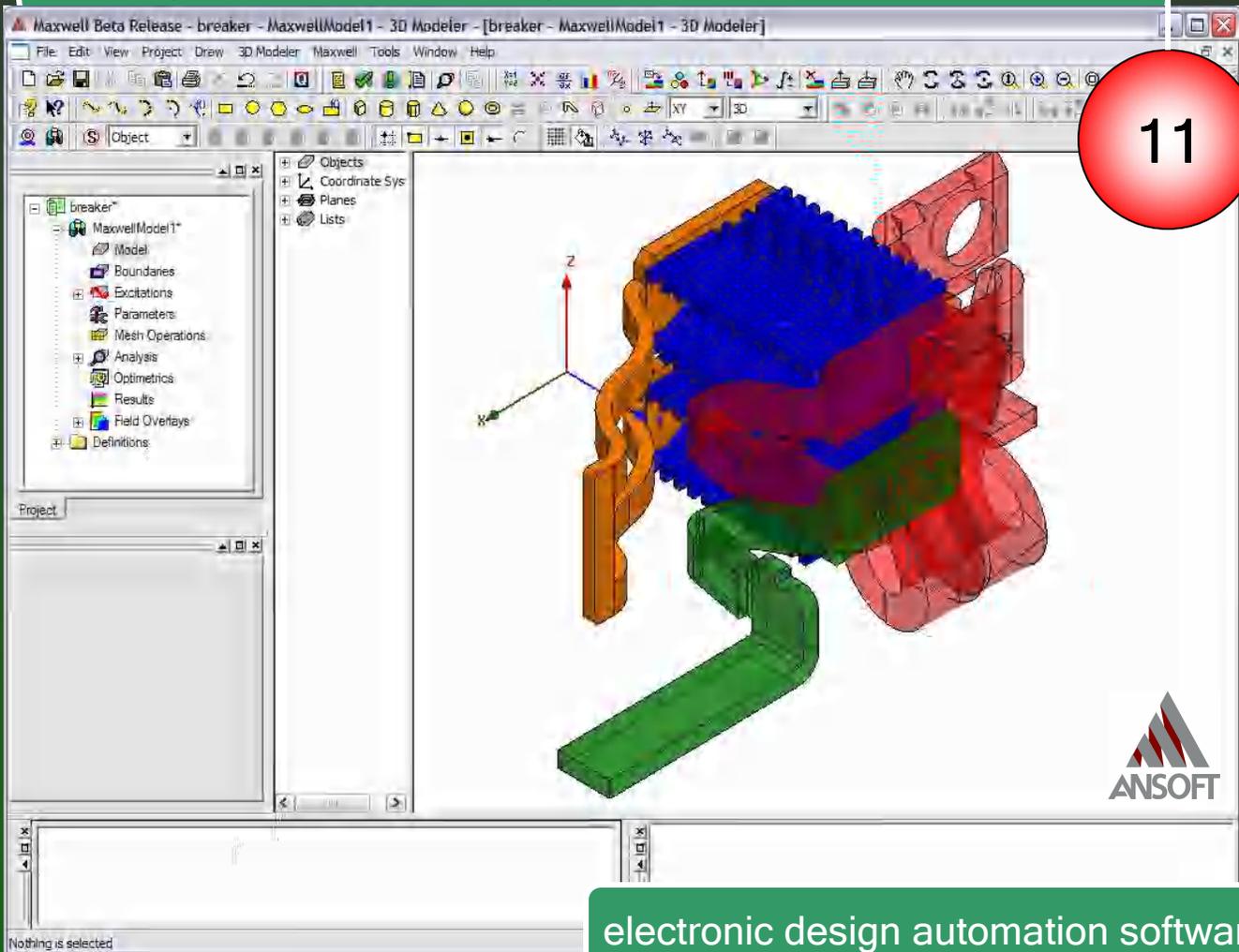


Ansoft

Maxwell 3D

Electromagnetic and Electromechanical Analysis



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

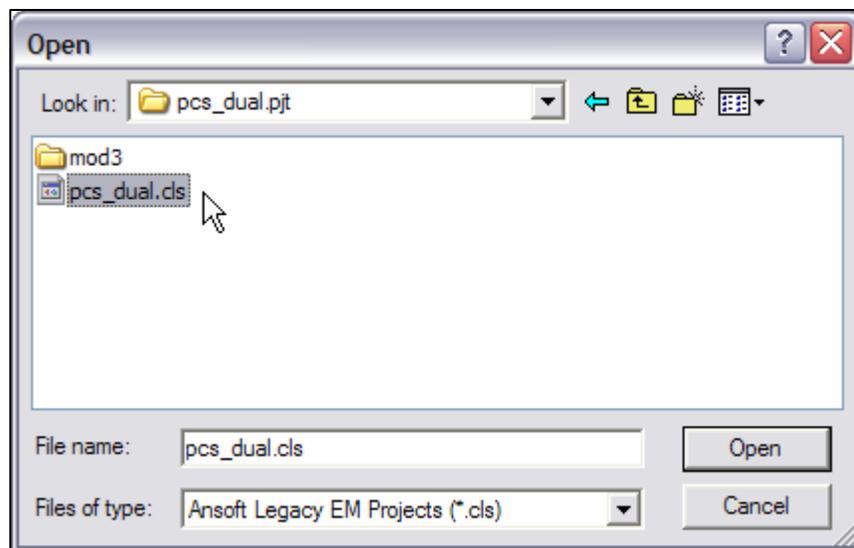
1. Click the Microsoft **Start** button, select **Programs**, and select the **Ansoft, Maxwell 11** program group. Click **Maxwell 11**.
2. Or Double click on the Maxwell 11 icon on the Windows Desktop



❖ 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**

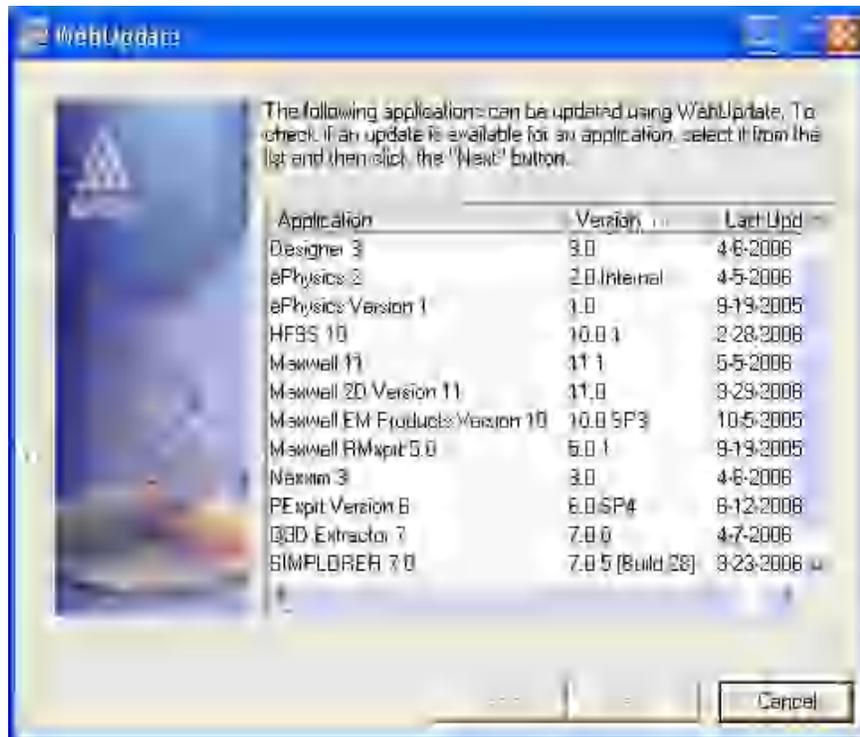
- (503) 906-7944 or (503) 906-7947

- **El Segundo, CA**

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

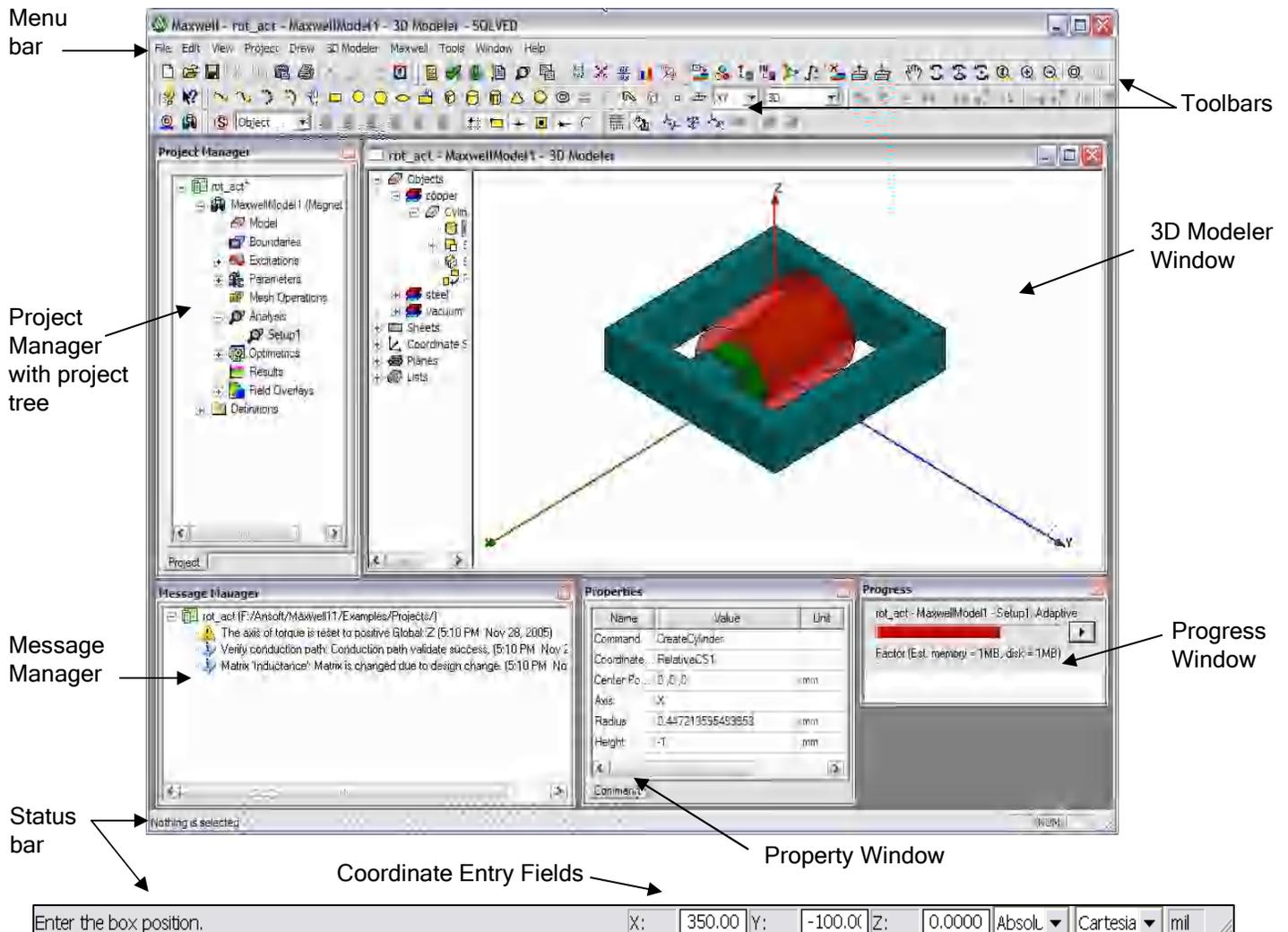
WebUpdate

- 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.



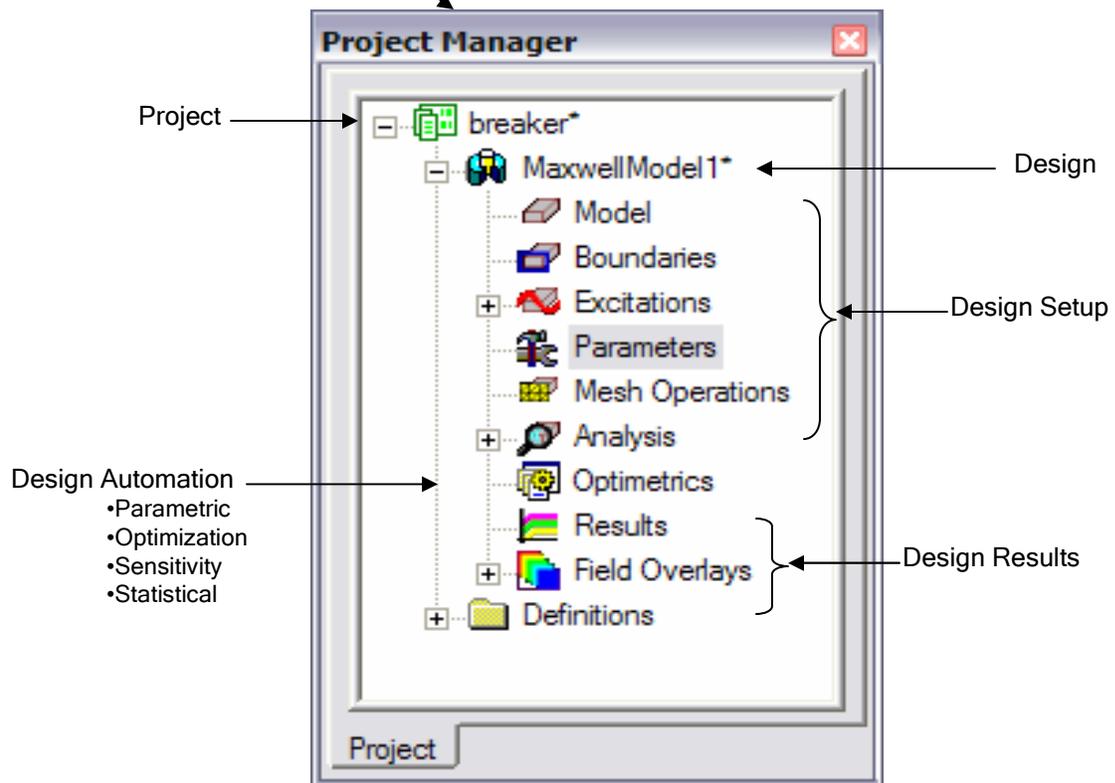
❖ Ansoft Terms

- ❖ 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.



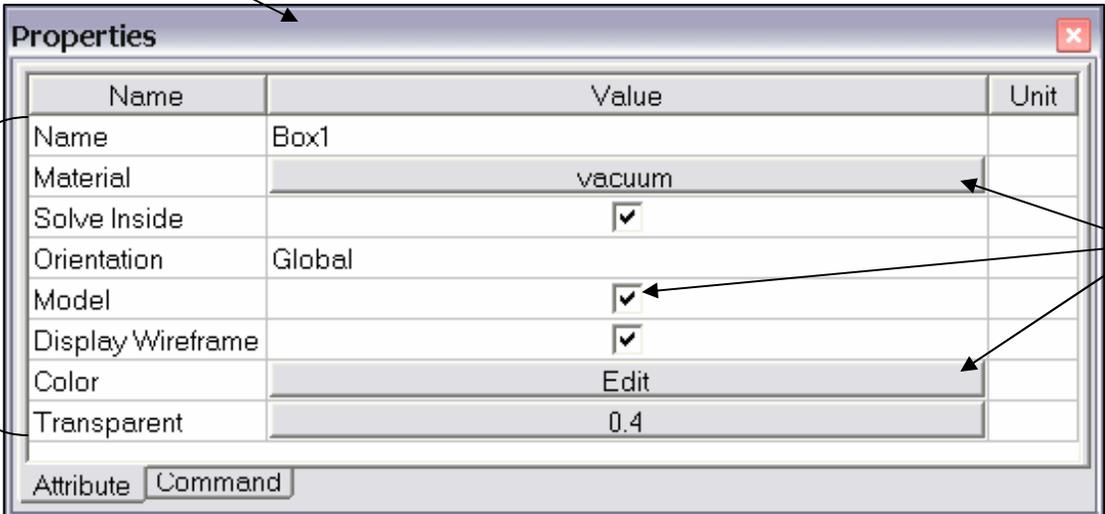
Project Manager

Project Manager Window



Property Window

Property Window



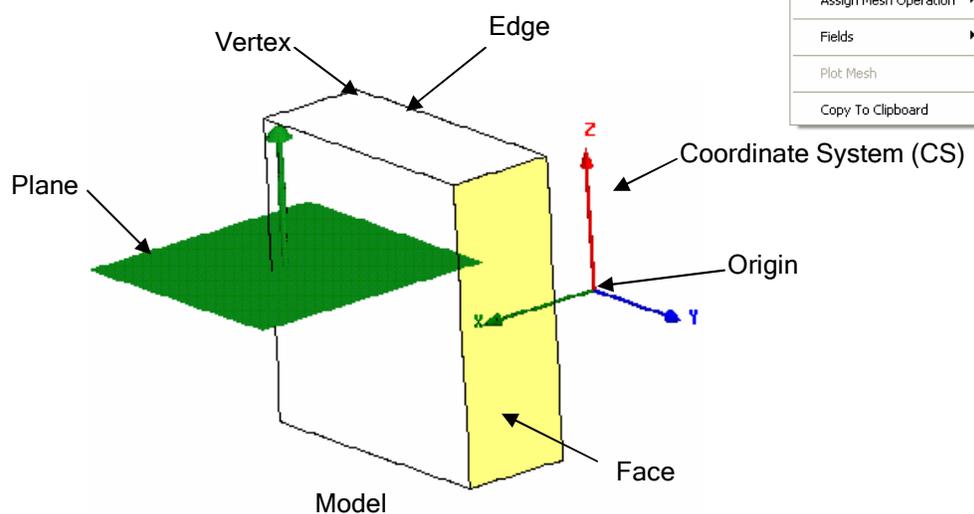
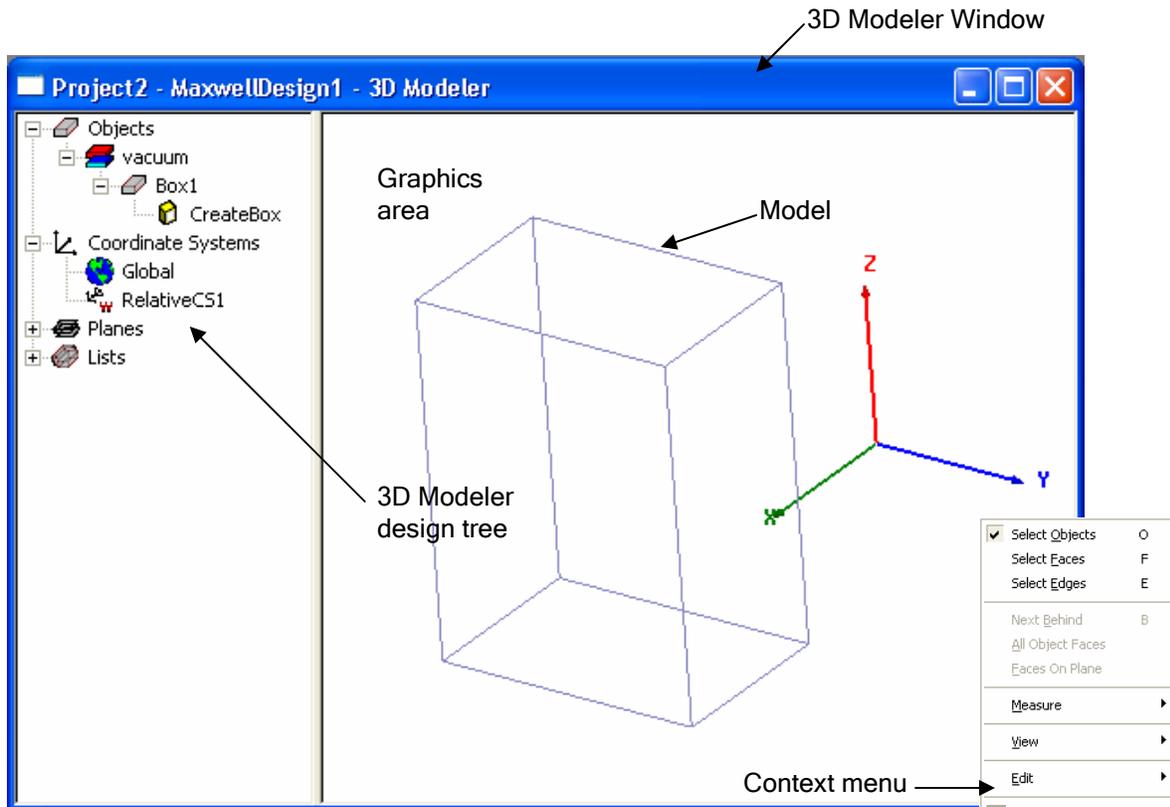
Name	Value	Unit
Name	Box1	
Material	vacuum	
Solve Inside	<input checked="" type="checkbox"/>	
Orientation	Global	
Model	<input checked="" type="checkbox"/>	
Display Wireframe	<input checked="" type="checkbox"/>	
Color	Edit	
Transparent	0.4	

Property table

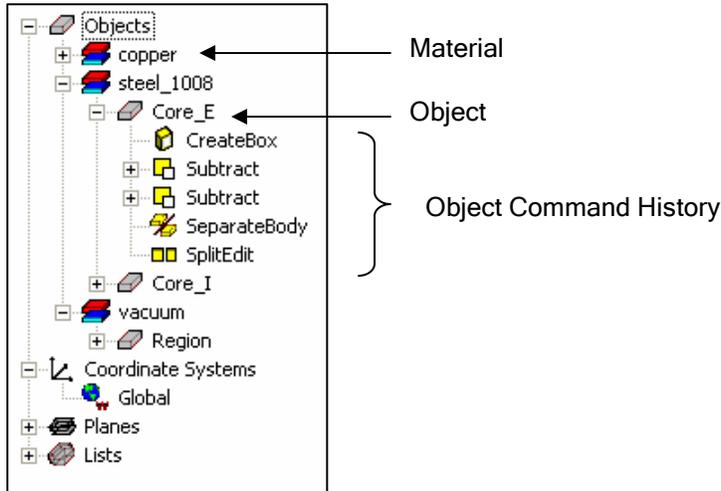
Property buttons

Property tabs

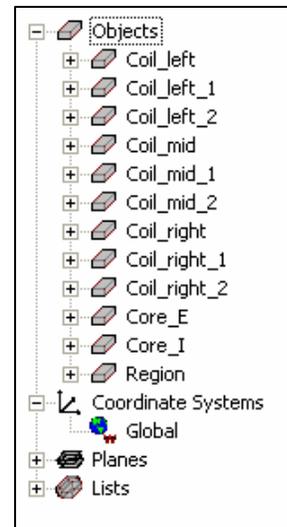
Ansoft 3D Modeler



3D Modeler Design Tree



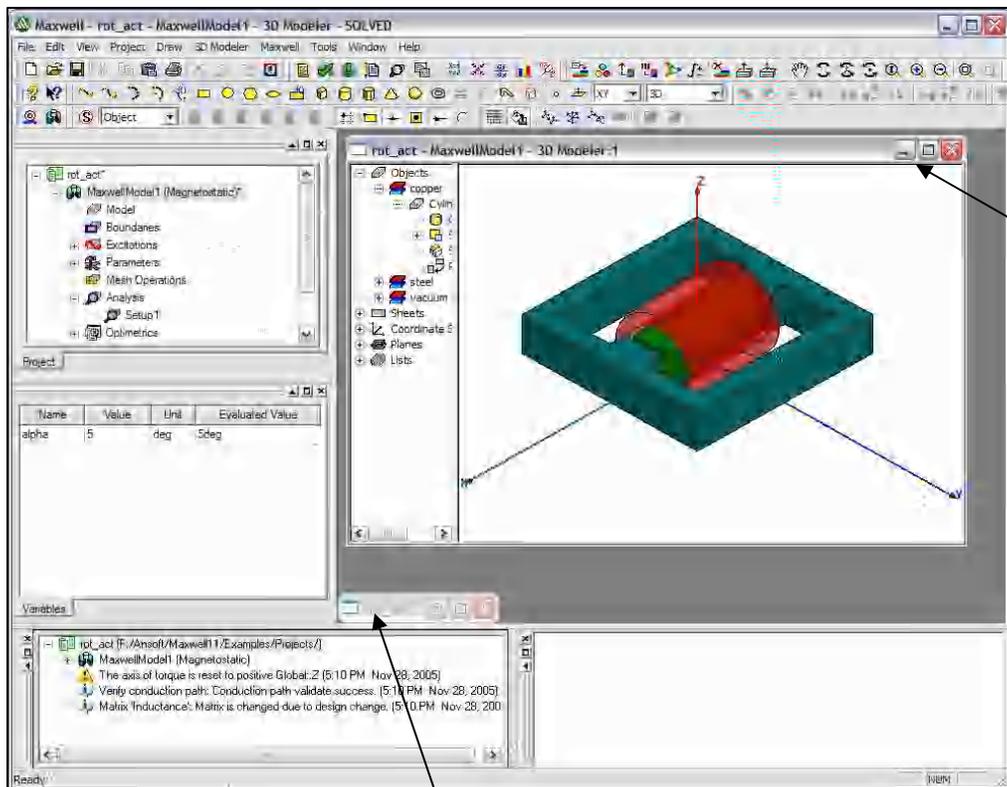
Grouped by Material



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.

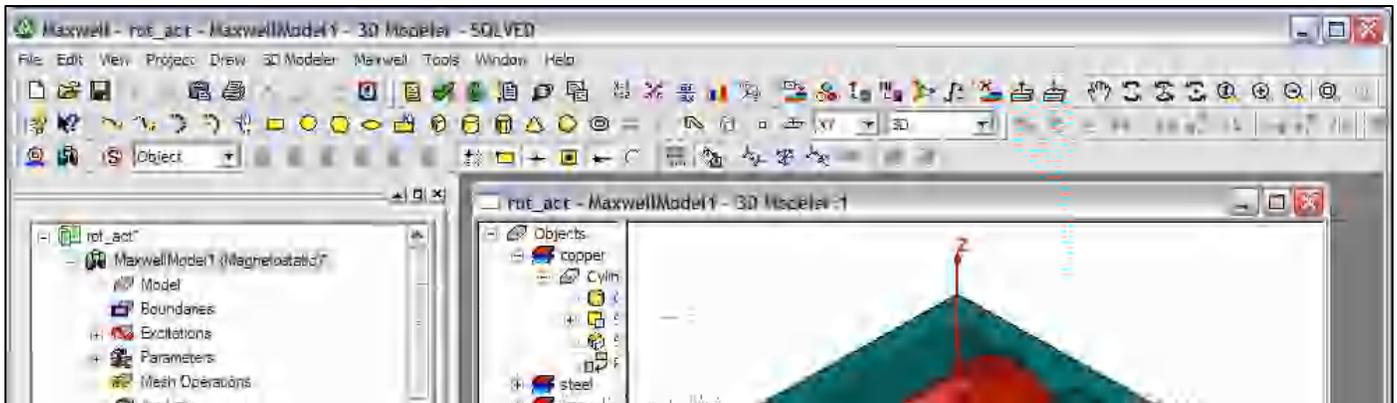


Iconize
Symbol

Design icons

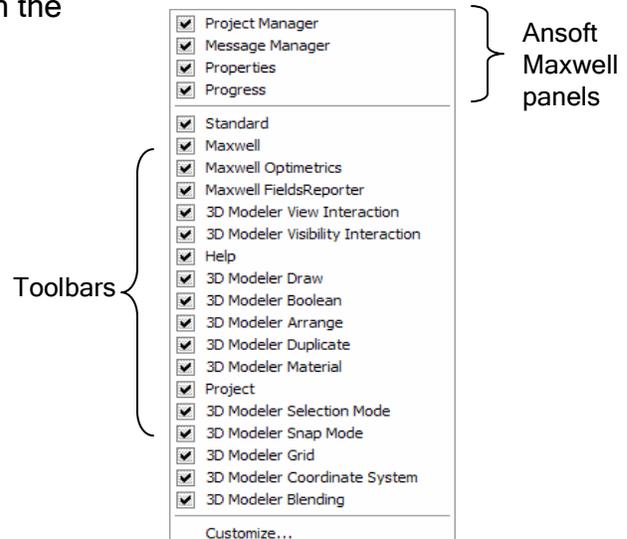
▲ Toolbars

- ▲ 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.



❖ 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.

❖ 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.

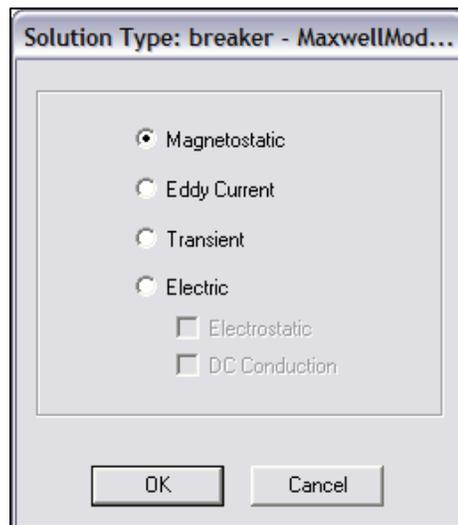
❖ To open a project from Windows Explorer, do one of the following:

- ❖ 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 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.

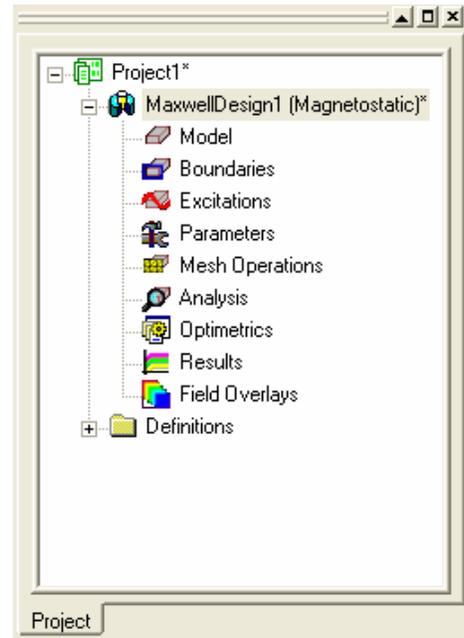
▸ Overview

- 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.

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.

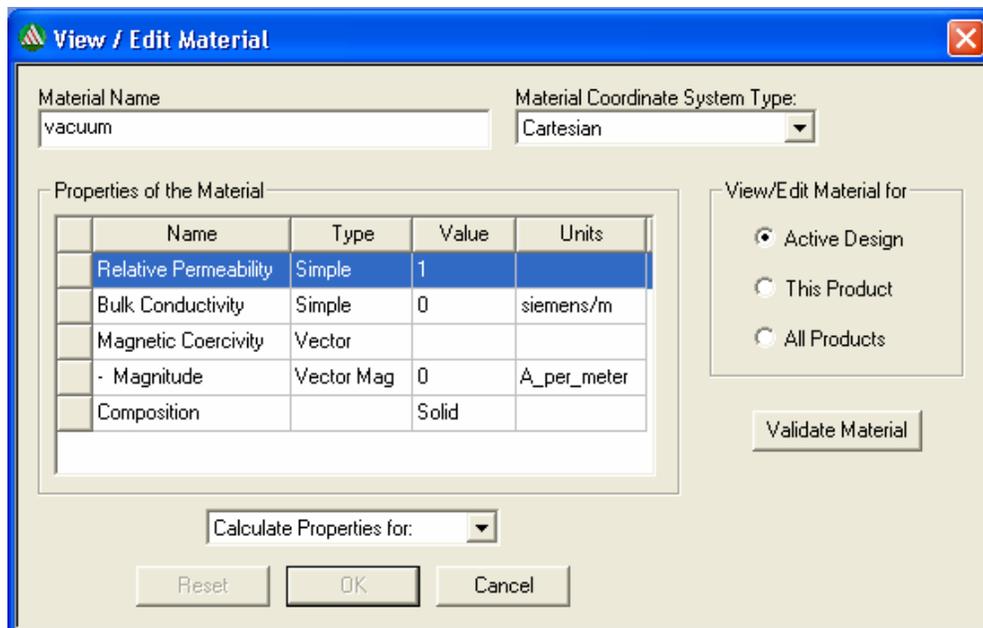
- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in a magnetostatic simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ 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.



❖ 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

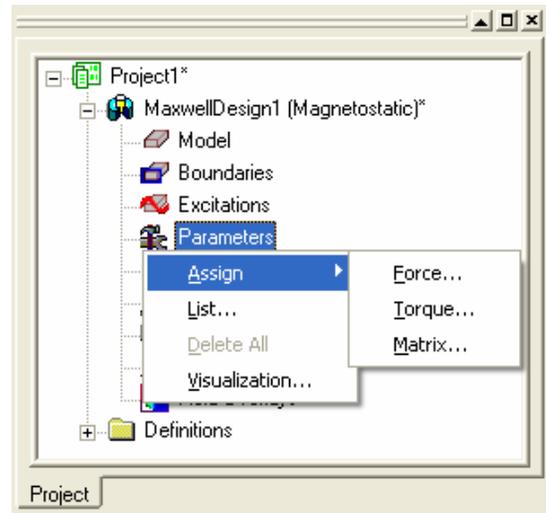
- ❖ **Master/Slave** - This boundary condition is also known as a matching boundary condition because it matches the magnetic field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this 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.

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).



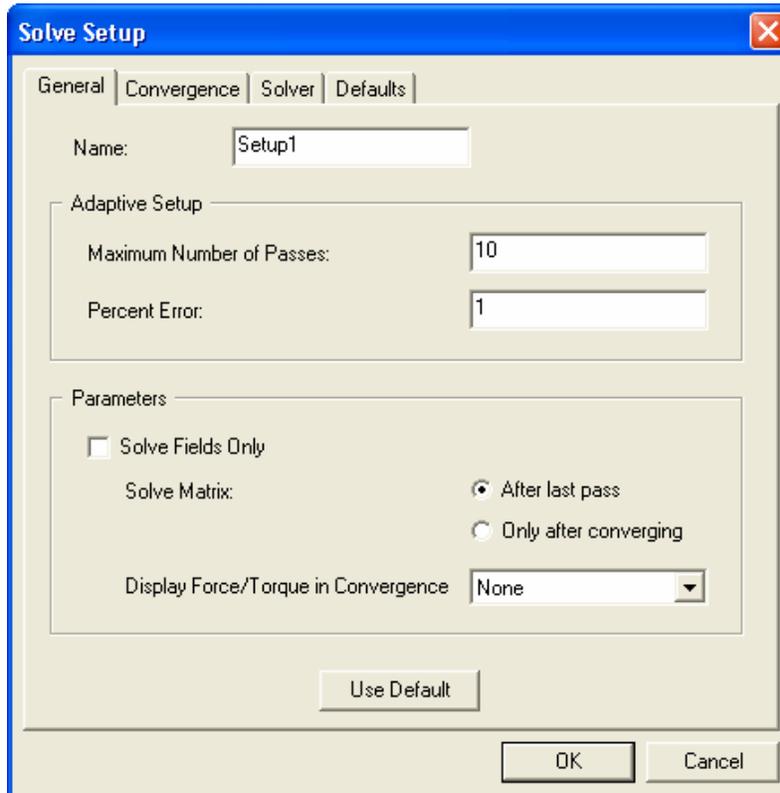
- ▲ Inductance Matrix has many post-processing options available - this is discussed in detail in the Magnetostatic Inductance Matrix example.
- ▲ The results of any parameters can be found by selecting *Maxwell > Results > Solution Data...* or by clicking on the  icon.
- ▲ Further results can be obtained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

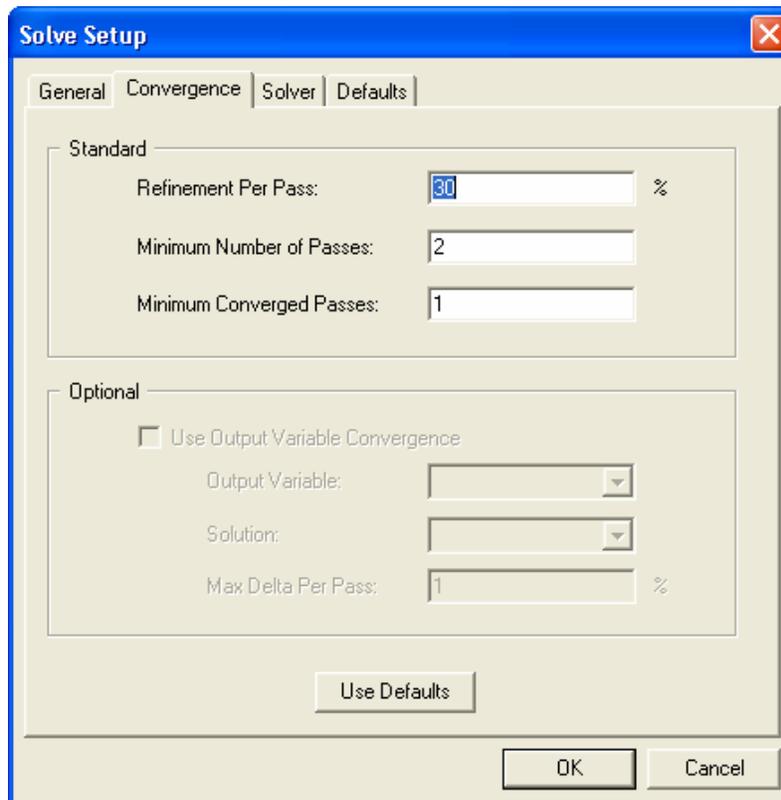
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
- The following window appears with the General Setup parameters.



- 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)

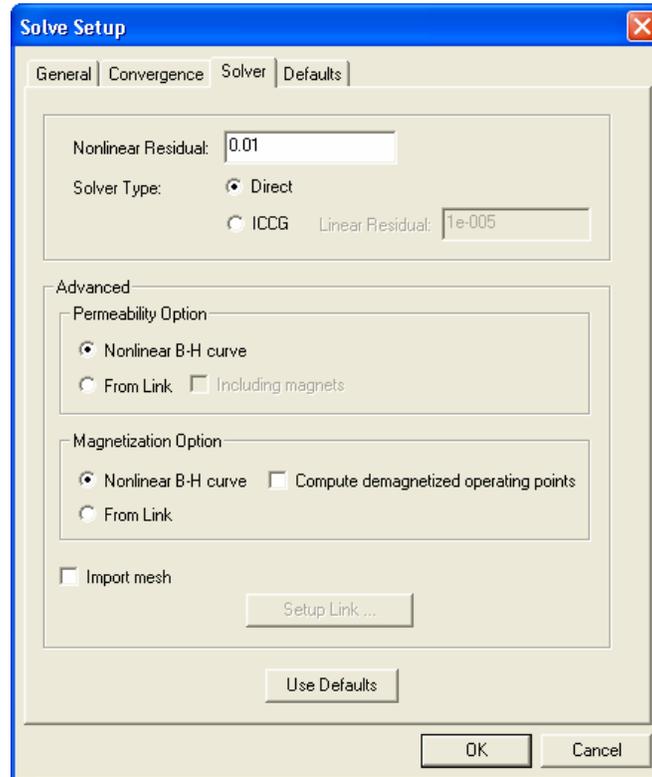
- The second tab of the Solution Setup contains information about Convergence.



- 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)

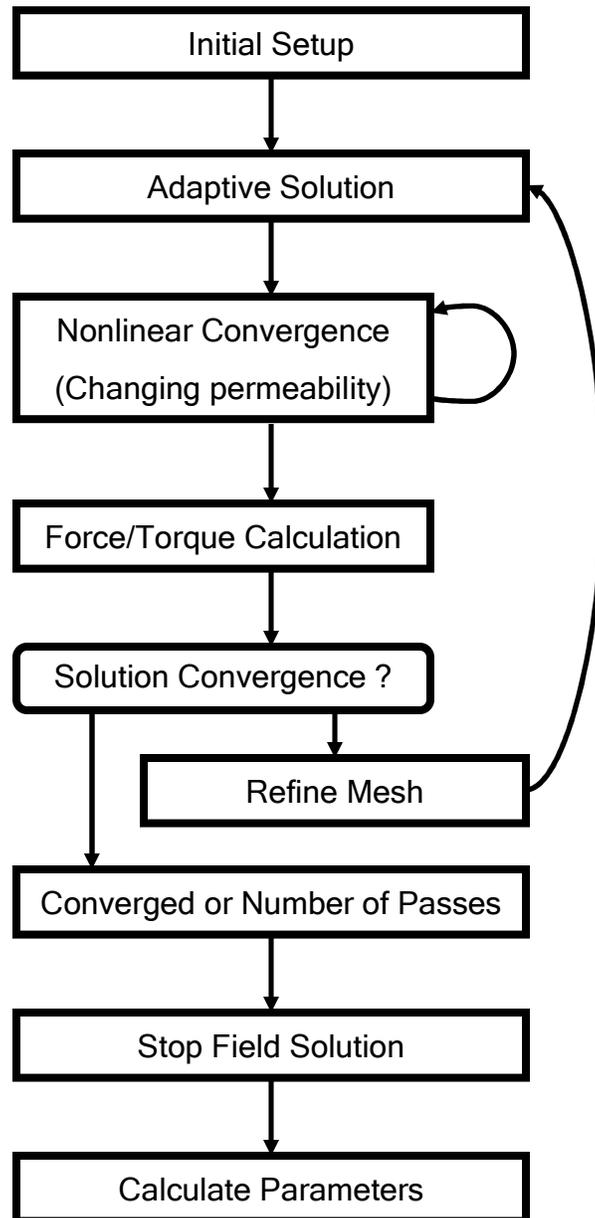
- The third tab of the Solution Setup contains information about the Solver.



- 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 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

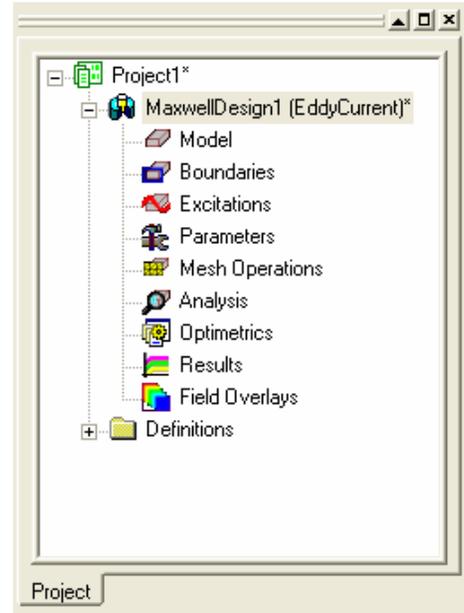
- ▀ The eddy current solver computes steady-state, time-varying (AC) magnetic fields at a given frequency - this is a frequency domain solution.
- ▀ All objects are stationary.
- ▀ The source of the static magnetic field can be:
 - ▀ Sinusoidal AC current (peak) in conductors.
 - ▀ Time-varying external 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.

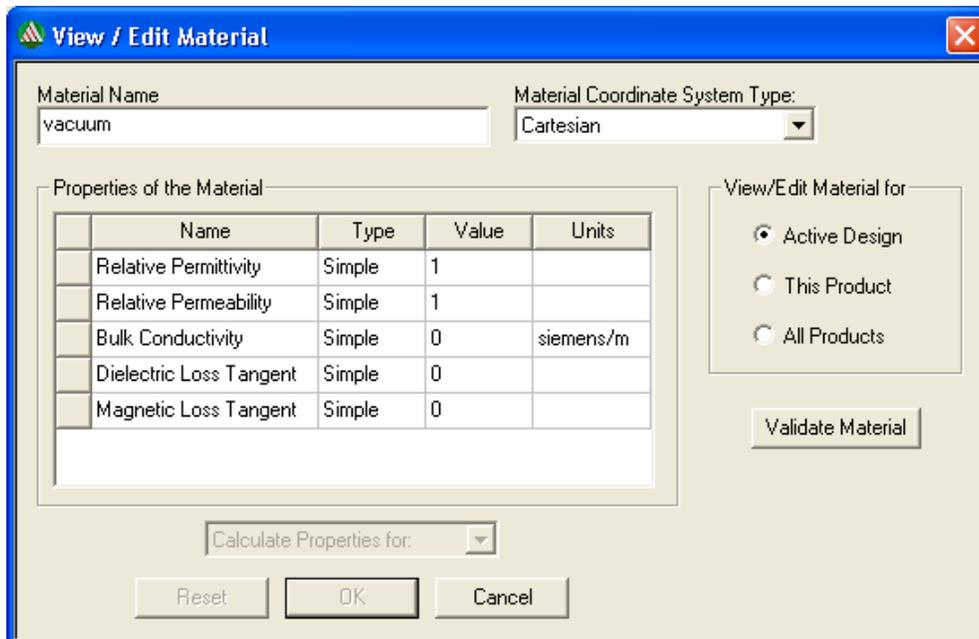


- ▲ These options are displayed in an order that can be followed in creating a new Eddy Current simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.

- ▲ 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.



▸ 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

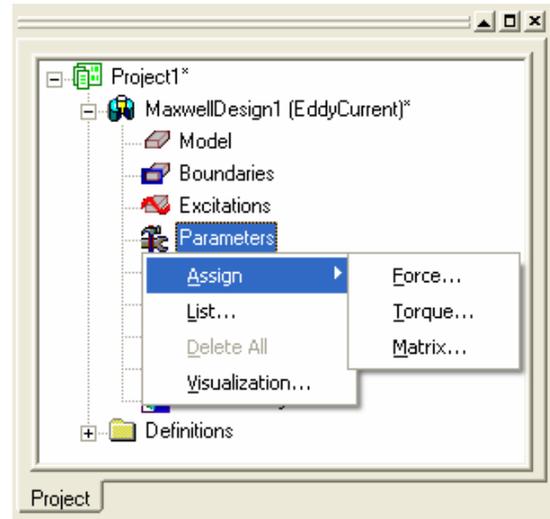
-  Typical sources for eddy current problems include current and current density. In applying the sources for the magnetic field problems, keep in mind that the applied current distribution must be divergence free in the entire space of the solution as it is physical for (quasi) stationary conduction 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.

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).



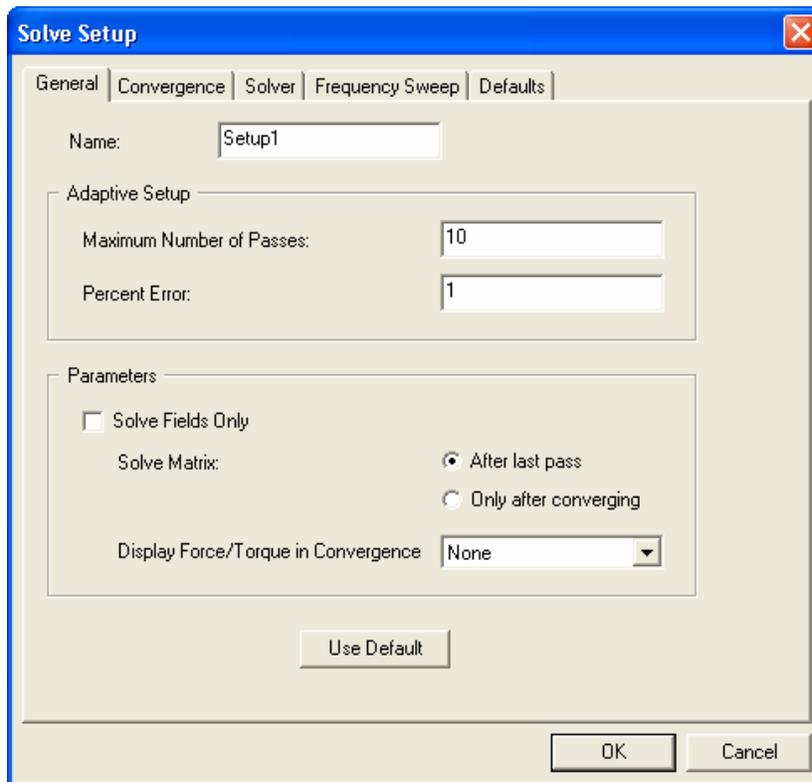
- ▲ The results of any parameters can be found by selecting **Maxwell > Results > Solution Data...** or by clicking on the  icon.
- ▲ Further results can be obtained manually through the field calculator.

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

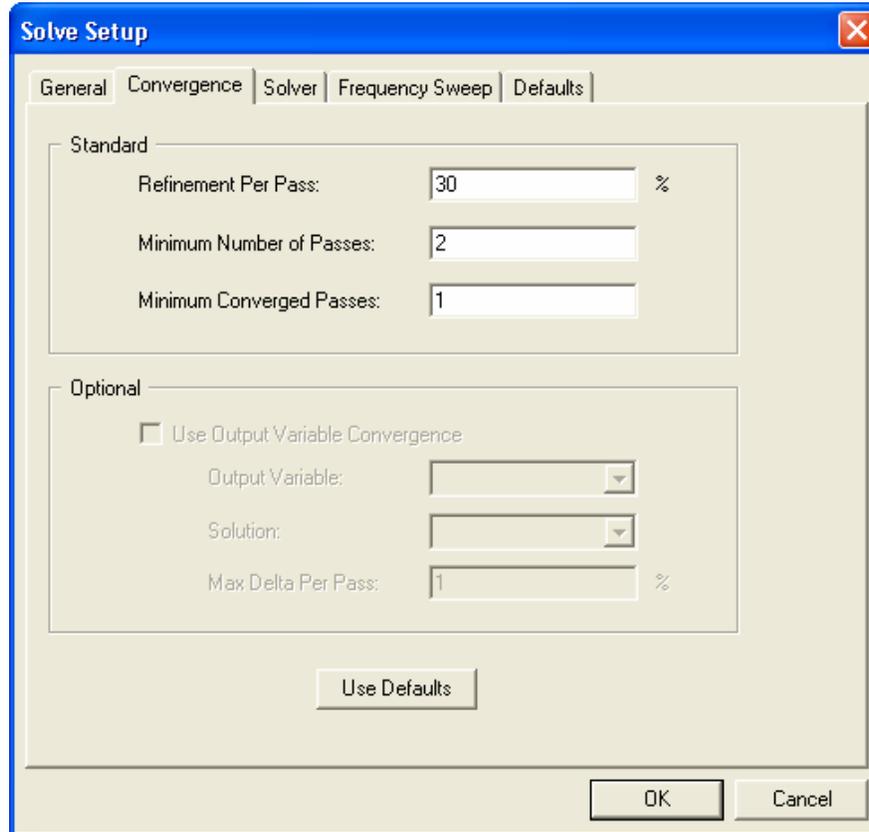
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
- The following window appears with the General Setup parameters.



- 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)**

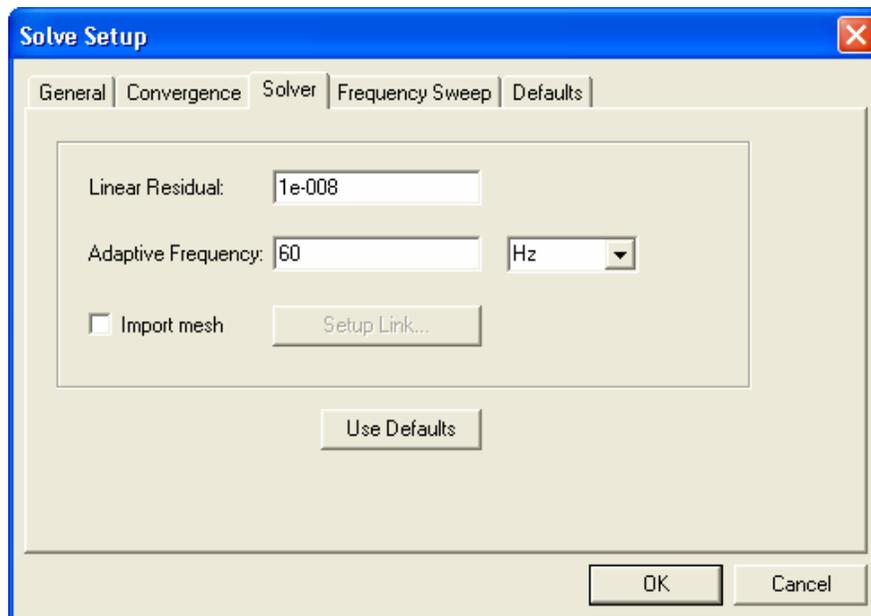
- ▲ The second tab of the Solution Setup contains information about Convergence.



- ▲ **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)

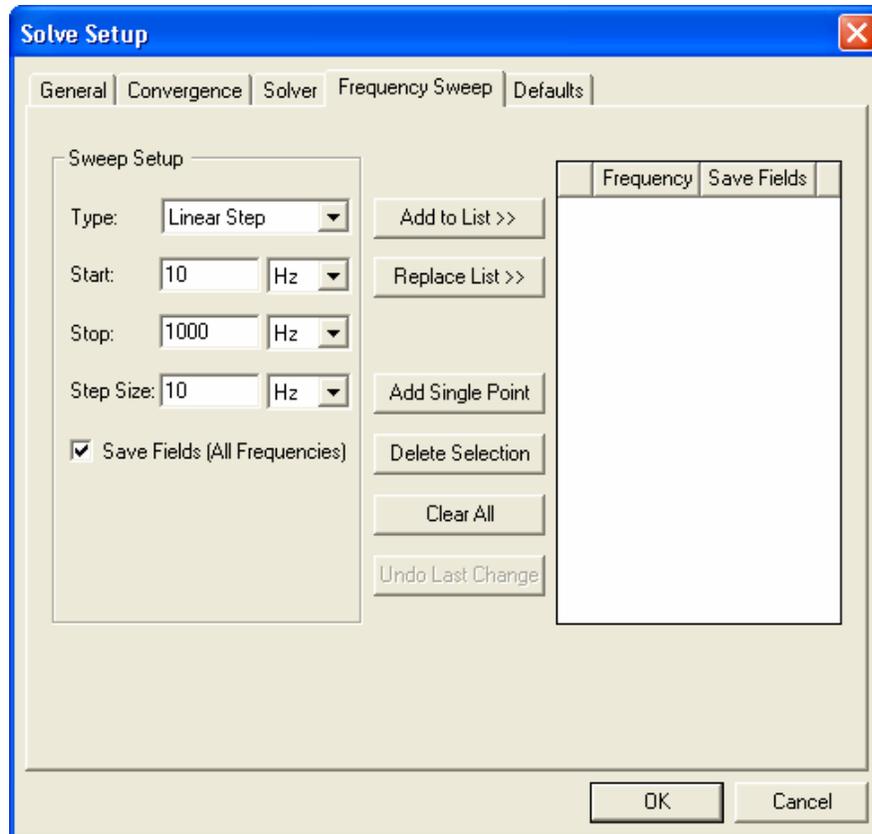
- The third tab of the Solution Setup contains information about the Solver.



- **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)

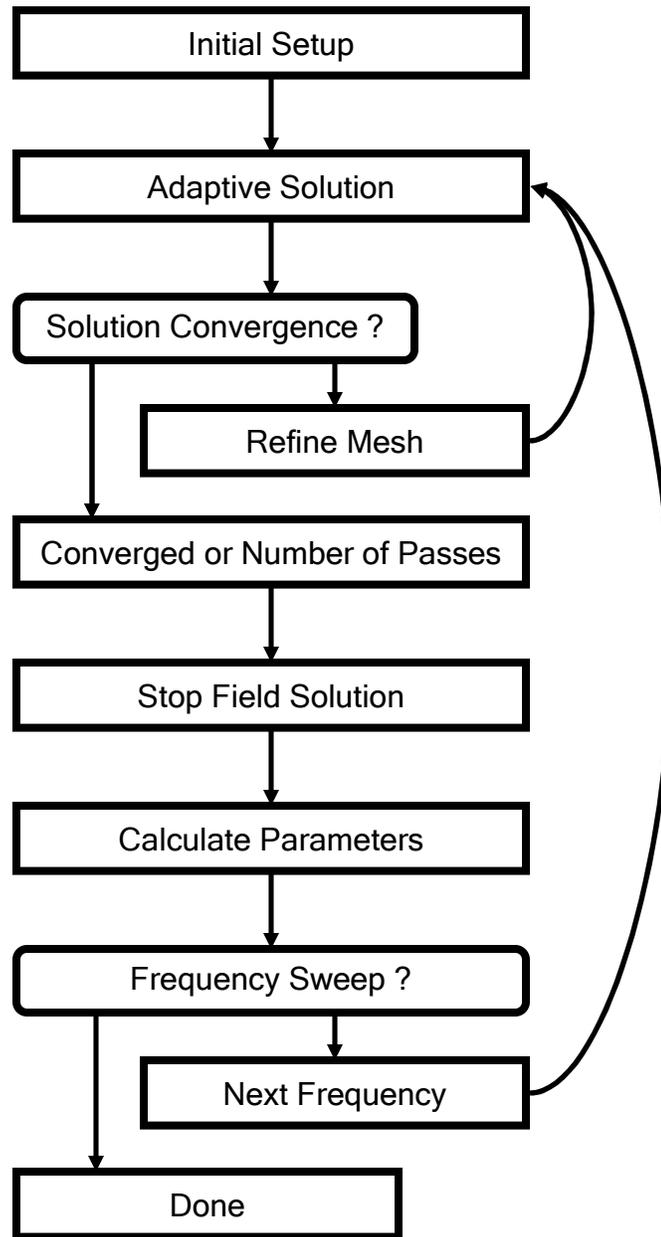
- The fourth tab of the Solution Setup contains information about an optional Frequency Sweep.



- 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.

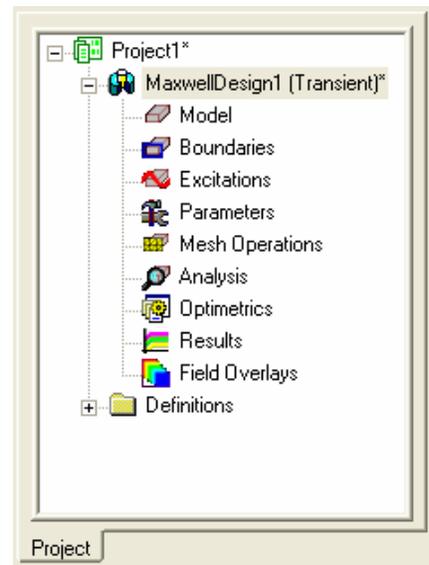
▶ Overview

- ▶ 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 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

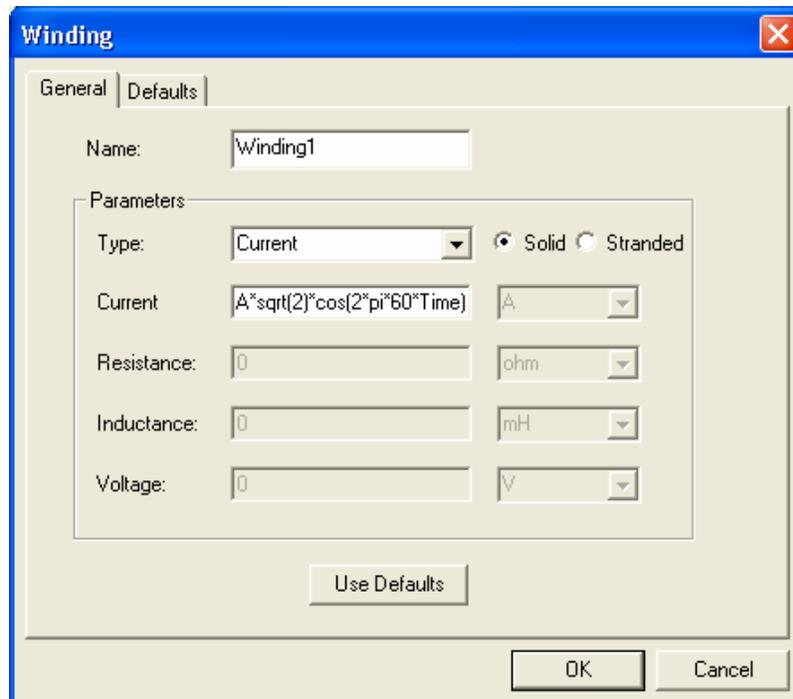
- ▀ **Windings** - Windings control the current flowing through the conductors and are therefore crucial to the magnetic solution. There are three different types of winding:
 - ▀ **Current** - defines a specified functional current through the conducting paths.
 - ▀ **Voltage** - defines a specified 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.

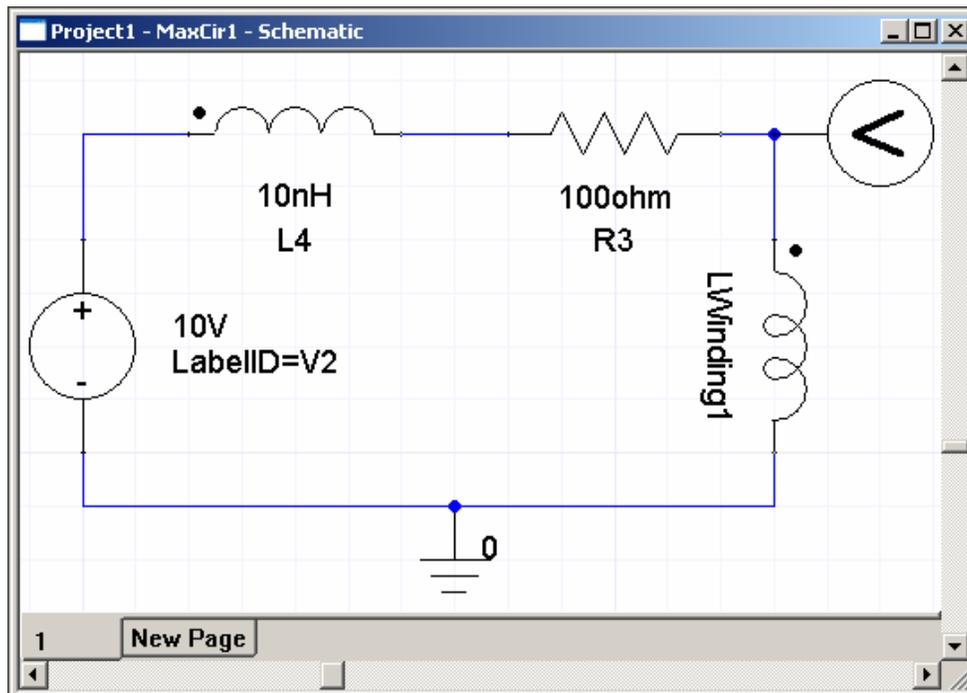


External Circuits

- Step 1: Open Maxwell Circuit Editor
 (*Windows Start Menu > Programs > Ansoft > Maxwell 11 > Maxwell Circuit Editor*)



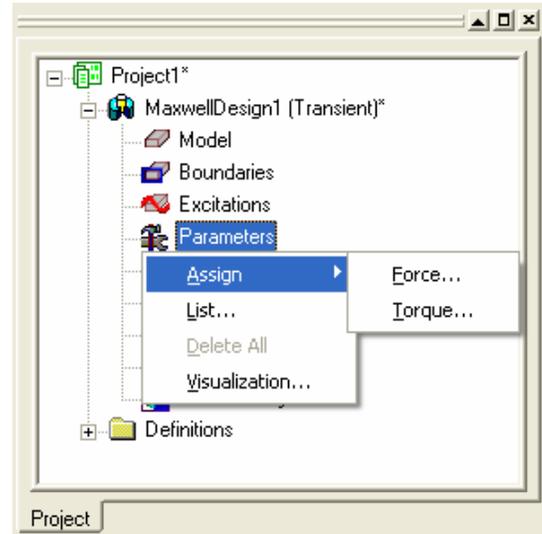
- Step 2: Create Circuit in Maxwell Circuit Editor Schematic
 - In the editor, each winding should appear as a Winding element, with a matching name for each winding - i.e. Winding1 in the transient simulation would require Winding1 in the schematic (displayed as LWinding1 on the schematic sheet).



- 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.

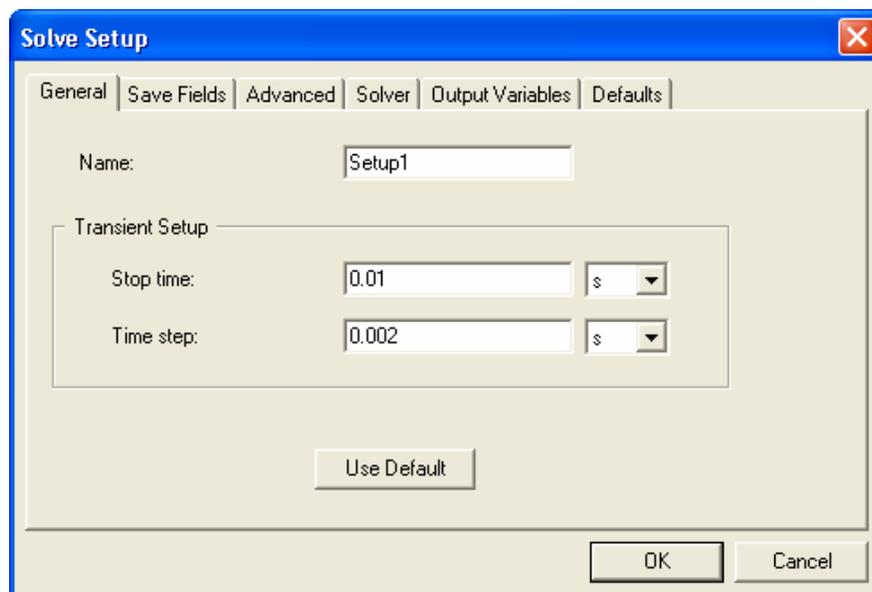


Mesh Operations

- ▲ 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

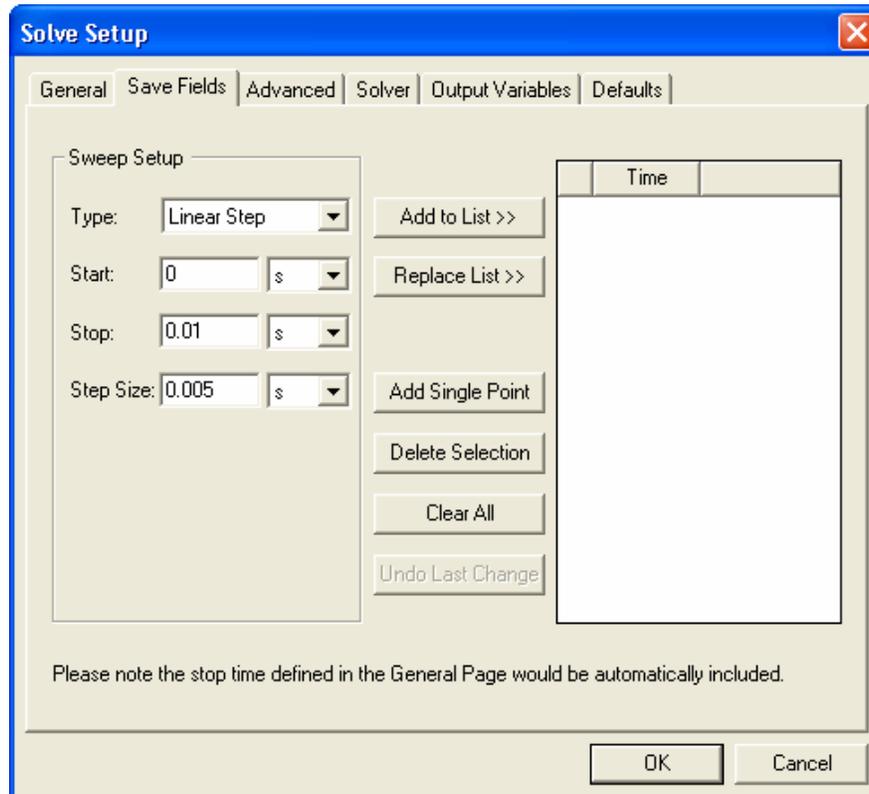
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
- The following window appears with the General Setup parameters.



- 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)**

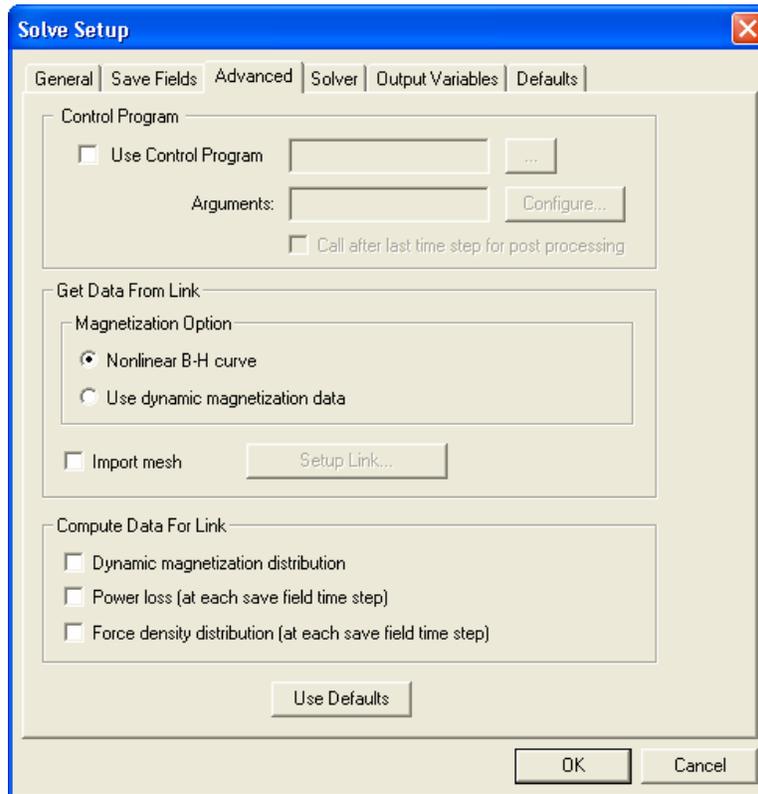
- ▲ The second tab of the Solution Setup contains information about Saving Fields.



- ▲ 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)

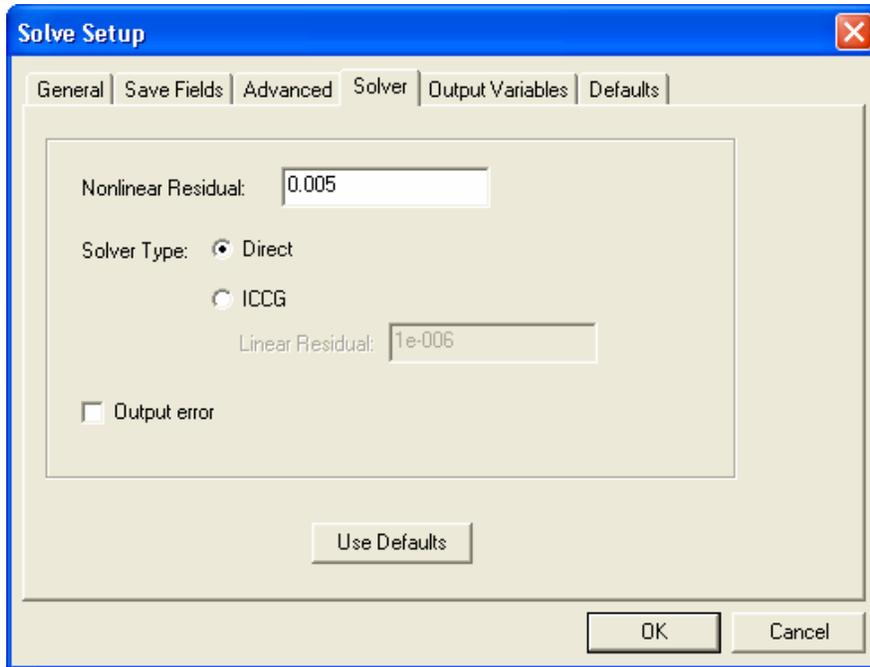
- The third tab of the Solution Setup contains Advanced options.



- 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)**

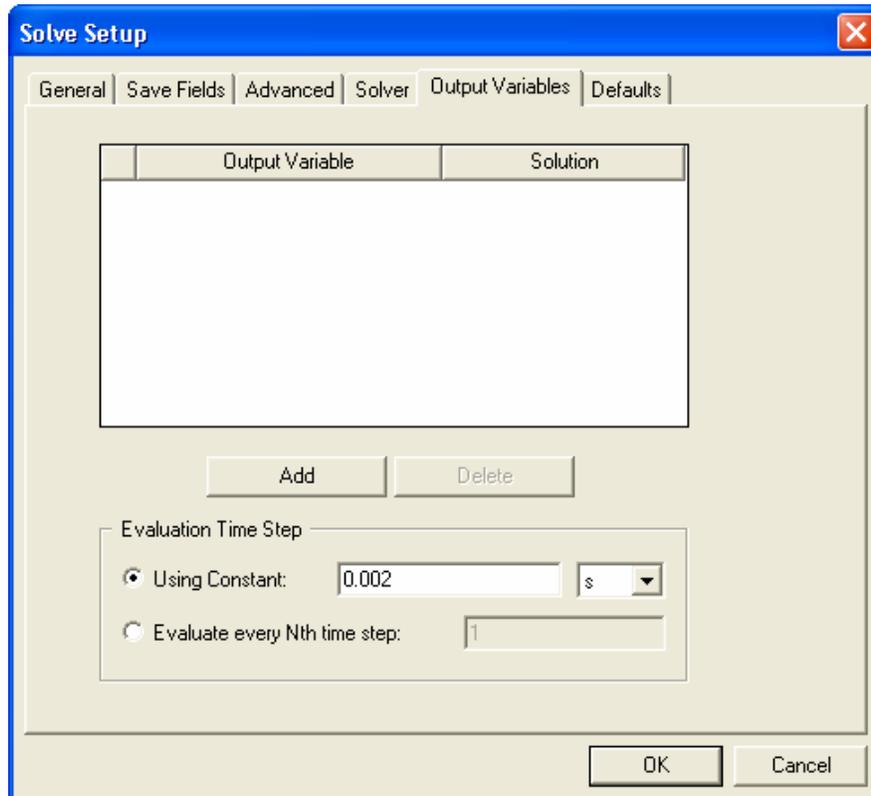
- ▲ The fourth tab of the Solution Setup contains information about the Solver.



- ▲ **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)

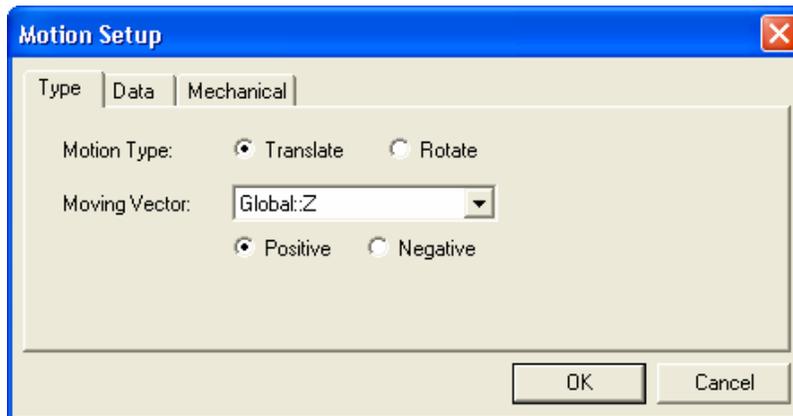
- The fifth tab of the Solution Setup contains Output Variable options.



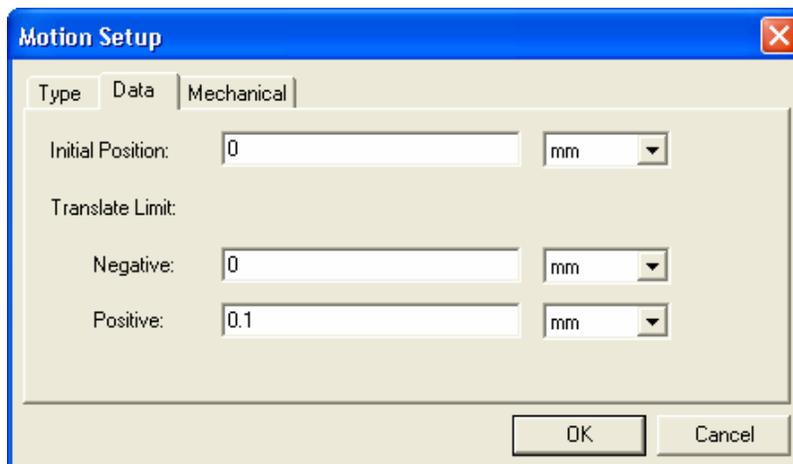
- Output Variables** are added to the list by selecting **Add** then selecting the available output variables from the pull down list.
- This output variable option requires that you first define an output variable before adding it (however, you often need to define the analysis setup before you can 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.

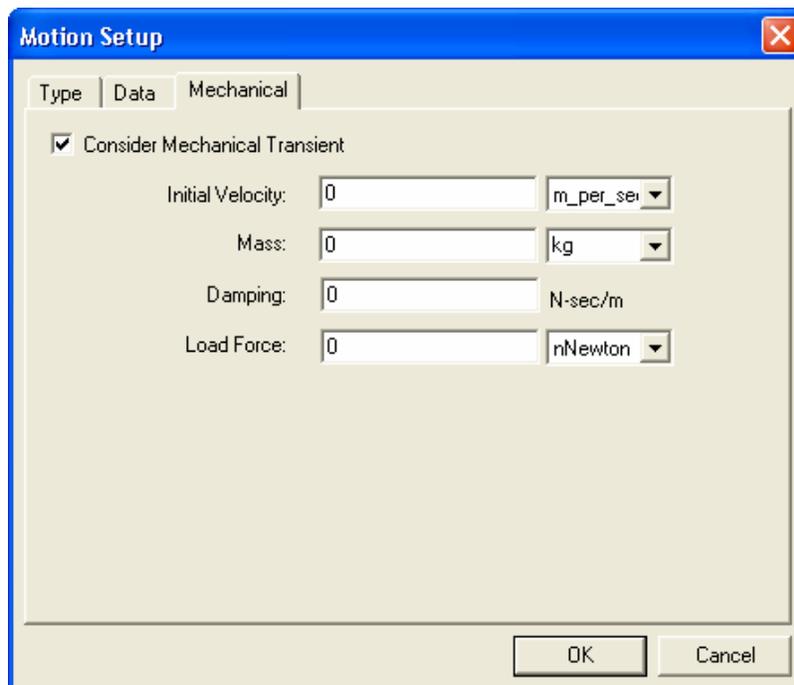


▄ Motion Setup (Continued)

- ▄ 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 - 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).

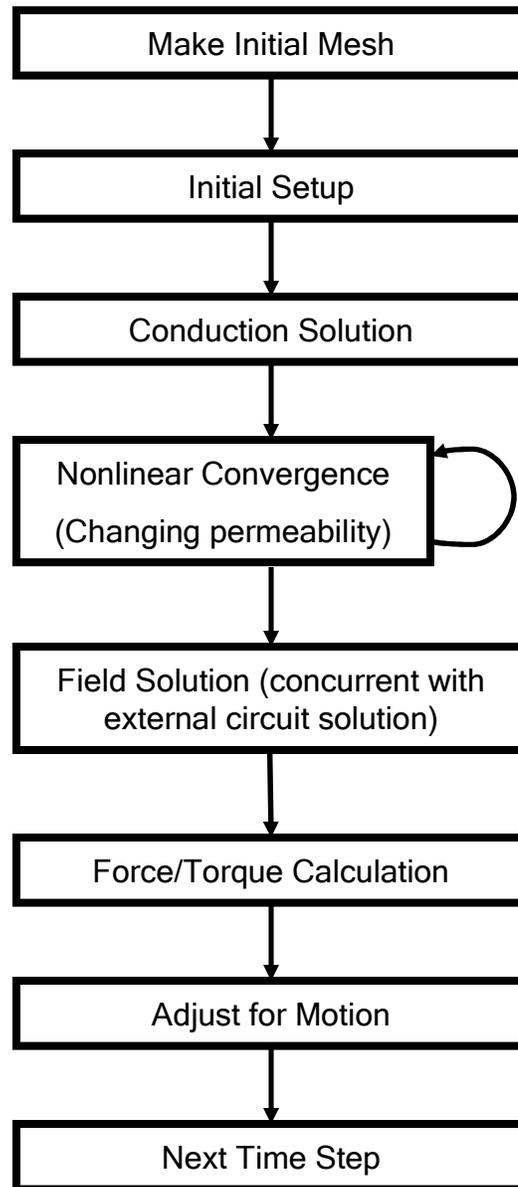
-  When conceptualizing cylindrical rotational motion, remember that the band object and all moving parts rotate without remeshing between.

-  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

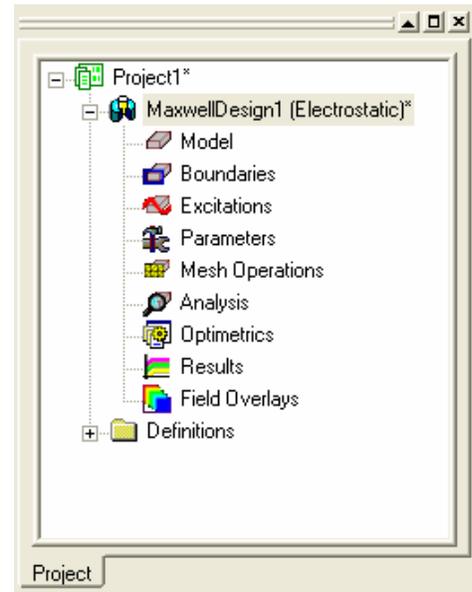
- ▣ The electrostatic solver computes static (DC) electric fields.
- ▣ All objects are stationary.
- ▣ The source of the static magnetic field can be:
 - ▣ Applied potentials.
 - ▣ Charge distributions.
- ▣ The quantity solved is the electric scalar potential (ϕ).
- ▣ Electric Field (E) and Electric 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.

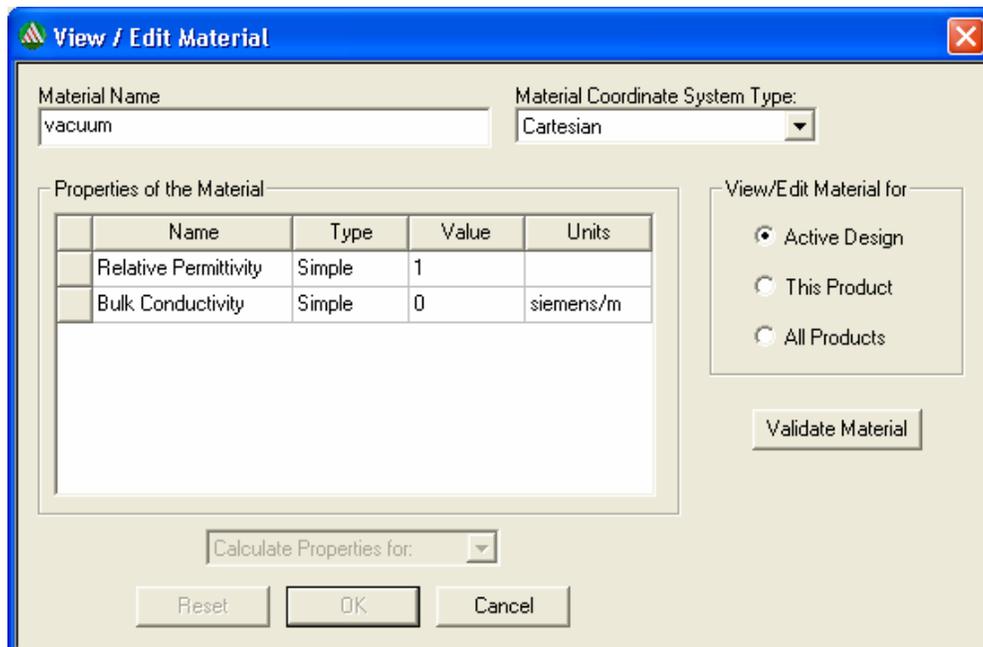
- ▲ The Model definition refers to the geometry and material definition.
- ▲ Boundaries and Excitations refer to the specific boundaries and excitations available in an electrostatic simulation.
- ▲ Parameters are values that the solver will automatically calculate after finding the field solution.
- ▲ 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.



❖ 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

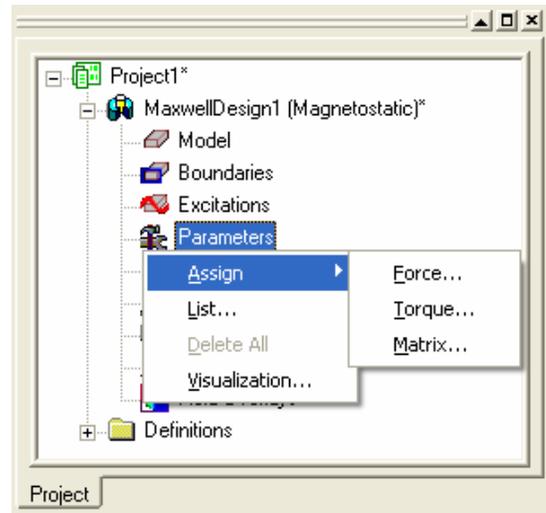
- ❖ **Master/Slave** - This boundary condition is also known as a matching boundary condition because it matches the electric field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this 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).



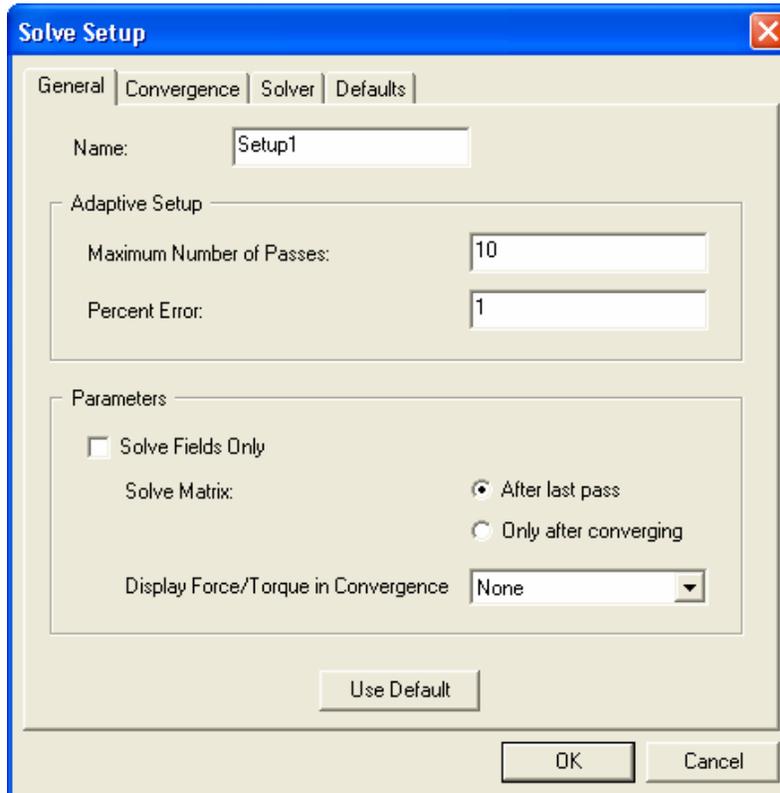
- The results of any parameters can be found by selecting **Maxwell > Results > Solution Data...** or by clicking on the  icon.
- Further results can be obtained manually through the field calculator.

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

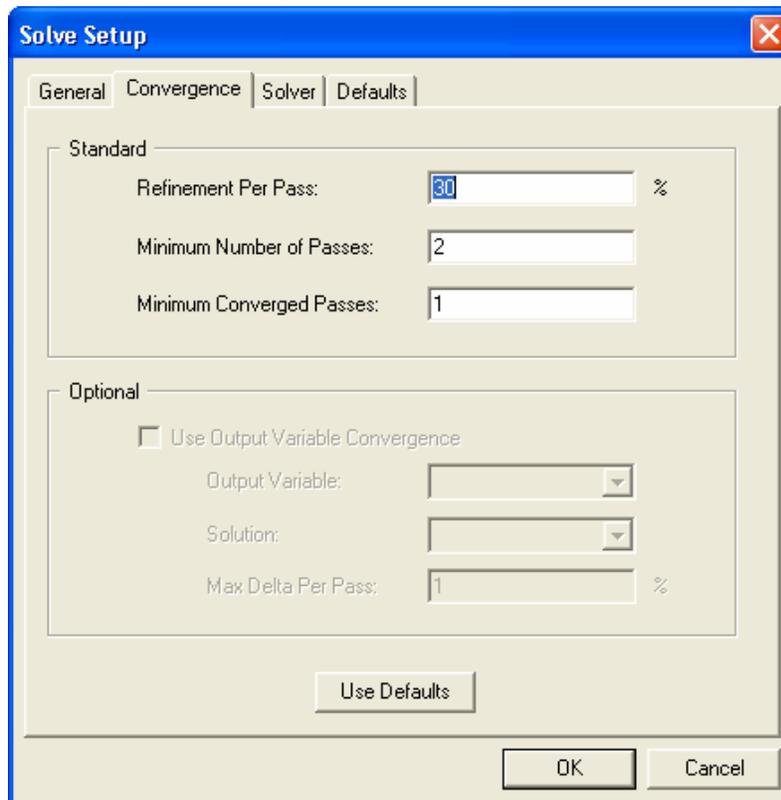
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
- The following window appears with the General Setup parameters.



- 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)

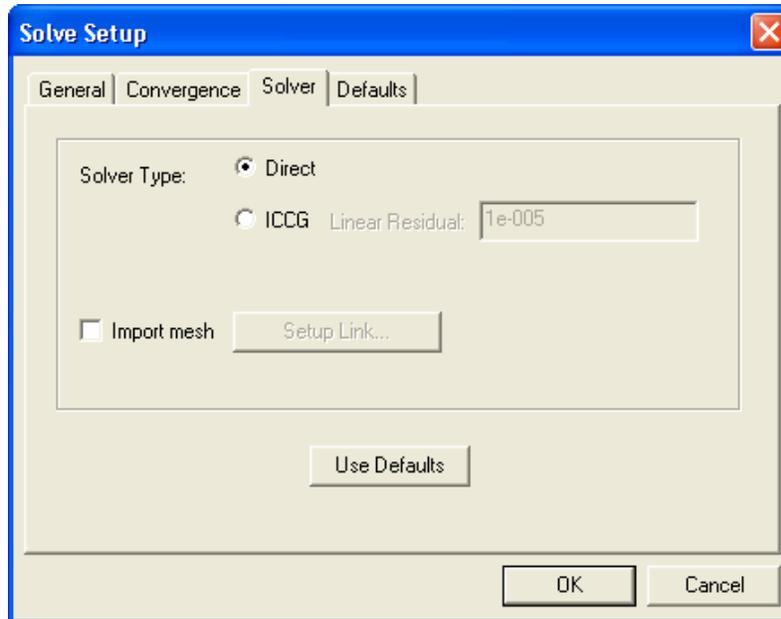
- ▲ The second tab of the Solution Setup contains information about Convergence.



- ▲ **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)

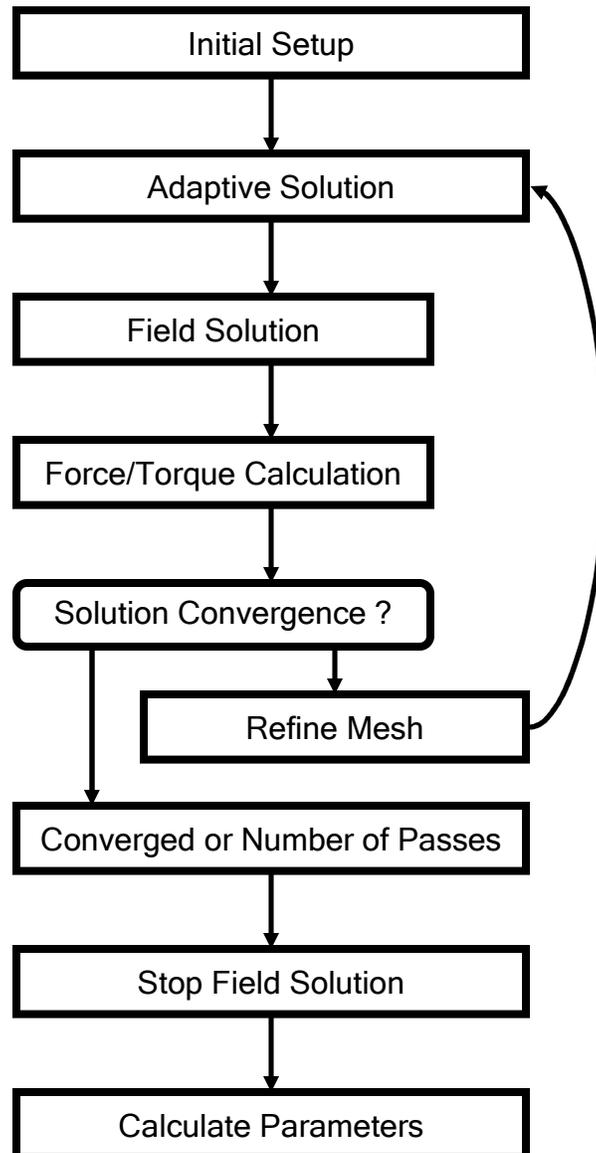
- The third tab of the Solution Setup contains information about the Solver.



- **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.

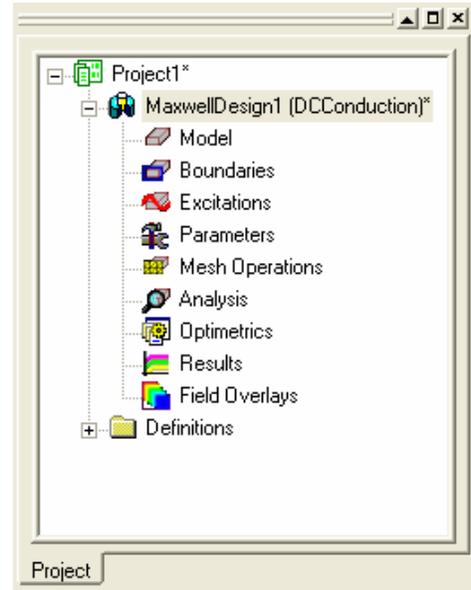
▀ Overview

- ▀ 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.

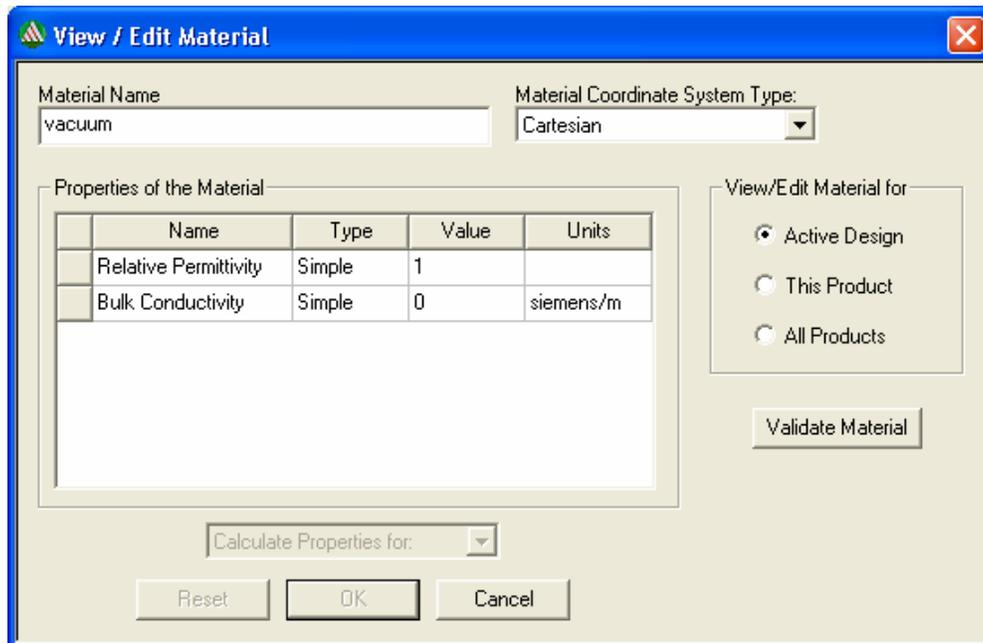


- ▲ These options are displayed in an order that can be followed in creating a new DC Conduction simulation. This is a general purpose order that goes linearly through simulation setup, analysis, and post-processing.

- ▲ 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.



❖ 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

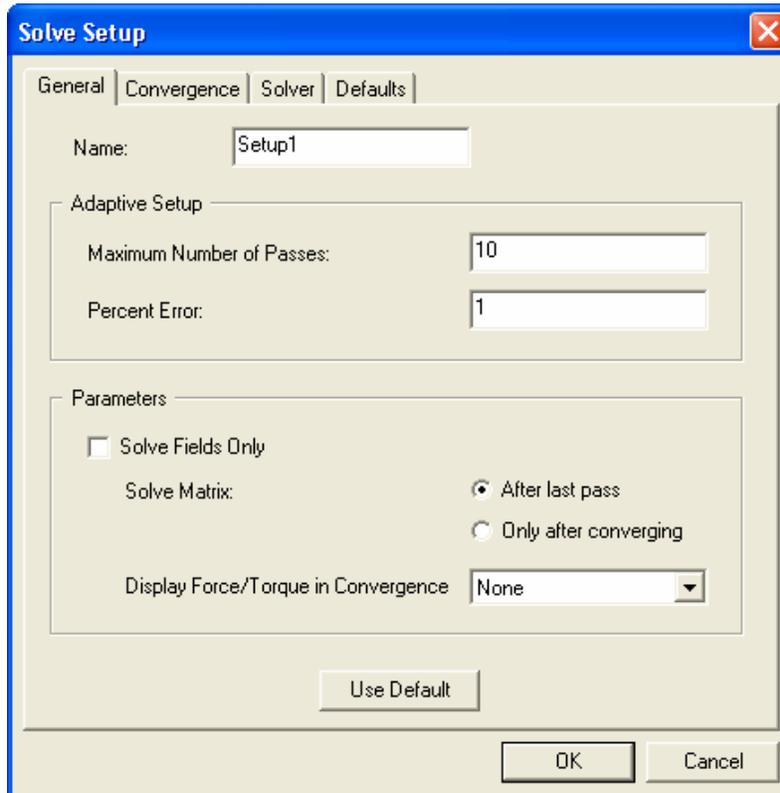
- ▀ Typical sources for DC current flow problems are currents applied on surfaces of conductors and voltages (electric potential applied to surfaces of conductors). The direction of the applied current is either “in” or “out”, always normal to the respective surfaces. Multiple conduction paths are allowed. 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

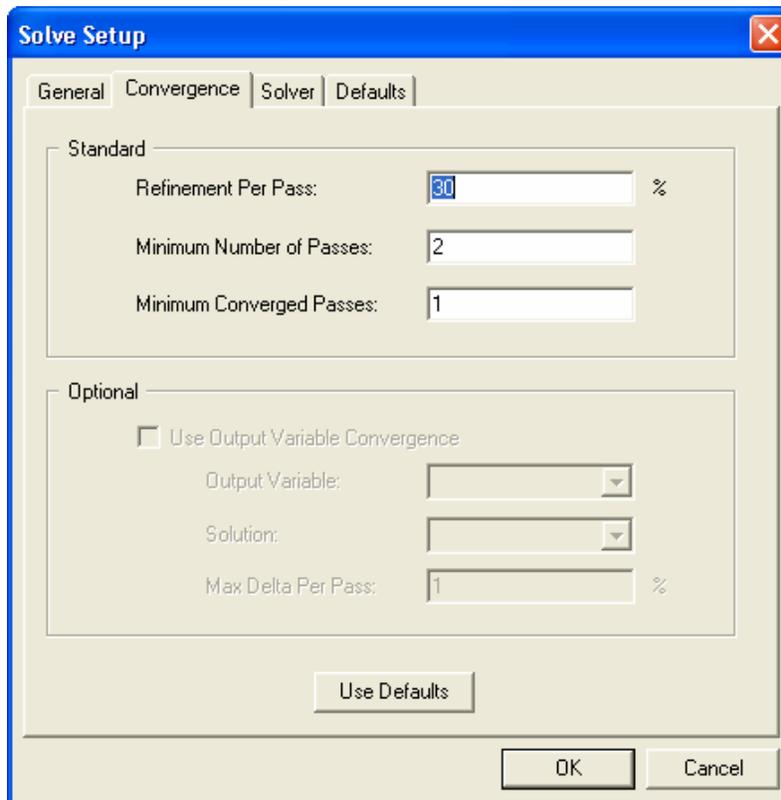
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting **Maxwell > Analysis Setup > Add Solution Setup...** or click on the  icon.
- The following window appears with the General Setup parameters.



- 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)**

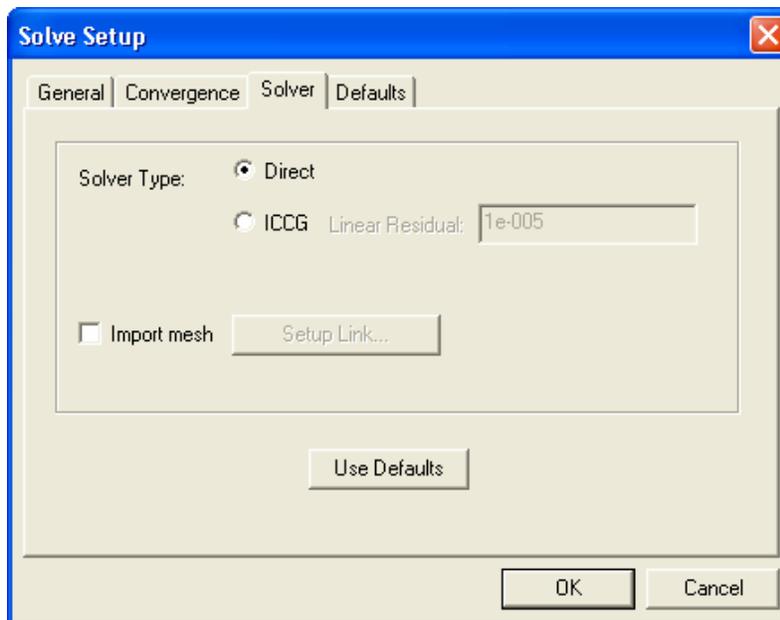
▲ The second tab of the Solution Setup contains information about Convergence.



- ▲ **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)

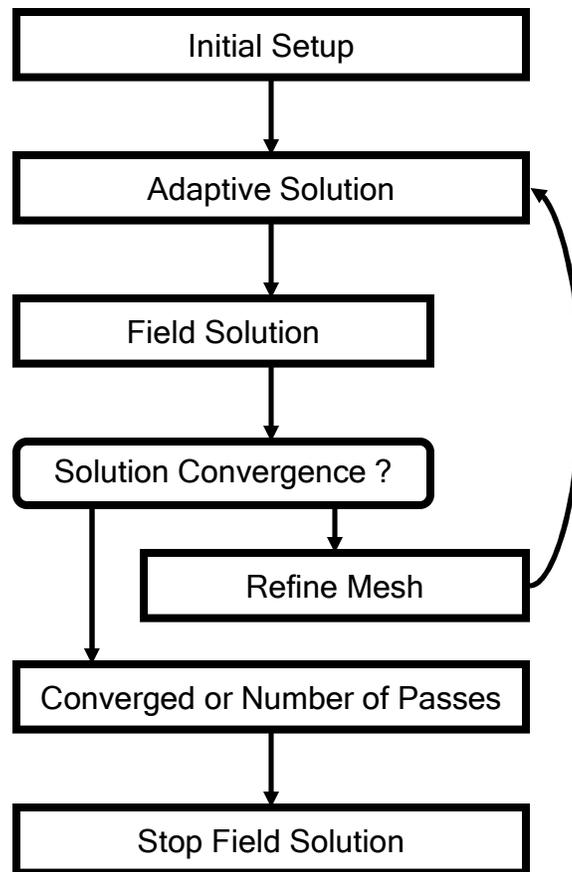
- ▲ The third tab of the Solution Setup contains information about the Solver.



- ▲ **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

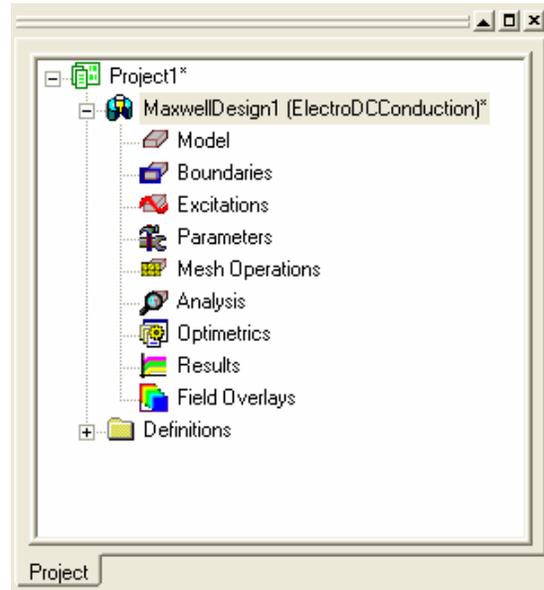
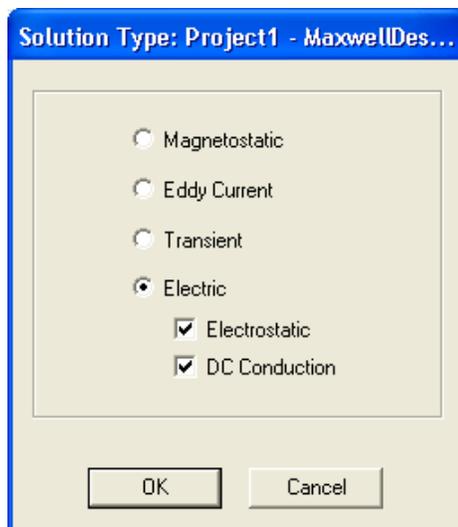
- ▀ Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.



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

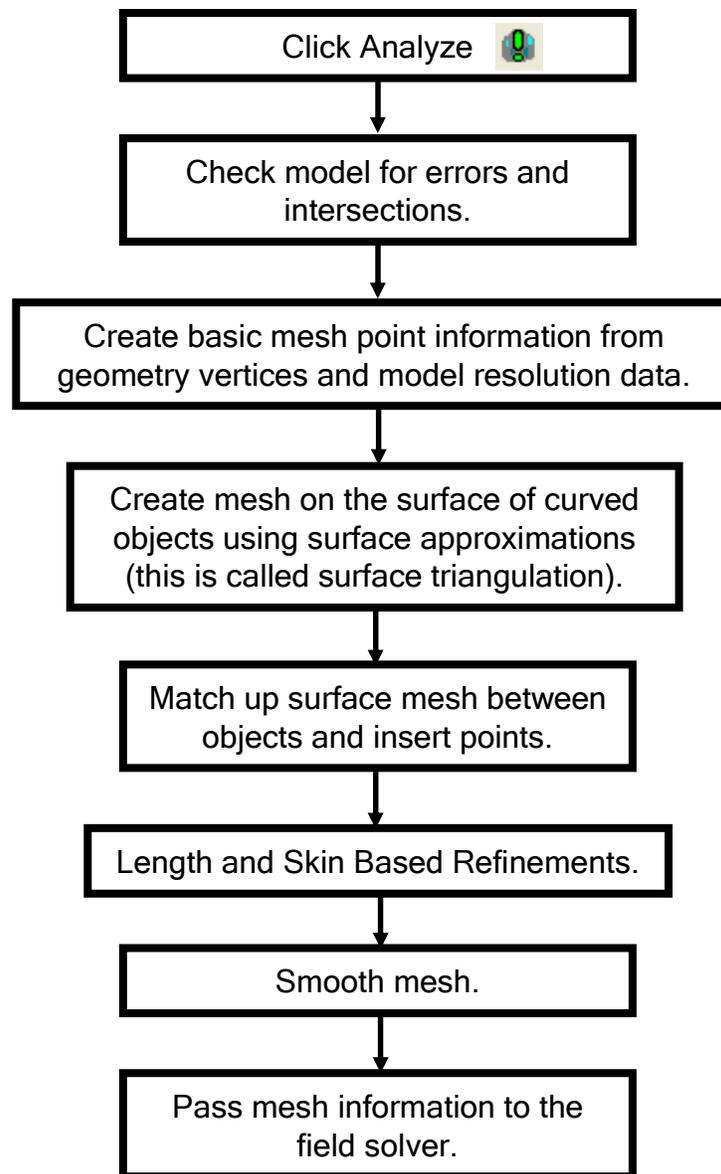
- Applying Mesh Operations
 - Surface Approximations
 - Curved Geometry Mesh Adaptation
 - Model Resolution
 - Length Based Mesh Operations
 - Skin Depth Based Mesh Operations

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

- Mesh failures and suggestions

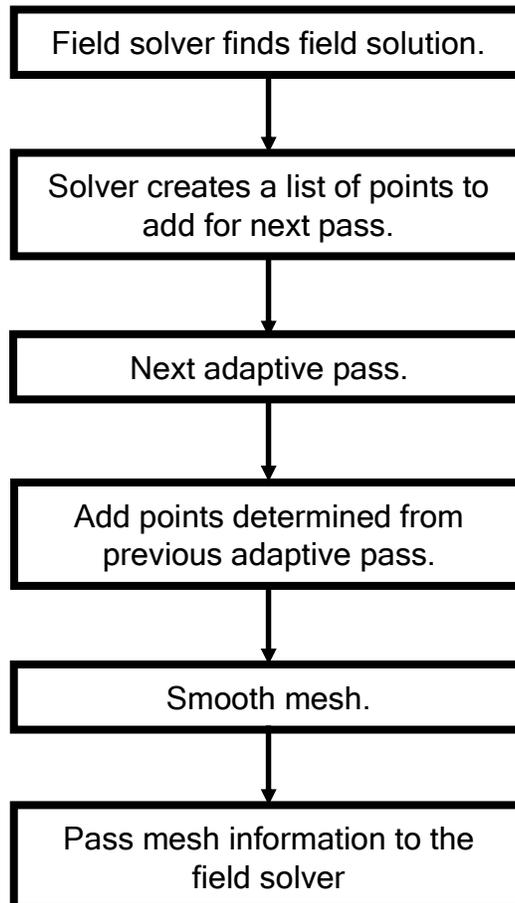
Initial Mesh Process

- 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**.

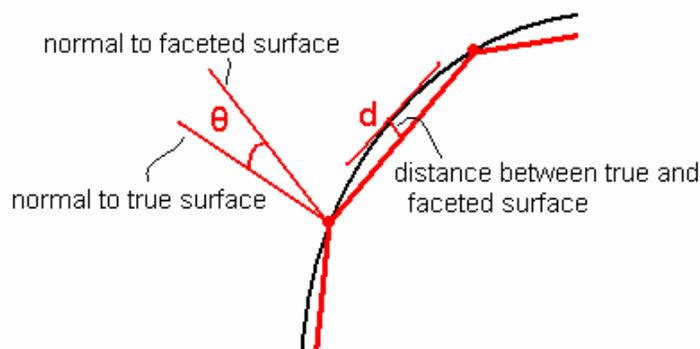
▀ Applying Mesh Operations (Continued)

▀ 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.

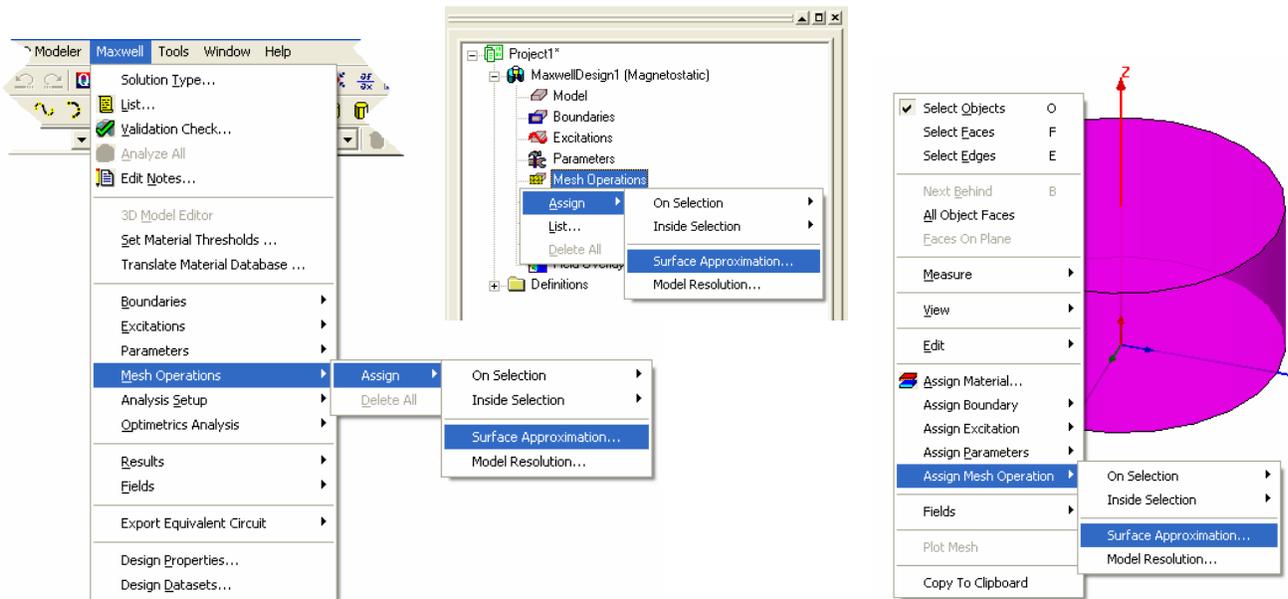
Surface Approximations

- ▲ 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 ".



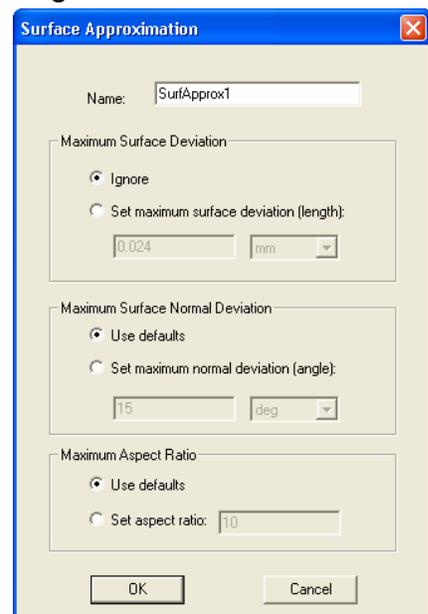
Surface Approximations (Continued)

- 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.

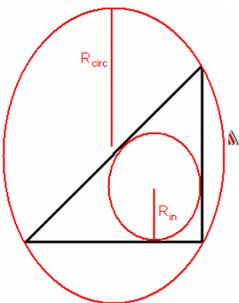


Surface Approximations - Usage Suggestions

- ▲ **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.

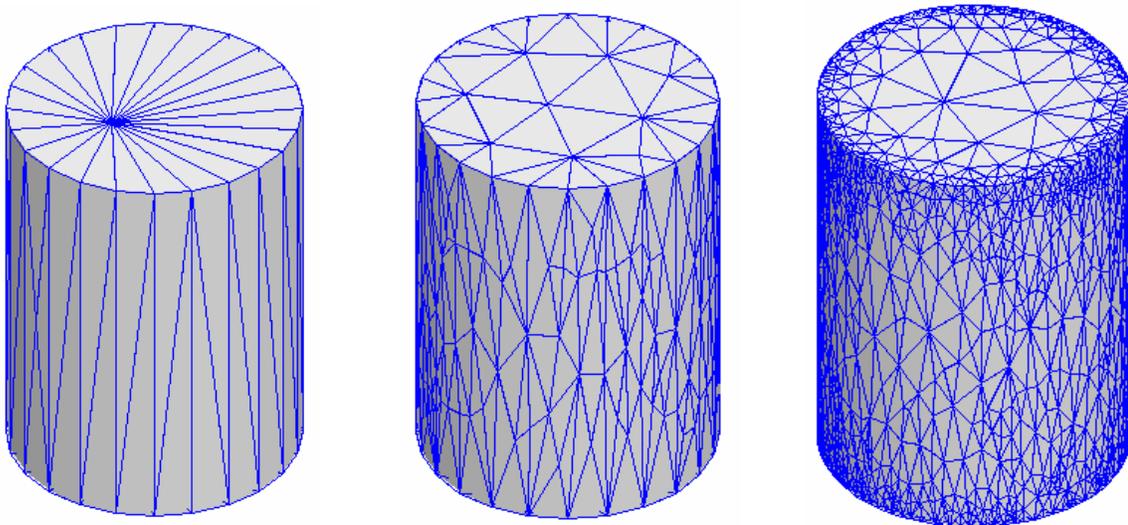
Surface Approximations - Aspect ratio comments

- ▲ 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

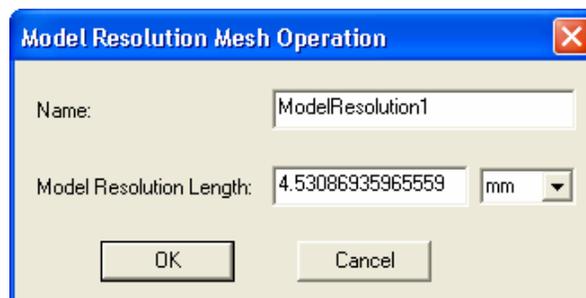
- ▲ The new graphical drawing interface encourages the use of true-curved drawing by removing the option to assign a facet count to the construction of primitives such as circles, cylinders, spheres, and ellipses.
 - ▲ Faceted primitives are available however as polyhedrons and polyhedral solids, if 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

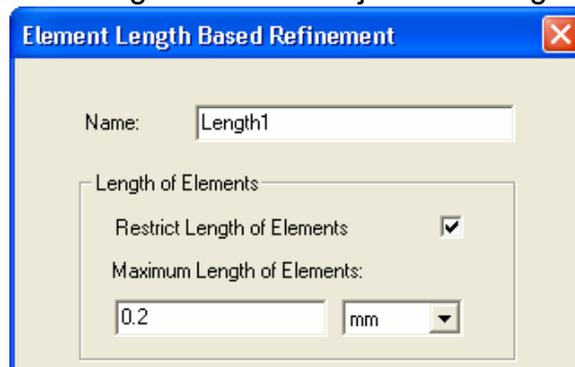
- This is part of a complex design that benefited from model resolution. The grey outline is the reduced model. The pink lines were the original model details that were removed due to the mesh operation. The original model had an initial mesh of 184,675 tets. After model resolution, the initial mesh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

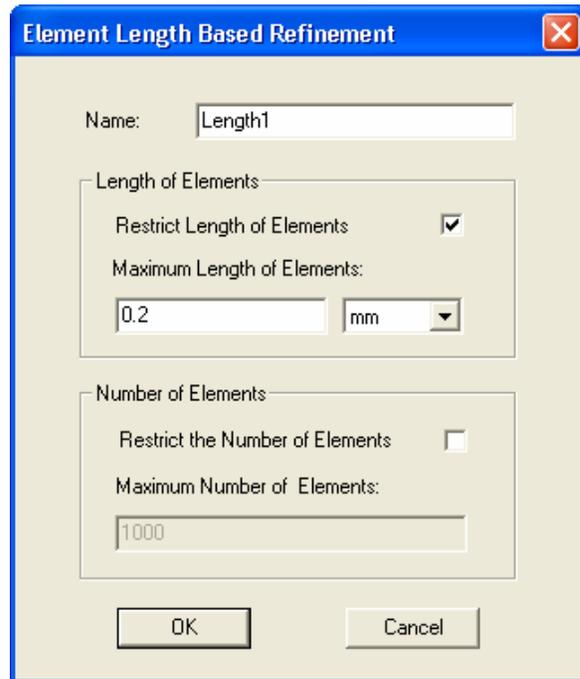
There is one other option that is provided when assigning a length based mesh operation that will limit the number of elements that are added into the simulation with that particular mesh refinement.

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).

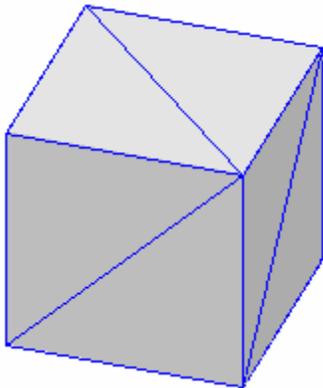
This second restriction brings another possible method of adding elements to objects without regard to the dimension of the object - you may uncheck the Length of Elements restriction and enter a value for the Number of Elements restriction, thereby adding a defined number of elements to the simulation.

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).

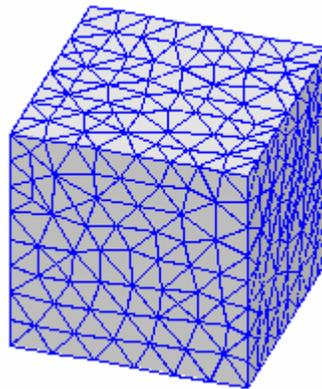


▀ Length Based Mesh Operations - Example

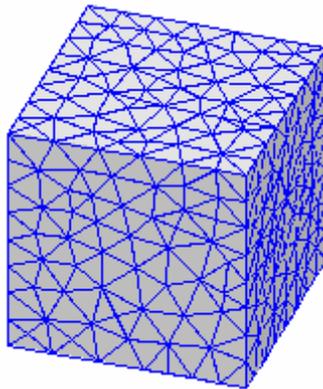
- ▀ This example uses a cube inside a Region with 100% padding. An initial mesh was created for four different cases of applying length-based mesh operations.
- ▀ First, a default initial mesh was created without defining any mesh operations.
- ▀ Second, an initial mesh was created by defining a length-based 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.



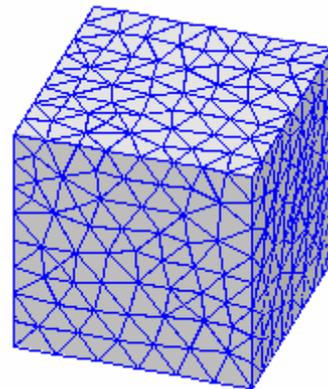
Default Initial mesh
42 elements



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.

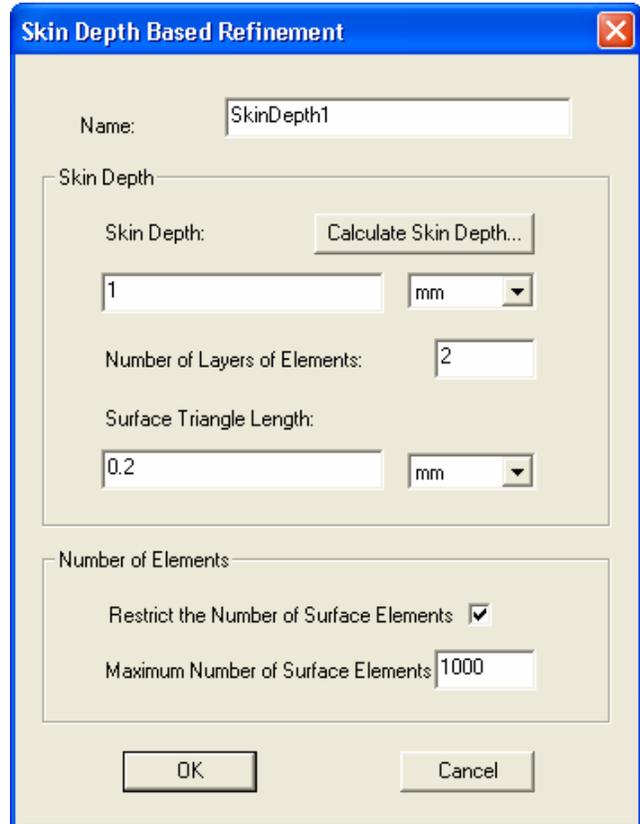
- This refinement method creates layers of mesh within the selected surfaces of objects - this is useful for modeling induced currents near the surface of a conducting object.

- 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.

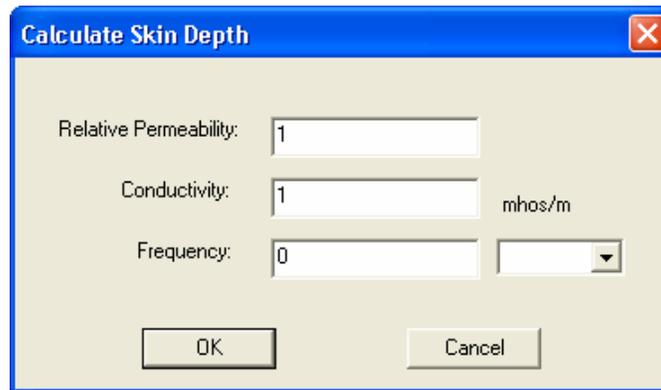
- The **Surface Triangle Length** is equivalent to the surface length restriction for length-based refinement.

- 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.



▲ 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 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.

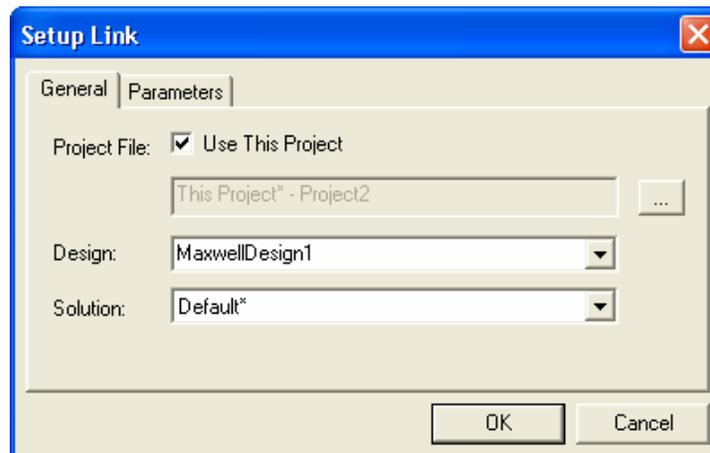
-  Some example places to use dummy objects are the following:
 -  In the gap of a magnetic core.
 -  As a container object for subgroups of objects (such as the inner rotor of an rotational machine).
 -  As a container object for all objects.
 -  Around a sensor.
 -  Around a surface that will be integrated 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.

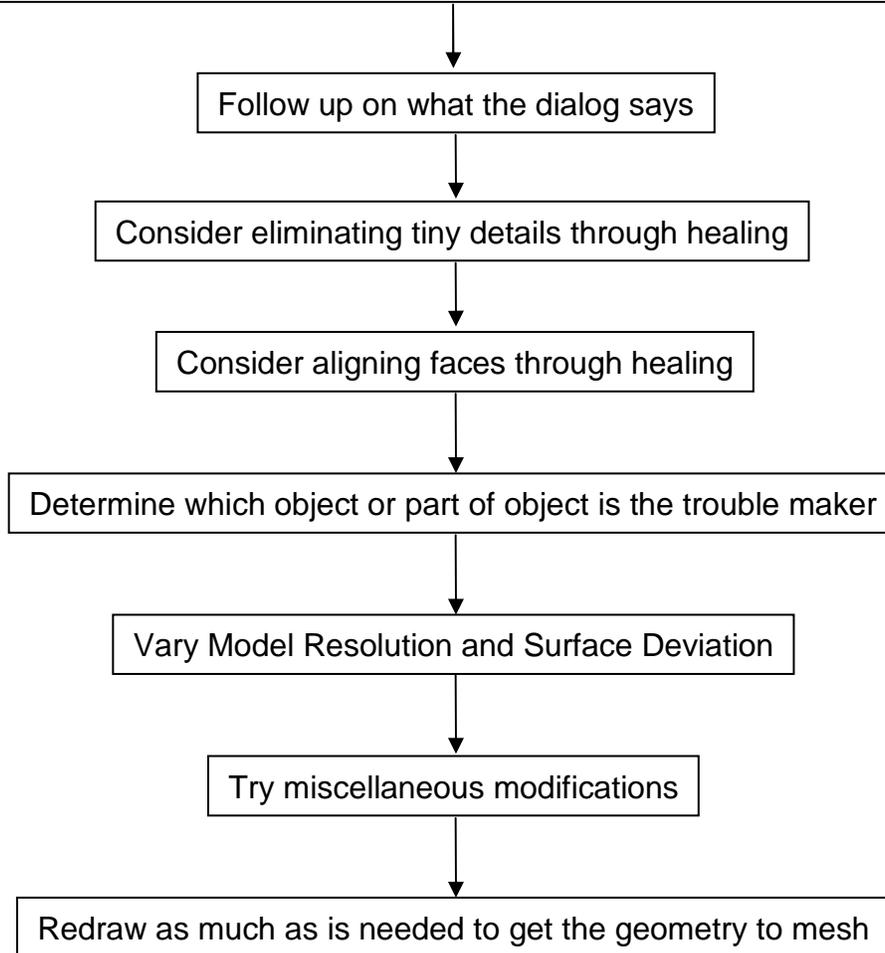
-  To re-initialize the linked data, select **Maxwell > Analysis Setup > Clear Linked Data** or right-click on **Analysis** and choose **Clear Linked Data**.

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.

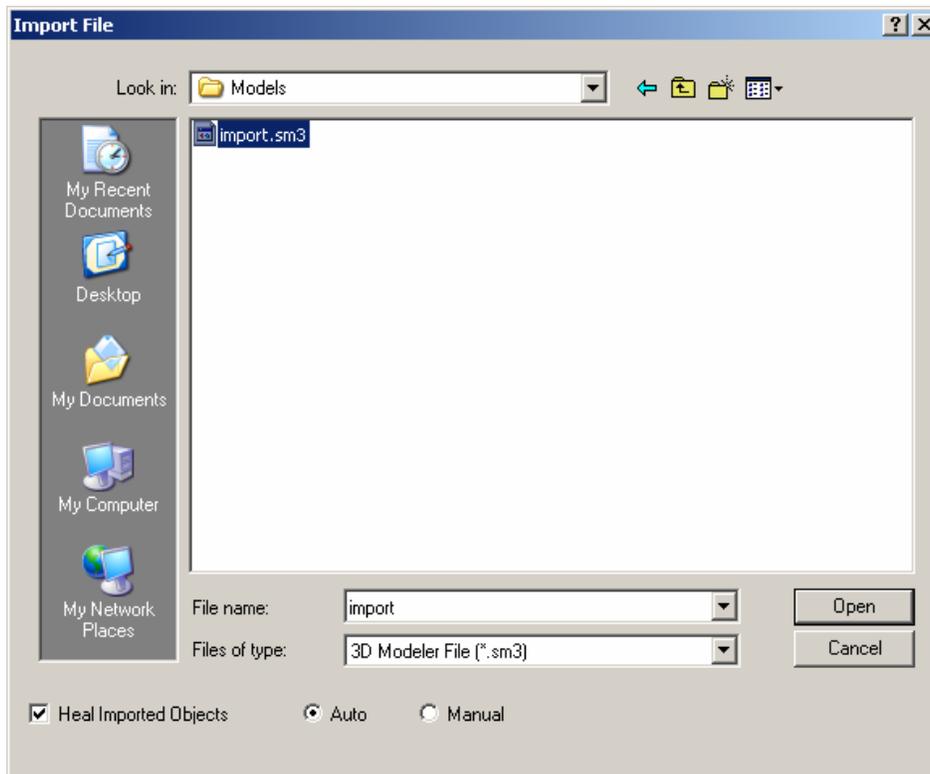
Mesh Failures and Suggestions (Continued)

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



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**

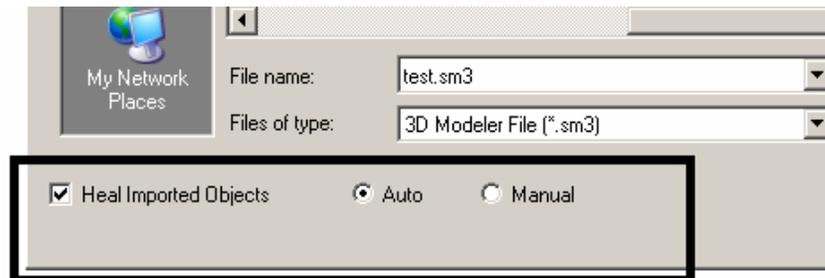
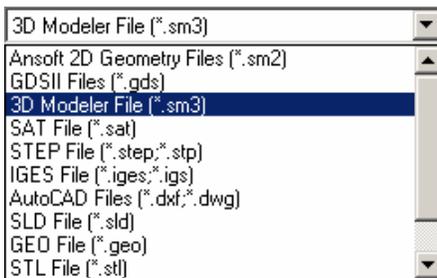
- **Analysis**

- **Validation Check**

Geometry Import/Healing

Importing

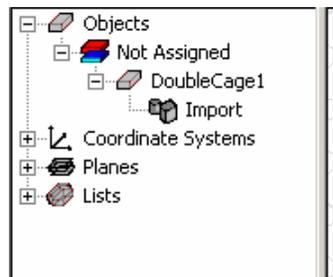
1. To import a model, select **3D Modeler > Import ...** .
 1. An **Import File** dialog pops up, allowing you to browse to the desired file.
-  On the left is a list of import files that we support. For some of these import options you will need an add-on translator feature in your license file.



-  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.

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.



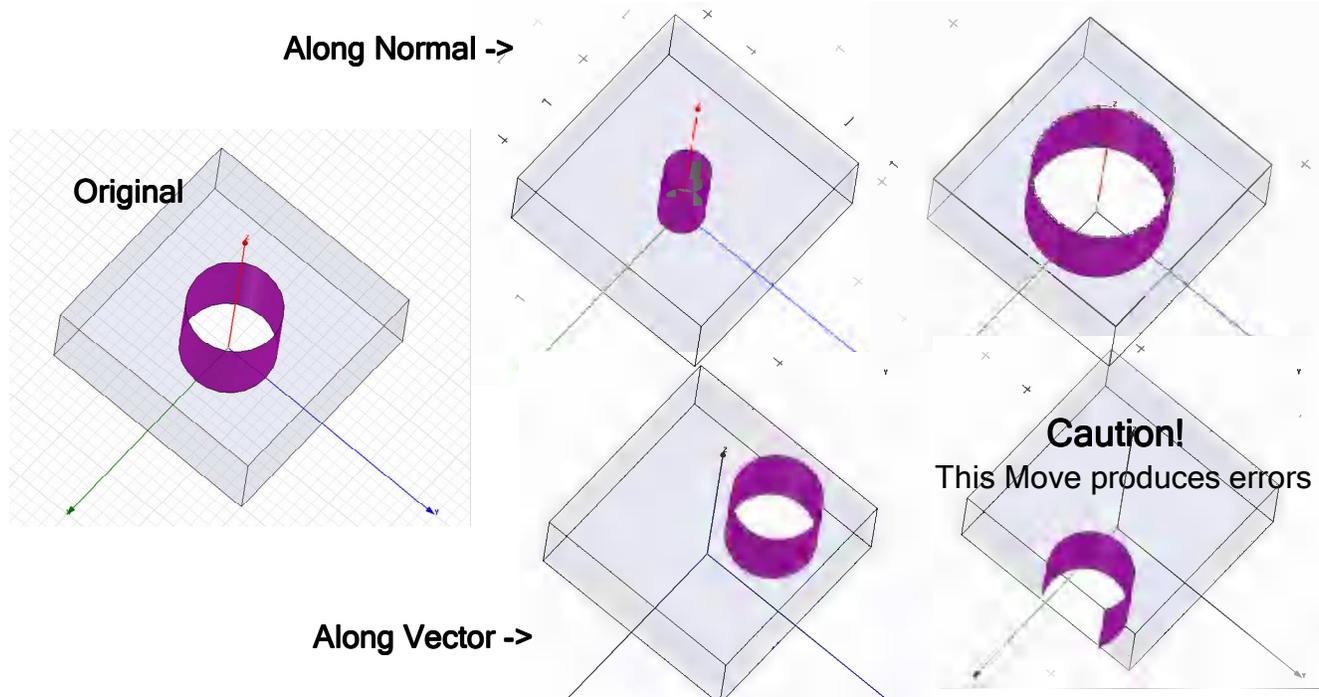
Suggestions for Exporting Geometries from Drawing Tools

- ▲ 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.



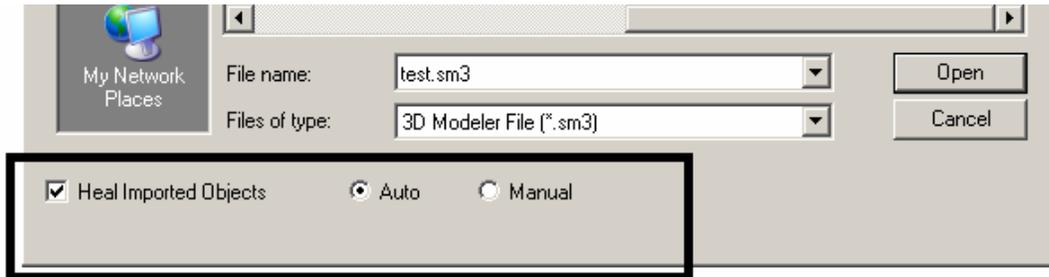
Common Model Errors

-  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



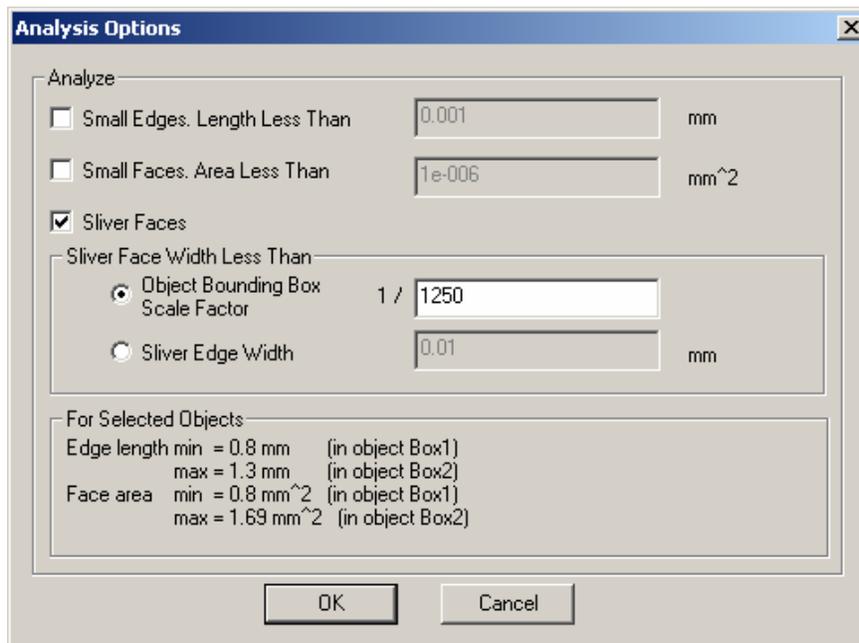
- ▲ In the Import File dialog there is a check box “**Heal Imported Objects**”
- ▲ This will heal small errors that occur when converting the imported file from its original format to the format of the 3D Modeler (ACIS SAT v14.0).
- ▲ There are two modes for healing the imported object - “auto” and “manual”.
 - ▲ 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.

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

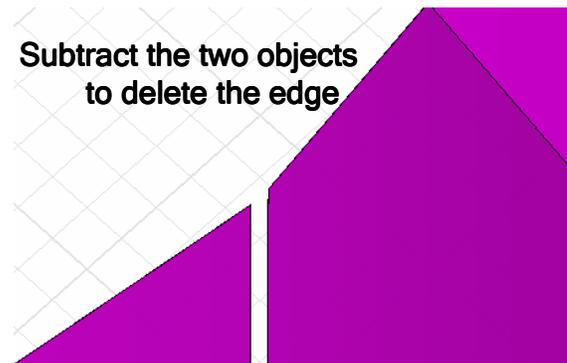
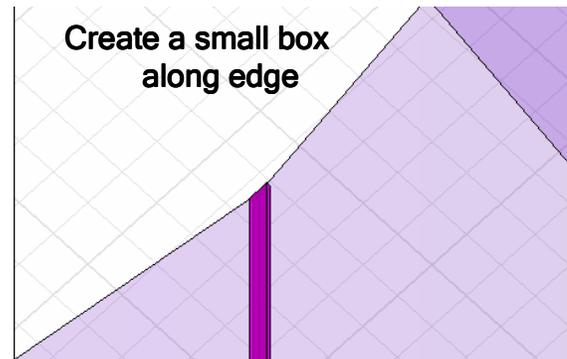
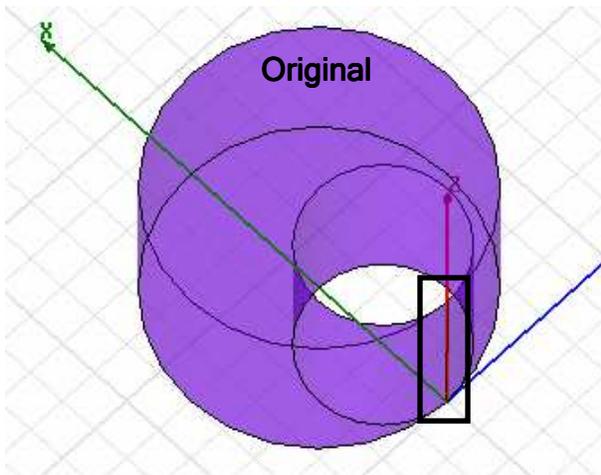
- ▲ The objects that fail api_check_entity() should be analyzed via the Analyze Objects menu item.
 1. Select the objects that have ACIS errors.
 2. Select **3D Modeler > Model Analysis > Analyze Objects**.
 3. The **Analysis Options** dialog box appears with options for small feature detection.



4. Select **OK** to proceed.

Fixing non-manifold errors

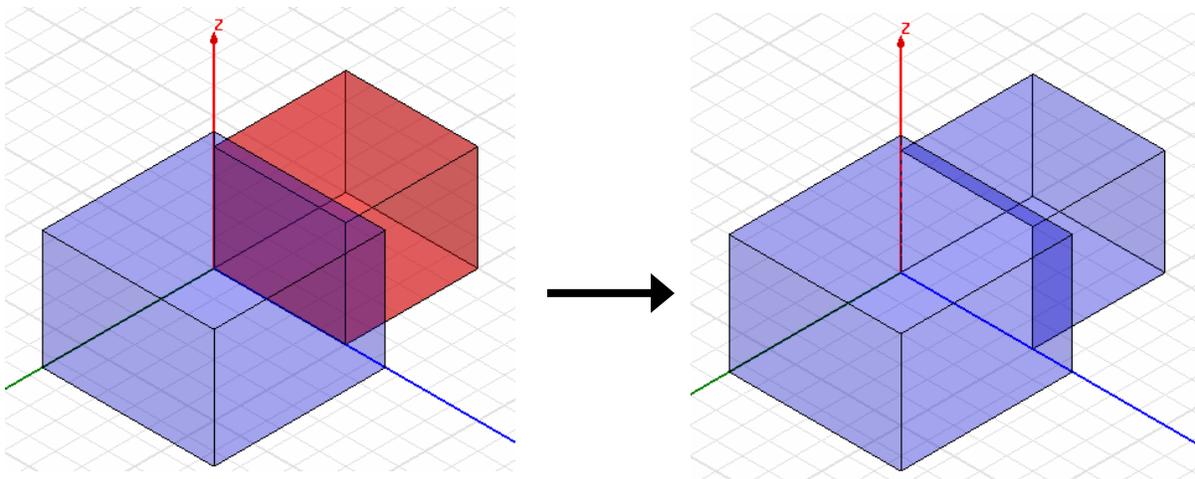
- ▶ The main strategy for fixing non-manifold errors is to slightly adjust the geometry to either add thickness to the object or delete the non-manifold edge.
- ▶ Find the non-manifold face/edge/vertex either visually or with the Analyze Objects menu item.
- ▶ Create a small box to contain the non-manifold 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

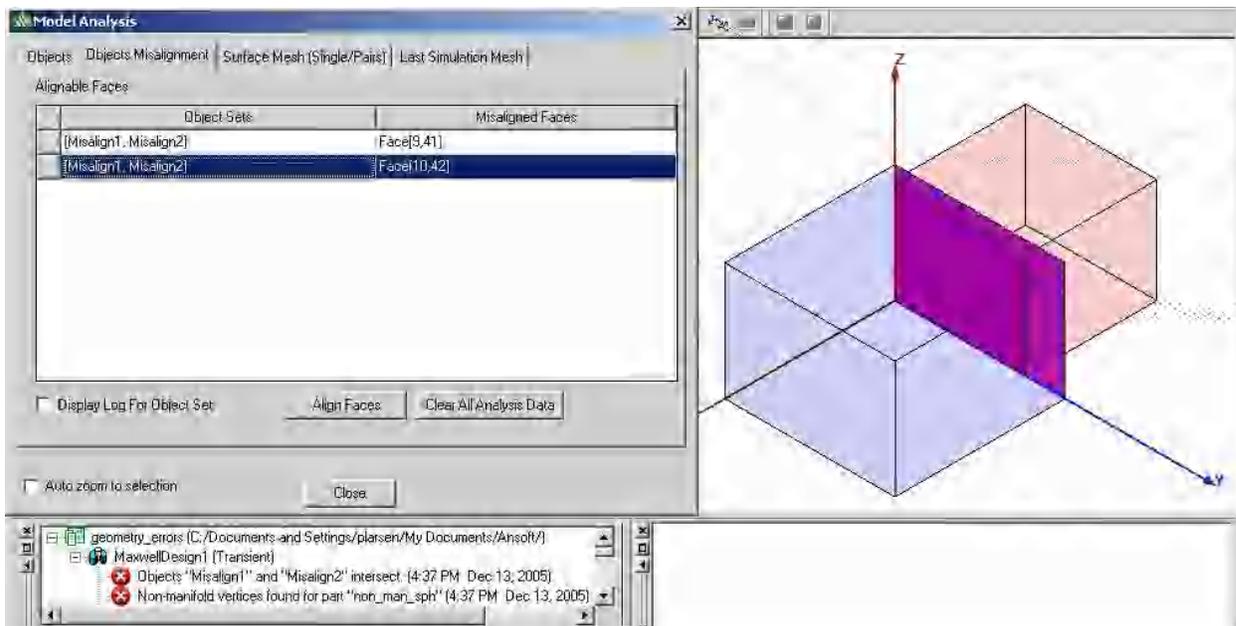
- ▲ This type of error is easily seen in a Validation Check, where the intersecting objects are designated with an error notification in the message window.
- ▲ There are several reasons why objects could be intersecting when they are imported. If care is taken with drawing and exporting the model from 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)

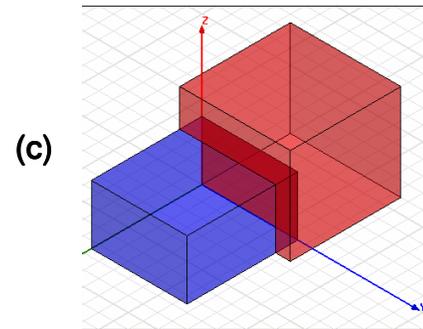
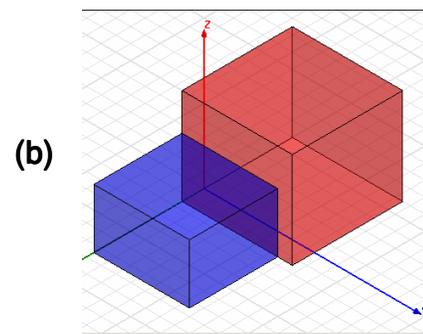
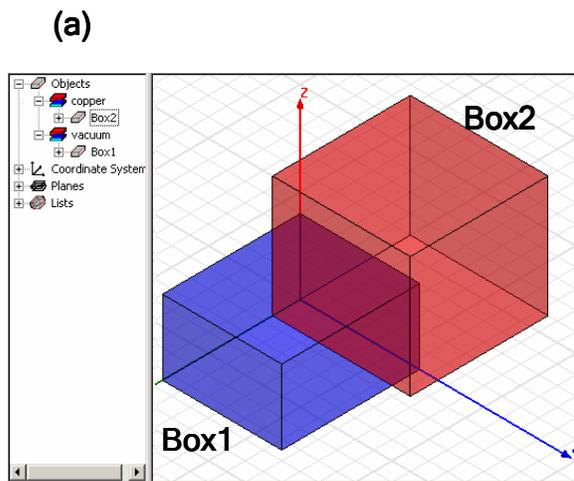
- ▲ In the circumstance that two adjacent objects are supposed to be coincident along their respective faces, yet the objects intersect, you can **Align Faces**.
- ▲ Aligning faces will take two different object faces that are very close together and make them coincident (more on mis-aligned objects 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)

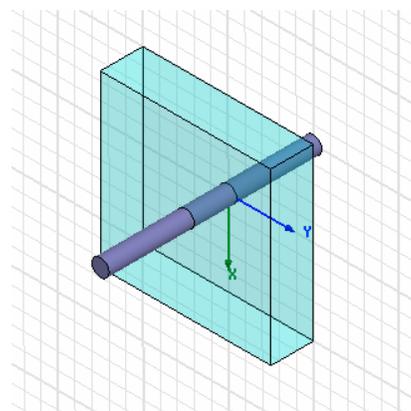
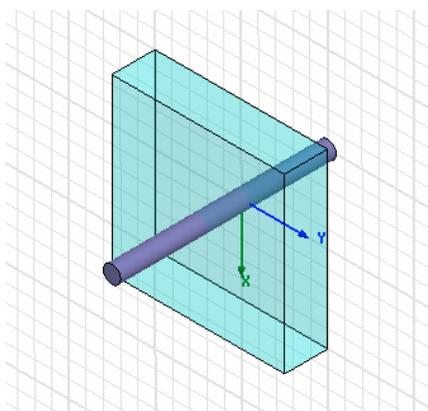
- ▲ Other circumstances for intersecting objects require manual Boolean manipulation.
- ▲ One Boolean method of fixing intersecting objects is to subtract a copy of one object from the other. In the subtract dialog, the tool object should be the object of which you want to keep the material definition 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

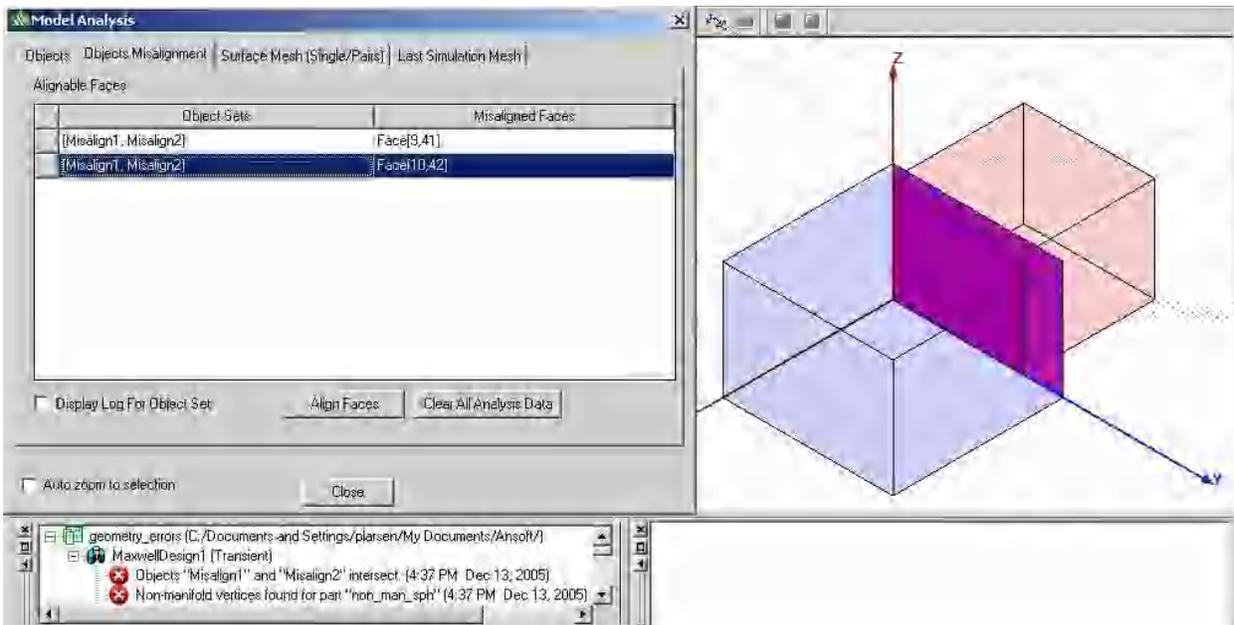
- ▶ When there are no ACIS errors in the model, no non-manifold objects, and no partial object intersections, the mesh generator can be invoked to create a valid mesh for the electromagnetic analysis.
- ▶ Even if the geometry is valid, mesh generation can still fail.
- ▶ Possible causes of mesh failure 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.

▄ 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

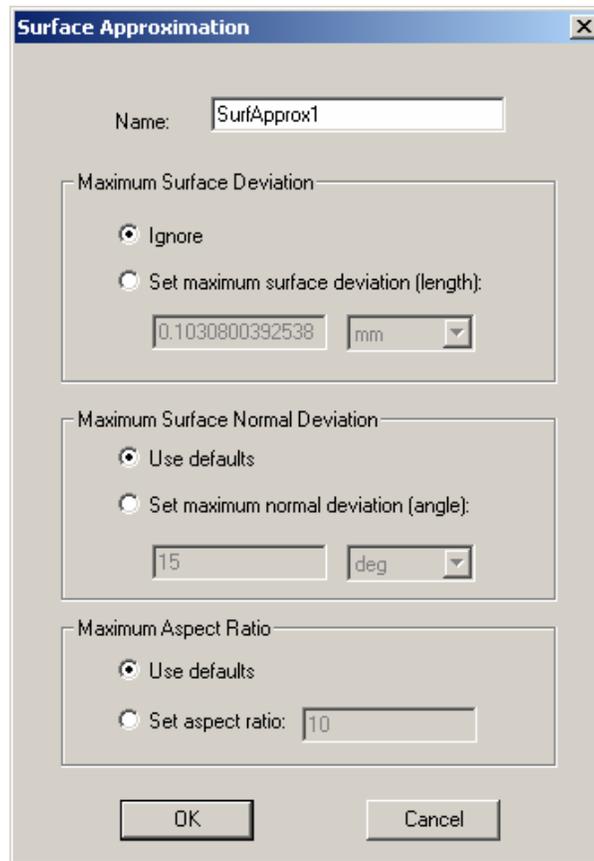
-  True Surfaces are curved surfaces in the 3D Modeler (e.g. spheres have a true surface, while boxes do not).
-  Due to the finite nature of the mesh, these true surfaces are approximated with facets and tetrahedra (as explained in the section on Mesh Operations).
-  When two true surfaces are coincident 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.