material setup - Anisotropic Material Properties

- ϵ_1 , μ_1 , and σ_1 are tensors in the X direction.
- ϵ_2 , μ_2 , and σ_2 are tensors in the Y direction.
- ϵ_3 , μ_3 , and σ_3 are tensors in the Z direction.
- Anisotropic permeability definitions can be either **LINEAR** or **NONLINEAR**.

$$\begin{array}{ccc}
 \begin{array}{c} \text{Electrostatic \& Eddy Current} \\ \left[\epsilon \right] = \begin{bmatrix} \epsilon_1 & 0 & 0 \\ 0 & \epsilon_2 & 0 \\ 0 & 0 & \epsilon_3 \end{bmatrix}, \end{array} & \begin{array}{c} \text{Magnetostatic, Eddy Current, \& Transient} \\ \left[\mu \right] = \begin{bmatrix} \mu_1 & 0 & 0 \\ 0 & \mu_2 & 0 \\ 0 & 0 & \mu_3 \end{bmatrix}, \end{array} & \begin{array}{c} \text{Eddy Current, \& Transient} \\ \left[\sigma \right] = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix} \end{array}
 \end{array}$$

Electric Boundary Conditions

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: Natural boundaries – The normal component of D changes by the amount of surface charge density. No special conditions are imposed. Neumann boundaries – E is tangential to the boundary. Flux cannot cross a Neumann boundary.	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: Even Symmetry (Flux Tangential) – E is tangential to the boundary; its normal components are zero. Odd Symmetry (Flux Normal) – E is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Matching (Master and Slave)	The E -field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field on the master boundary.	Planes of symmetry in periodic structures where E is oblique to the boundary.
Note: No Balloon Boundary available		

Electric Sources

Source	Type of Excitation
Floating Conductor	Used to model conductors at unknown potentials.
Voltage	The DC voltage on a surface or object.
Charge	The total charge on a surface or object (either a conductor or dielectric).
Charge Density	The charge density in an object.
Note: DC conduction solution (voltage) can also be used as input for an electric field solution	

Magnetostatic Boundaries

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: - Natural boundaries – $\mathbf{H}$ is continuous across the boundary. - Neumann boundaries – $\mathbf{H}$ is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Magnetic Field (H-Field)	The tangential components of $\mathbf{H}$ are set to pre-defined values. Flux is perpendicular.	External magnetic fields.
Symmetry	Field behaves as follows: - Odd Symmetry (Flux Tangential) – $\mathbf{H}$ is tangential to the boundary; its normal components are zero. - Even Symmetry (Flux Normal) – $\mathbf{H}$ is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Matching (Master and Slave)	The $\mathbf{H}$ -field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the $\mathbf{H}$ -field on the master boundary.	Planes of symmetry in periodic structures where $\mathbf{H}$ is oblique to the boundary.
Note: No Balloon Boundary available		

Magnetostatic Sources

Source	Type of Excitation
Voltage	The DC voltage on a surface or object.
Voltage Drop	The voltage drop across a sheet object.
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Current Density Terminal	The terminal source current.
Notes: - Current and H-field are RMS (or DC) values - Current sources require one or more 2D sheet objects (terminal) - Permanent Magnets can also be a source	

Eddy-current boundaries

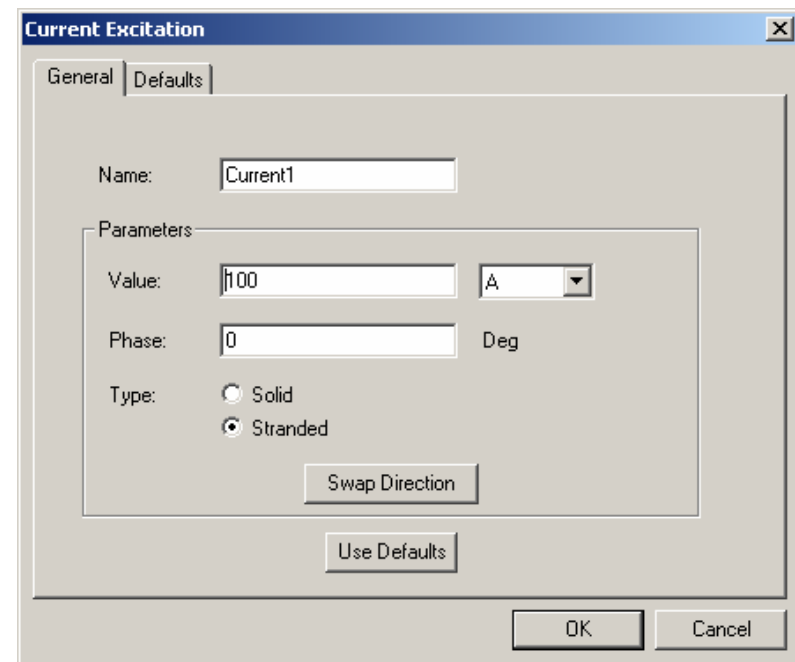
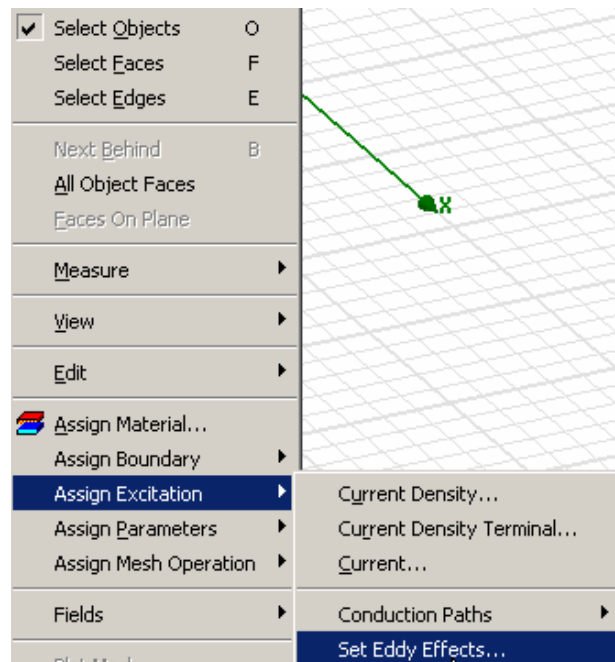
Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: - Natural boundaries – $\mathbf{H}$ is continuous across the boundary. - Neumann boundaries – $\mathbf{H}$ is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Magnetic Field (H-Field)	The tangential components of $\mathbf{H}$ are set to pre-defined values. Flux is perpendicular.	External AC magnetic fields.
Symmetry	Field behaves as follows: - Odd Symmetry (Flux Tangential) – $\mathbf{H}$ is tangential to the boundary; its normal components are zero. - Even Symmetry (Flux Normal) – $\mathbf{H}$ is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Impedance	Includes the effect of induced currents beyond the boundary surface.	Conductors with very small skin depths.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Radiation	No restrictions on the field behavior.	Unbounded eddy currents.
Matching (Master and Slave)	The $\mathbf{H}$ -field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the $\mathbf{H}$ -field on the master boundary.	Planes of symmetry in periodic structures where $\mathbf{H}$ is oblique to the boundary.
Note: Radiation (Balloon) Boundary available		

Eddy-current Sources

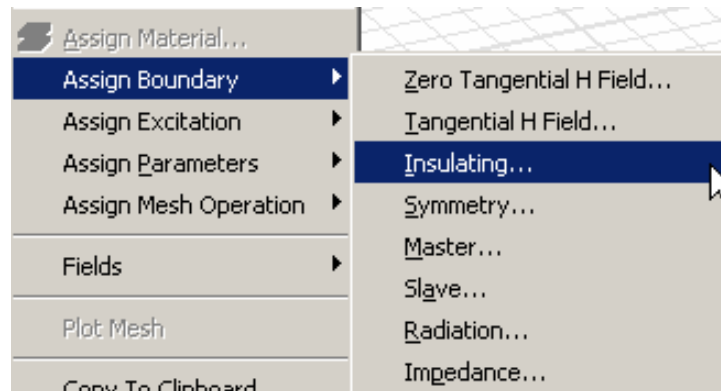
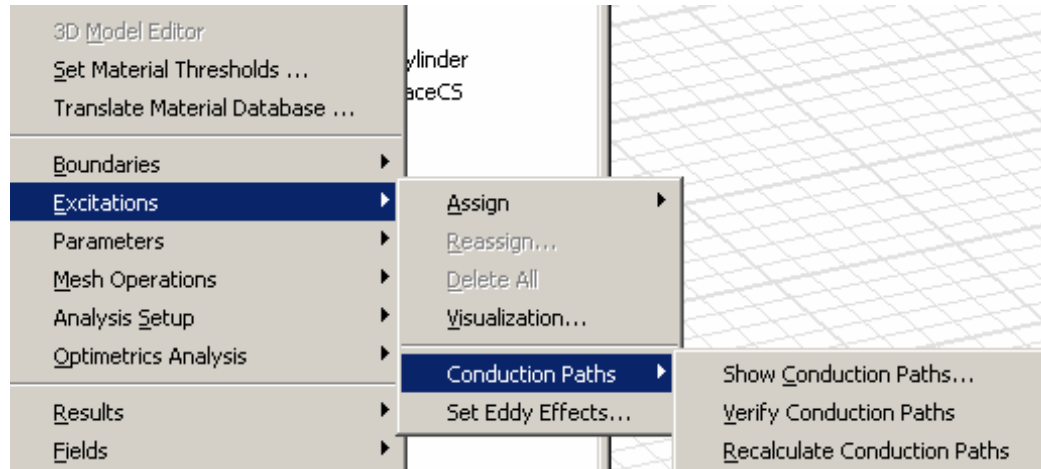
Source	Type of Excitation
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Current Density Terminal	The current density terminals in a conductor.
Notes:  Current and H-field are Peak (Not RMS) values  Current sources requires a 2D sheet objects (terminal) which lie in the cross-section of the conductor	

Setup Boundaries/Sources

- Select Object (for solid sources and current terminals)
- Select Face (for insulating boundaries)
- Select item **Maxwell > Excitations > Assign** for sources
- Select item **Maxwell > Boundaries > Assign** for boundary conditions
- Turn on eddy or displacement current calculation in eddy current solver for materials of interest: **Maxwell > Excitations > Set Eddy Effects ...**
- Stranded coil option available in Magnetostatic and Eddy Current solvers

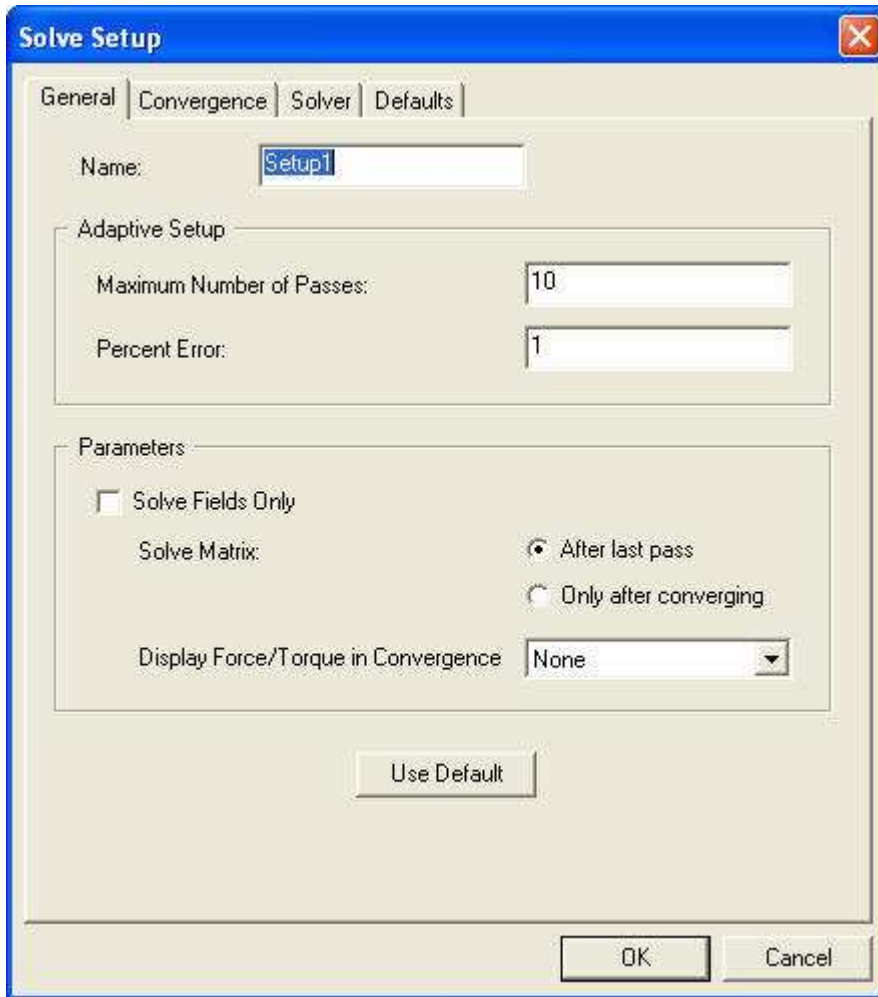


- ▲ View conduction paths by selecting: **Maxwell > Excitations > Conduction Paths > Show Conduction Paths**
- ▲ Assign insulating boundaries on faces between touching conductors if necessary



▲ Magnetostatic and Electric Solution Setup

- ▲ Start the menu of solution setup by: **Maxwell > Analysis Setup > Add Solution Setup ...**
- ▲ For Magnetostatic solver on Solver tab, suggest setting nonlinear residual = 0.001. On default tab choose Save Defaults to set this value for all future projects.



Solve Setup

General | Convergence | Solver | Defaults

Name:

Adaptive Setup

Maximum Number of Passes:

Percent Error:

Parameters

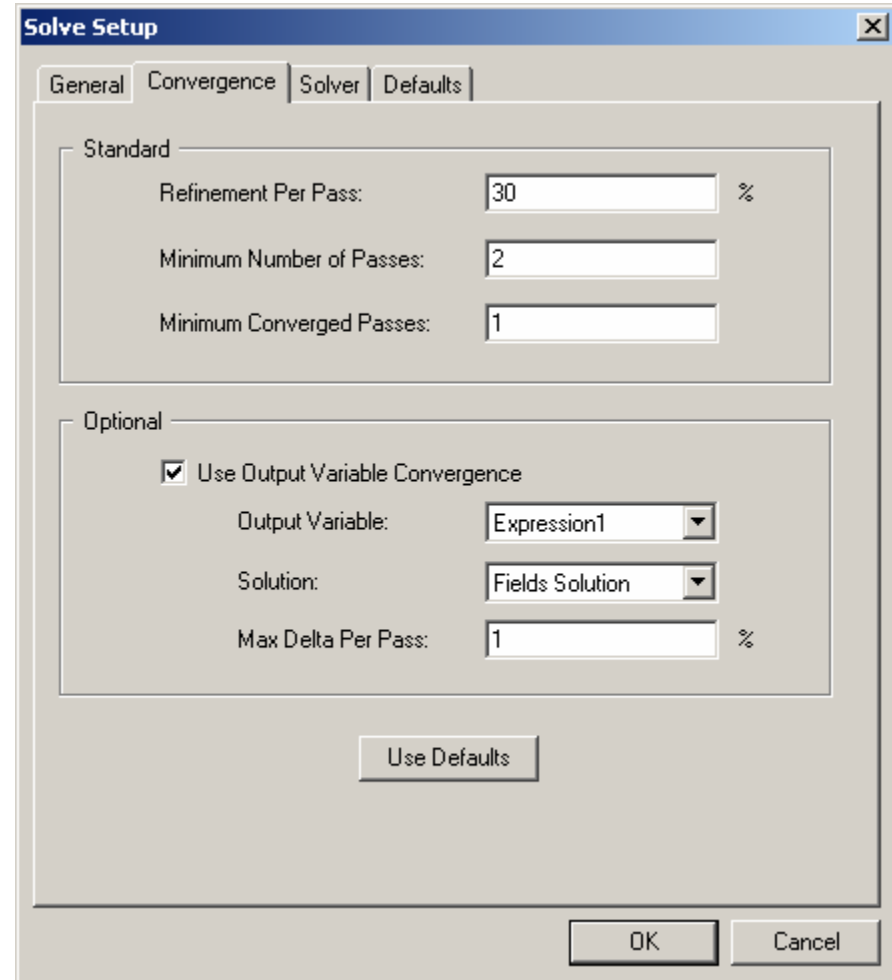
☐ Solve Fields Only

Solve Matrix: ☒ After last pass ☐ Only after converging

Display Force/Torque in Convergence:

Use Default

OK Cancel



Solve Setup

General | Convergence | Solver | Defaults

Standard

Refinement Per Pass: %

Minimum Number of Passes:

Minimum Converged Passes:

Optional

☒ Use Output Variable Convergence

Output Variable:

Solution:

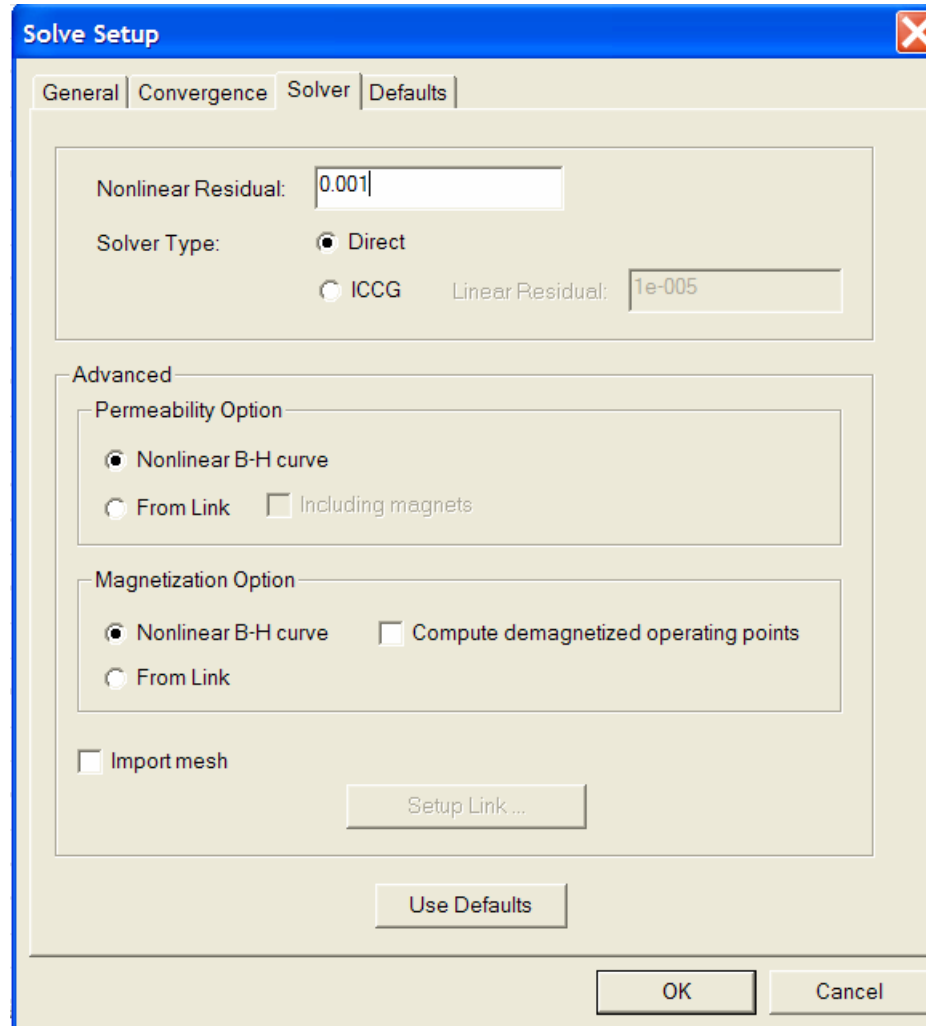
Max Delta Per Pass: %

Use Defaults

OK Cancel

Magnetostatic Setup

- For Magnetostatic solver on Solver tab, suggest setting nonlinear residual = 0.001.
- On default tab choose Save Defaults to set this value for all future projects.



Eddy Current Solution Setup

The screenshot shows the "Solve Setup" dialog box with the "Frequency Sweep" tab selected. The "Sweep Setup" section contains the following settings:

- Type: Linear Step
- Start: 10 Hz
- Stop: 1000 Hz
- Step Size: 10 Hz
- ☒ Save Fields (All Frequencies)

To the right of these settings are five buttons: "Add to List >>", "Replace List >>", "Add Single Point", "Delete Selection", and "Clear All". Below these buttons is a button labeled "Undo Last Change". On the far right is a large white rectangular area representing a frequency list, which currently contains no entries. At the top of this area are two column headers: "Frequency" and "Save Fields". At the bottom of the dialog are "OK" and "Cancel" buttons.

Transient Solution Setup

Solve Setup

General | Save Fields | **Advanced** | Solver | Output Variables | Defaults

Sweep Setup

Type: **Linear Step** ▼

Start: s ▼

Stop: s ▼

Step Size: s ▼

Add to List >>

Replace List >>

Add Single Point

Delete Selection

Clear All

Undo Last Change

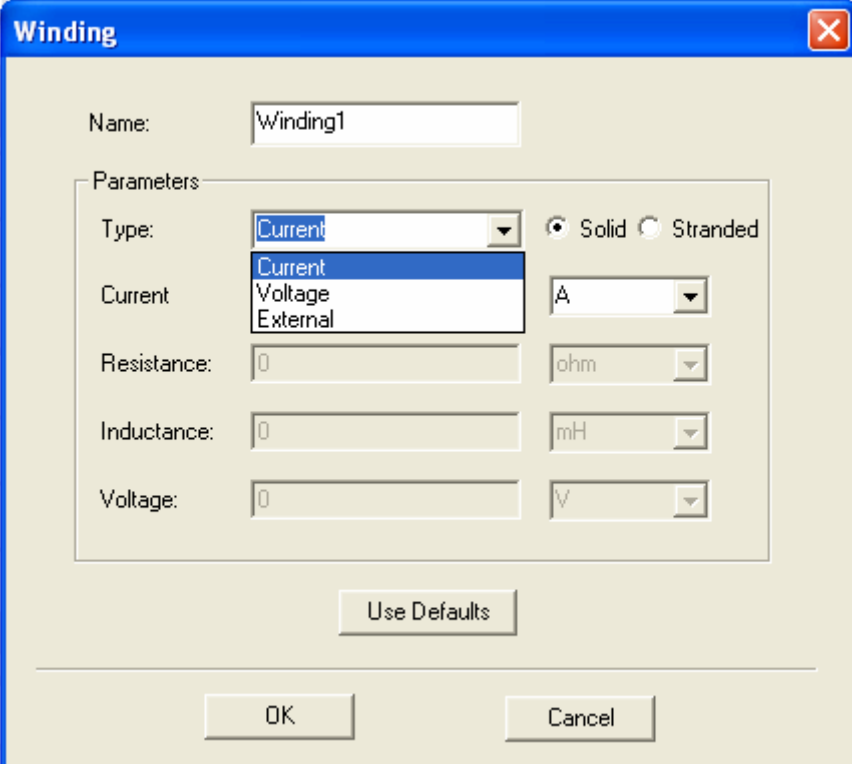
Time

Please note the stop time defined in the General Page would be automatically included.

OK Cancel

Transient Sources

- ▶ **Coil Terminal** - used to define one or more model windings
- ▶ **Winding With Current** - stranded and solid conductor
- ▶ **Winding With Voltage** - stranded and solid conductor
- ▶ **Winding With External Circuit Connection** - stranded and solid conductor
- ▶ In addition, **permanent magnets** serve as sources of magnetic fields



The image shows a 'Winding' dialog box from the Maxwell 3D software. The dialog has a title bar with a close button. Inside, there is a 'Name' field containing 'Winding1'. Below this is a 'Parameters' section. The 'Type' dropdown is set to 'Current', with a list showing 'Current', 'Voltage', and 'External'. To the right of the 'Type' dropdown are radio buttons for 'Solid' (selected) and 'Stranded'. Below the 'Type' dropdown is a 'Current' field with a unit dropdown set to 'A'. Below that is a 'Resistance' field with a unit dropdown set to 'ohm'. Below that is an 'Inductance' field with a unit dropdown set to 'mH'. Below that is a 'Voltage' field with a unit dropdown set to 'V'. At the bottom of the dialog are 'Use Defaults', 'OK', and 'Cancel' buttons.

Name: Winding1

Parameters

Type: Current (Solid selected)

Current: A

Resistance: 0 ohm

Inductance: 0 mH

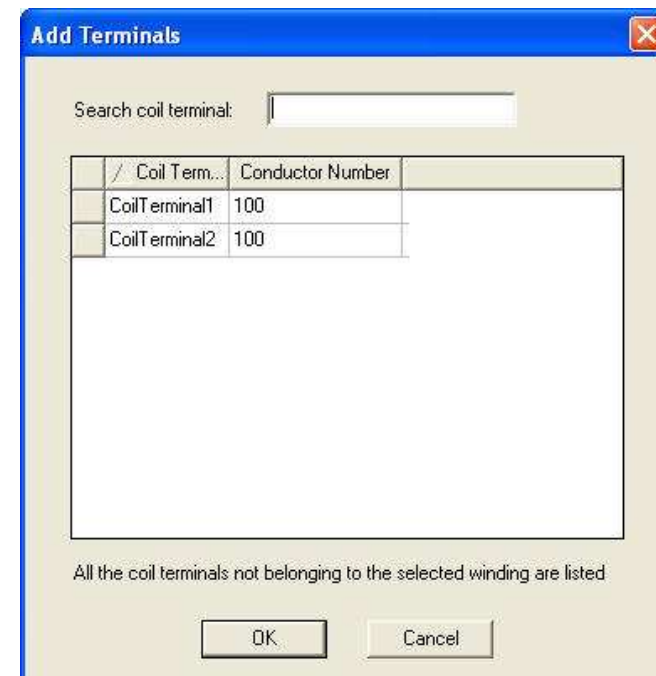
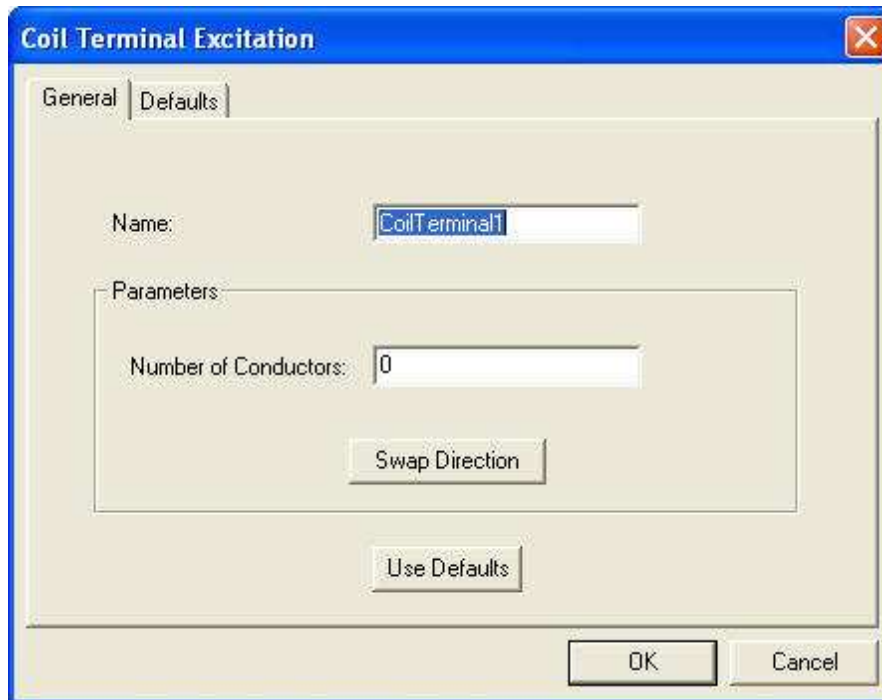
Voltage: 0 V

Use Defaults

OK Cancel

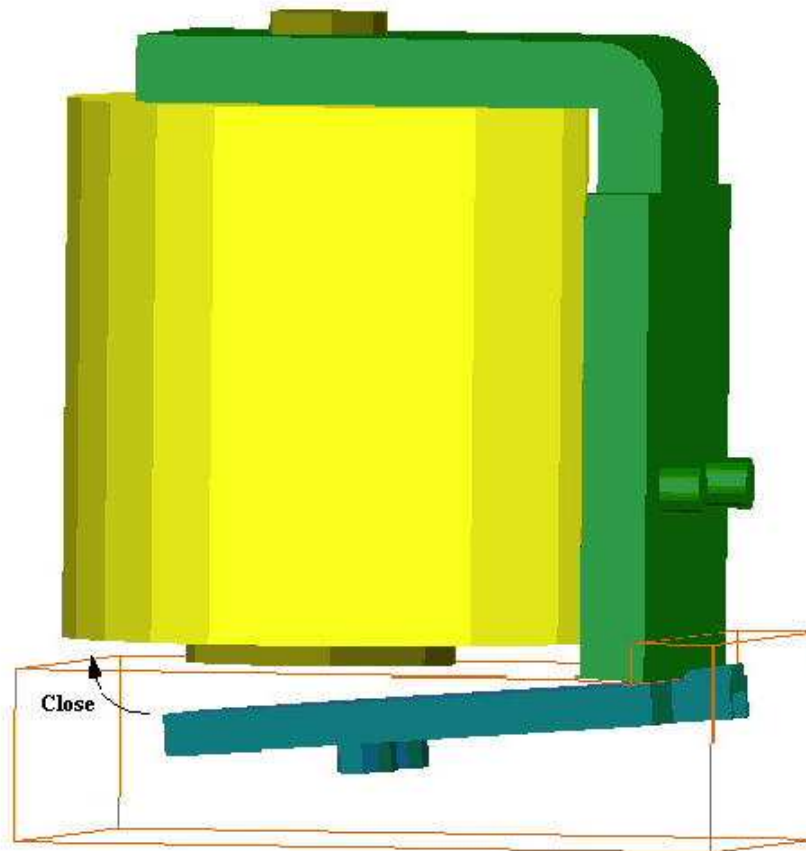
To define a winding for a Transient Source

1. Define the terminal(s): **Maxwell > Excitations > Assign > Coil Terminal**
 1. Select the planar surface(s) as appropriate for the model
 2. Assign the coil terminal excitation by specifying the orientation of the current with respect to the terminal (current in or current out, graphically represented by an arrow).
 3. Specify the *actual* number of conductors (turns) intersecting *the chosen* planar surface (coil terminal), regardless of the symmetry (if any) of the problem.
2. Add a new winding: **Maxwell > Excitations > Add Winding**
3. Specify the coil terminal(s) that belong to the winding: Right-click the winding in the property window and select **Add Terminals**



Transient Motion

- ▶ Translational motion (motion along a user specified, linear direction).
- ▶ Rotational motion (non-cylindrical such as the pivoting rotation around an axis encountered in the armature of a relay).
- ▶ Rotational motion (cylindrical, such as the type of rotation encountered in an electric machine type of application).



Ex: Pivoting Rotation

Band

-  Regardless of the particular type of motion involved, all types of motion applications require a band object that must contain the moving part(s).
-  If there are multiple moving objects, all of the moving parts must be included in one all-inclusive object – this is because they all must be moving as one rigid body, with a single force acting on the assembly.
-  Regardless of the type of motion, the user must create a mesh density capable of capturing the physical effects characteristic for the specific application, such as field gradients, skin and proximity effects, etc.

Translational and non-cylindrical rotation types of motion:

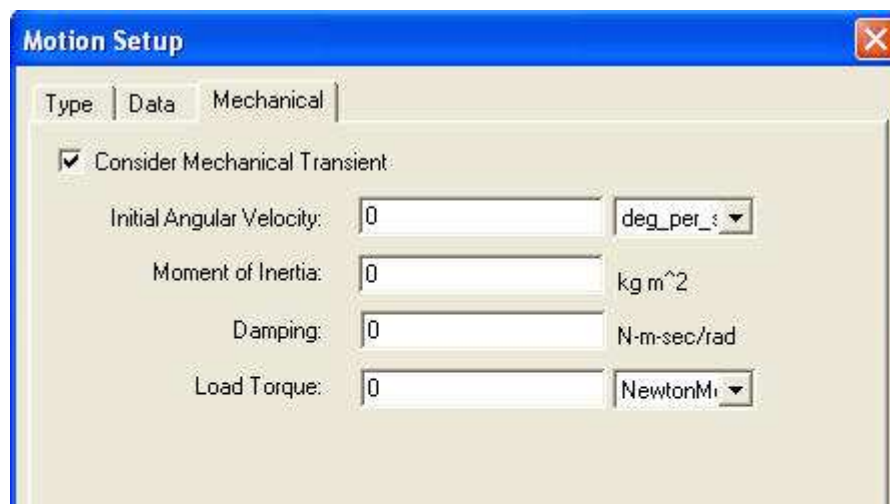
-  The band object can touch the symmetry plane if any exists.
-  The moving object cannot touch a stationary object during motion (the gap between the moving object and the band can never become zero during analysis).
-  The band cannot have true surface faces; all faces must be segmented (for example created with the regular polyhedron primitive).
-  The band object cannot separate the stationary part into unconnected sub-regions.
-  Hollow objects cannot be used for band objects.

For the cylindrical type of rotational motion applications:

-  Always use a faceted (regular polyhedron) type of cylindrical object or a wedge object if symmetry is used. The angular aperture of each facet depends on the problem; however, an opening of 2-3 degrees per facet is usually sufficient.
-  Hollow cylinders cannot be used for band objects.
-  The band object can separate the stationary part into unconnected sub-regions (a rotor sandwiched between two stators is allowed).
-  Non-cylindrical band objects are allowed.

Motion Setup

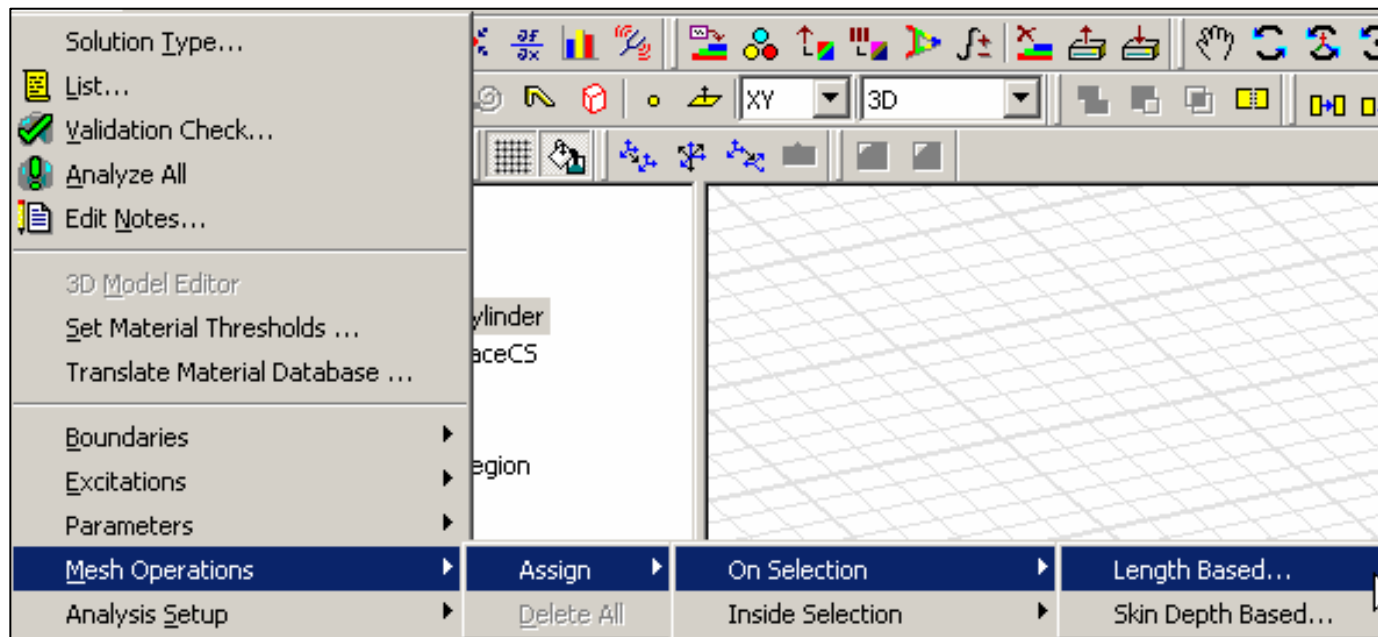
 To assign a band: **Maxwell > Model > Motion Setup > Assign Band**



Manual Mesh Refinement

To assign Mesh operations to Objects:

1. Select the Menu item: **Maxwell > Mesh Operations > Assign**



- ▲ Element length based refinement: **Maxwell > Mesh Operations > Assign > On Selection > Length Based**
- ▲ Skin Depth based refinement: **Maxwell > Mesh Operations > Assign > On Selection > Skin Depth Based**
 - ▲ Note: Adaptive Meshing will not refine these elements in this case

Element Length Based Refinement

Name: Length2

Length of Elements

Restrict Length of Elements ☒

Maximum Length of Elements:

1.2 mm

Number of Elements

Restrict the Number of Elements ☒

Maximum Number of Elements:

1000

OK Cancel

Skin Depth Based Refinement

Name: SkinDepth1

Skin Depth

Skin Depth: Calculate Skin Depth...

1 mm

Number of Layers of Elements: 2

Surface Triangle Length:

1.2 mm

Number of Elements

Restrict the Number of Surface Elements ☒

Maximum Number of Surface Elements: 1000

OK Cancel

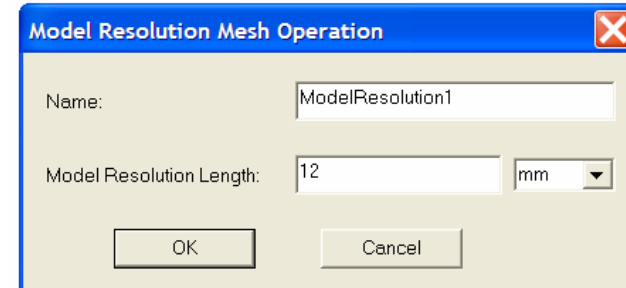
Manual Mesh Refinement

To de-feature the mesh:

1. Specify the maximum dimension to be used in the mesh under: **Mesh Operations > Assign > On Selection > Model Resolution**
2. Improves mesh quality and reduces mesh size for complex models
3. Increases the mesh speed
4. Reduces the load on the solver with reductions in mesh size

For true surfaces, perform faceting control on a face-by-face basis select: **Mesh operation > Assign > Surface approximation** and specify one or more settings:

- Maximum surface deviation (length) which is the difference between the shortest and longest edge of a triangle on the true surface. This encourages equilateral triangles on the surface.
- Maximum Surface Normal Deviation (degrees) which allows you to limit the maximum angle between normal vectors to adjacent tetrahedrons which break up a true surface. This can force more tetrahedron to break up a curved true surface like a cylinder.
- Maximum Aspect Ratio is the ratio between the largest and shortest edges of triangles on a given face. A ratio = 1 would attempt to create equilateral triangles.

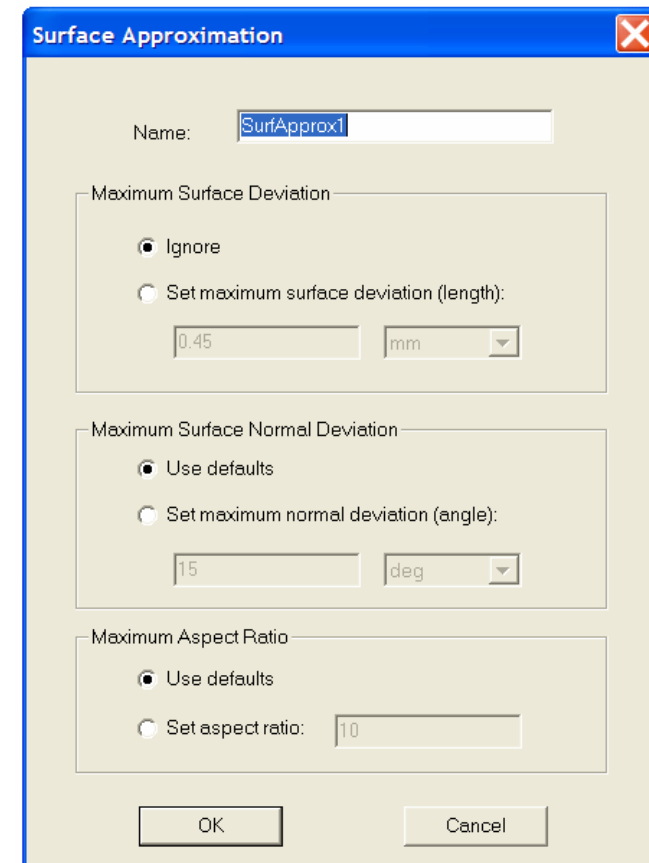


Model Resolution Mesh Operation

Name: ModelResolution1

Model Resolution Length: 12 mm

OK Cancel



Surface Approximation

Name: SurfApprox1

Maximum Surface Deviation

☒ Ignore

☐ Set maximum surface deviation (length): 0.45 mm

Maximum Surface Normal Deviation

☒ Use defaults

☐ Set maximum normal deviation (angle): 15 deg

Maximum Aspect Ratio

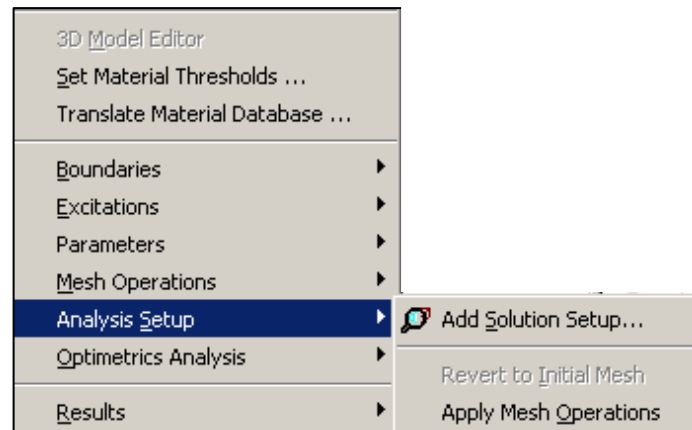
☒ Use defaults

☐ Set aspect ratio: 10

OK Cancel

Manual mesh creation

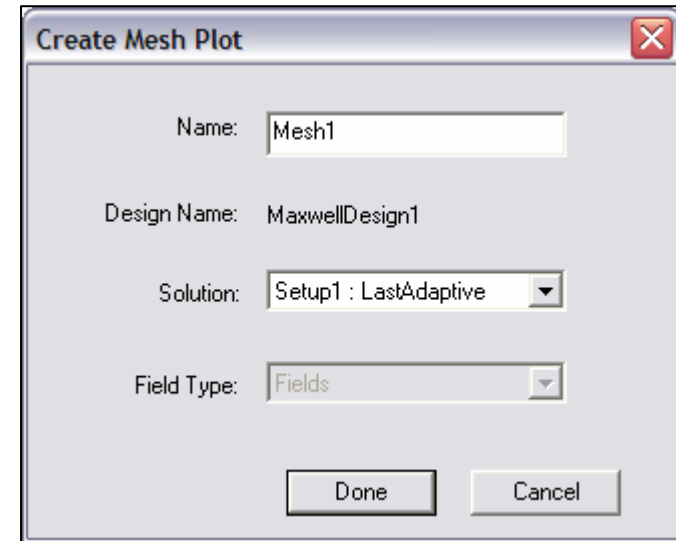
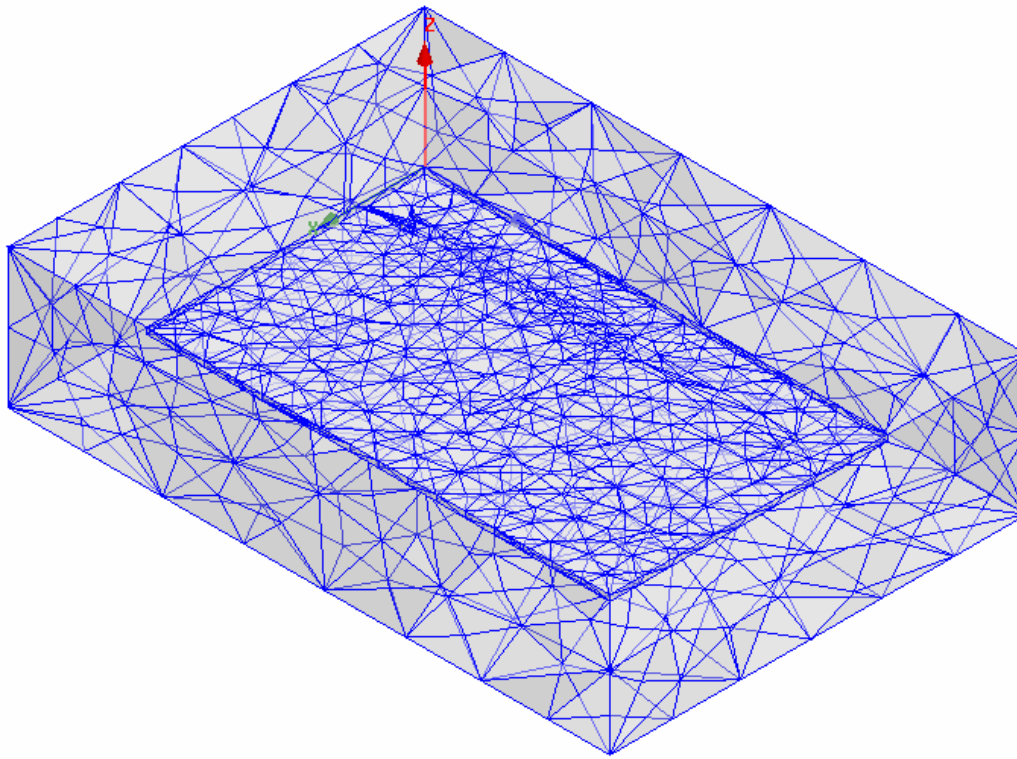
- ▲ To create the initial mesh: Click **Maxwell > Analysis Setup > Apply Mesh Operations**
- ▲ To refine the mesh without solving
 1. Define mesh operations as previously discussed
 2. Click **Maxwell > Analysis Setup > Apply Mesh Operations**



- ▲ If a mesh has been previously generated, Maxwell refines it using the defined mesh operations.
- ▲ If an initial mesh has not been generated, Maxwell generates it and applies the mesh operations to the initial mesh.
- ▲ If the defined mesh operations have been previously applied to the selected face or object, the current mesh is not altered.
- ▲ Hint: When modifying an existing mesh operation, you should revert to initial mesh and then re-apply mesh operations.
- ▲ To view mesh information: Click **Maxwell > Results > Solution Data** and click on the tab **Mesh Statistics**

Mesh Display

1. Select an object
2. Select the menu item *Maxwell > Fields > Plot Mesh*



Post Processing

Two Methods of Post Processing Solutions:

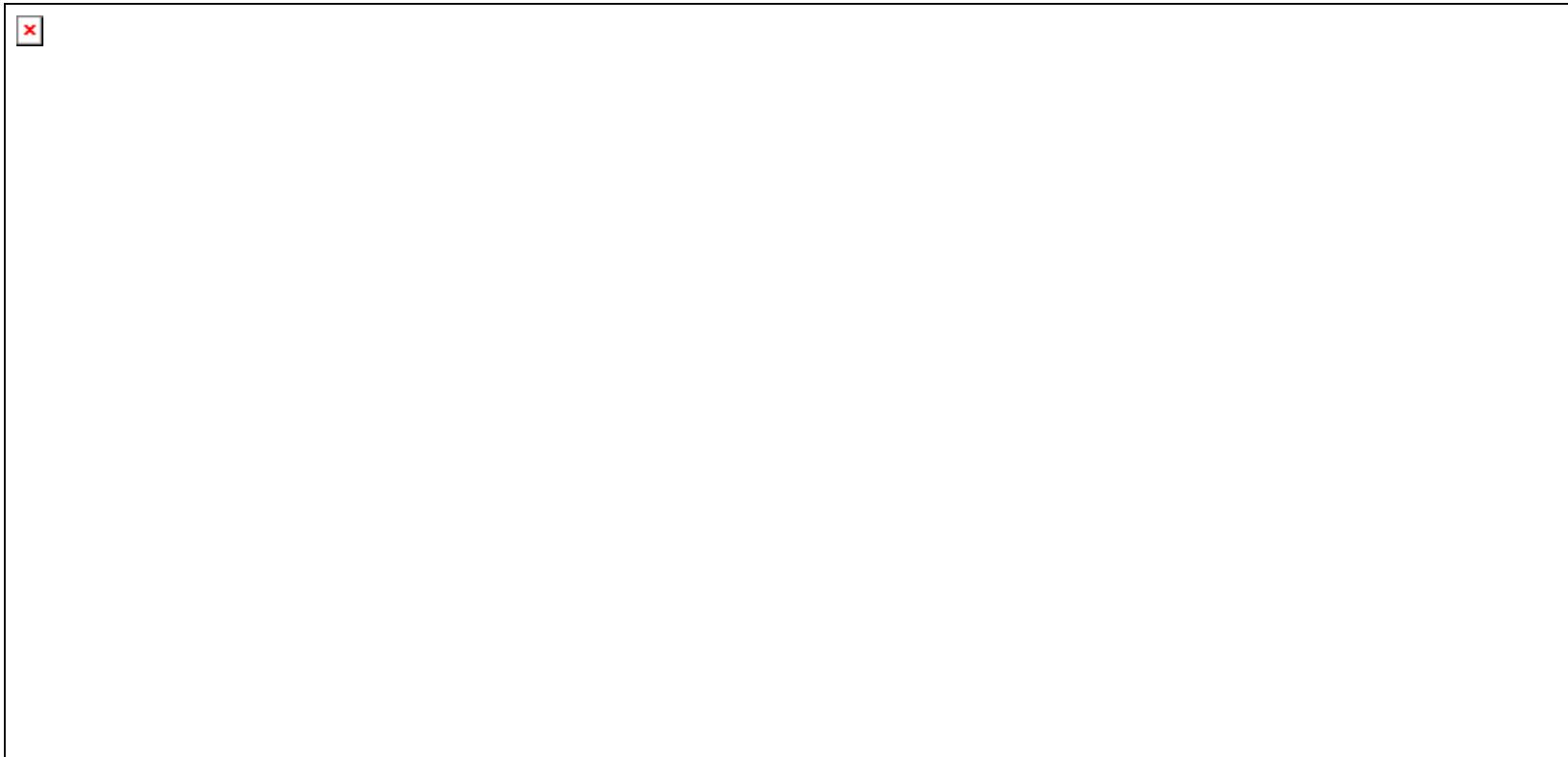
- ▀ Viewing Plots
- ▀ Manipulating Field Quantities in Calculator

Six Types of Plots:

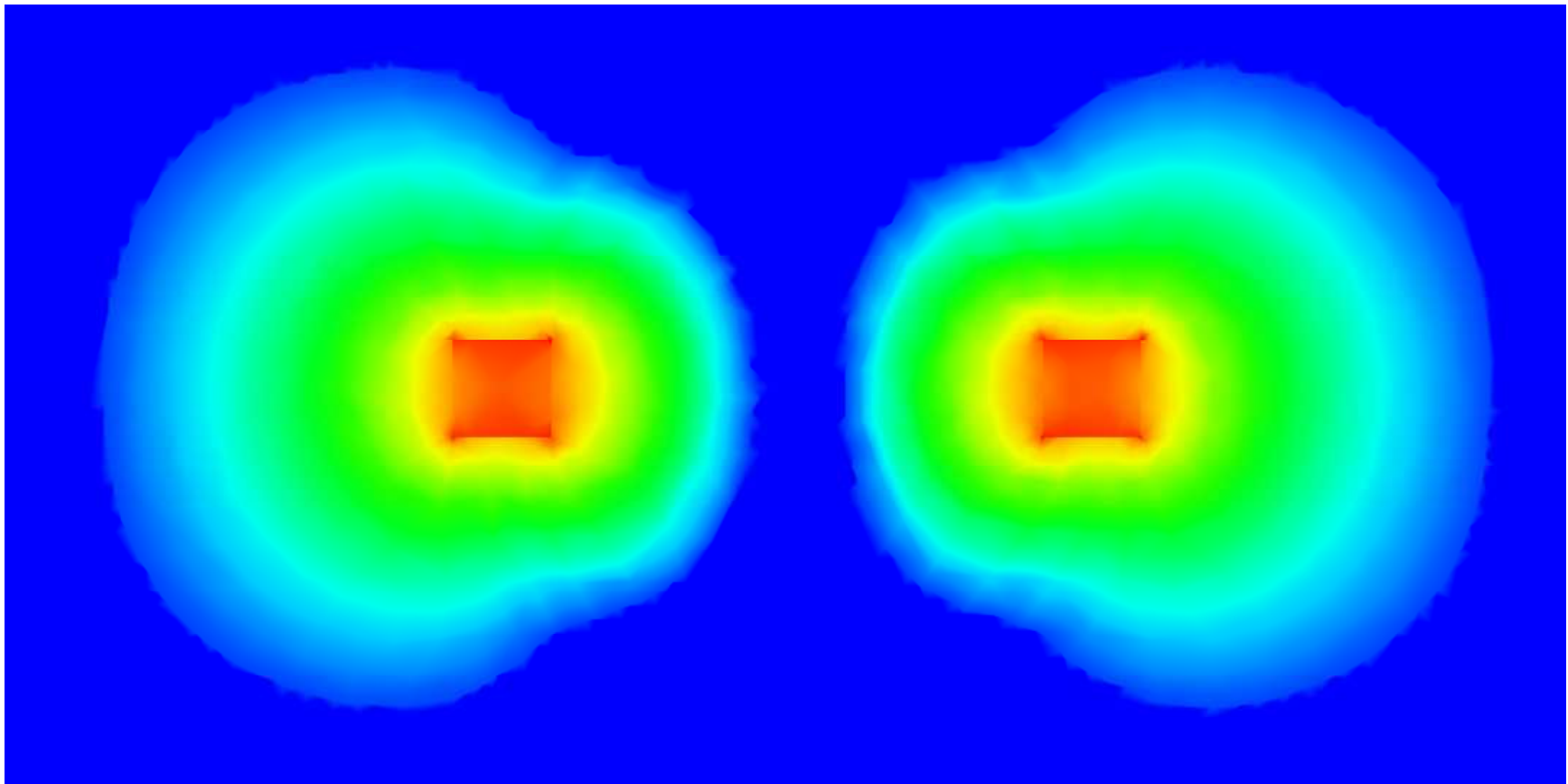
1. Contour plots (scalars): equipotential lines, ...
2. Shade plots (scalars): Bmag, Hmag, Jmag, ...
3. Scatter plots (scalars): Bmag, Hmag, Jmag, Voltage
4. Arrow plots (vectors): B vector, H vector, ...
5. Line plots (scalars): magnitude vs. distance along a predefined line
6. Animation Plots



Contour plot

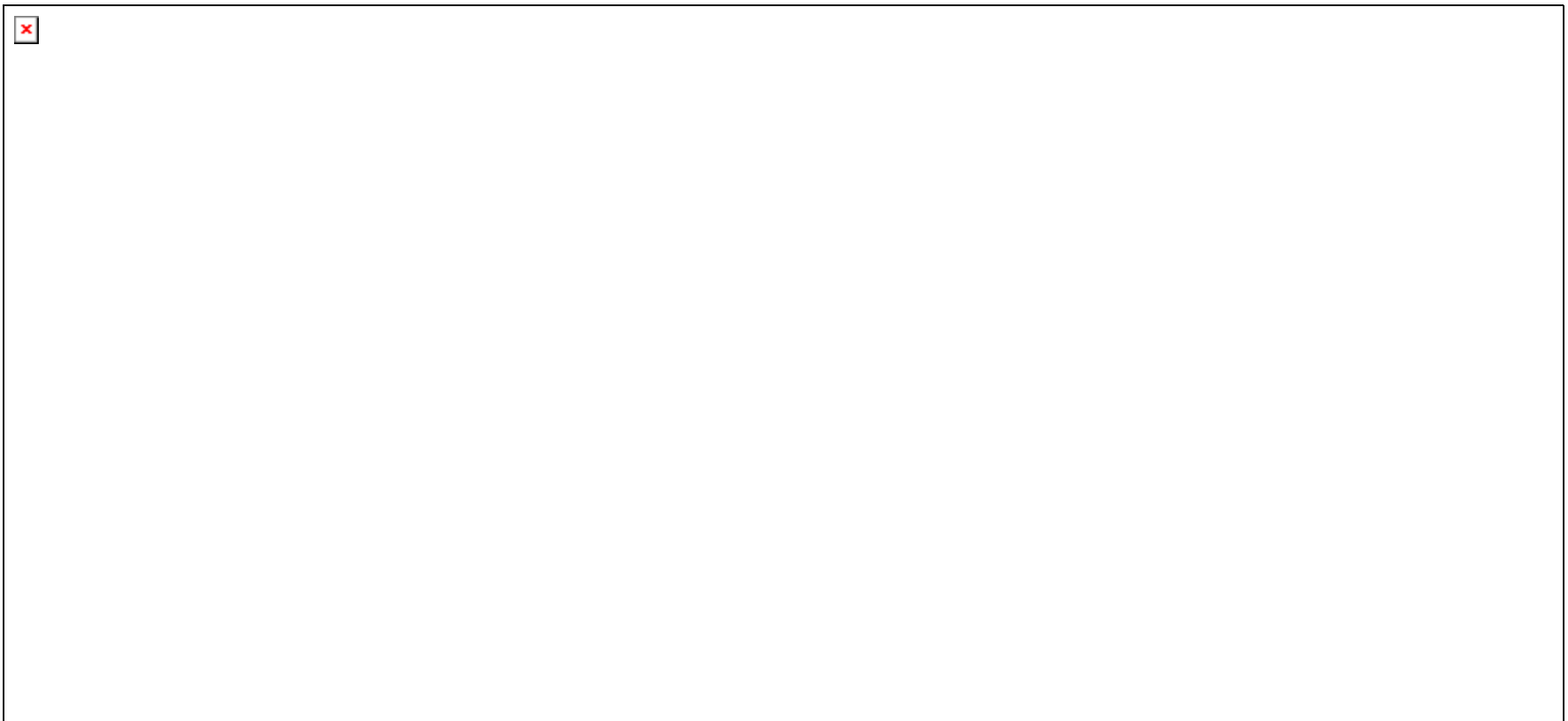


 Shade plot (tone)

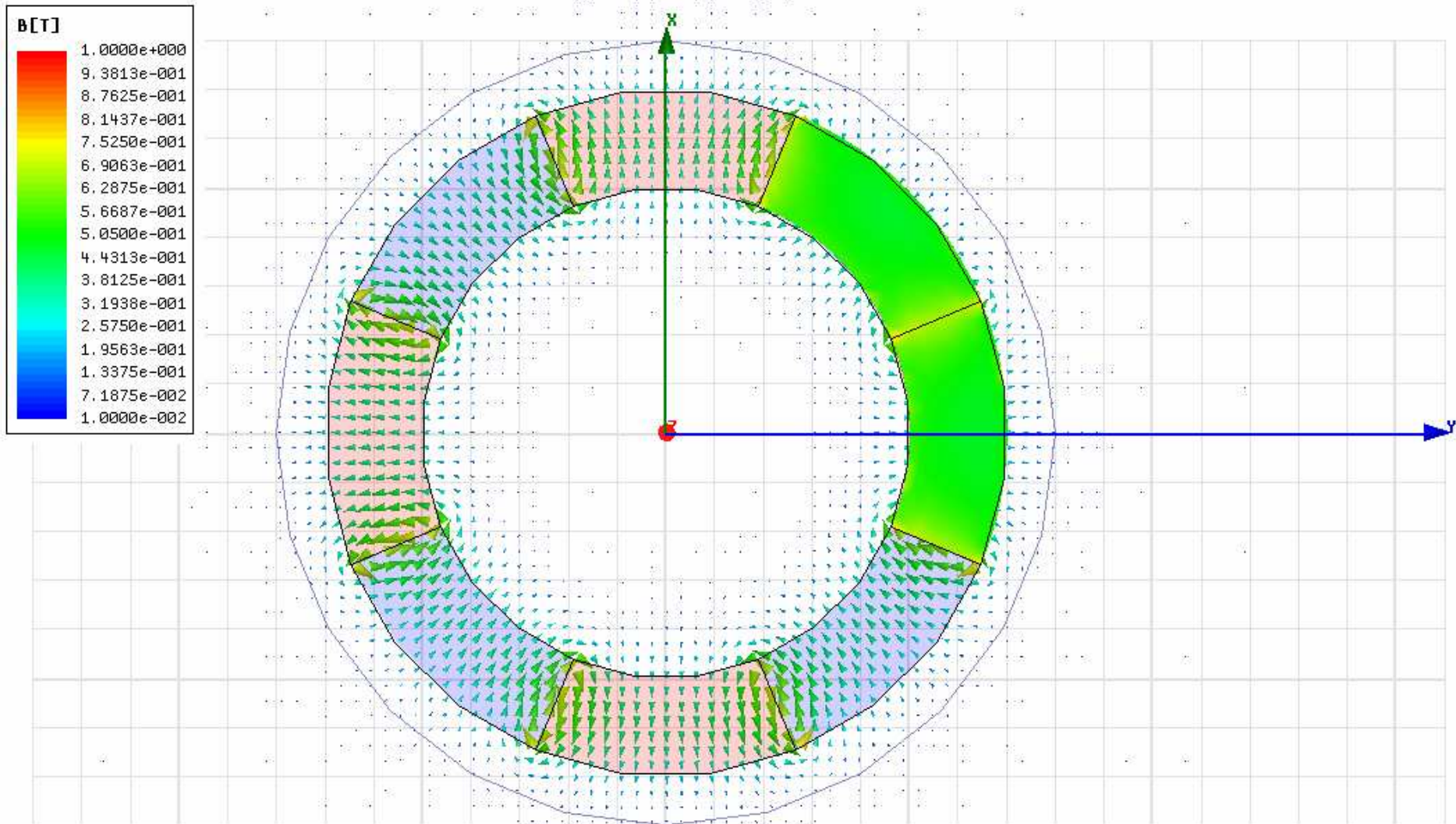




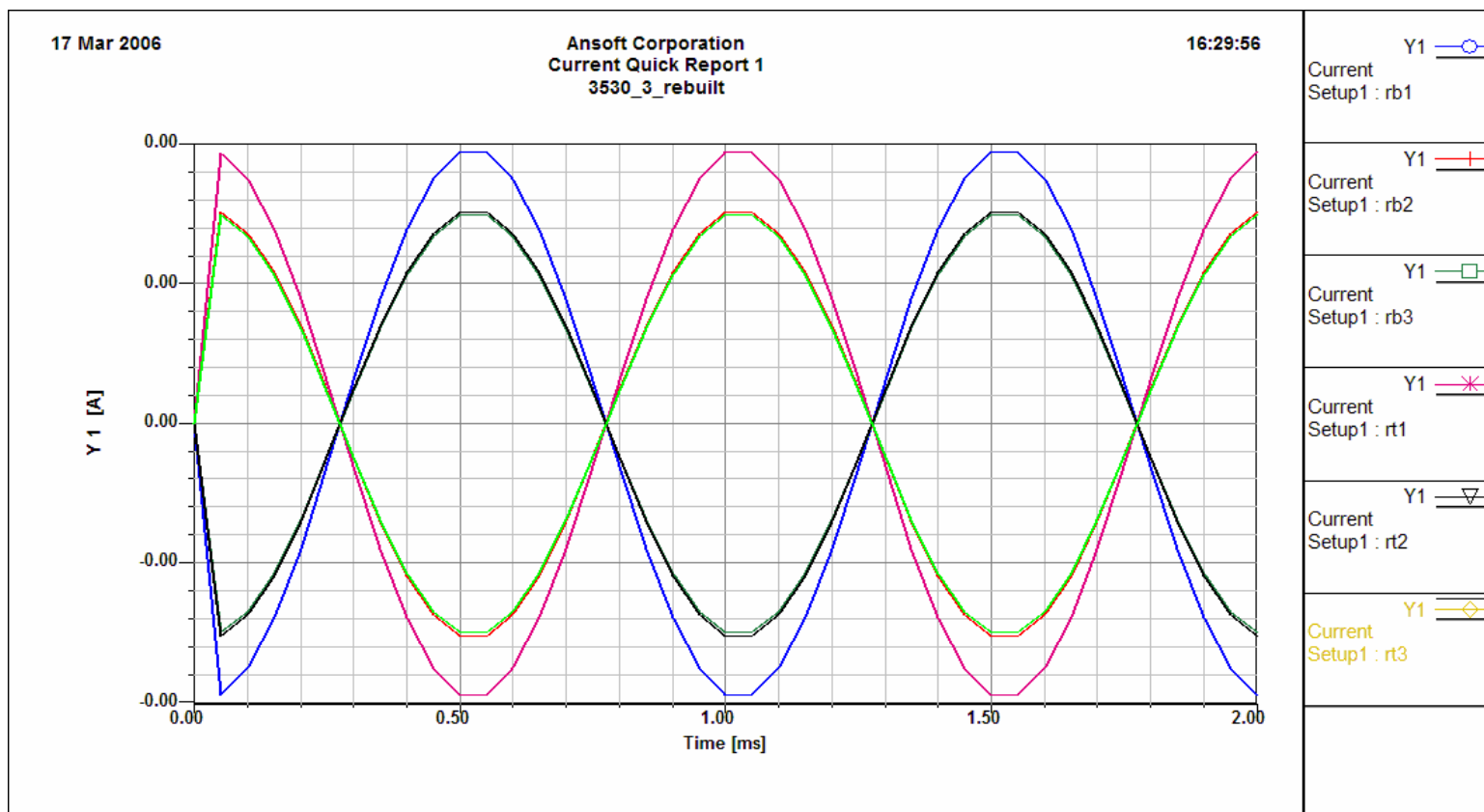
 Shade plot (fringe with outline)



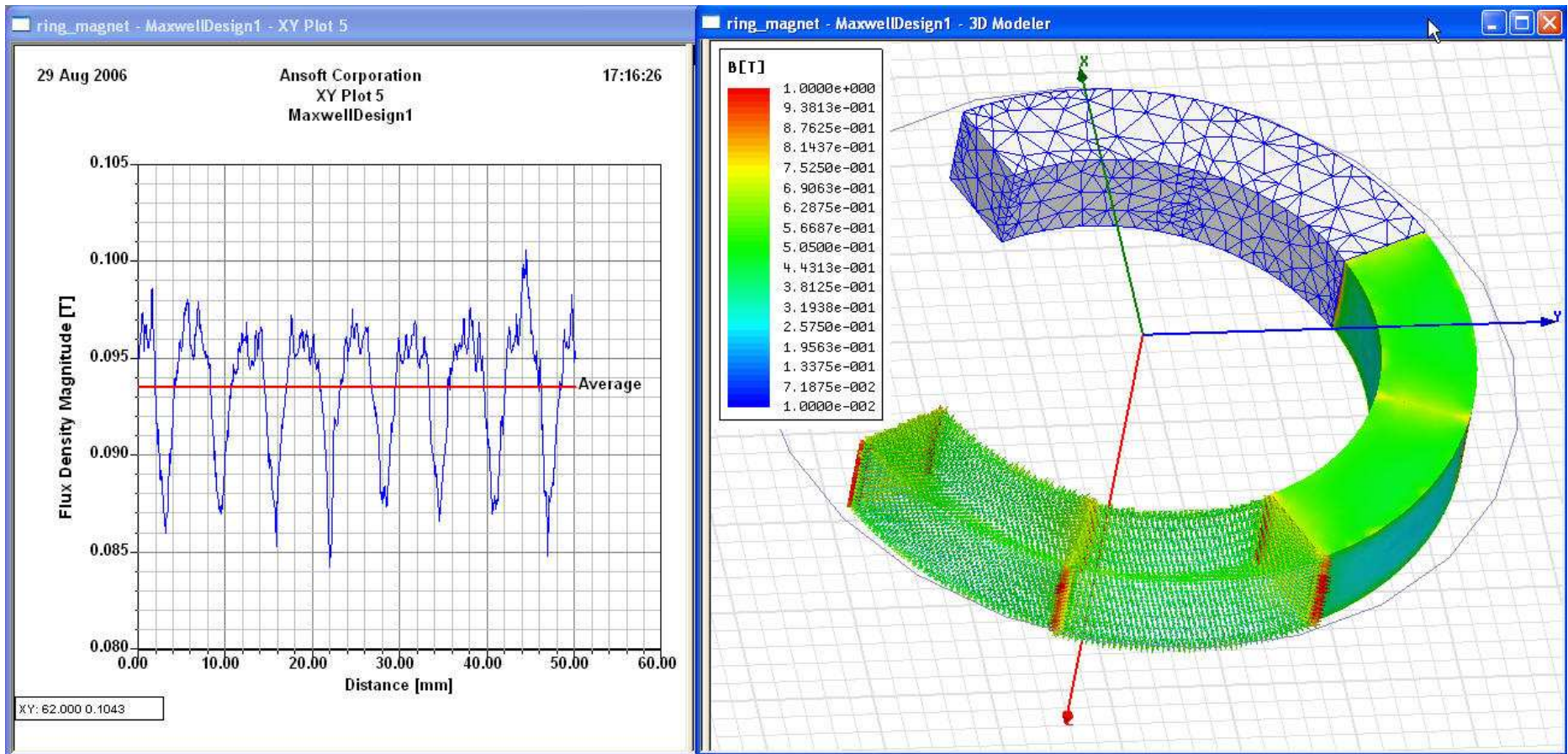
Arrow plot



Line plot

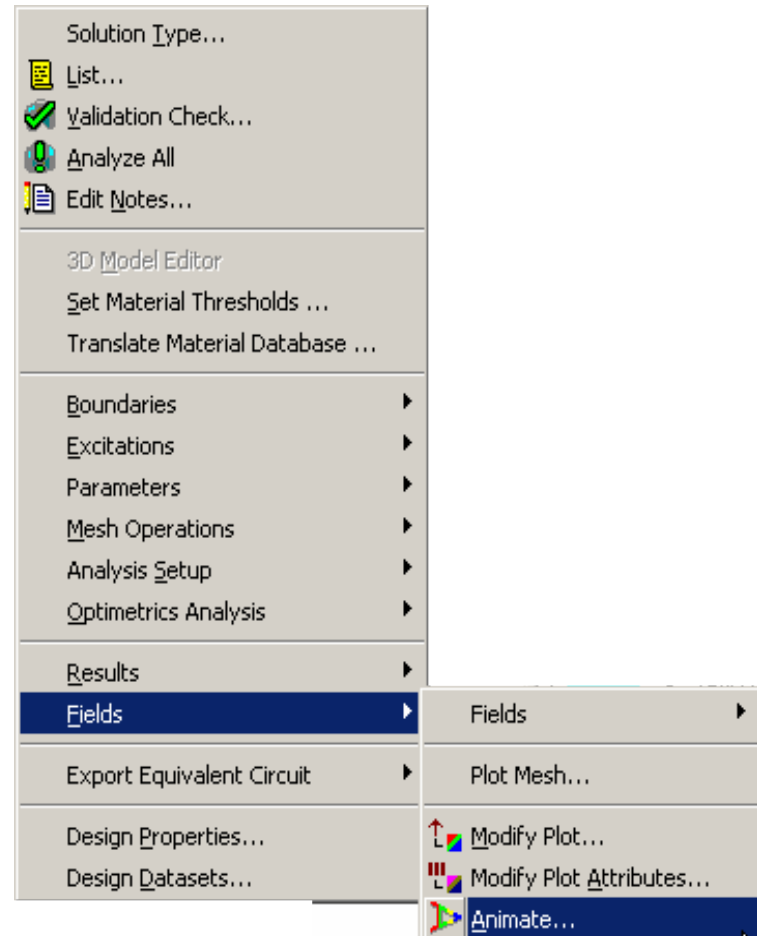


Multiple windows and multiple plots



Animation plot

-  Animate across different surfaces or with different phase angles (eddy solver)
-  Export to .gif or .avi format



Fields Calculator

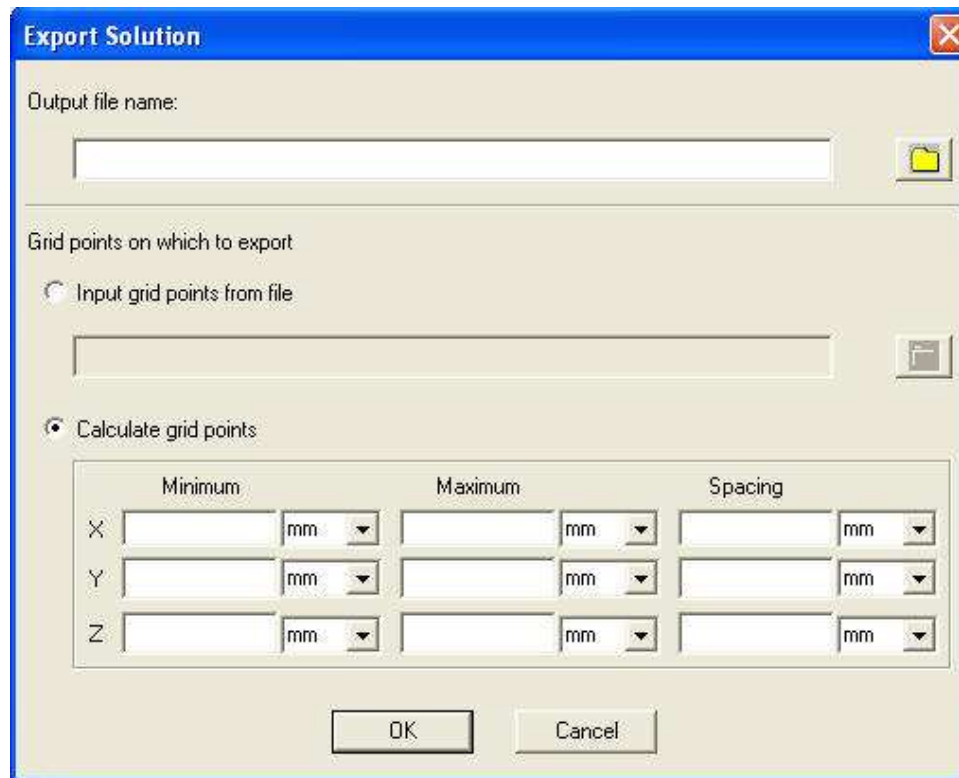
-  To bring up the Fields Calculator tool
 1. Select the menu item *Maxwell->Fields->Calculator*

-  Typical quantities to analyze:
 1. Flux through a surface
 2. Current Flow through a surface
 3. Tangential Component of E-field along a line
 4. Average Magnitude of B-field in a core
 5. Total Energy in an object

Fields Calculator - Export Command

Exports the field quantity in the top register to a file, mapping it to a grid of points. Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available:

1. **Grid points from file:** Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points.
2. **Calculate grid points:** Maps the field quantity to a three-dimensional Cartesian grid. You specify the dimensions and spacing of the grid in the x, y, and z directions.



The dialog box titled "Export Solution" contains the following fields and controls:

- Output file name:** A text input field with a folder icon to its right.
- Grid points on which to export:**
 - ☐ Input grid points from file: A text input field with a file icon to its right.
 - ☒ Calculate grid points: A section containing a table for grid parameters.

	Minimum		Maximum		Spacing
X		mm		mm	mm
Y		mm		mm	mm
Z		mm		mm	mm

At the bottom of the dialog are "OK" and "Cancel" buttons.

Export to Grid

- Vector data <Ex,Ey,Ez>
- Min: [0 0 0]
- Max: [2 2 2]
- Spacing: [1 1 1]
- Space delimited ASCII file saved in project subdirectory

Vector data "<Ex,Ey,Ez>"

Grid Output Min: [0 0 0] Max: [2 2 2] Grid Size: [1 1 1]

```

0 0 0 -71.7231 -8.07776 128.093
0 0 1 -71.3982 -1.40917 102.578
0 0 2 -65.76 -0.0539669 77.9481
0 1 0 -259.719 27.5038 117.572
0 1 1 -248.088 16.9825 93.4889
0 1 2 -236.457 6.46131 69.4059
0 2 0 -447.716 159.007 -8.6193
0 2 1 -436.085 -262.567 82.9676
0 2 2 -424.454 -236.811 58.8847
1 0 0 -8.91719 -241.276 120.392
1 0 1 -8.08368 -234.063 94.9798
1 0 2 -7.25016 -226.85 69.5673
1 1 0 -271.099 -160.493 129.203
1 1 1 -235.472 -189.125 109.571
1 1 2 -229.834 -187.77 84.9415
1 2 0 -459.095 -8.55376 2.12527
1 2 1 -447.464 -433.556 94.5987
1 2 2 -435.833 -407.8 70.5158
2 0 0 101.079 -433.897 -18.5698
2 0 1 -327.865 -426.684 95.8133
2 0 2 -290.824 -419.471 70.4008
2 1 0 -72.2234 -422.674 -9.77604
2 1 1 -495.898 -415.461 103.026
2 1 2 -458.857 -408.248 77.6138
2 2 0 -470.474 -176.115 12.8698
2 2 1 -613.582 -347.994 83.2228
2 2 2 -590.326 -339.279 63.86

```

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.
- ▲ **Tooltips** 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**

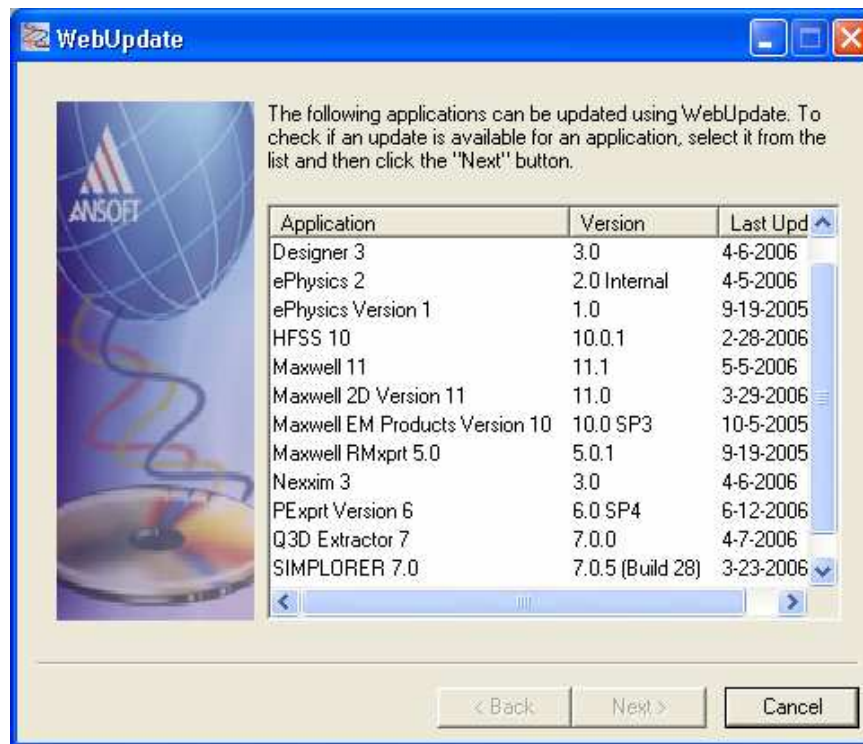
- ▲ To contact Ansoft technical support staff in your geographical area, please log on to the Ansoft corporate website, www.ansoft.com and select **Contact**.
- ▲ Your Ansoft sales engineer may also be contacted in order to obtain this information.

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

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.



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>

Application Support for North America

-  The names and numbers in this list may change without notice
 -  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

THIS PAGE INTENTIONALLY LEFT BLANK