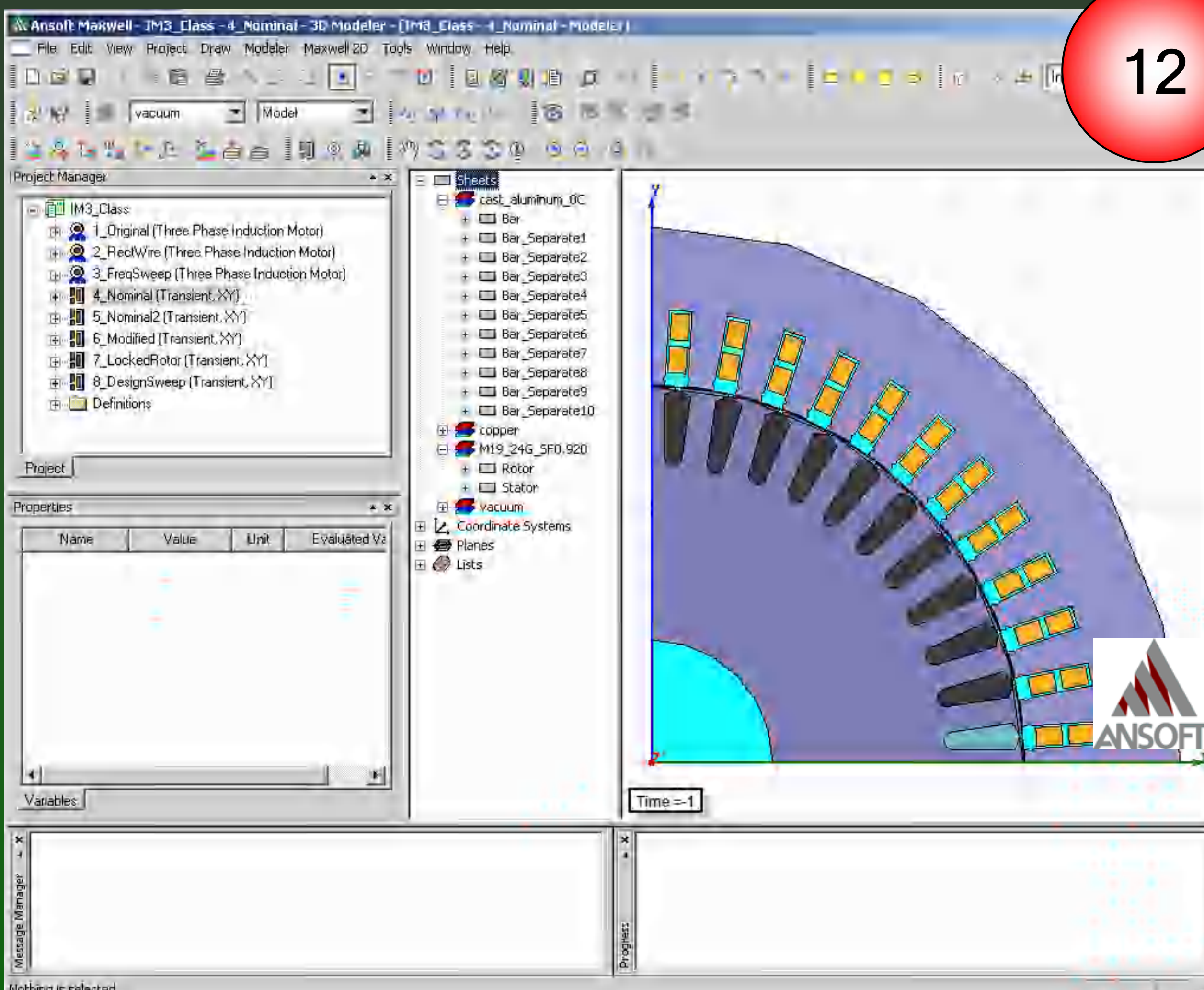
Ansoft

Maxwell 2D

Electromagnetic and Electromechanical Analysis

12



electronic design automation software

user’s guide - Maxwell 2D

ANSOFT CORPORATION • 225 West Station Square Dr. Suite 200 • Pittsburgh, PA 15219-1119

MAXWELL®

ANSOFT

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.

© 2009 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

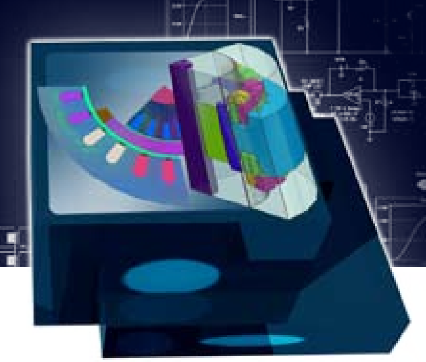
Maxwell, ePhysics and Optimetrics are registered trademarks or trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.

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: 15 January 2009
Software Version: 12.1

Contents

- This document discusses some basic concepts and terminology used throughout the Ansoft Maxwell application. It provides the following information:



MAXWELL® v12

3D/2D Electromagnetic Field Simulation

Maxwell® is a comprehensive electromagnetic field simulation software package for engineers tasked with designing and analyzing 3D/2D structures, such as motors, actuators, transformers and other electric and electromechanical devices common to automotive, military/aerospace and industrial systems. Based on the Finite Element Method (FEM), Maxwell can solve static, frequency-domain and time-varying electromagnetic and electric fields. In addition, the software can be dynamically linked with Simplorer® to create a powerful, system-level electromagnetic-based design flow. This flow enables users to combine complex circuits with accurate component models to design high-performance electromechanical and power electronic systems. Additionally, Maxwell's 3D solvers have dynamic links to ePhysics™. This allows engineers to perform complex 3D multi-physics studies by linking Maxwell to ePhysics' thermal and structural solvers.

KEY BENEFITS

Electromagnetic field simulation
Maxwell includes 3D/2D Transient, AC Electromagnetic, Magnetostatic, Electrostatic and Electrotransient solvers that accurately solve for force, torque, capacitance, inductance, resistance, and impedance, as well as generate state-space models.

Automatic adaptive meshing
Maxwell uses the Ansoft-pioneered automatic adaptive meshing techniques. This robust meshing algorithm automatically creates and refines the finite element mesh as the solution converges, streamlining the solution process and making the software very easy to use.

Dynamic link - ePhysics
The Maxwell 3D solvers can be dynamically linked with ePhysics' thermal and stress analysis and are the ideal solution for every electromechanical device requiring cross-disciplinary design analysis.

Dynamic link - Simplorer
Dynamic links with Simplorer multi-domain system simulation allow accurate high-fidelity component models to be combined with circuits and system architecture to create a powerful, electromagnetic-based design flow.

Import
CAD files can be imported in Maxwell streamlining the design process.

Multi-processing and distributed analysis
Maxwell can leverage available computing power with multi-processing and distributed analysis options for fast turnaround of your largest designs.

Optimization
Optimetrics™ provides parametric, optimization, sensitivity, and statistical analysis capabilities to Maxwell. Optimetrics automates the design-optimization process by quickly identifying optimal values for design parameters that satisfy user-specified constraints.

Customized pre-processors
RMxprt (electric machine design) and PExprt™ (magnetic component design) are used to design devices based on a traditional analytical approach. They also can be directly linked to Maxwell and provide fully automated design creation and analytical analysis. Users can perform preliminary studies of design concepts prior to performing rigorous electromagnetic analysis with Maxwell.

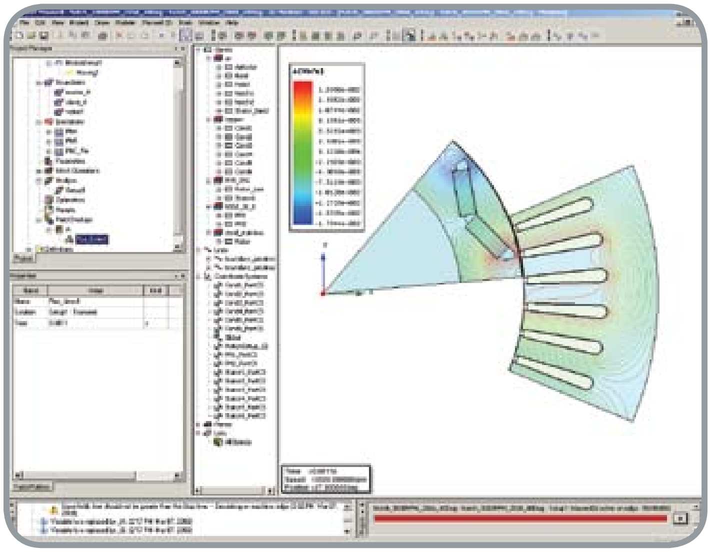
APPLICATIONS

Electromechanical
- Motors and generators
- Linear or rotational actuators
- Relays
- MEMS
- Magnetic recording heads

Electromagnetic
- Coils
- Permanent magnets
- Sensors

Power electronic
- Transformers
- Converters
- Bus bars
- IGBTs and similar devices

EM behavior
- Insulation studies
- Electrostatic discharge
- Electromagnetic shielding
- EMI/EMC
- Semiconductor
- Biomedical



The new 2D interface provides strong coupling with 3D and many new usability features.

KEY FEATURES

Low-frequency electromagnetic field simulation and analysis using FEM for 3D/2D structures

- Transient - nonlinear analysis with:
  - Motion—rotation, translational, non-cylindrical rotation
  - External circuit coupling
  - Permanent magnet demagnetization analysis
  - Core loss computation
  - Lamination modeling for 3D
- AC Electromagnetic—Analysis of devices influenced by skin/proximity effects, eddy/displacement currents
- Magnetostatic—Nonlinear analysis with automated equivalent circuit model generation
- Electric Field—Transient, Electrostatic/Current flow analysis with automated equivalent circuit model generation

Display of data/visualization of results

- Field visualization and animations (shaded, contour and vector plots)
- Mesh visualization (full, partial)
- Current, induced voltage, flux linkage
- Power loss, stored energy
- Core loss, eddy, excess, hysteresis loss (including the minor loop effects)
- Impedance, inductance, capacitance
- Force, torque
- Custom reports of user-defined solution data

Performance and integration

- Distributed Analysis* for parallel computing of parameterized models
- 64-bit operating system support
- Links to Simplorer®*, ePhysics™*, HFSS™*, RMxprt™*, PExprt™*

Integrated 3D modeler featuring ACIS v16 and MFC technology

- Standard primitives and multi-sweep functions
- Boolean operations: union, subtraction, intersection
- Direct import of SAT and DXF files
- AnsoftLinks™* for import of STEP, IGES and Pro/E files

Automatic, adaptive mesh technology

- Fault-tolerant meshing algorithms
- Mesh-generation feedback
  - GUI performs validation and integrity checks
  - Software identifies artifacts within the imported geometry
- Mesh-based model resolution

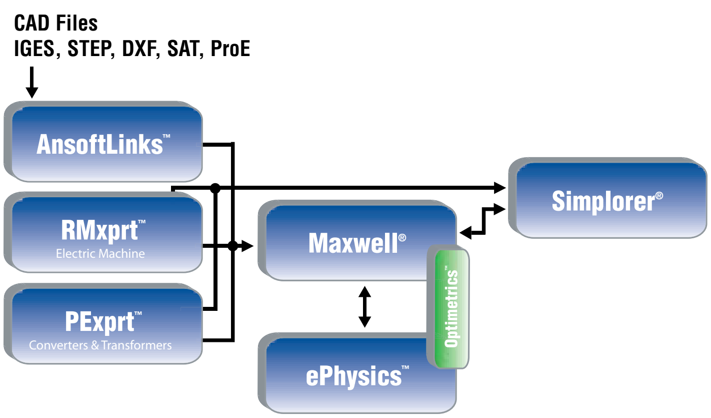
Versatile material manager and material types

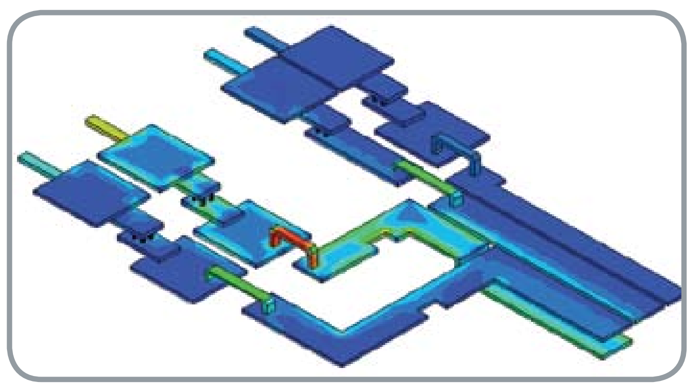
- User, group and system libraries
- Linear, nonlinear anisotropic materials
- Material assignment by coordinate type: cartesian, cylindrical or spherical

Integrated Optimetrics™*

- Geometry and material parameterization
- Optimization, sensitivity and statistical analysis

*Option available at additional charge.





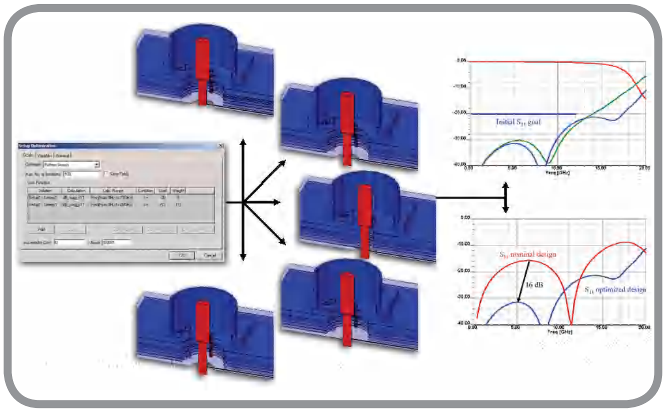
Current density in a busbar system as calculated by Maxwell 3D.

Maxwell, Simplorer, ePhysics, Optimetrics, PExprt, AnsoftLinks, and HFSS are trademarks of Ansoft Corporation.
All other trademarks are the property of their respective owners.
© 2008 Ansoft Corporation 0308

225 West Station Square Drive • Suite 200 • Pittsburgh, PA 15219-1119 USA
T 412-261-3200 **F** 412-471-9427 **E** info@ansoft.com **W** www.ansoft.com

OPTIMETRICS™

Parametric Analysis and Optimization



Optimetrics™ is an optional software module that adds parametric capabilities, optimization algorithms, sensitivity and statistical analyses to Ansoft's best-in-class electromagnetic-field simulation products—HFSS™, Maxwell® 3D and Q3D Extractor®. Optimetrics automates the design-optimization process for high-performance electronics, such as microwave/RF devices, printed circuit boards, on-chip passives, IC packages and electromechanical components, by quickly identifying optimal values for design parameters that satisfy user-specified constraints and goals.

This example is a connector designed with HFSS and Optimetrics. The control panel displays design variables (i.e., cost functions, parameters), launches design perturbations and converges to the optimal performance criterion.

OVERVIEW

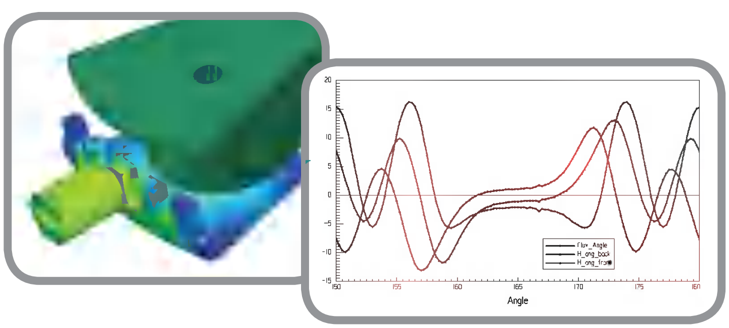
Optimetrics™ enables users to study the effects of geometry and materials on a design by creating parameters for the dimensions and material constants of the model to be analyzed. Optimetrics then varies these parameters and adjusts the geometry and materials to achieve the desired, user specified, performance goal.

Leveraging previously computed parametric simulation results within its optimizer, Optimetrics enables engineers to understand device characteristics over a large design space and quickly identify the best performing design that is least sensitive to manufacturing tolerances.

Optimetrics, when used in conjunction with HFSS™, Maxwell® 3D and Q3D Extractor®, delivers an innovative and robust design platform from which users gain a greater understanding of the design space and the ability to make insightful design choices.

FEATURED CAPABILITIES

- Parametric Analysis
  - User-specified range and number of steps for parameters
  - Automatic analysis of parameter permutations
  - Distributed Analysis (cost option)
    - Automated parser management across multiple hardware platforms and reassembly of data for parametric tables and studies
- Sensitivity Analysis
  - Design variations to determine sensitivities
    - Manufacturing tolerances
    - Material properties
- Optimization
  - User-selectable cost functions and goal objective
    - Quasi-Newton method
    - Sequential Nonlinear Programming (SNLP)
    - Integer-only Sequential Nonlinear Programming
  - Automatic analysis of parameter variants until optimum goal obtained
- Tuning
  - User-controllable slide bar for real-time tuning display and results
- Statistical Analysis
  - Design performance distribution versus parameter values



Current sensor optimization results using Maxwell 3D and Optimetrics

Ansoft Corporation • 225 West Station Square Drive • Pittsburgh, PA 15219-1119 USA
TEL 1.412.261.3200 FAX 1.412.471.9427 EMAIL info@ansoft.com WEB www.ansoft.com

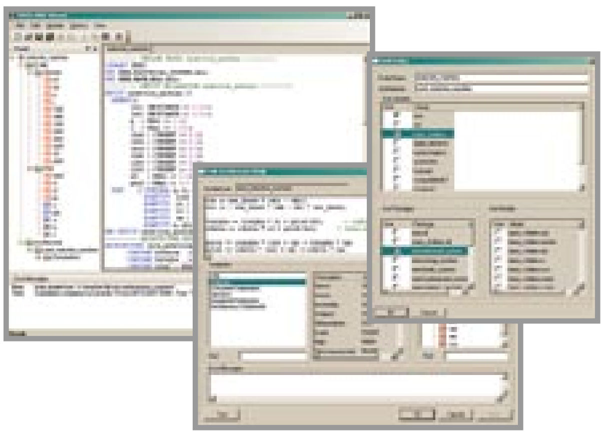
Please consult your local sales representative for pricing and information on this and on other Ansoft products. HFSS, Maxwell, Optimetrics and Q3D Extractor are trademarks of Ansoft Corporation.
All other trademarks are the property of their respective owners. © 2005 Ansoft Corporation PH15-1105

SIMPLORER® v7.0

Multi-domain simulation software

Overview

SIMPLORER® is the premier software program for the design and analysis of complex, multi-domain systems commonly found in automotive, aerospace/defense and industrial systems.



SIMPLORER v7 offers VHDL-AMS wizard technology, making it easy to leverage the IEEE multi-domain modeling standard.

Multi-domain system design is challenging and complex. It consists of many interdisciplinary and nonlinear components from multiple domains: electrical, mechanical, thermal and control. The close interaction across domains renders single-domain system simulation tools ineffective.

SIMPLORER is the only system engineering tool to offer multiple standard modeling techniques (VHDL-AMS, circuits, block diagrams, state machines, C/C++) that can be used concurrently. It also utilizes the concept of "natures," allowing components of different engineering domains to interact.

SIMPLORER is the ideal tool for system designs such as:

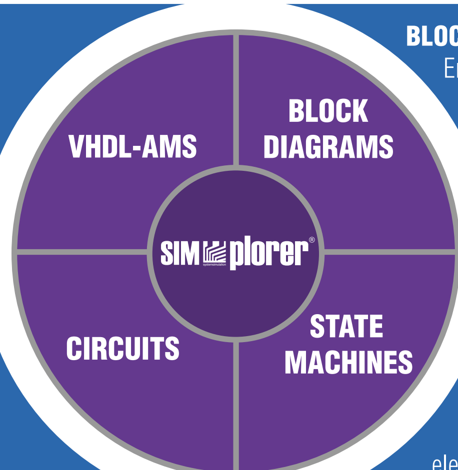
- Power Systems
- Electric Motors and Drives
- Powertrains
- Hybrid-electric Propulsion
- Other Multi-domain Systems

Modeling Techniques

SIMPLORER allows components to be described as behavioral or physical models using one or any combination of SIMPLORER's modeling techniques. This eliminates error-prone mathematical transformations and model analogies often employed by single-domain simulation tools.

VHDL-AMS
IEEE-endorsed modeling language (standard 1076.1) created to provide a general-purpose, easily exchangeable and open language for multi-domain analog mixed-signal designs.

BLOCK DIAGRAMS
Enables the description of signal-flow-based linear, nonlinear, continuous, time-discrete and hybrid systems, making it ideal for dynamic system simulation and closed-loop-control applications.



CIRCUITS
Numerically stable and fast circuit simulator with support for multi-level semiconductor modeling that is optimized to the needs of demanding power-electronic and high-switching-frequency circuit design.

STATE MACHINES
Highly efficient modeling technique for event-driven systems, such as logical control found in embedded control systems, space vector controls or PWM for power-electronic circuits.

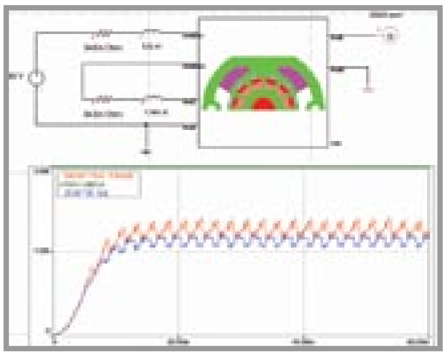
Ansoft Corporation • 225 West Station Square Drive • Pittsburgh, PA 15219-1119 USA
TEL: 412.261.3200 • FAX: 412.471.9427 • EMAIL: info@ansoft.com • WEB: www.ansoft.com

Models

SIMPLORER Model Libraries

SIMPLORER offers optional application-specific model libraries to enhance productivity and reduce design time:

- Alternative Power
- Automotive
- Hydraulic
- Machine
- Mechanical
- Power
- SMPS
- Sensor



SIMPLORER v7 now includes a transient simulation coupling link. Users can simultaneously solve a transient FEA project with a transient system simulation.

FEA-Based Models

For models requiring the highest level of fidelity, SIMPLORER provides a direct link to Ansoft's industry-leading electromagnetic field simulation and design programs: Maxwell®, RMxprt™, and PExprt™. Users can easily create equivalent circuit models from the finite-element analysis (FEA) results and import them directly to SIMPLORER.

Alternatively, users can employ the Transient Simulation coupling link to couple transient FEA directly to SIMPLORER. This powerful feature provides the ultimate in accuracy and flexibility and is ideal for detailed analysis of electromechanical components operating within a system.

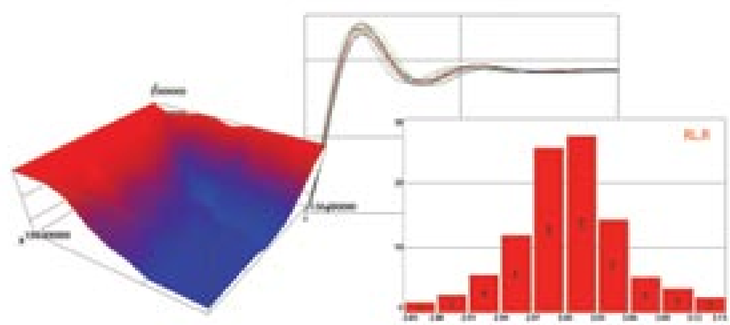
Manufacturers' Models

SIMPLORER users can access up-to-date manufacturer-specific components online at www.model.simplorer.com. MOSFET, IGBT, ultra capacitors and other components are available to customers as a free download.

Statistical Analysis and Optimization

SIMPLORER includes many advanced analysis capabilities such as parametric sweeps and optimization routines to provide insight into design variations and "trade-offs."

- Parameter Sweep/Table
- Monte Carlo
- 3D Graphic
- Genetic Algorithm
- Successive Approximation
- SIMPLEX
- Frequency Sweep
- Worst Case
- Sensitivity



SIMPLORER v7 includes many new statistical design and optimization routines.

Integration

Scripting

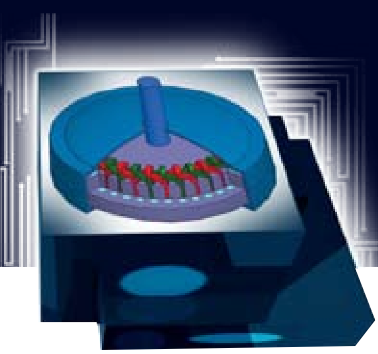
This powerful feature opens APIs in the SIMPLORER environment, allowing SIMPLORER to be embedded into existing design flows. The scripting capability is language independent so users can work with popular scripting languages, such as Visual Basic®, Java® or Tcl/Tk and interact easily with other tools supporting the Microsoft Com interface, such as MS Office and LabView®

Co-Simulation

SIMPLORER allows the integration of proprietary C/C++ programs, MATLAB®/Simulink®, Mathcad® and other specialized programs, allowing SIMPLORER to utilize customized code and existing design control. The direct integration of models in their native environment avoids model translation, saves design time and allows communication and model exchange across departments and between suppliers and OEMs.

Please consult your local sales representative for pricing and information on this and other Ansoft products.

SIMPLORER, PExprt and RMxprt are trademarks of Ansoft Corporation. All other trademarks are the property of their respective owners.
PL37-0407 © 2004 Ansoft Corporation



RMxprt™ v12

Design Software for Electric Machines

RMxprt™ is a versatile software program that speeds the design and optimization process of rotating electric machines. With RMxprt, users can calculate machine performance, make initial sizing decisions, and perform hundreds of "what if" analyses in a matter of seconds. As the entry point for the Ansoft motor and drive design methodology, RMxprt automatically produces both system-level models and geometric data, allowing the preliminary design to be refined and integrated with power electronic and control circuitry.

KEY BENEFITS

Fast design
RMxprt offers numerous machine-specific, template-based interfaces for induction, synchronous, and electronically and brush-commutated machines that allow users to easily enter design parameters and to evaluate design tradeoffs early in the design process.

Performance metrics
Critical performance data, such as torque versus speed, power loss, flux in the air gap, power factor and efficiency can be quickly calculated.

Robust calculation methods
RMxprt uses classical analytical motor theory and equivalent magnetic circuit methods to compute performance metrics for a specific machine design and accounts for nonlinear magnetic characteristics and 3D effects, such as skew and end-turn.

Model pre-processor
RMxprt is a key part of Ansoft's motor design methodology. In addition to providing classical motor performance calculations, RMxprt can automatically create 3D and 2D geometry and assign material properties and other necessary problem definition data necessary to perform rigorous finite element analysis on the design using Maxwell®.

Wire library
RMxprt includes a comprehensive database of ANSI and IEC wires.

High-fidelity system models
RMxprt creates high-fidelity, state-space system models incorporating machines' physical dimensions, winding characteristics and nonlinear material properties. Engineers can use the resulting behavioral model to explore electronic control topologies, loads, and interactions with drive-system components in Simplorer®.

Convenient design sheet output
Design sheets list all the relevant input parameters and calculated parameters and graphically display waveforms including current, voltage, torque and back EMF as well as a detailed winding layout. RMxprt also can output Excel-format design sheets based on the user-defined template.

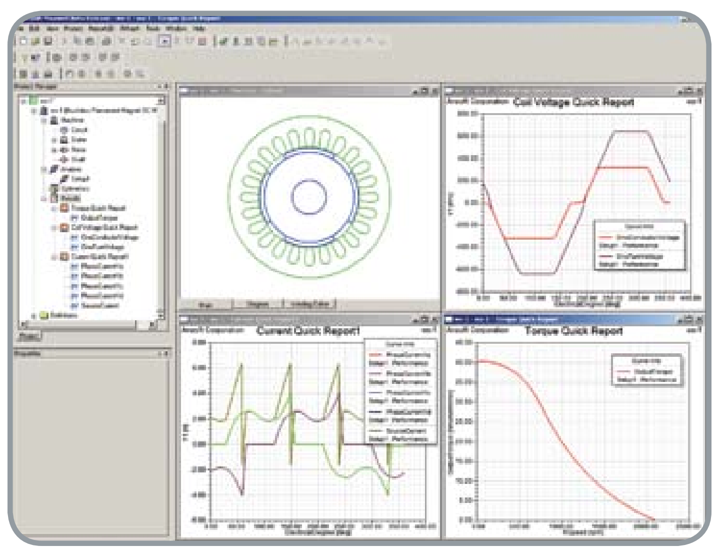
Design optimization
RMxprt can perform hundreds of "what if" analyses in a matter of minutes, making it a valuable tool for designers needing to make initial sizing and material decisions quickly.

Powerful scripting
RMxprt can be integrated with third party development programs through scripting languages such as VB script, Tcl/TK, JavaScript®, Perl, Excel and MATLAB®. This allows users to customize the design flow and leverage internally developed programs and historical data.

DESIGN TEMPLATES

Machine types

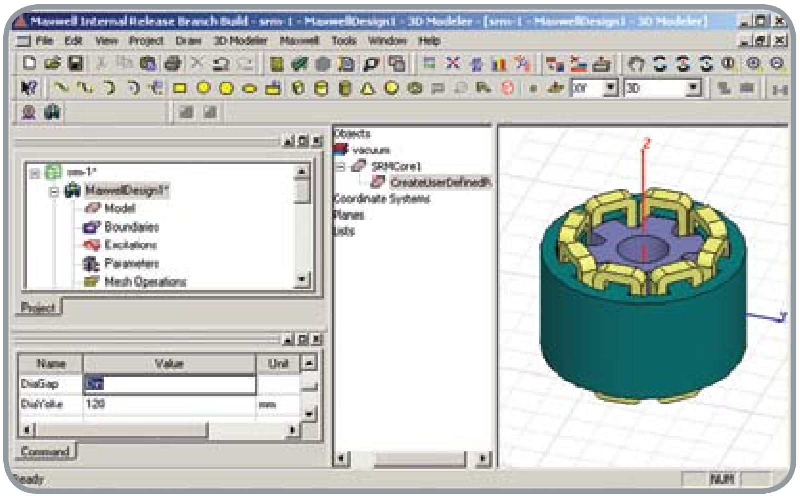
- Induction machines
  - o Single-phase motors
  - o Three-phase motors
- Synchronous machines
  - o Line-start PM motors
  - o Salient-pole motors and generators
  - o Non-salient pole motors and generators
- Brush commutated machines
  - o DC motors and generators
  - o Permanent magnet DC motors
  - o Universal motors
- Electronically commutated machines
  - o Brushless DC motors
  - o Adjustable-speed PM motors and generators
  - o Switched reluctance motors
  - o Claw-pole generators



RMxprt delivers the reports you need to quickly analyze and tune your design.

KEY FEATURES

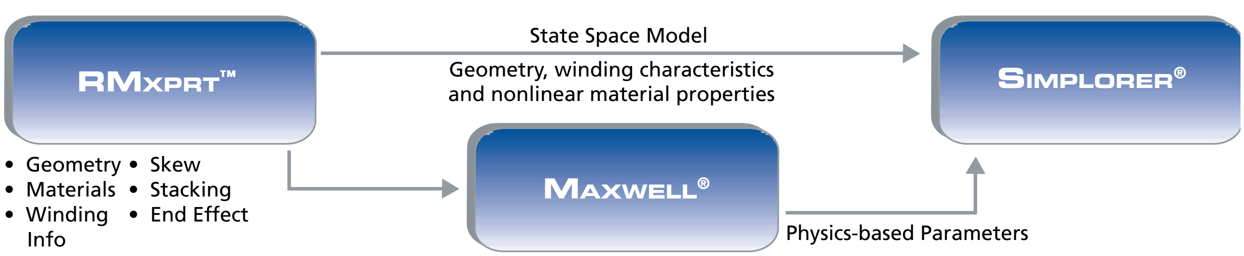
- Machine-specific template editor
  - o Rotor
  - o Stator
  - o Running strategies
  - o Drive circuits
- Auto-design feature
  - o Slot size
  - o Coil turns and wire diameter
  - o Starting capacitance
  - o Winding arrangement
- Performance curves
  - o Torque
  - o Power
  - o Efficiency
- Output waveforms
  - o Current
  - o Cogging torque
  - o Flux in the air gap
- Graphical winding editor
- Cross section Editor
- Customizable design sheet
- Cost evaluation
- Integrated parametrics and optimization
- State-space model export to Simplorer®
- Automated project setup for Maxwell® 2D
- Automated geometry and material setup for Maxwell 3D



RMxprt™ creates 3D and 2D geometry, assigns materials and sets up boundary conditions for Maxwell. Additionally, any parameter changed in RMxprt is automatically updated in the finite element project.

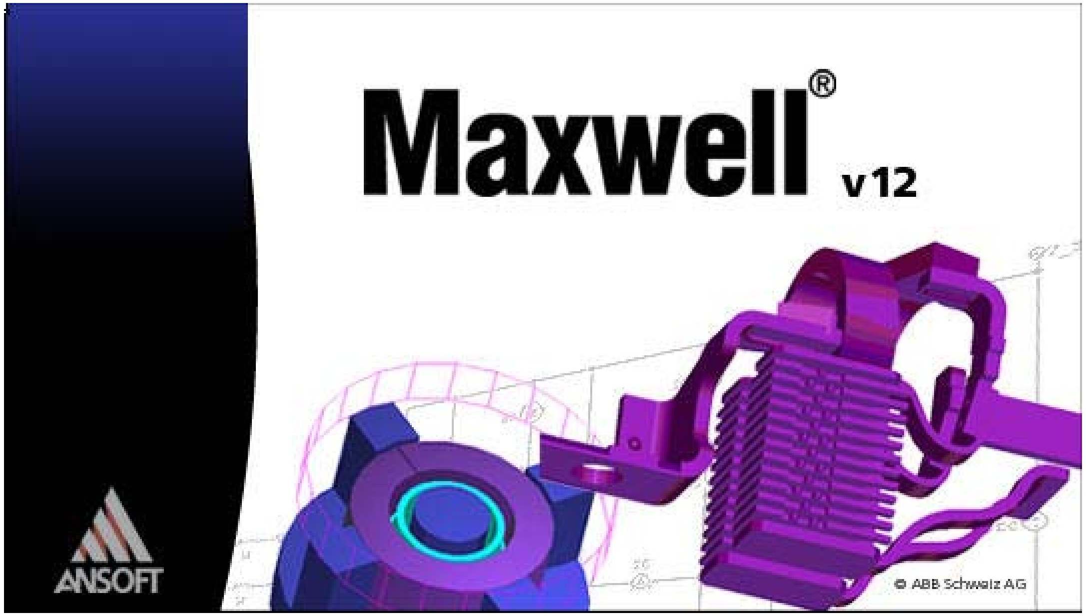
DESIGN FLOW

RMxprt is the ideal starting point for a comprehensive electric machine design flow. RMxprt with Maxwell and Simplorer provides an efficient and accurate methodology to design and optimize an electric machine and related electric drive and control system.



RMxprt, Simplorer and Maxwell are trademarks of Ansoft Corporation.
All other trademarks are the property of their respective owners.
© 2008 Ansoft Corporation 0308

225 West Station Square Drive • Suite 200 • Pittsburgh, PA 15219-1119 USA
T 412-261-3200 **F** 412-471-9427 **E** info@ansoft.com **W** www.ansoft.com



- Maxwell 2D is a high-performance interactive software package that uses finite element analysis (FEA) to solve electric field and magnetic field problems.

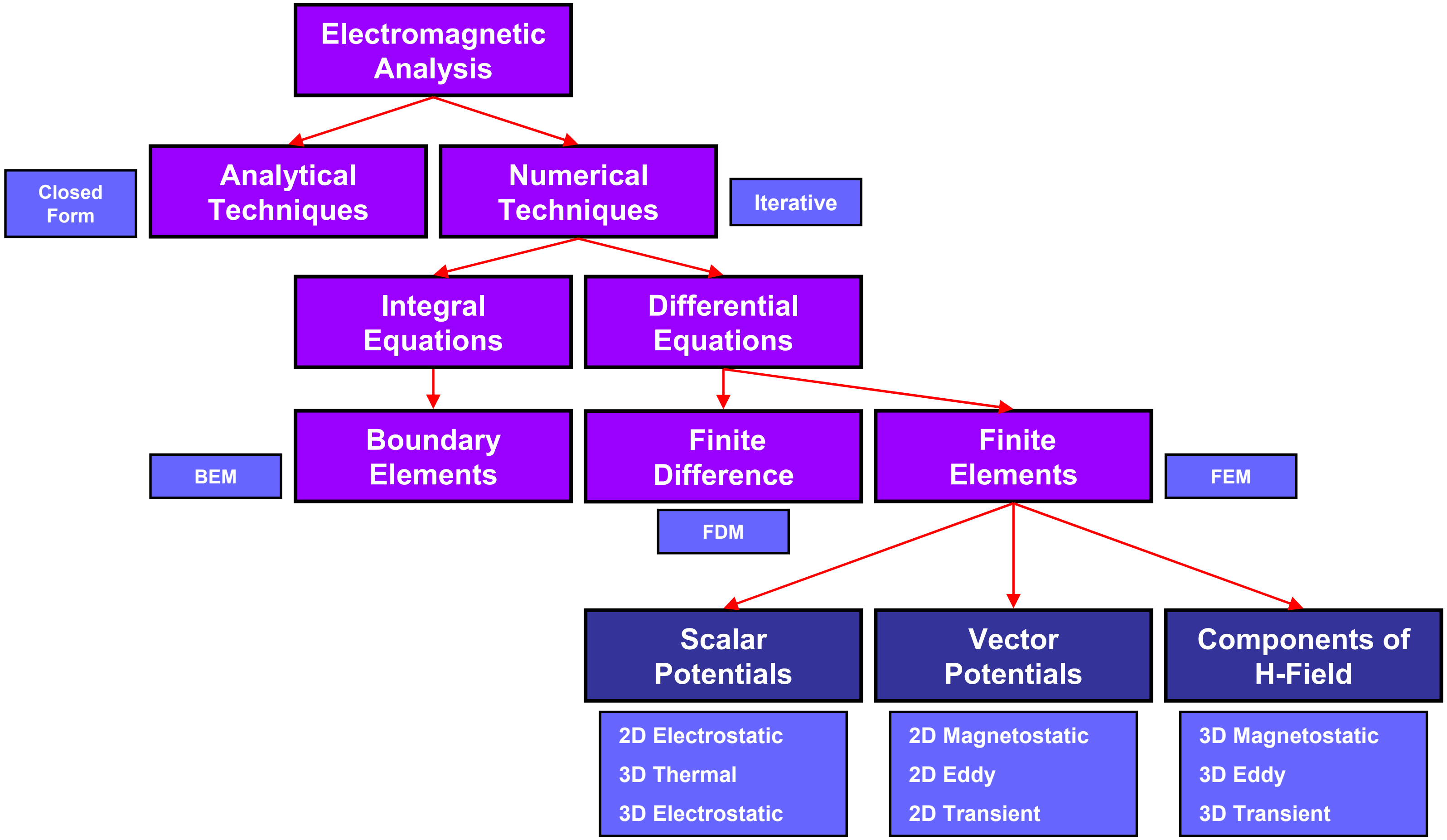
Maxwell® v12

- Maxwell 2D solves the electromagnetic field problems for a given model with appropriate materials, boundaries and source conditions applying Maxwell's equations over a finite region of space.
- There are two geometry modes available in Maxwell 2D:
  - Cartesian (XY) model
  - Axisymmetric (RZ) model
- There are six solvers available in Maxwell 2D:
  - Electrostatic
  - AC Conduction
  - DC Conduction
  - Magnetostatic
  - Eddy Current
  - Transient Magnetic

Electric Fields

Magnetic Fields

Different Methods of Electromagnetic Analysis



Differential Form of Maxwell's Equations

Faraday's Law of Induction $\nabla \times E = -\frac{\partial B}{\partial t}$

Gauss's Law for Magnetism $\nabla \bullet B = 0$

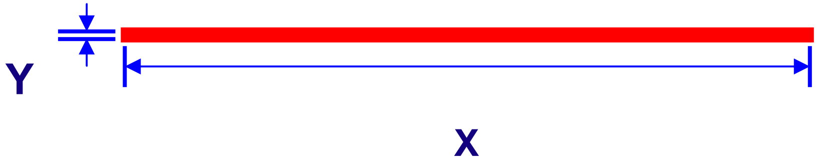
Ampere's Law $\nabla \times H = J + \frac{\partial D}{\partial t}$

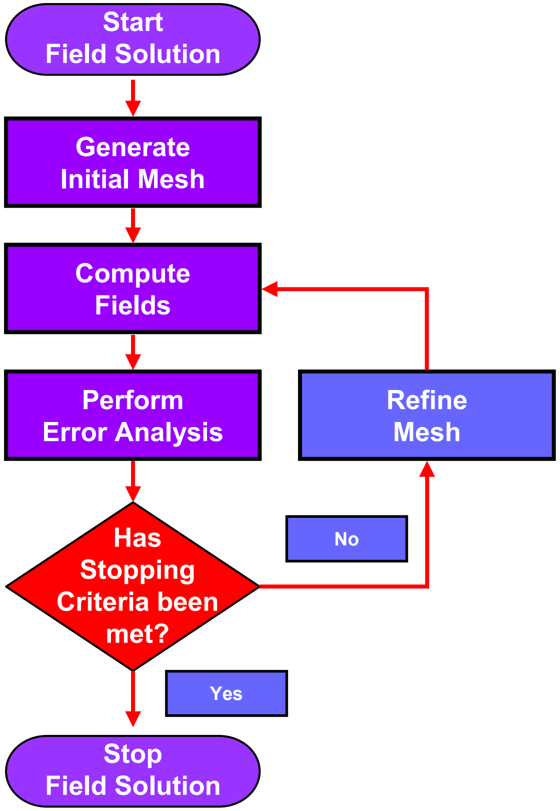
Gauss's Law for Electricity $\nabla \bullet D = \rho$

FEM and adaptive meshing

- In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into small elements (e.g., triangles in 2D).
- All the model solids are meshed automatically by the mesher.
- The assembly of all triangles is referred to as the finite element mesh of the model or simply the mesh.
- Approximate aspect ratio limit in 2D:

$$X = 10{,}000Y$$



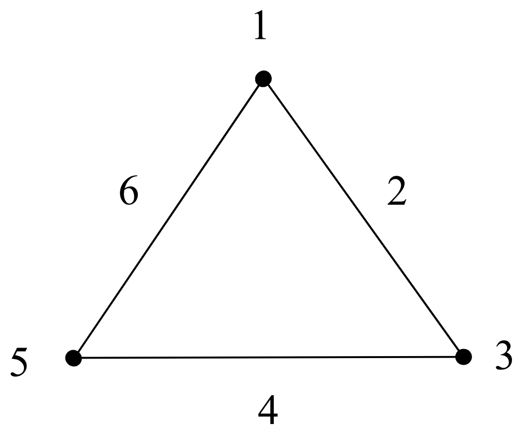


FEM Approximation Functions

- The desired field in each element is approximated with a 2nd order quadratic polynomial

$$A_z(x,y) = a_o + a_1 x + a_2 y + a_3 x^2 + a_4 xy + a_5 y^2$$

- Field quantities are calculated for 6 points (3 corners and 3 midpoints) in 2D
- Field quantities inside of the triangle are calculated using a 2nd order quadratic interpolation scheme



FEM Variational Principle

- Poisson's equation: $\nabla^2 A = -\mu J$

  is replaced with energy functional: $F(A) = \frac{1}{2}\int\left(\frac{\nabla A \bullet \nabla A}{\mu} + A \bullet J\right)dV$

- This functional is minimized with respect to value of A at each node in every triangle

FEM Matrix Equation

- Now, over all the triangles, the result is a large, sparse matrix equation

$$[S][A]=[J]$$

- This can be solved using standard matrix solution techniques such as:
  - Sparse Gaussian Elimination (direct solver)
  - Incomplete Choleski Conjugate Gradient Method (ICCG iterative solver)

FEM Error Evaluation

- Put the approximate solution back into Poisson's equation

$$\nabla^2 A^{approx} + \mu J = R$$

- Since A is a quadratic function, R is a constant in each triangle.
- The local error in each triangle is proportional to R.

FEM Percent Error Energy

- Summation of local error in each triangle divided by total energy

$$\text{Percent Error Energy} = \sum_{i=1}^{n} \frac{\left| R(\text{local})_i \right|}{\text{Total Energy}} \times 100\%$$

- Local errors can exceed Percent Error Energy

Transient Solver
Fully Coupled Dynamic Physics Solution

Current Source Density

Electric Scalar Potential

Velocity

$$\nabla \times \upsilon \nabla \times A = J_s - \sigma \frac{\partial A}{\partial t} - \sigma \nabla V + \nabla \times H_c + \sigma v \times \nabla \times A$$

Magnetic Vector Potential

Permanent Magnet

Time-varying Electric and Magnetic Fields

Transient Solver - Magnetic Field Diffusion

- Magnetic fields "diffuse" into materials at different rates depending on:
  - Material properties of the component
  - Physical size of the component
- For a cylindrical conductor, diffusion time is:

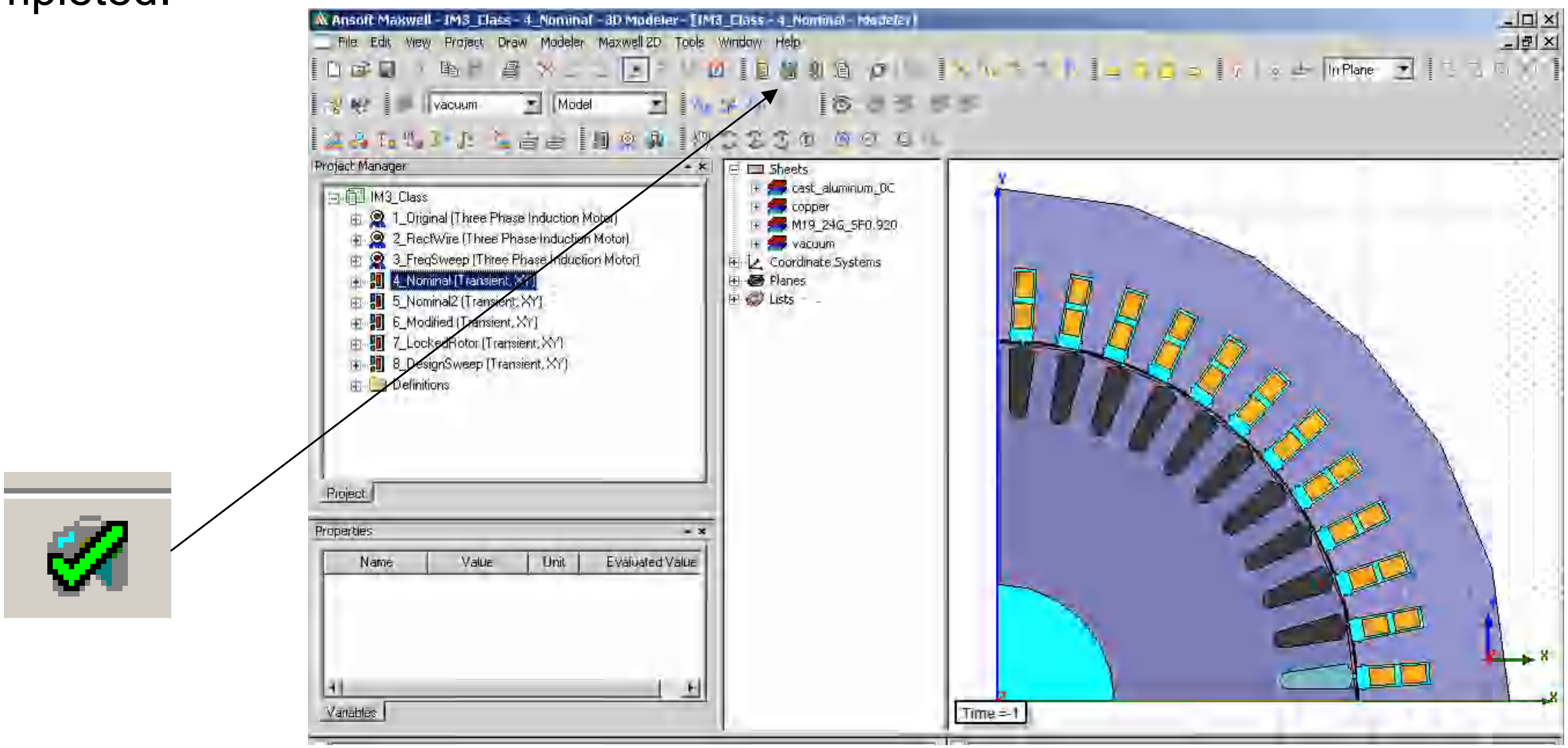
$$\tau = \frac{u\sigma a^2}{2.4048^2} \quad (\mathrm{sec})$$

$$where : u \;\; = perm, \sigma = conductivity, a = radius \;\; in \;\; meters$$

- Induced eddy currents always occur in conducting objects due to time-varying fields; however, they may not always be significant

GUI - Desktop

- The complex functionality built into the Maxwell solvers is accessed through the main user interface (called the desktop).
- Problem can be setup in a fairly arbitrary order.
- A new “validation check” has been added to insure that all required steps are completed.



ACIS solid modeling kernel

- The underlying solid modeling technology used by Ansoft products is provided by ACIS geometric modeler. ACIS version 16 is presently used.
- Users can create directly models using primitives and operations on primitives.
- In addition, users can import models saved in a variety of formats (sm2 .gds .sm3 .sat .step .iges .dxf .dwg .sld .geo .stl .prt .asm)
- When users import models into Ansoft products, translators are invoked that convert the models to an ACIS native format (sat format).
- Exports directly .sat, .dxf, .sm3, .sm2, .step, .iges

Supported platforms

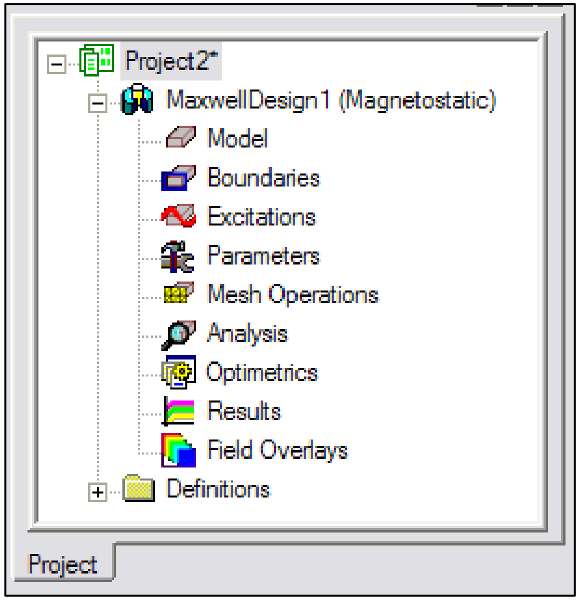
- Windows XP Pro
- Windows XP Pro x64 Edition
- Windows Server 2003
- Windows Server 2003 x64 Edition
- Red Hat Enterprise Linux 3, 4
- SuSE Linux Enterprise Server 9.3
- Solaris 8 -10

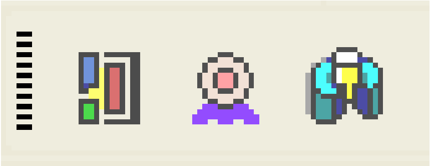
Starting Maxwell

- Click the Microsoft **Start** button, select **Programs**, and select the *Ansoft > Maxwell 12 > Maxwell 12*
- **Or** Double click on the Maxwell 12 icon on the Windows Desktop

Adding a Design

- When you first start Maxwell a new project will be automatically added to the Project Tree.
- To insert a Maxwell Design to the project, select the menu item *Project > Insert Maxwell 2D Design*



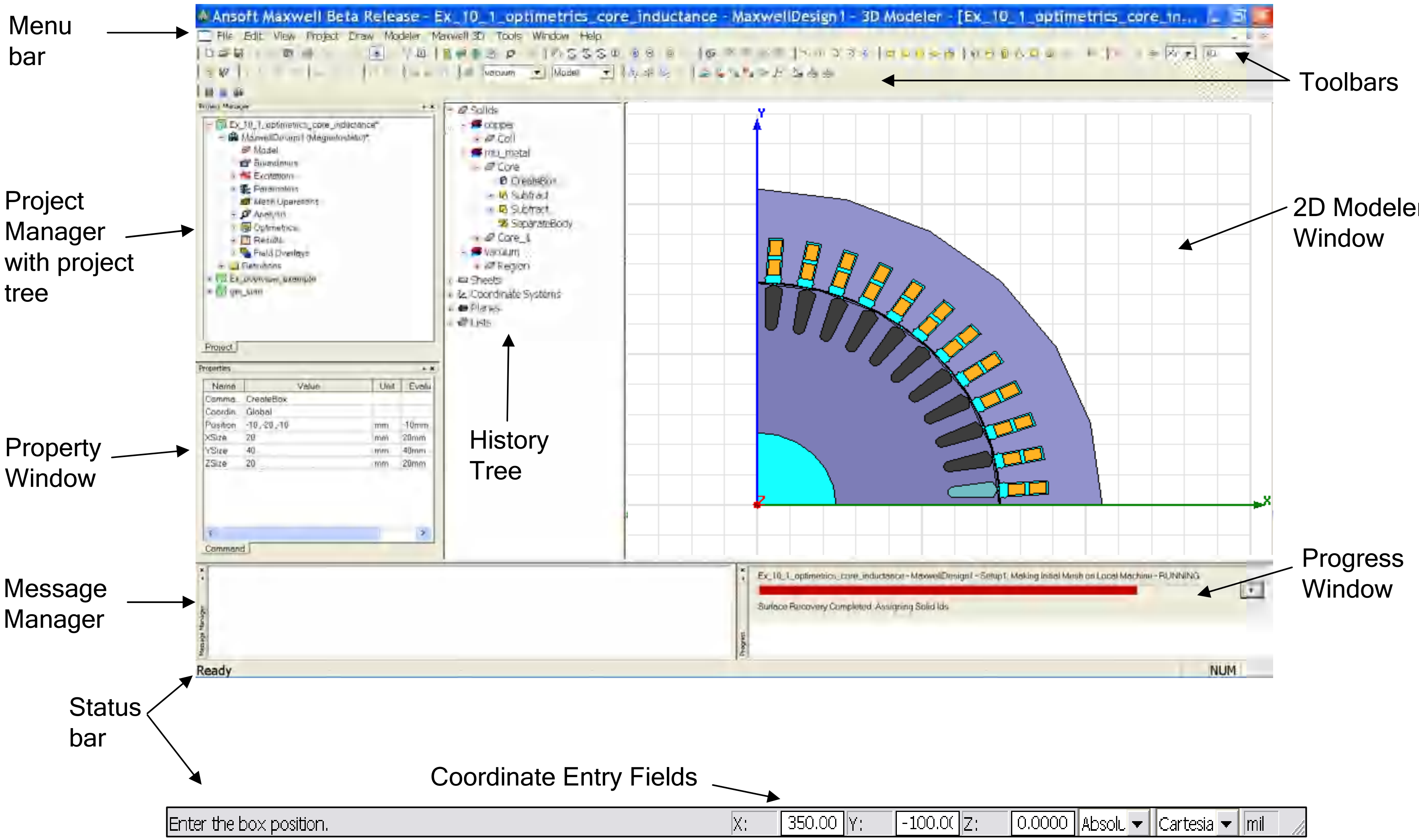


Toolbar:
Insert Maxwell 2D Design

Insert Maxwell 3D Design

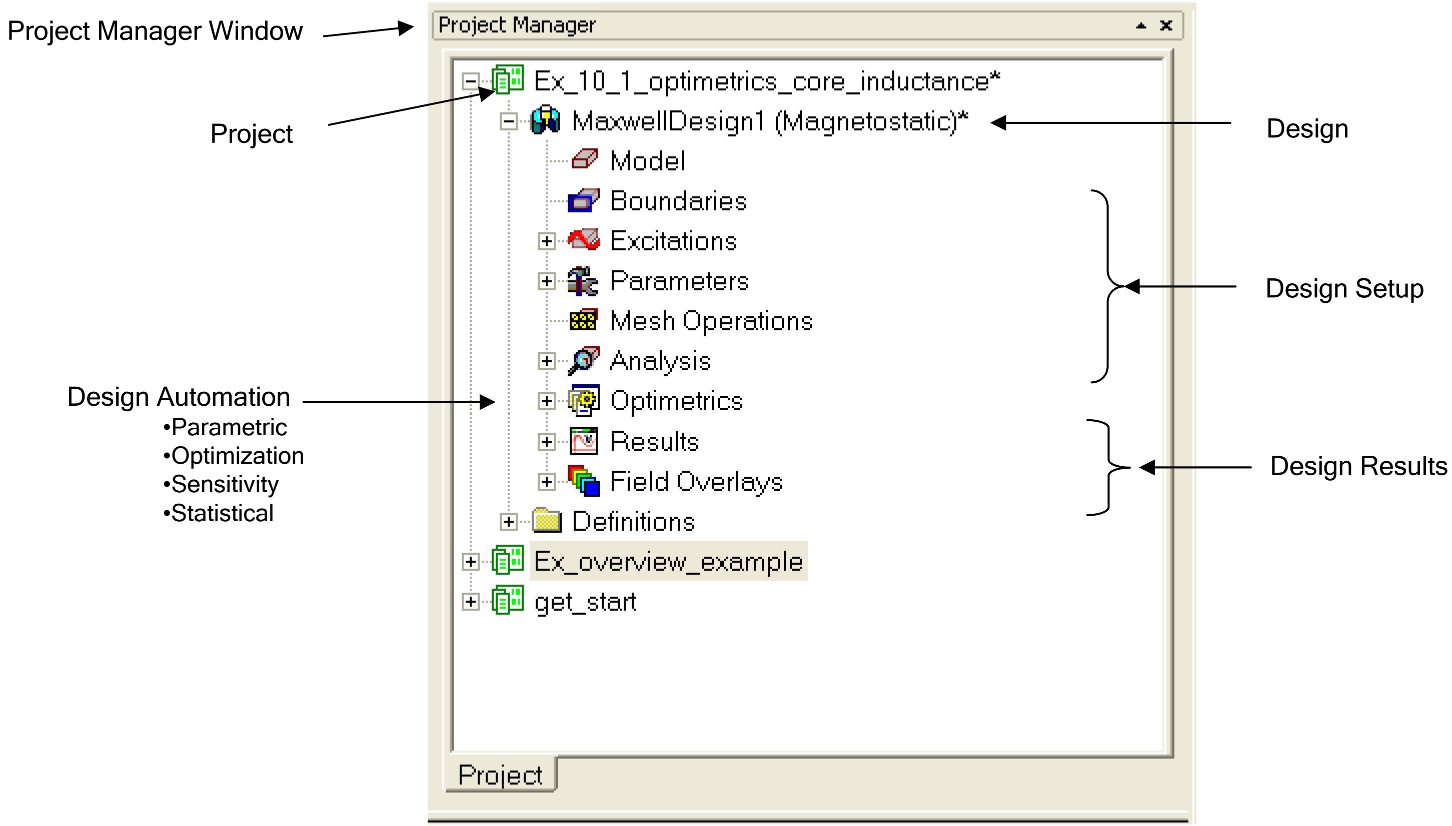
Insert RMxprt Design

Maxwell Desktop

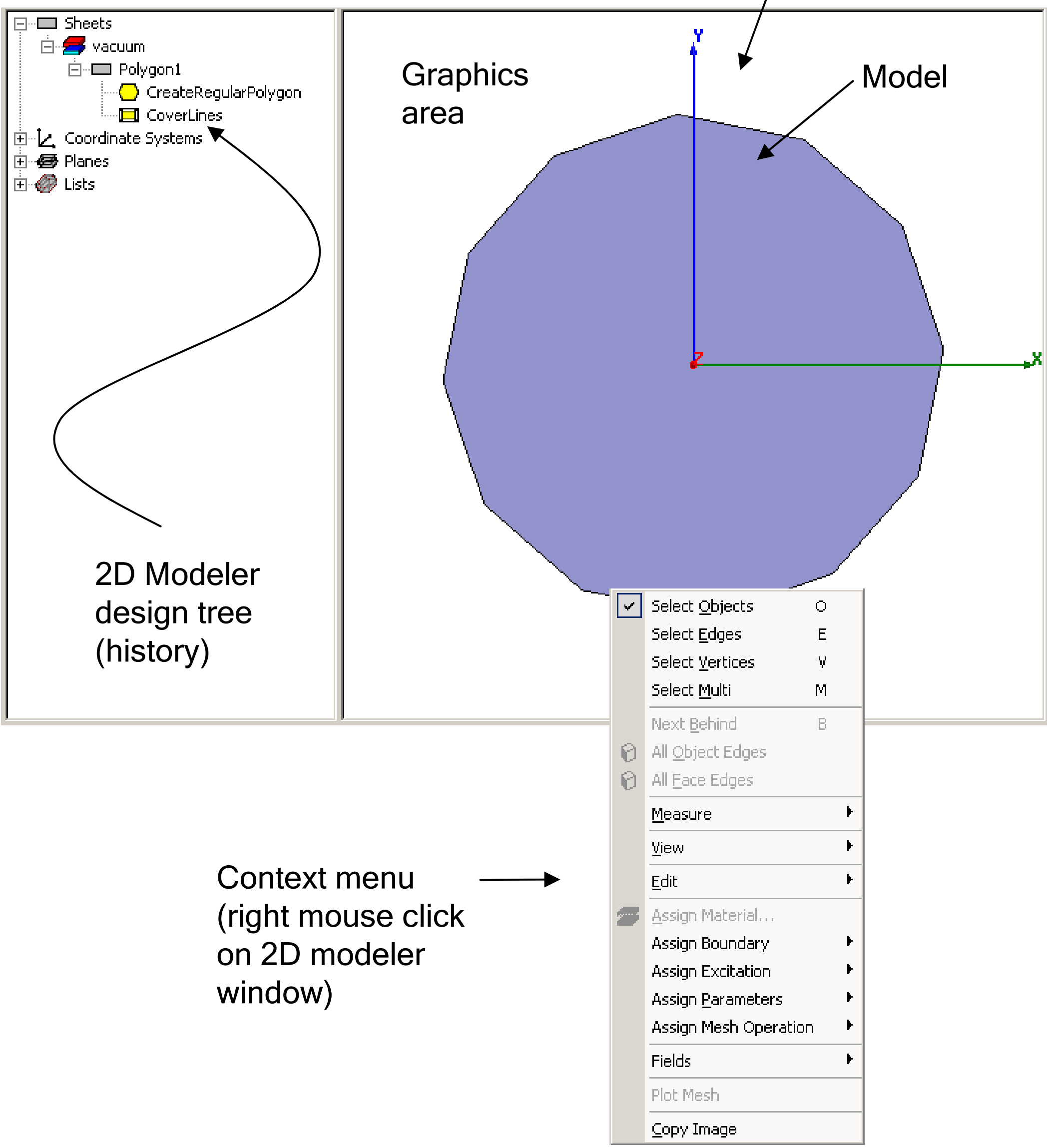


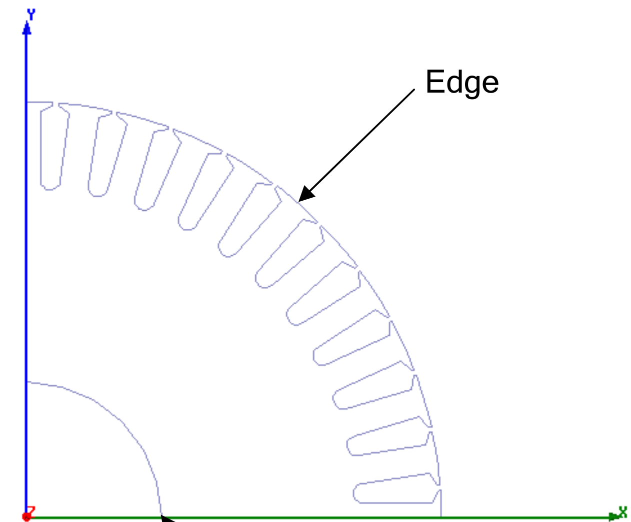
Maxwell Desktop - Project Manager

- Multiple Designs per Project
- Multiple Projects per Desktop
- Integrated Optimetrics Setup (requires license for analysis)



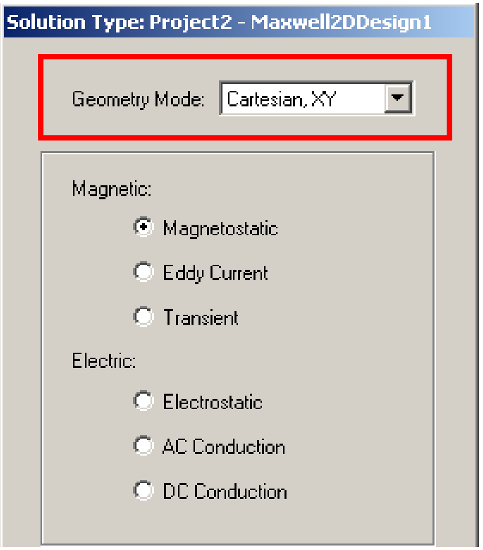
Maxwell Desktop - 2D Modeler





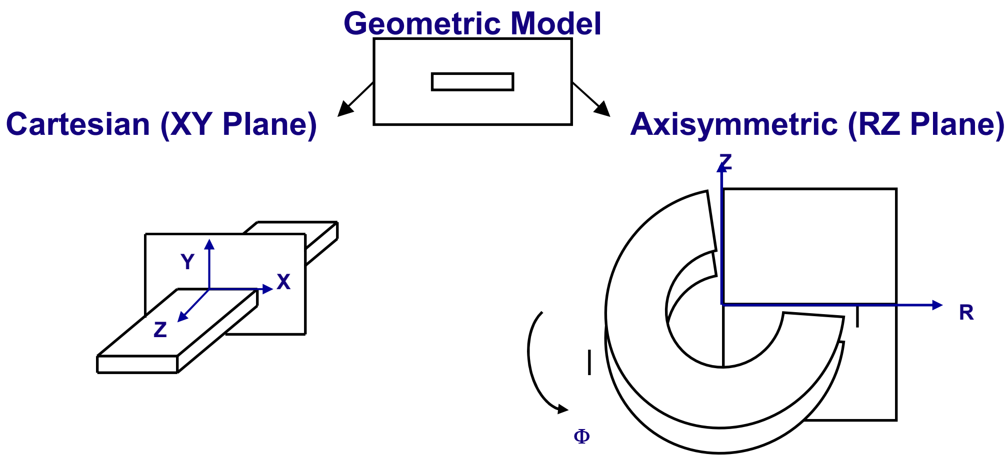
Geometry Mode

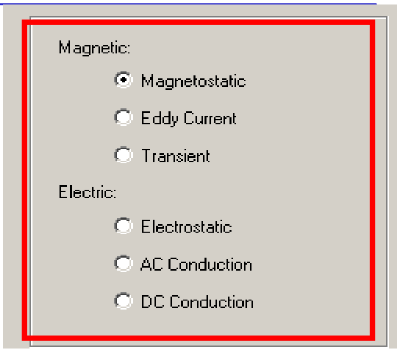
- To set the geometry mode:
  1. Select the menu item *Maxwell 2D > Solution Type*
  2. Solution Type Window:
     - Choose Geometry Mode: Cartesian XY



Maxwell - Geometry Modes

- A Cartesian (XY) model represents a cross-section of a device that extends in the z-direction. Visualize the geometric model as extending perpendicular to the plane being modeled.
- An Axisymmetric (RZ) model represents a cross-section of a device that is revolved 360° around an axis of symmetry (the z-axis). Visualize the geometric model as being revolved around the z-axis.





Set Solution Type

- To set the solution type: select the menu item *Maxwell 2D > Solution Type*

Magnetic Solution Types

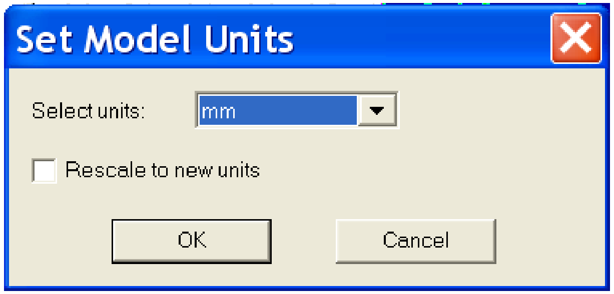
- Magnetostatic
  Computes the static magnetic field that exists in a structure given a distribution of DC currents and permanent magnets. The magnetic field may be computed in structures with both nonlinear and linear materials. An inductance matrix, force, torque, and flux linkage may also be computed from the energy stored in the magnetic field.
- Eddy Current
  Computes the oscillating magnetic field that exists in a structure given a distribution of AC currents. Also computes current densities, taking into account all eddy current effects (including skin effects). An impedance matrix, force, torque, core loss, and current flow may also be computed from the computed field solution.
- Transient
  Computes transient (Time Domain) magnetic fields caused by permanent magnets, conductors, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time, position and speed. It can also be coupled with external circuits. Rotational or translational motion effects can be included in the simulation. Uses a time-stepping solver. Considers source induced and motion inducted eddy effects.

Electric Solution Types

- Electrostatic
  Computes the static electric field that exists in a structure given a distribution of DC voltages and static charges. A capacitance matrix, force, torque, and flux linkage may also be computed from the electric field.
- AC Conduction
  Computes the AC voltages and current density distribution in a material having both conductive and dielectric properties given a distribution of AC voltages. An admittance matrix and current flow may also be computed from the calculated fields.
- DC Conduction
  Computes the DC currents that flow in a lossy dielectric given a distribution of DC voltages. A conductance matrix and current flow may also be computed from the computed electric field solution.

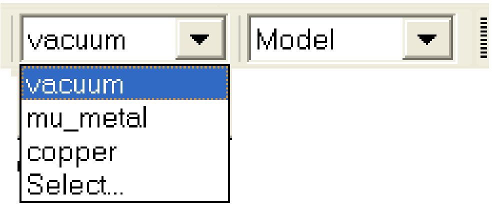
Set Model Units

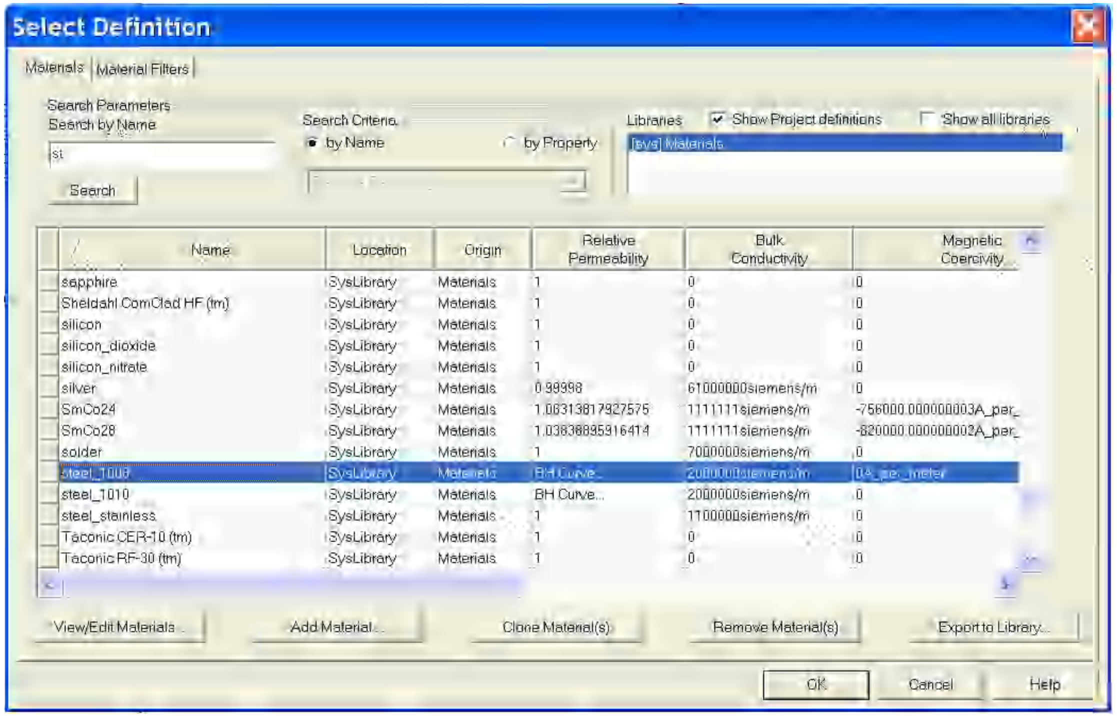
- To set the units:
  1. Select the menu item *Modeler > Units*
  2. Set Model Units:
     1. Select Units: mm
     2. Click the OK button



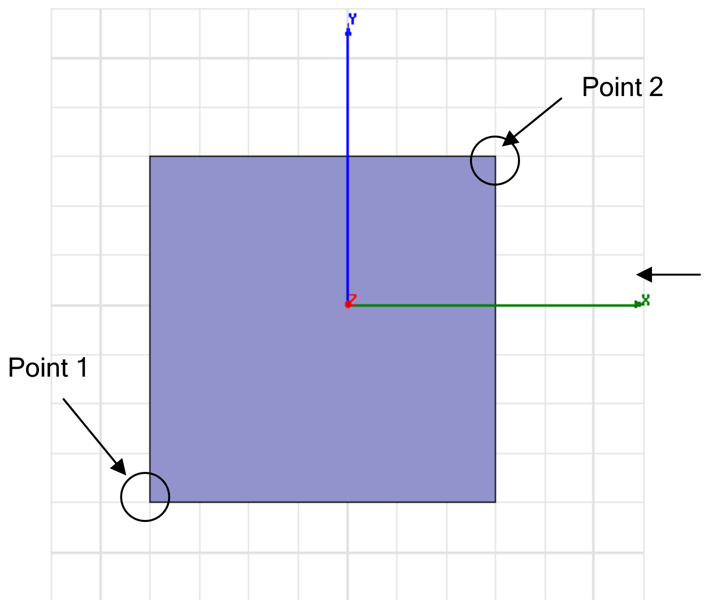
Set Default Material

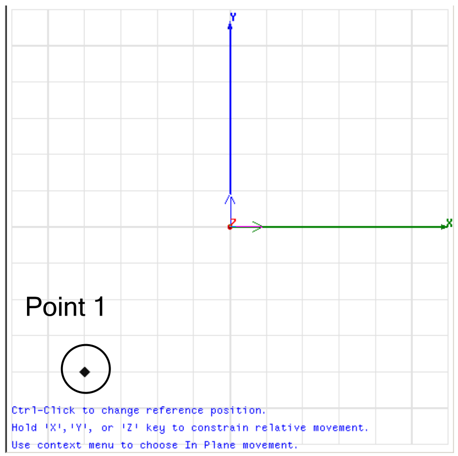
- To set the default material:
  1. Using the Modeler Materials toolbar, choose **Select**
  2. Select Definition Window:
     1. Type **steel_1008** in the **Search by Name** field
     2. Click the OK button

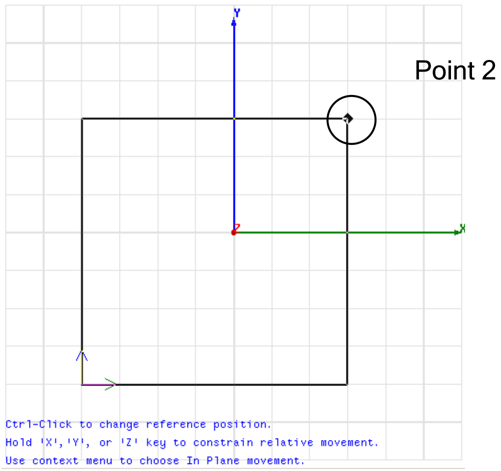


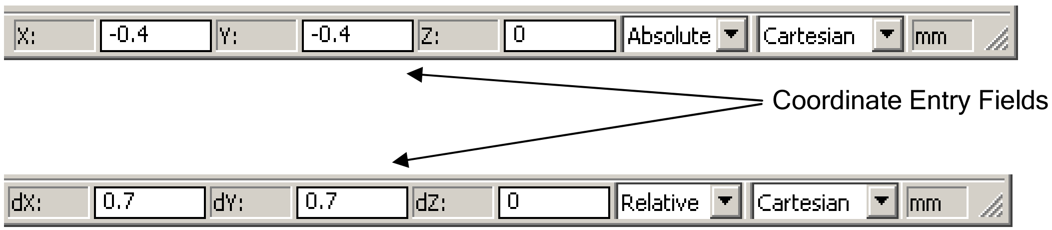


Modeler - Draw a Rectangle





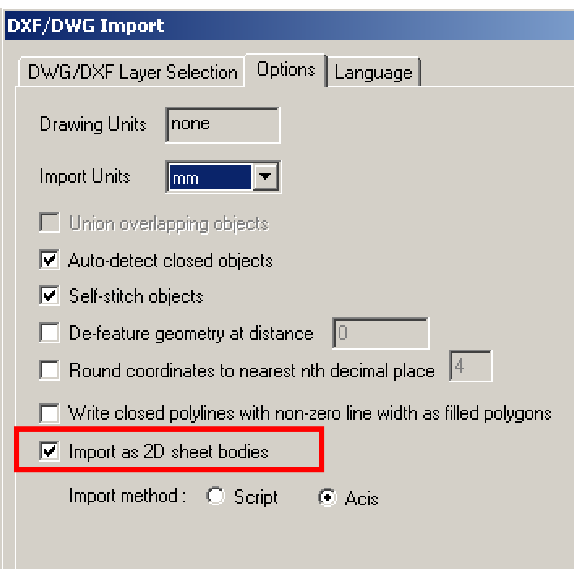


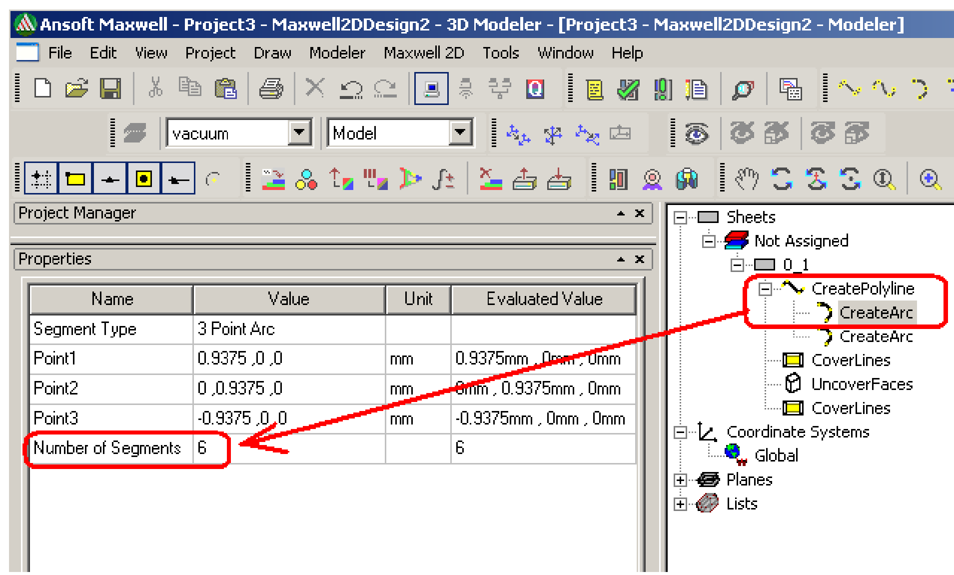


- The Coordinate Entry fields allow equations to be entered for position values.
  - Examples: 2*5, 2+6+8, 2*cos(10*(pi/180)).
- Variables are not allowed in the Coordinate Entry Field
- Note: Trig functions are in radians

Modeler – Importing .dxf and .dwg CAD files

- Check *"Import as 2D sheet bodies"* so objects come in as sheets and not solids
- To change the number of segments on an imported curve:
  - Change to face select mode: Edit > Select > Faces and click on face
  - Modeler > Surface > Uncover Faces
  - Change to object select mode: Edit > Select > Objects and click on open polyline
  - Modeler > Purge History
  - Modeler > Generate History
  - Expand the history tree for that polyline and change number of segments as desired
  - Select the polyline and: Modeler > Surface > Cover Lines

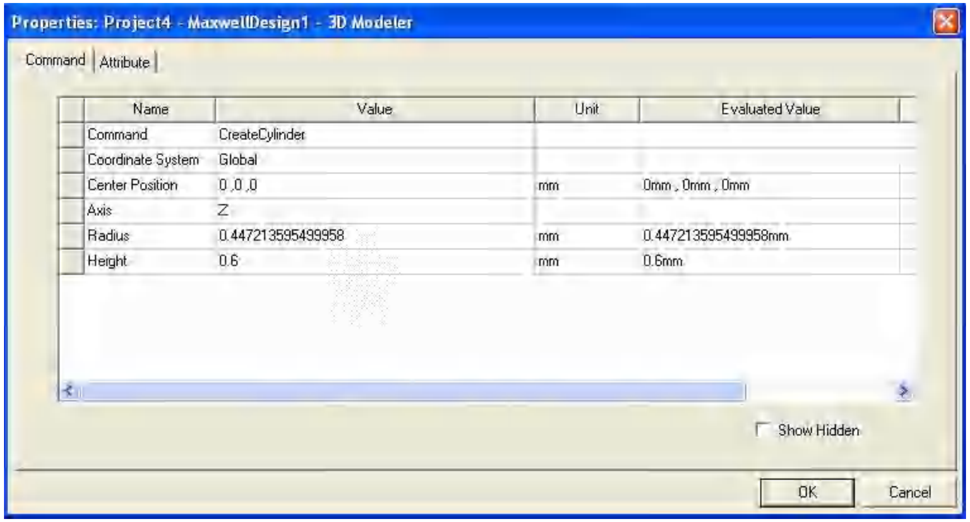




Modeler – Object Properties

Commands

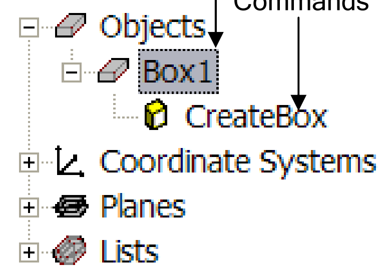
(dimensions
and history)



In History Tree:

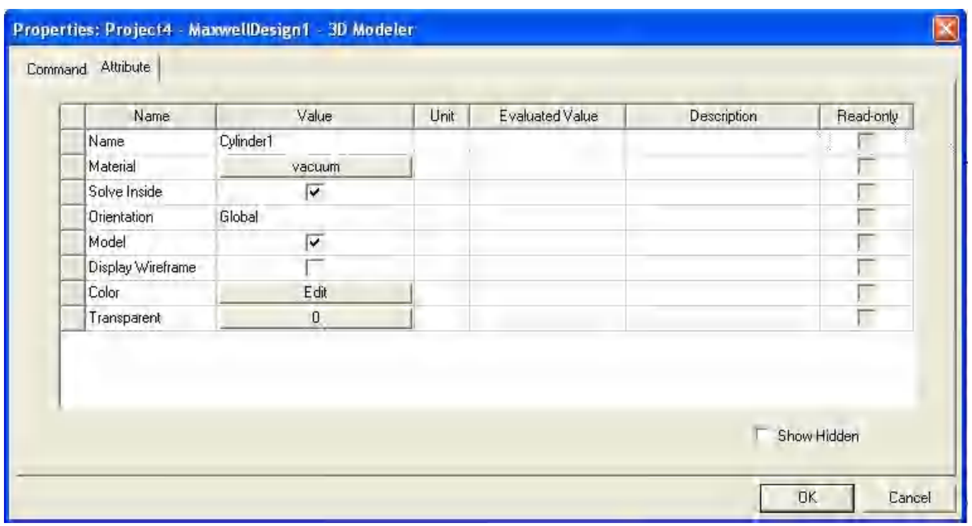
Attributes

Commands

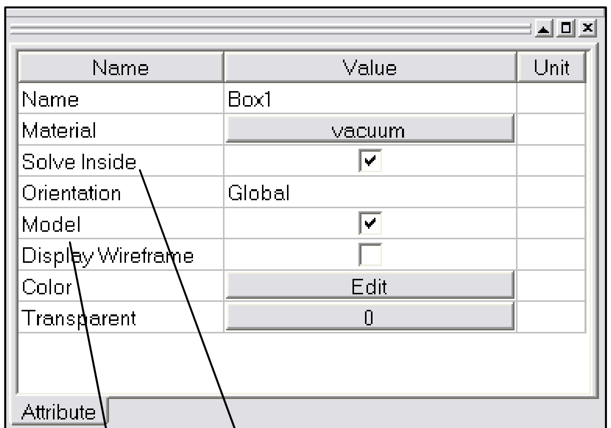


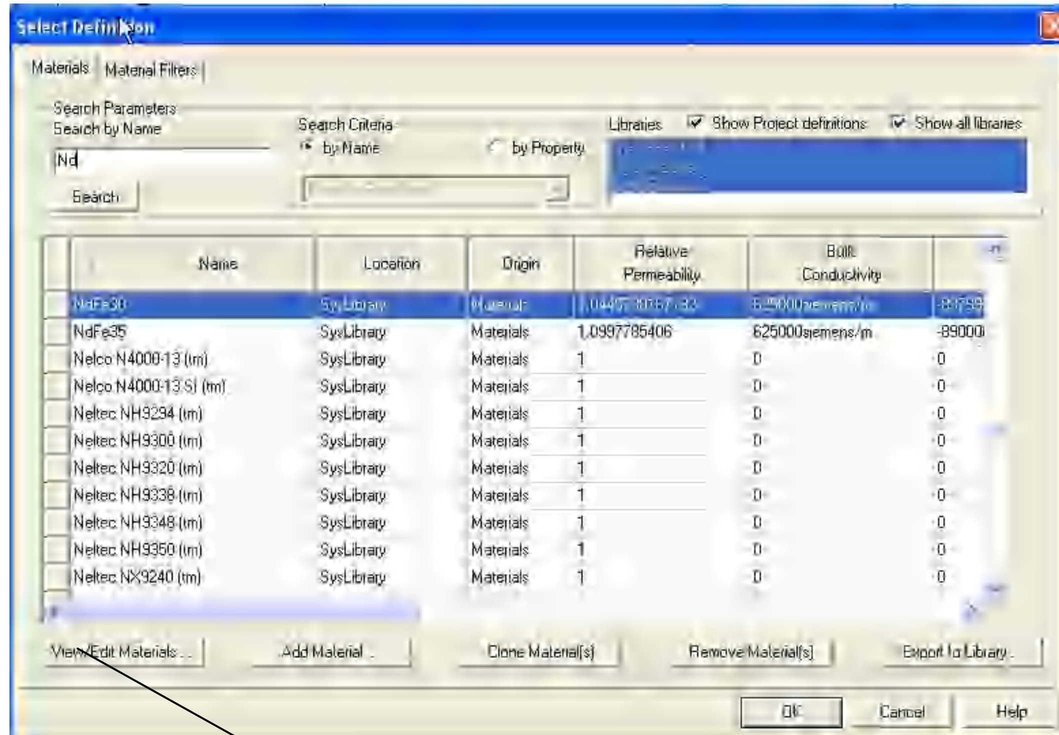
Attributes

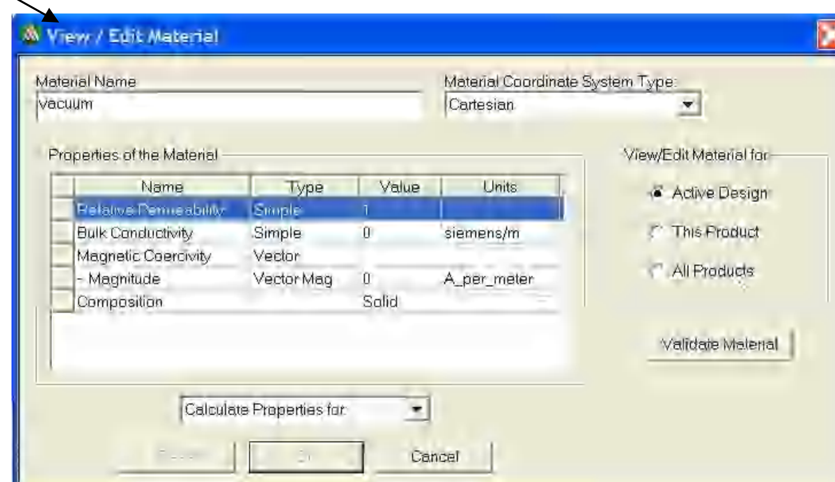
(properties
of the object)



Modeler - Attributes







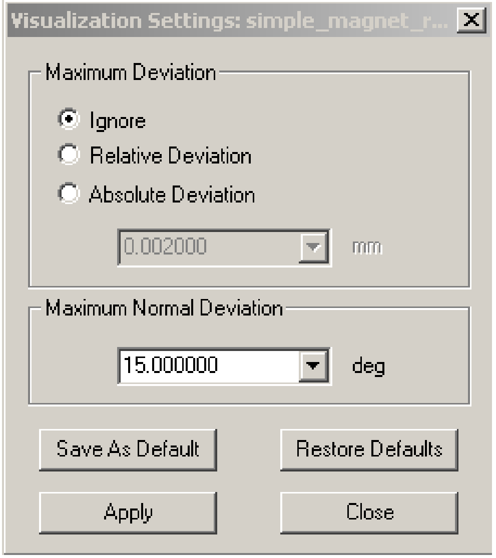
Solve Inside - if unchecked meshes but no solution inside (like the old exclude feature in material manager)

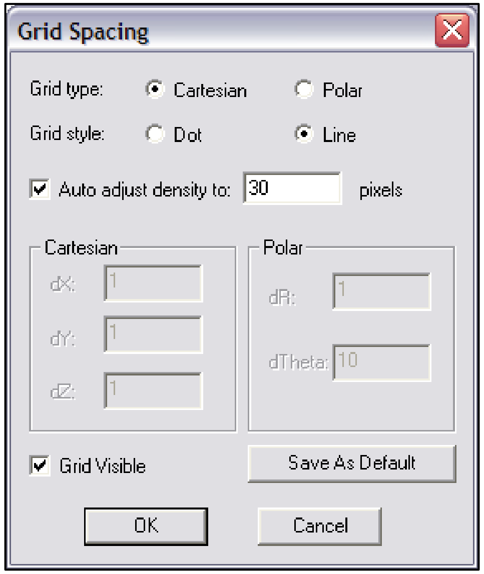
Model - if unchecked, the object is totally ignored outside of modeler with no mesh and no solution

Modeler - Views

- *View > Modify Attributes >*
  - *Orientation* - Predefined/Custom View Angles
  - *Lighting* - Control angle, intensity, and color of light
  - *Projection* - Control camera and perspective
  - *Background Color* - Control color of 3D Modeler background
- *View > Visualization Settings* - displayed resolution of curves
- *View > Active View Visibility* - Controls the display of: 3D Modeler Objects, Color Keys, Boundaries, Excitations, Field Plots
- *View > Options* - Stereo Mode, Drag Optimization, Color Key Defaults, Default Rotation
- *View > Render > Wire Frame* or *Smooth Shaded* (Default)
- *View > Coordinate System > Hide* or *Small (Large)*
- *View > Grid Setting* - Controls the grid display

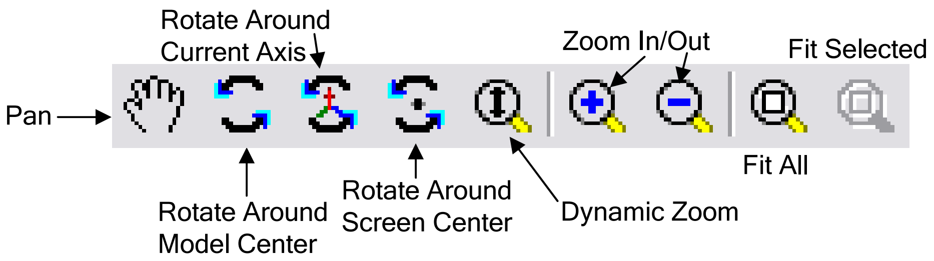
Toolbar: Toggle Grid Visibility



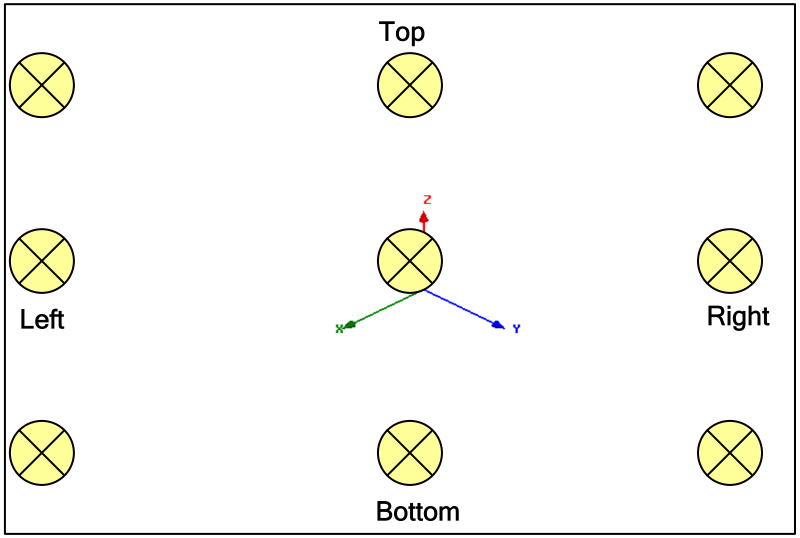


Changing the View

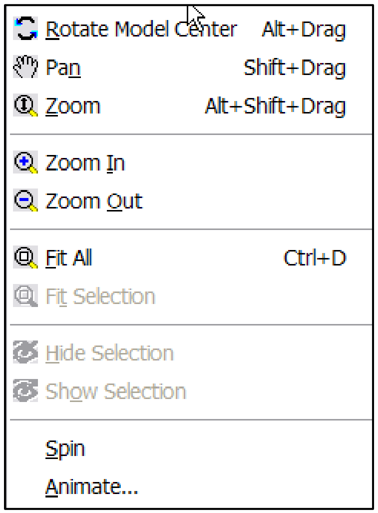
- **Toolbar**



Predefined View Angles



- **Context Menu**



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

Maxwell V12 Keyboard Shortcuts

General Shortcuts

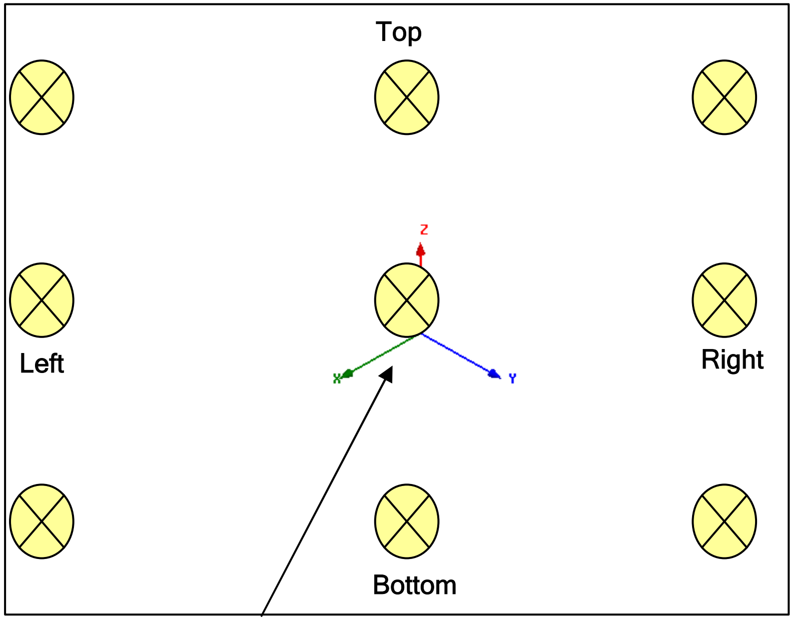
- F1: Help
- Shift + F1: Context help
- CTRL + F4: Close program
- CTRL + C: Copy
- CTRL + N: New project
- CTRL + O: Open...
- CTRL + S: Save
- CTRL + P: Print...
- CTRL + V: Paste
- CTRL + X: Cut
- CTRL + Y: Redo
- CTRL + Z: Undo
- CTRL + 0: Cascade windows
- CTRL + 1: Tile windows horizontally
- CTRL + 2: Tile windows vertically

Modeller Shortcuts

- B: Select face/object behind current selection
- F: Face select mode
- O: Object select mode
- CTRL + A: Select all visible objects
- CTRL + SHIFT + A: Deselect all objects
- CTRL + D: Fit view
- CTRL + E: Zoom in, screen center
- CTRL + F: Zoom out, screen center
- CTRL + Enter: Shifts the local coordinate system temporarily
- SHIFT + Left Mouse Button: Drag
- Alt + Left Mouse Button: Rotate model
- Alt + SHIFT + Left Mouse Button: Zoom in / out
- F3: Switch to point entry mode (i.e. draw objects by mouse)
- F4: Switch to dialogue entry mode (i.e. draw object solely by entry in command and attributes box.)
- F6: Render model wire frame
- F7: Render model smooth shaded

- Alt + Double Click Left Mouse Button at points on screen: Sets model projection to standard isometric projections (see diagram below).
- ALT + Right Mouse Button + Double Click Left Mouse Button at points on screen: give the nine opposite projections.

Predefined View Angles

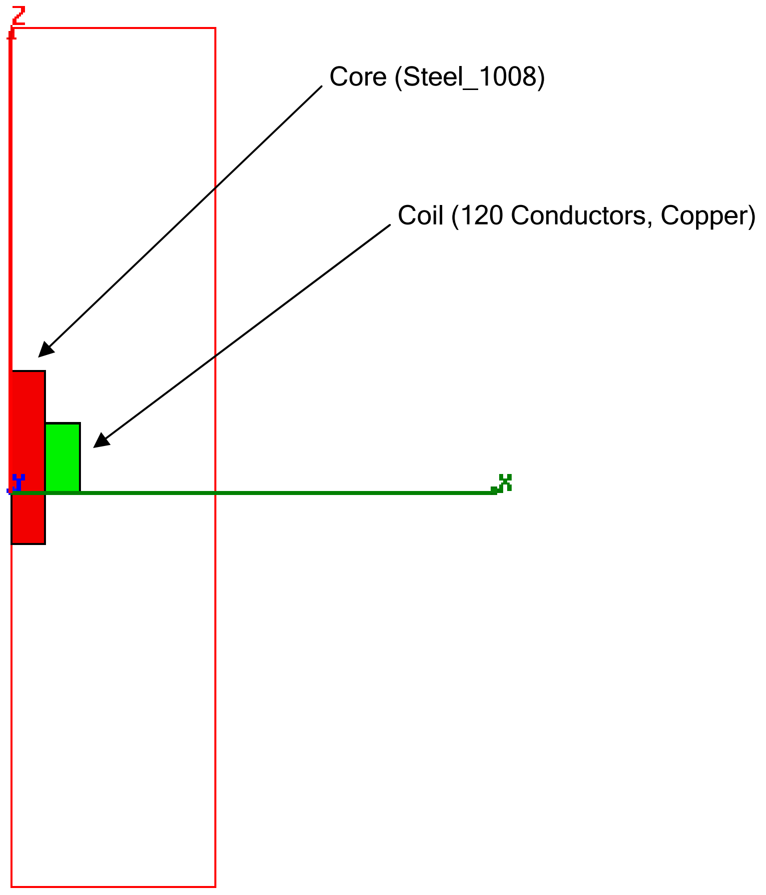


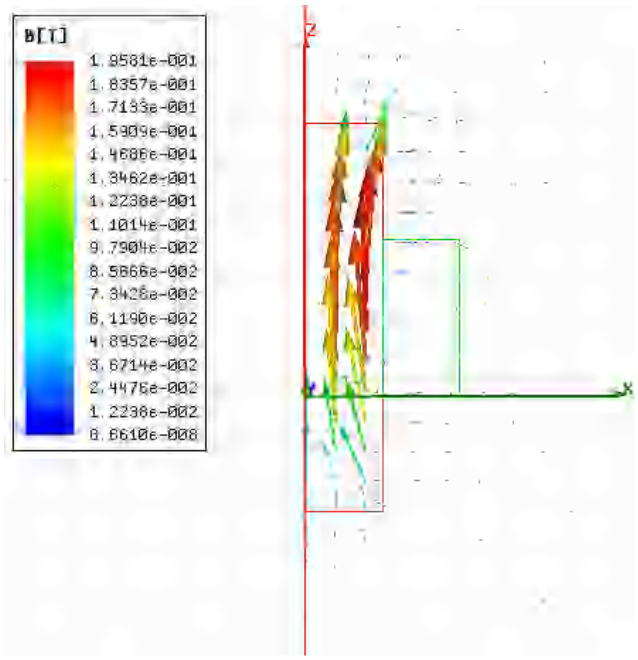
Alt + double left Click here to restore view in an RZ model

Alt + double left Click here to restore view in an XY model

Simple Example

- Magnetic core with coil
- Use 2D RZ Magnetostatic Solver



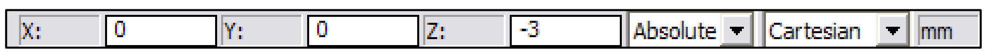


Setup the geometry mode and solver

- Choose *Cylindrical about Z* under *Maxwell 2D > Solution Type*
- Choose **Magnetostatic**
- Click the **OK** button

Create Core

- **To create the core:**
  1. Select the menu item *Draw > Rectangle*
  2. Using the coordinate entry fields, enter the center position
     - X: **0.0**, Y: **0.0**, Z: **-3.0**, Press the **Enter** key

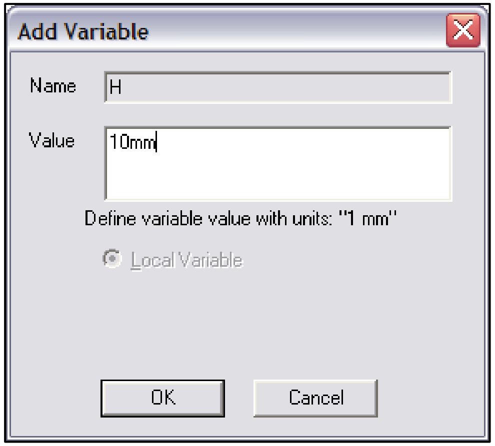


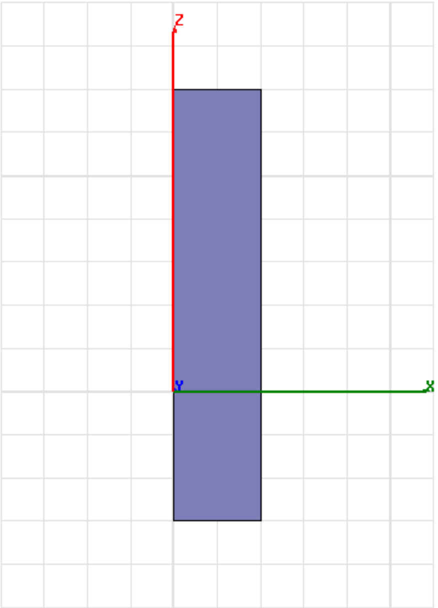
  3. Using the coordinate entry fields, enter the opposite corner of the rectangle
     - dX: **2.0**, dY: **0.0**, dZ: **10.0**, Press the **Enter** key

- **Continued on Next Page**

Create Core (Continued)

- To Parameterize the Height
  1. Select the **Command** tab from the **Properties** window
  2. ZSize: **H**
  3. Press the **Tab** key
  4. Add Variable Window
     1. Value: **10mm**
     2. Click the **OK** button
- To set the name:
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Core**
- To set the material:
  1. Select the **Attribute** tab from the **Properties** window
  2. Click on the button in Material value: set to steel_1008
- To set the color:
  1. Select the **Attribute** tab from the **Properties** window.
  2. Click the **Edit** button
- To set the transparency:
  1. Select the **Attribute** tab from the **Properties** window.
  2. Click the **OK** button
- To finish editing the object properties
  1. Click the **OK** button
- To fit the view:
  1. Select the menu item *View > Fit All > Active View*





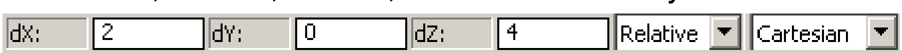
Set Default Material

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

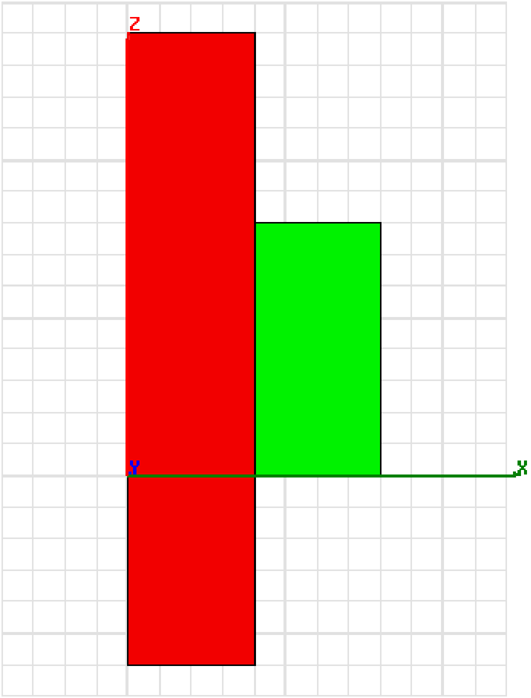
Create Coil

- **To create the coil for the current to flow:**
  1. Select the menu item *Draw > Rectangle*
  2. Using the coordinate entry fields, enter the center position
     - X: **2.0**, Y: **0.0**, Z: **0.0**, Press the **Enter** key

  3. Using the coordinate entry fields, enter the opposite corner of the re
     - dX: **2.0**, dY: **0.0**, dZ: **4.0**, Press the **Enter** key



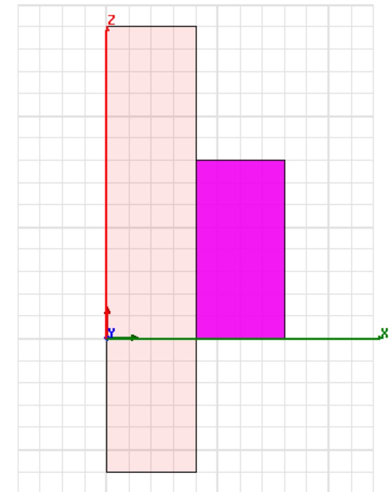
- **To set the name:**
  1. Select the **Attribute** tab from the **Properties** window.
  2. For the **Value** of **Name** type: **Coil**
  3. Click the **OK** button
- **To fit the view:**
  1. Select the menu item *View > Fit All > Active View*

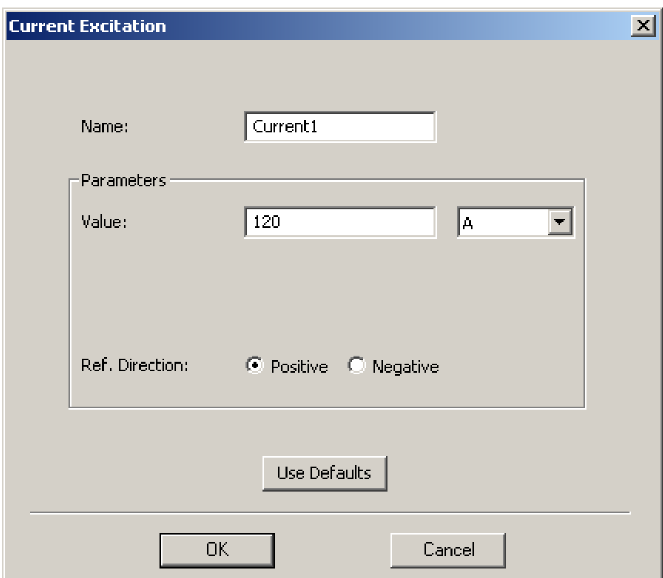


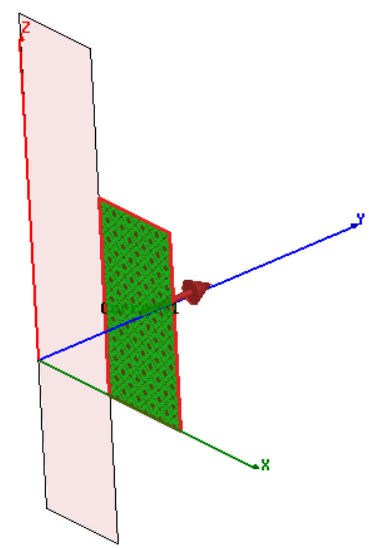
Create Excitation

Assign Excitation

1. Click on the coil.
2. Select the menu item *Maxwell 2D > Excitations > Assign > Current*
3. Current Excitation : General
   1. Name: **Current1**
   2. Value: **120 A** (Note: this is 120 Amp-turns)
   3. Ref. Direction: **Positive**
4. Click the **OK** button
5. Note that for RZ models, positive current flows into the screen, however for XY models, positive current flows out of the screen.

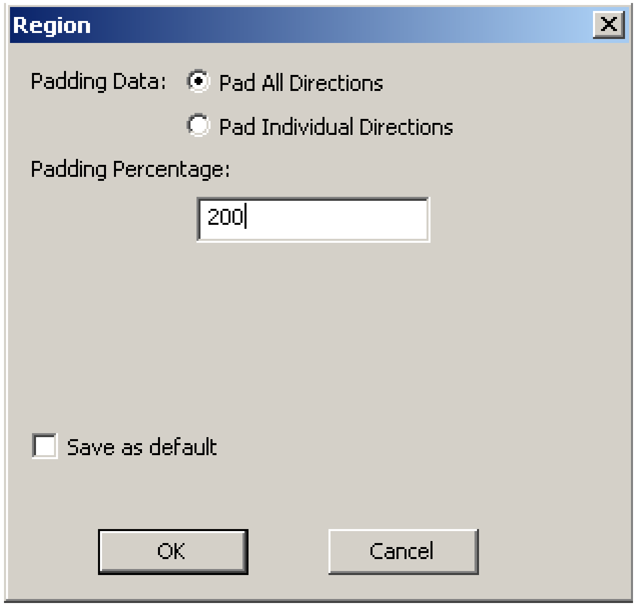


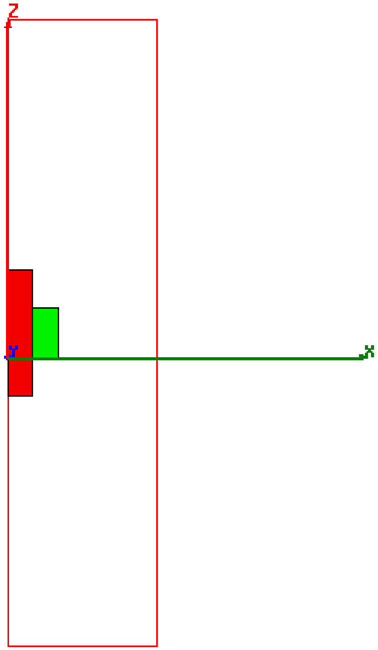




Define a Region

- Before solving a project a region has to be defined. A region is basically an outermost object that contains all other objects. The region can be defined by a special object in *Draw > Region*. This special region object will be resized automatically if your model changes size.
- A ratio in percents has to be entered that specifies how much distance should be left from the model.
  - **To define a Region:**
    1. Select the menu item *Draw > Region*
        1. Padding Data: **One**
        2. Padding Percentage: **200**
        3. Click the **OK** button



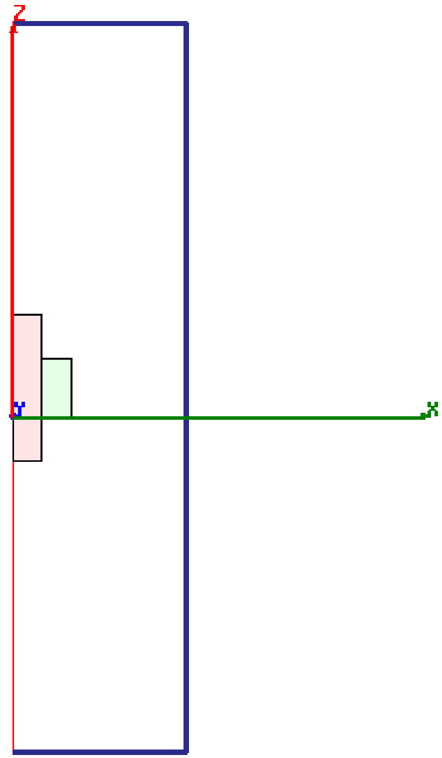


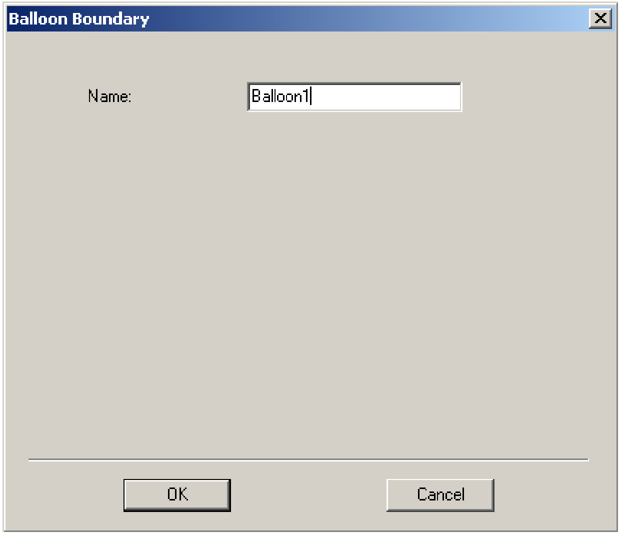
Note: Since there will be considerable fringing in this device, a padding percentage of at least 2 times, or 200% is recommended

Setup Boundary

Assign Boundary

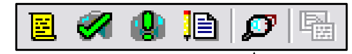
1. Change to edge selection mode by choosing: *Edit > Select > Edges*
2. Using the mouse, click on the top, right and bottom edges while holding down the CTRL key.
3. Select the menu item *Maxwell 2D > Boundary > Assign > Balloon*
4. Click the OK button



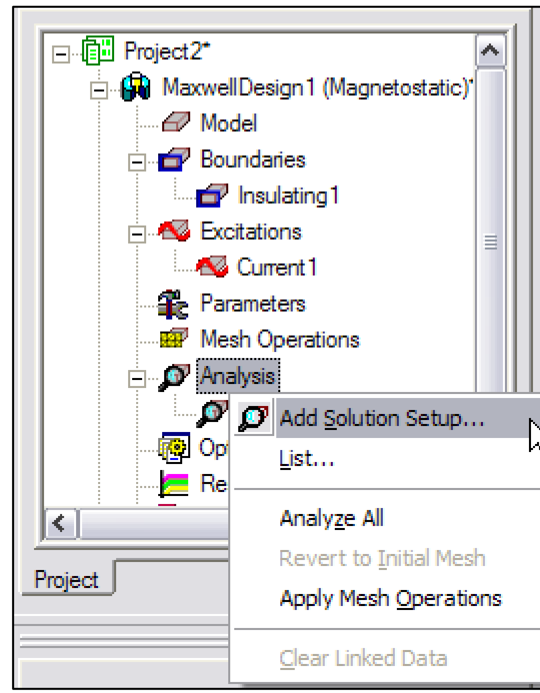


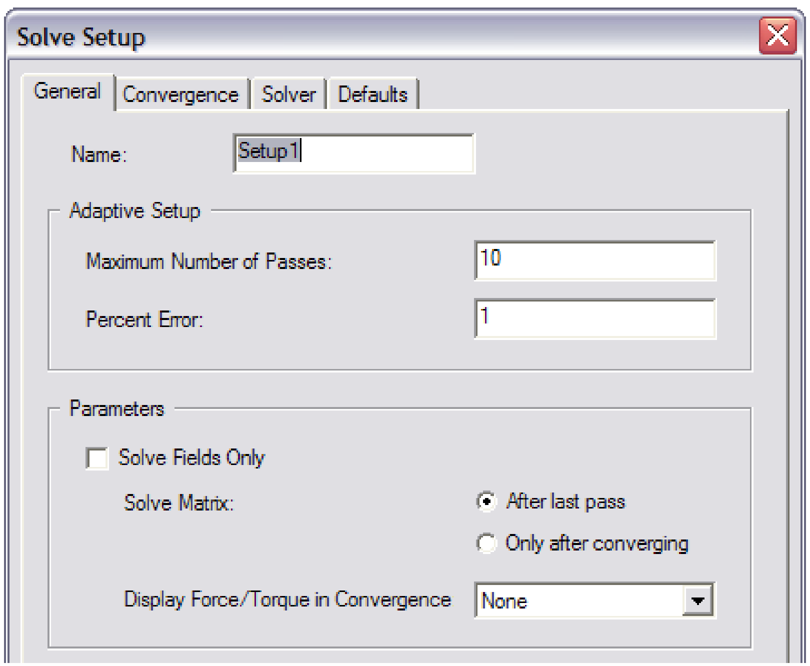
Solution Setup - Creating an Analysis Setup

- To create an analysis setup:
  1. Select the menu item *Maxwell 2D> Analysis Setup > Add Solution Setup*
  2. Solution Setup Window:
     1. Click the **General** tab:
        - Maximum Number of Passes: 10
        - Percent Error: 1
     2. Click the **OK** button



Add Solution Setup



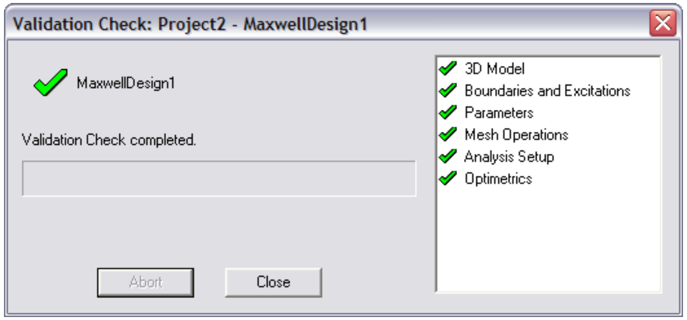


Save Project

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

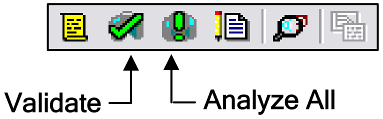
Model Validation

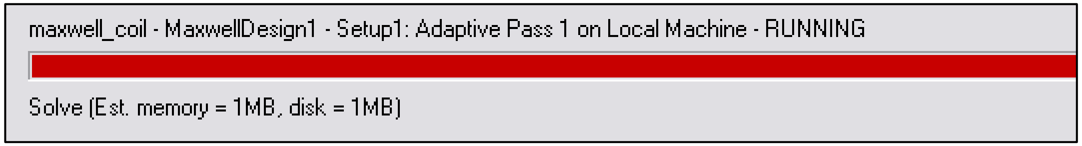
- To validate the model:
  1. Select the menu item *Maxwell 3D> Validation Check*
  2. Click the **Close** button
     - **Note:** To view any errors or warning messages, use the Message Manager.



Analyze

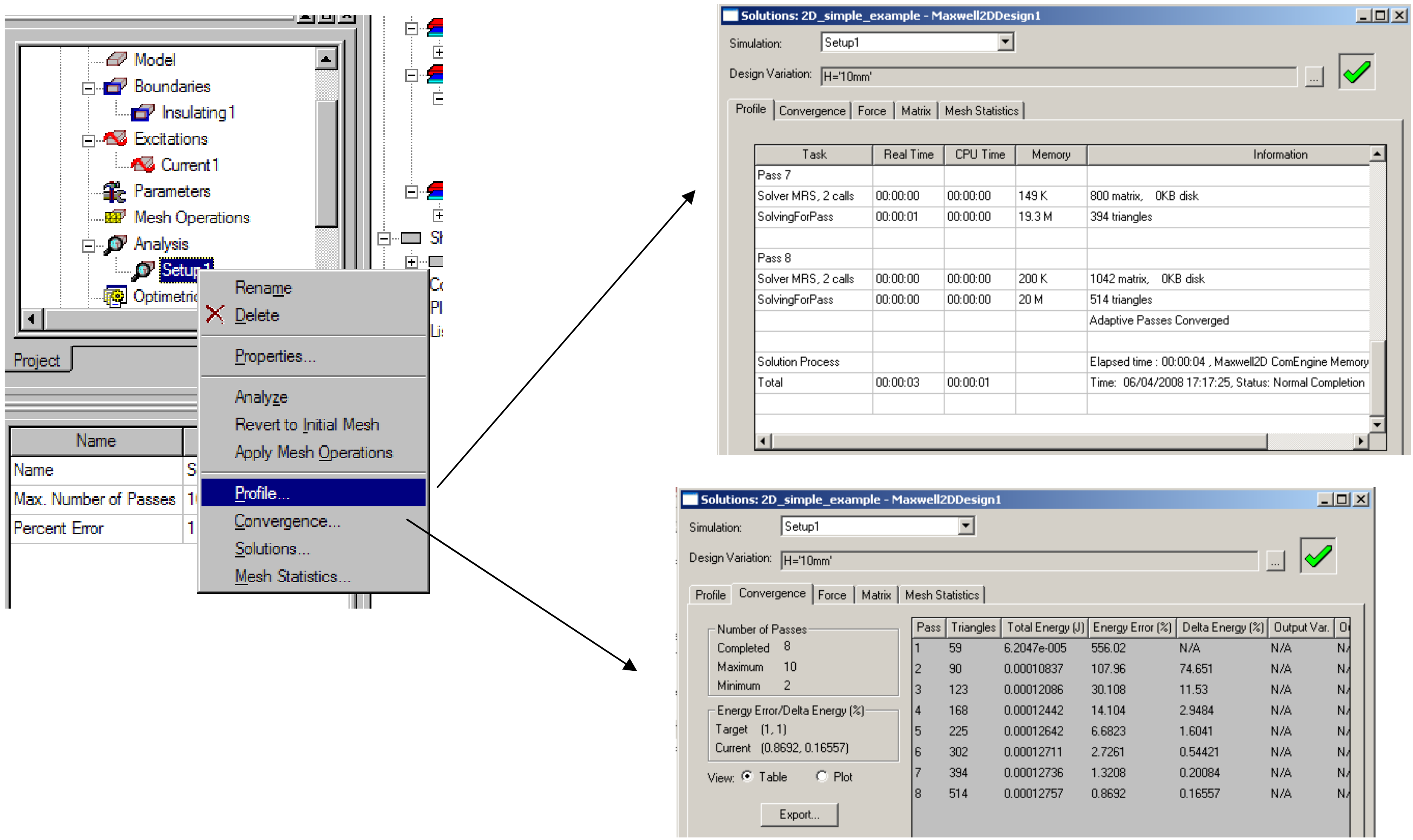
- To start the solution process:
  1. Select the menu item *Maxwell 2D> Analyze All*





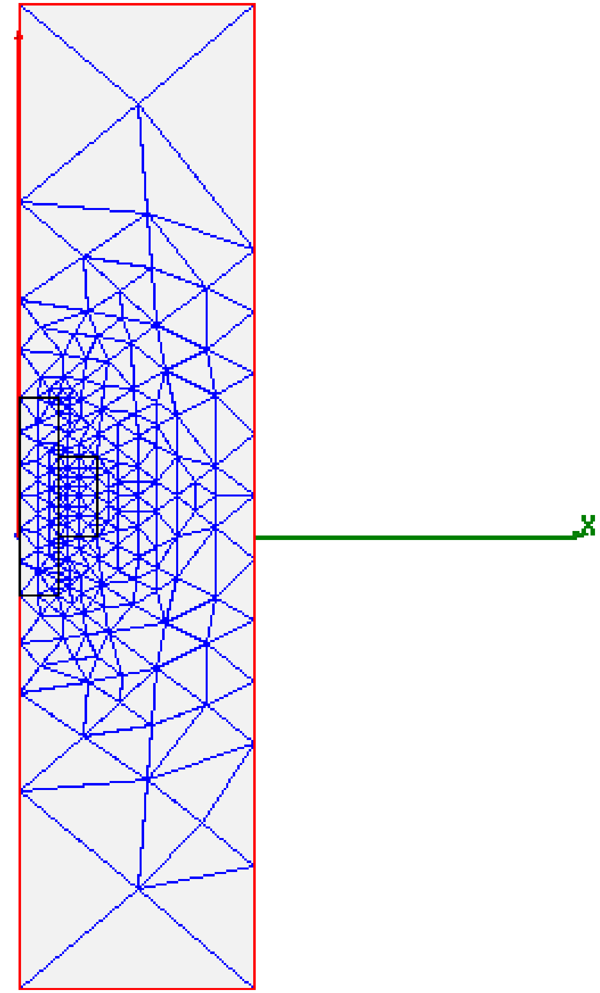
View detailed information about the progress

- In the Project Tree click on **Analysis > Setup1** with the right mouse button und select **Profile**



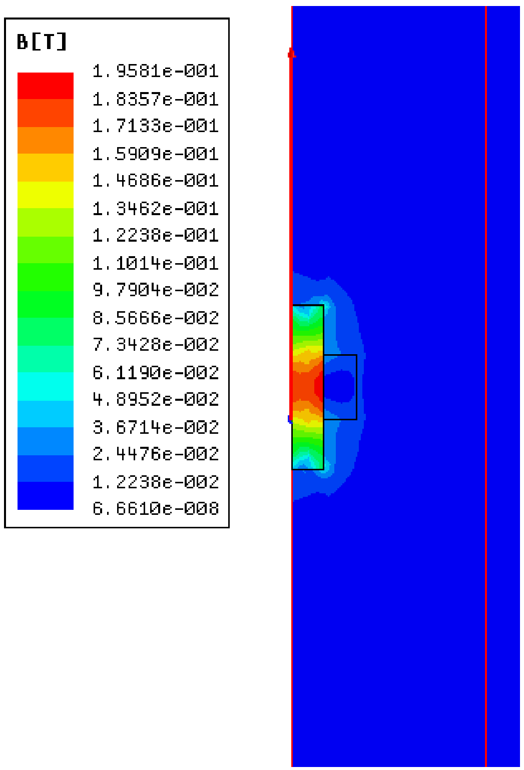
Mesh Overlay

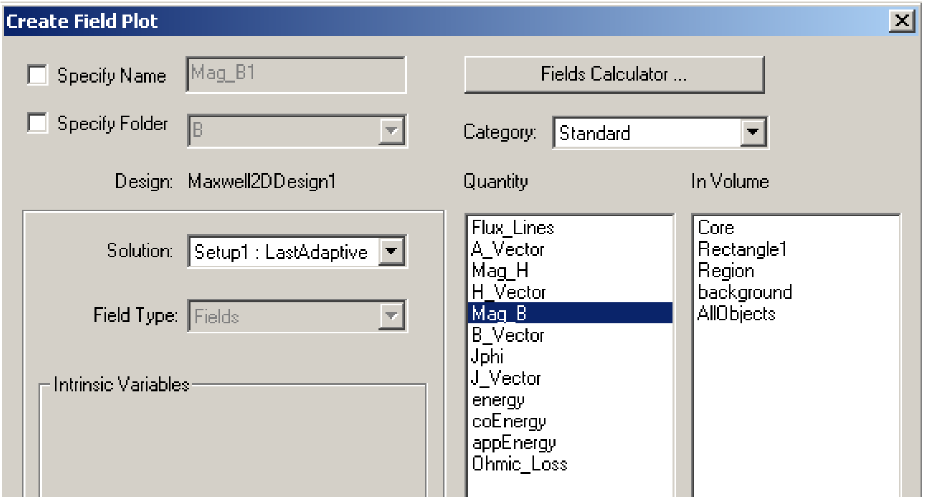
- Create a plot of the mesh
  1. Select the menu item *Edit > SelectAll*
- To create a mesh plot:
  1. Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  2. Create Mesh Window:
     1. Click the **Done** button



Field Overlays

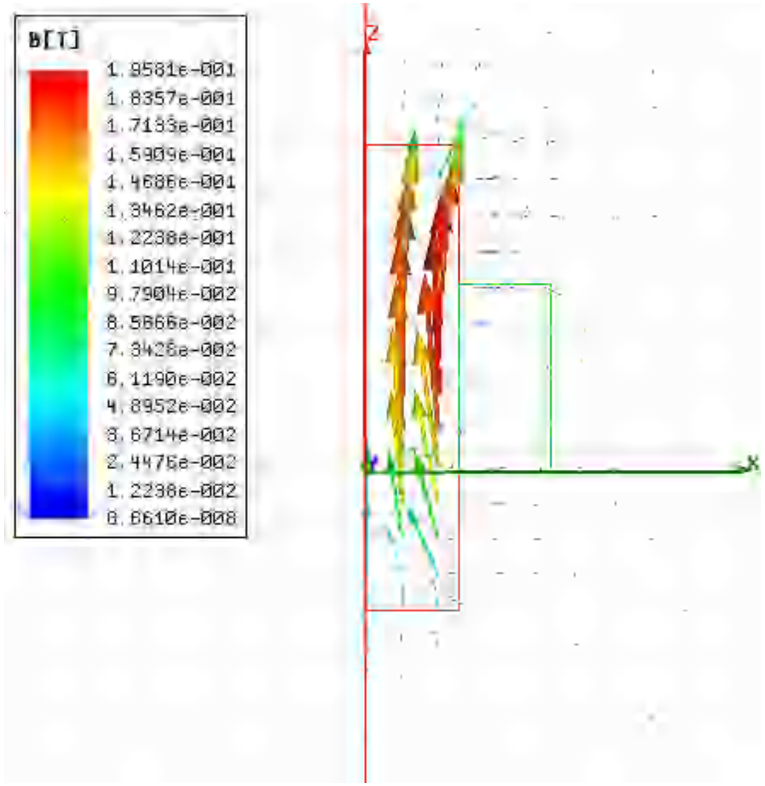
- **To create a field plot:**
  1. In the object tree, select the plane for plotting:
     1. Using the Model Tree, expand **Planes**
     2. Select **Global:XZ**
  2. Select the menu item *Maxwell 2D> Fields > Fields > B > Mag_B*
  3. Create Field Plot Window
     1. Solution: **Setup1 : LastAdaptive**
     2. Quantity: **Mag_B**
     3. In Volume: **Allobjects**
     4. Click the **Done** button
  4. When done, turn off the plot using:
     View > Active View Visibility > Filed Reporter





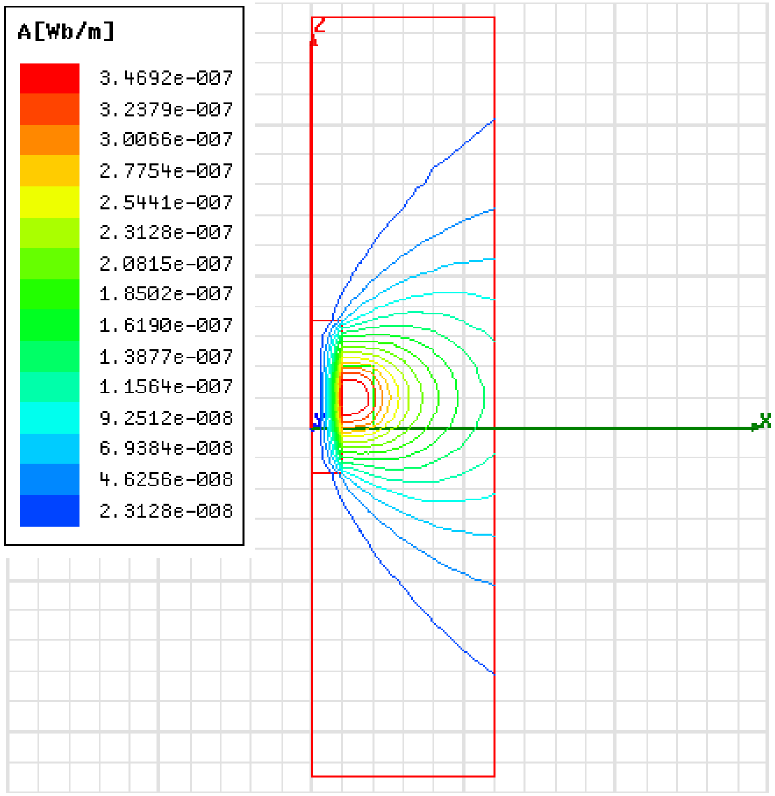
Field Overlays (cont)

- Create another field plot:
  1. In the object tree, select the plane for plotting:
     1. Using the Model Tree, expand **Planes**
     2. Select **Global:XZ**
  2. Select the menu item *Maxwell 2D> Fields > Fields > B > B_Vector*
  3. Create Field Plot Window
     1. Solution: **Setup1 : LastAdaptive**
     2. Quantity: **B_Vector**
     3. In Volume: **Allobjects**
     4. Click the **Done** button
  4. When done, turn off the plot using:
     View > Active View Visibility > Filed Reporter



Field Overlays (cont)

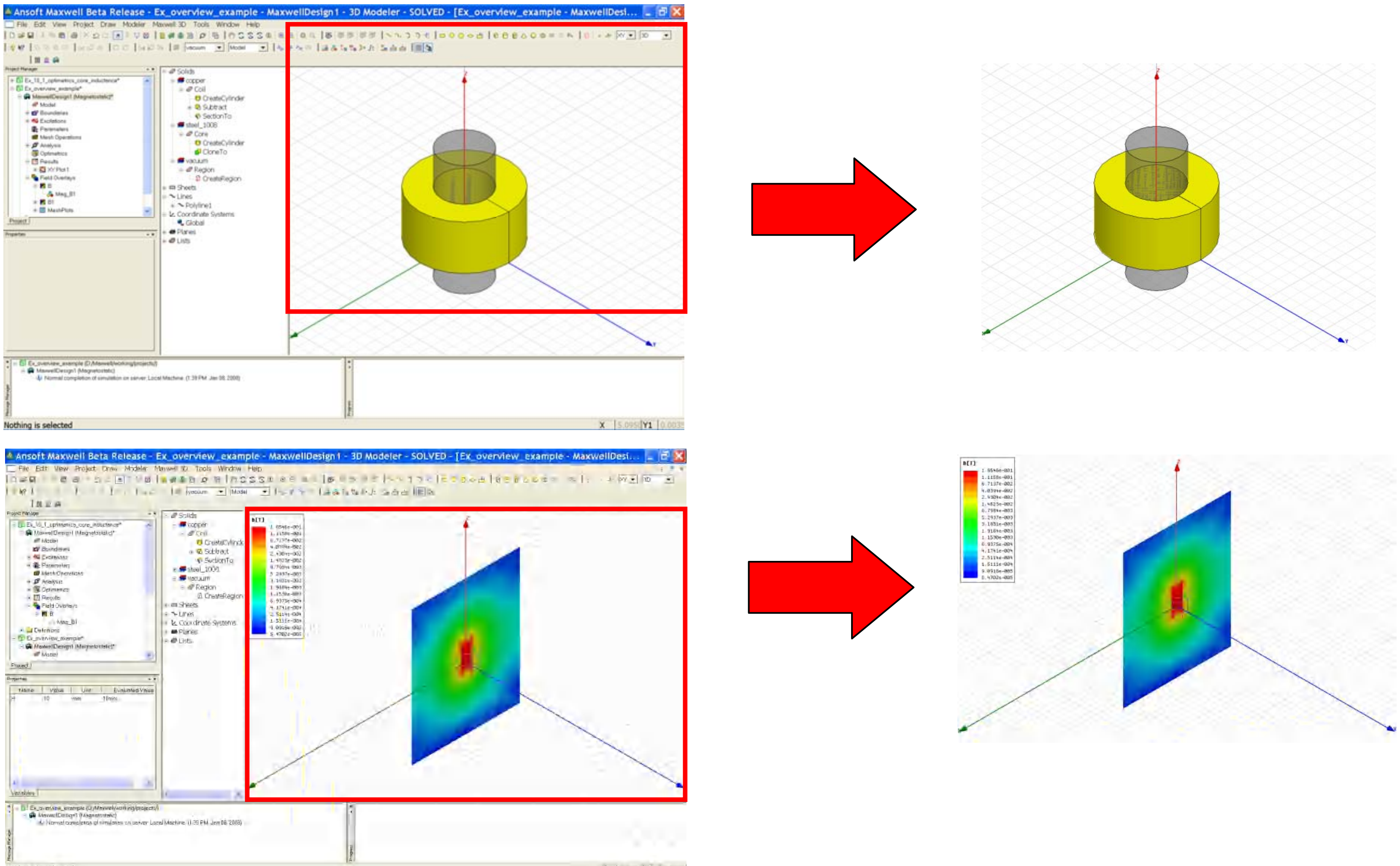
- **Create another field plot:**
  1. In the object tree, select the plane for plotting:
     1. Using the Model Tree, expand **Planes**
     2. Select **Global:XZ**
  2. Select the menu item *Maxwell 2D> Fields > Fields > A > Flux_Lines*
  3. Create Field Plot Window
     1. Solution: **Setup1 : LastAdaptive**
     2. Quantity: *Flux_Lines*
     3. In Volume: **Allobjects**
     4. Click the **Done** button
  4. When done, turn off the plot using:
     View > Active View Visibility > Filed Reporter



This completes the simple example.

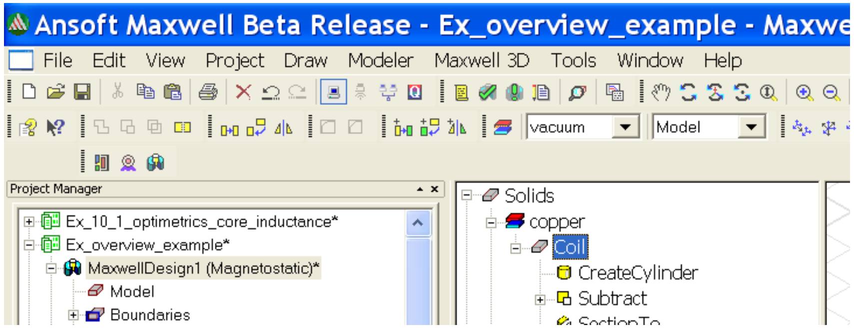
Screen Capturing

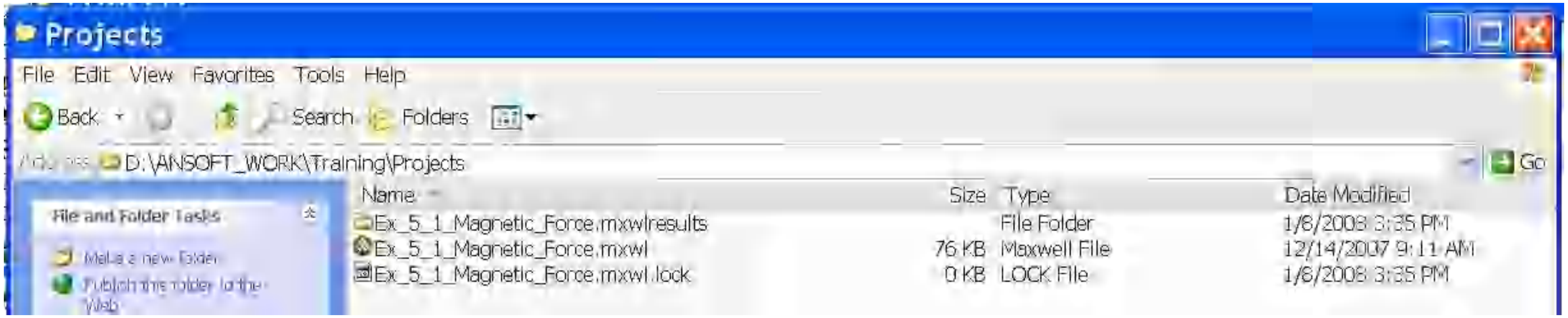
- To save the drawing Window or a plot to the clipboard select the menu item: *Edit > Copy Image*
- In any Windows application, select: *Edit > Paste* to paste the image



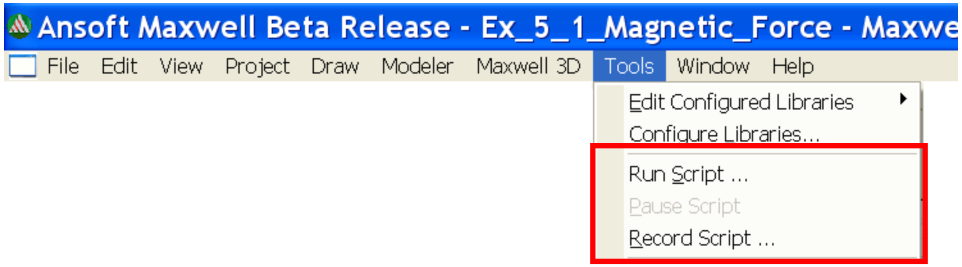
File Structure

- Everything regarding the project is stored in an ascii file
  - File: <project_name>.mxwl
  - Double click from Windows Explorer will open and launch Maxwell v12
- Results and Mesh are stored in a folder named <project_name>.mxwlresults
- Lock file: <project_name>.lock.mxwl
  - Created when a project is opened
- Auto Save File: <project_name>.mxwl.auto
  - When recovering, software only checks date
    - If an error occurred when saving the auto file, the date will be newer then the original
    - Look at file size (provided in recover dialog)

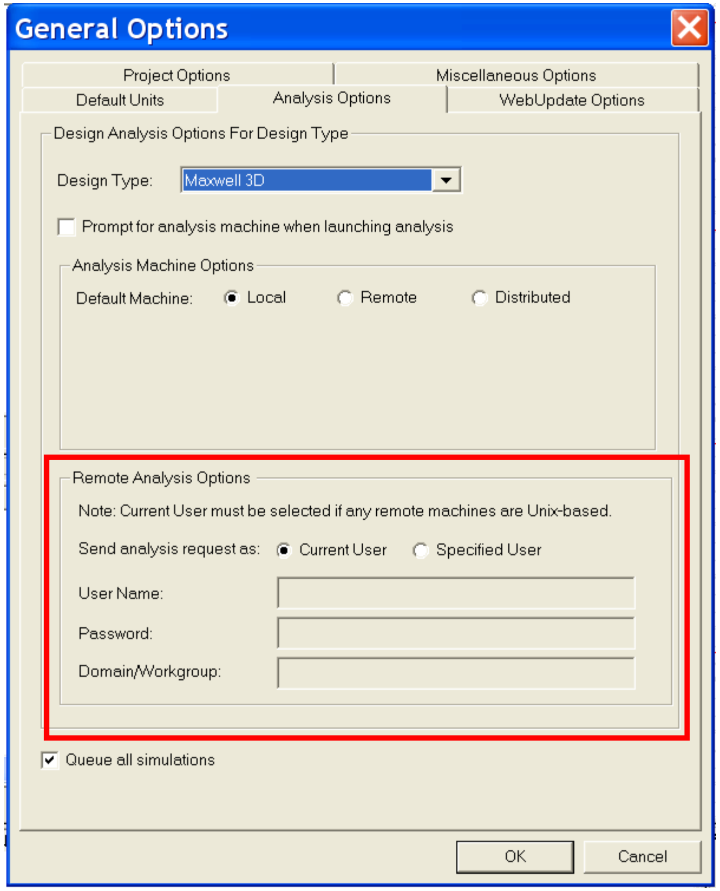




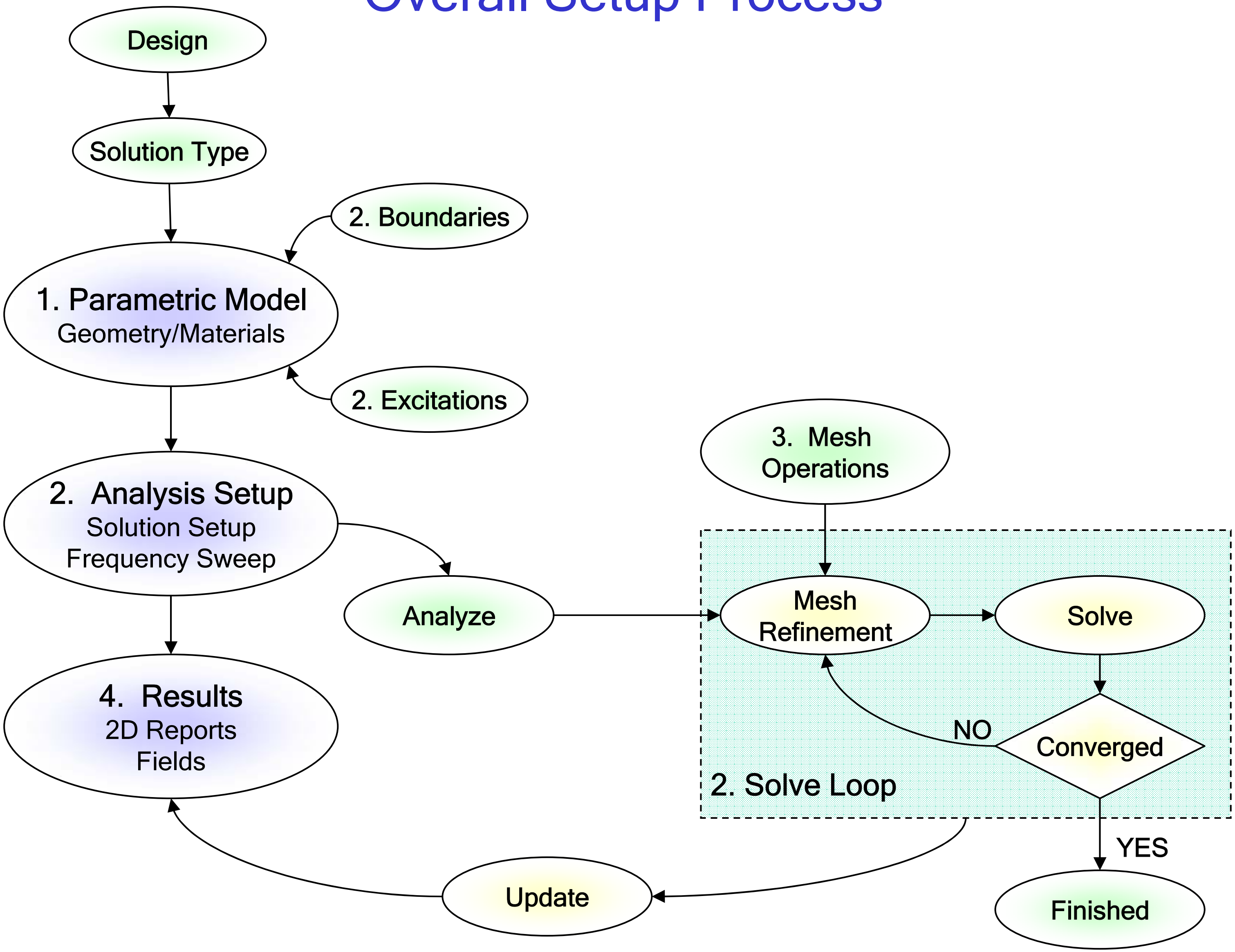
- **Scripts**
  - Default Script recorded in v12
    - Visual Basic Script



- **Remote Solve (Windows Only)**
  - *Tools > Options > General Options > Analysis Options*

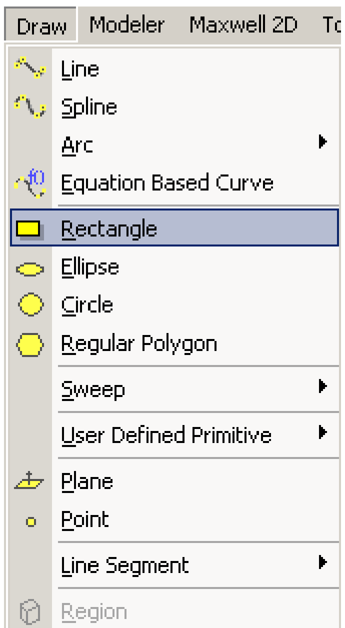


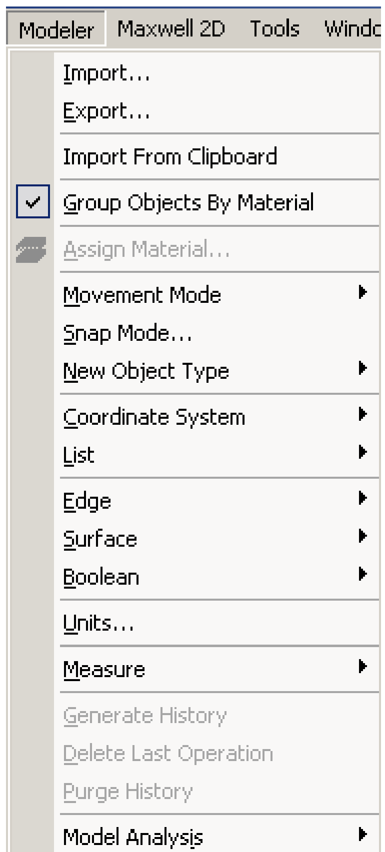
Overall Setup Process

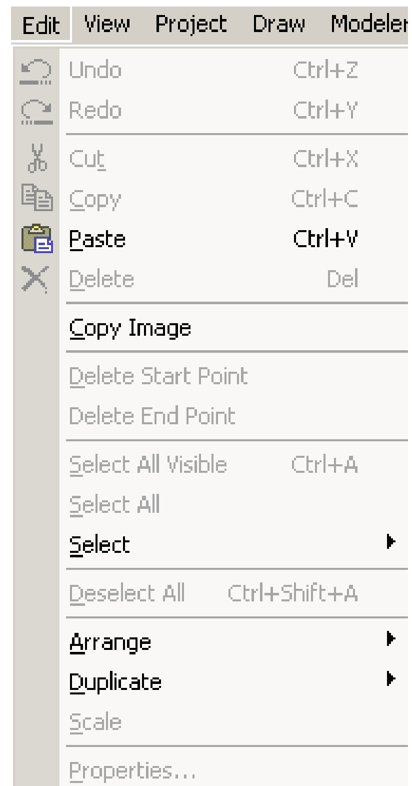


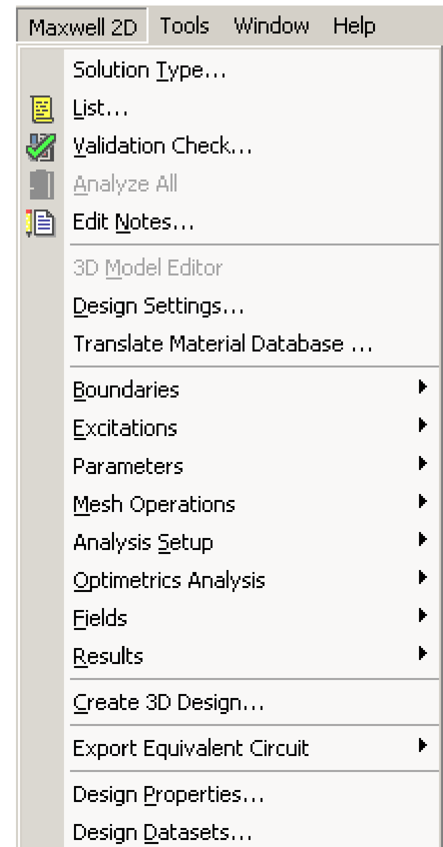
Menu Structure

- **Draw** - Primitives
- **Modeler** - Settings and Boolean Operations
- **Edit** - Copy/Paste, Arrange, Duplicate
- **Maxwell 2D** - Boundaries, Excitations, Mesh Operations, Analysis Setup, Results



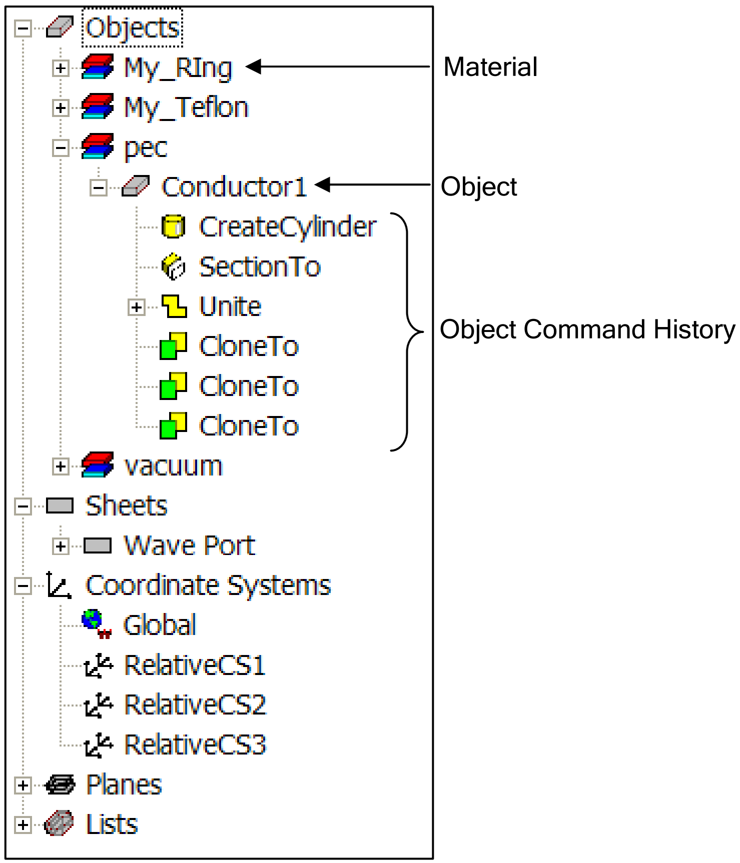




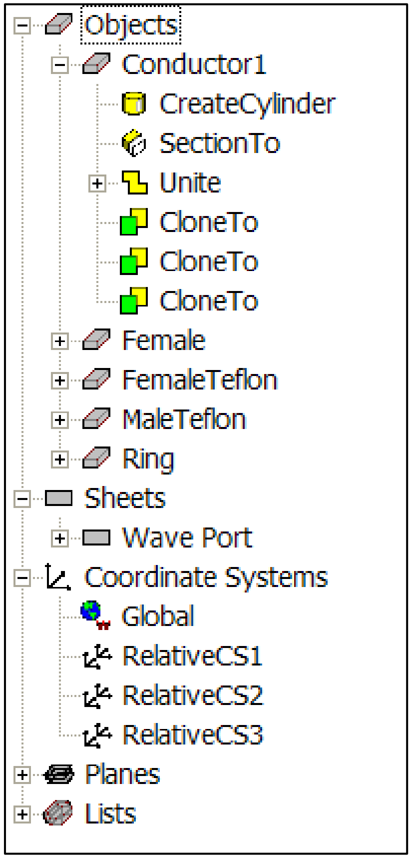


Modeler - Model Tree

- Select menu item *Modeler > Group by Material*



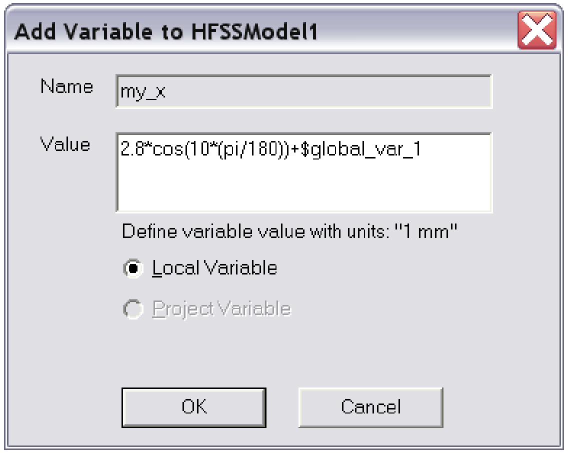
Grouped by Material



Object View

Modeler - Commands

- Parametric Technology
  - Dynamic Edits - Change Dimensions
  - Add Variables
    - Project Variables (Global) or Design Variables (Local)
    - Animate Geometry
    - Include Units - Default Unit is meters
  - Supports mixed Units



Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	-1 , -1.6 , 0	mm
XSize	2.6	mm
YSize	2.8	mm
ZSize	1	mm

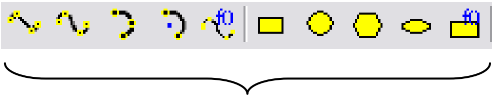
Command

→

Name	Value	Unit
Comma...	CreateBox	
Coordin...	Global	
Position	-1 , -1.6 , 0	mm
XSize	my_x	
YSize	2.8	mm
ZSize	1	mm

Command

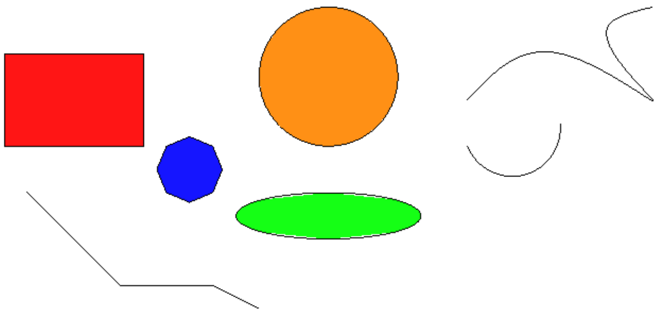
Modeler - Primitives



Toolbar: 2D Objects

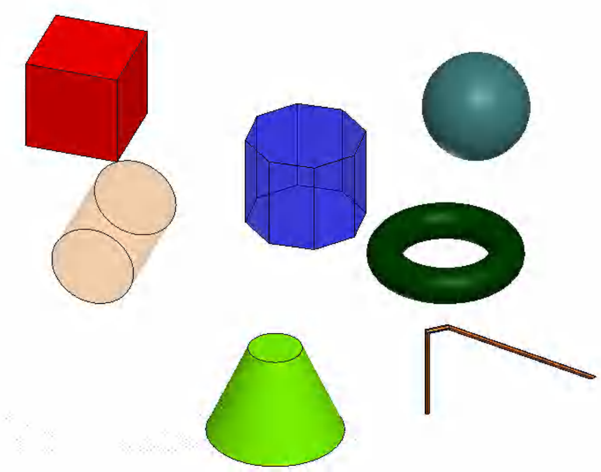
- 2D Draw Objects
  - The following 2D Draw objects are available:
    - Line, Spline, Arc, Equation Based Curve,

      Rectangle, Ellipse, Circle, Regular Polygon, Equation Based Surface



- 3D Draw Objects
  - Note that 3D objects can be pasted into the 2D model window, but they are ignored by the solution
  - The following 3D Draw objects are available (in Maxwell 3D):
    - Box, Cylinder, Regular Polyhedron

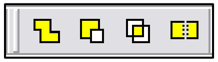
      Cone, Sphere, Torus, Helix, Spiral, Bond Wire



- True Surfaces
  - Circles, Cylinders, Spheres, etc are represented as true surfaces. In versions prior to release 11 these primitives would be represented as faceted objects. If you wish to use the faceted primitives, select the Regular Polyhedron or Regular Polygon.

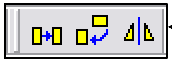
Modeler - Boolean Operations/Transformations

- *Modeler > Boolean >*
  - **Unite** - combine multiple primitives
    - Unite disjoint objects (**Separate Bodies** to separate)
  - **Subtract** - remove part of a primitive from another
  - **Intersect**- keep only the parts of primitives that overlap
  - **Split** - break primitives into multiple parts along a plane (XY, YZ, XZ)
  - **Split Crossing Objects** - splits objects along a plane (XY, YZ, XZ) only where they intersect
  - **Separate Bodies** - separates objects which are united but not physically connected into individual objects



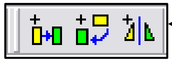
Toolbar: Boolean

- *Edit > Arrange >*
  - **Move** - Translates the structure along a vector
  - **Rotate** - Rotates the shape around a coordinate axis by an angle
  - **Mirror** - Mirrors the shape around a specified plane
  - **Offset** - Performs a uniform scale in x, y, and z.



Toolbar: Arrange

- *Edit > Duplicate >*
  - **Along Line** - Create multiple copies of an object along a vector
  - **Around Axis** - Create multiple copies of an object rotated by a fixed angle around the x, y, or z axis
  - **Mirror** - Mirrors the shape around a specified plane and creates a duplicate



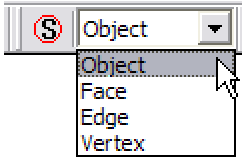
Toolbar: Duplicate

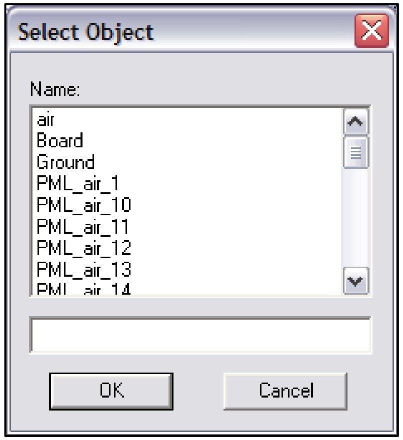
- *Edit > Scale* - Allows non-uniform scaling in the x, y, or z direction

Modeler - Selection

Selection Types

- Object (Default)
- Face
- Edge
- Vertex





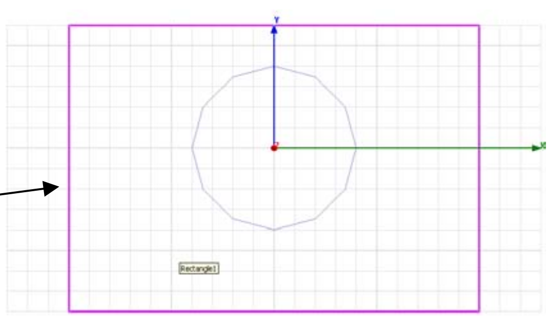
Selection Modes

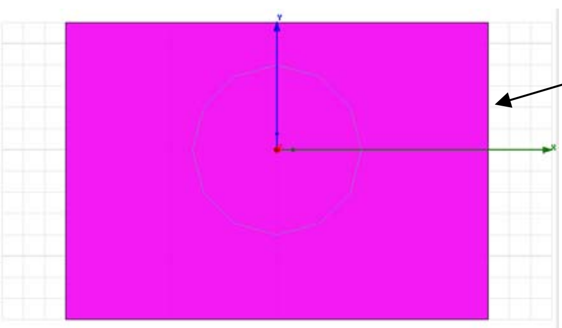
- All Objects
- All Visible Object
- By Name

Highlight Selection Dynamically - By default, moving the mouse pointer over an object will dynamically highlight the object for selection. To select the object simply click the left mouse button.

- **Multiple Object Selection** - Hold the **CTRL** key down to graphically select multiple objects
- **Next Behind** - To select an object located behind another object, select the front object, press the **b** key to get the next behind. Note: The mouse pointer must be located such that the next behind object is under the mouse pointer.
- **To Disable:** Select the menu item *Tools > Options > Modeler Options*
  - From the **Display Tab**, uncheck **Highlight selection dynamically**

Dynamically Highlighted
(Only frame of object)

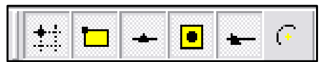




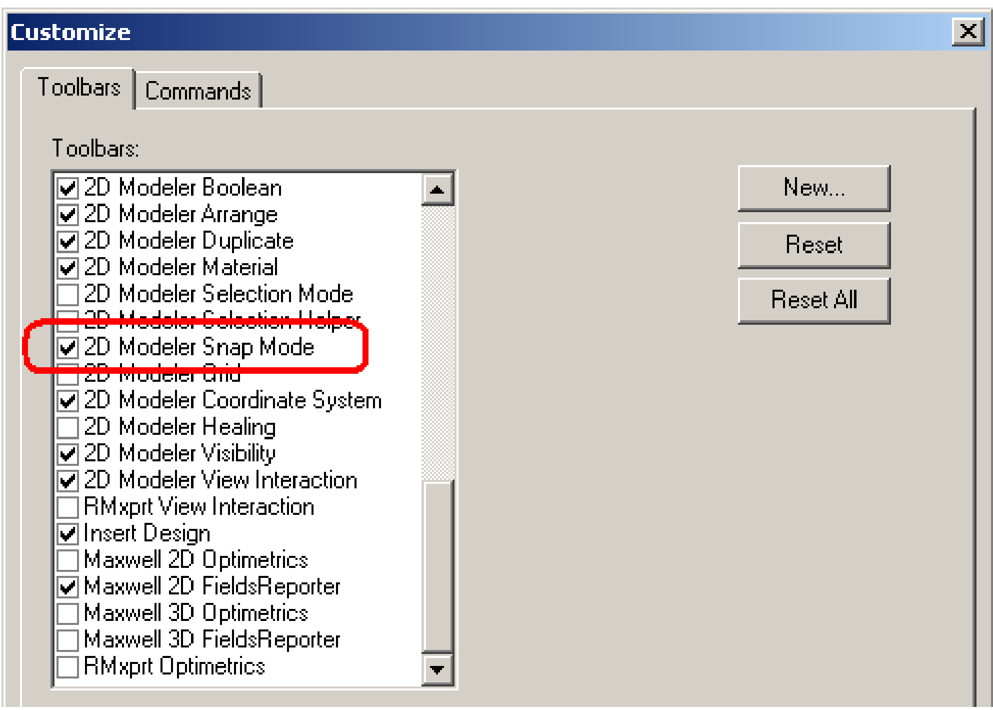
Selected

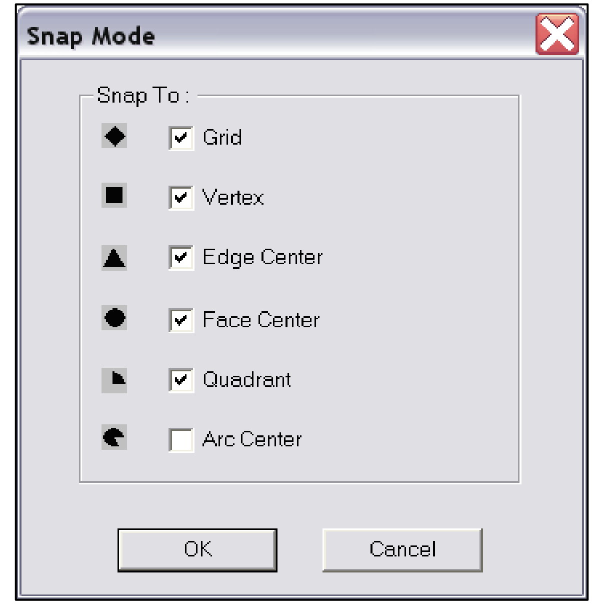
Modeler - Moving Around

- Modeler > Snap Mode to set the snaps
- Tools > Customize…
  Snap Mode to view Snap Mode toolbar



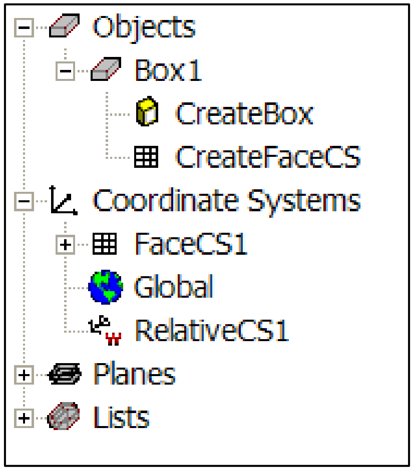
Toolbar: Snap Mode

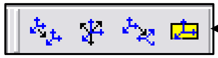




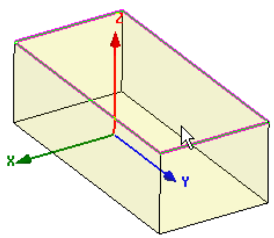
Modeler - Coordinate Systems

- Can be Parameterized
- Working Coordinate System
  - Currently selected CS. This can be a local or global CS
- Global CS
  - The default fixed coordinate system
- Relative CS
  - User defined local coordinate system.
    - Offset
    - Rotated
    - Both
- Face CS (setting available to automatically switch to face coordinate system in the Modeler Options)

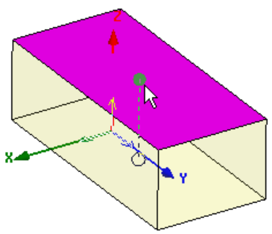




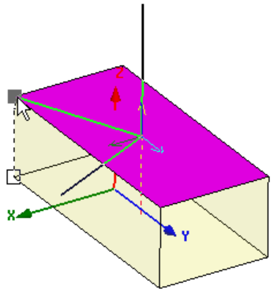
Toolbar: Coordinate System



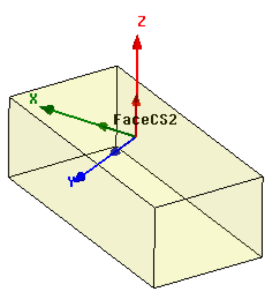
Step 1: Select Face



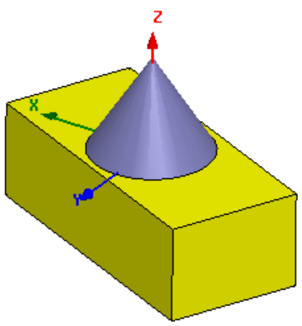
Step 2: Select Origin



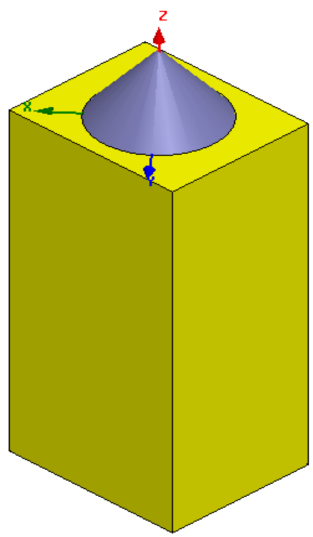
Step 3: Set X-Axis



New Working CS



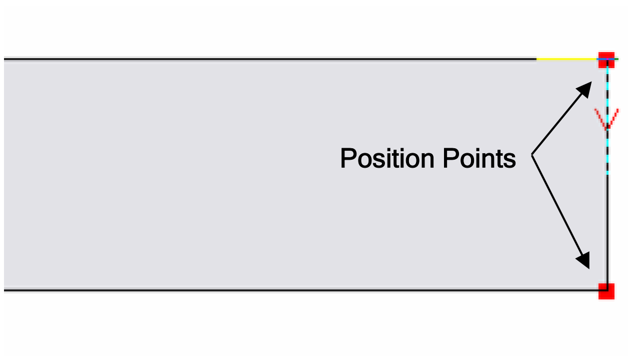
Cone created with Face CS

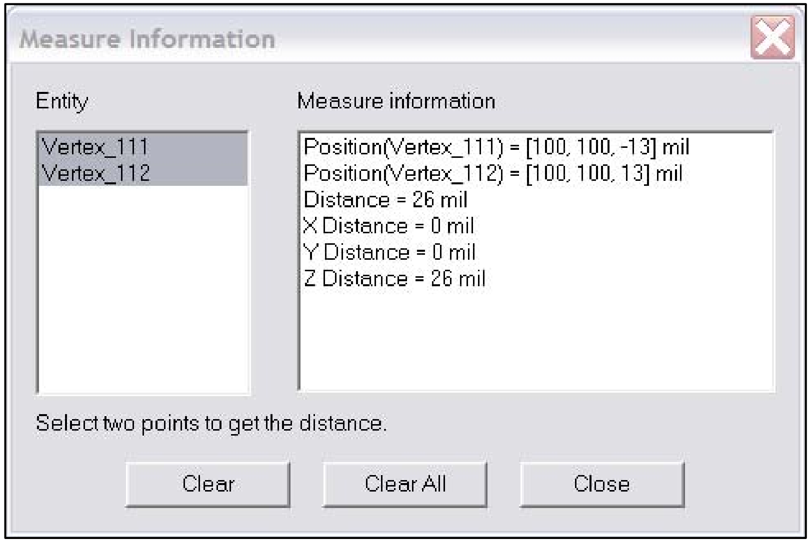


Change Box Size and Cone is automatically positioned with the top face of the box

2D Measure

- *Modeler > Measure >*
  - *Position* - Location, Distance, and Area
  - *Edge* - Edge Length
  - *Face* - Surface Area
  - *Object* - Surface Area, Object Volume





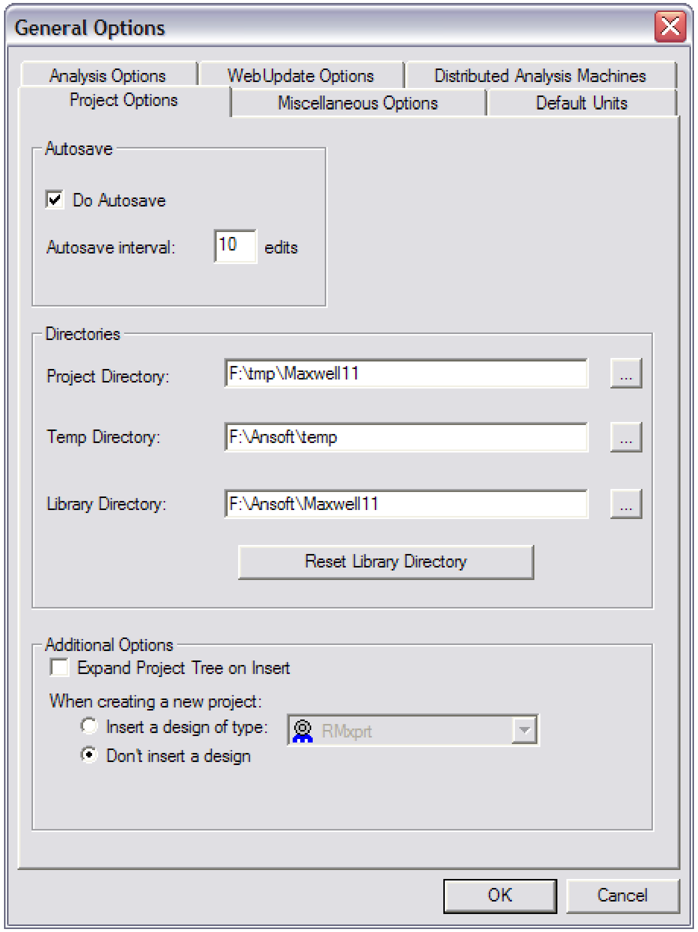
Options - General

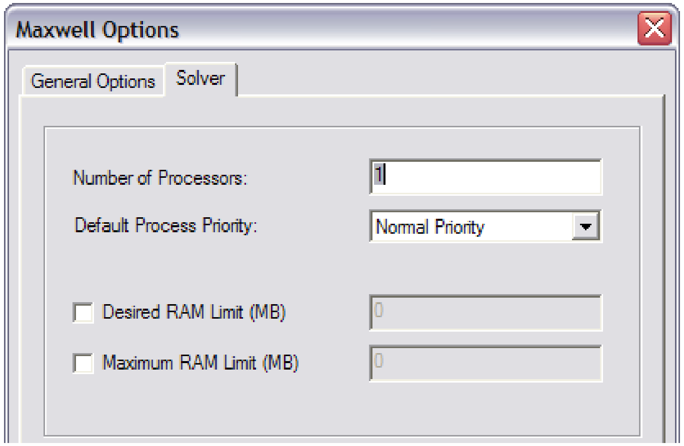
- *Tools > Options > General Options > Project Options*
  - **Temp Directory** – Location used during solution process
    - Make sure it has at least 512MB free disk.

Options - Maxwell

- *Tools > Options > Maxwell Options > Solver*
  - Set Number of Processors = 2 for 1 dual-core processor or two single-core processors. Requires additional license
  - Default Process Priority – set the simulation priority from Critical

    (highest) to Idle (lowest)

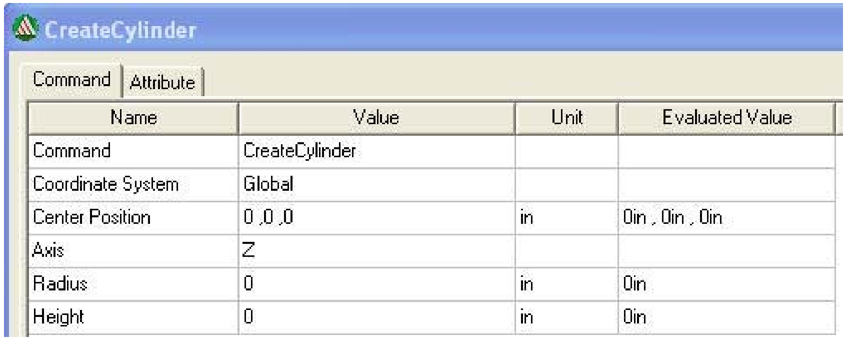




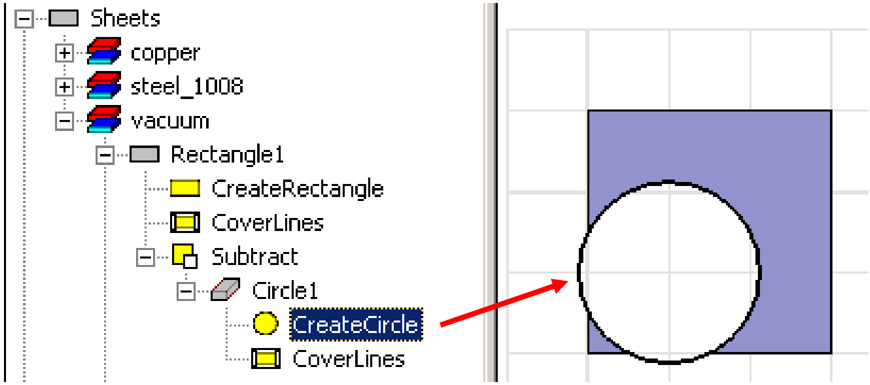
Options - Modeler Options

- *Tools > Options > Modeler Options > Drawing* for Point and Dialog Entry Modes
- Can enter in new dimensions using either Point (mouse) or Dialog entry mode
- Alternatively use F3 and F4 to switch between Point and Dialog entry modes

Typical "Dialog" entry mode window

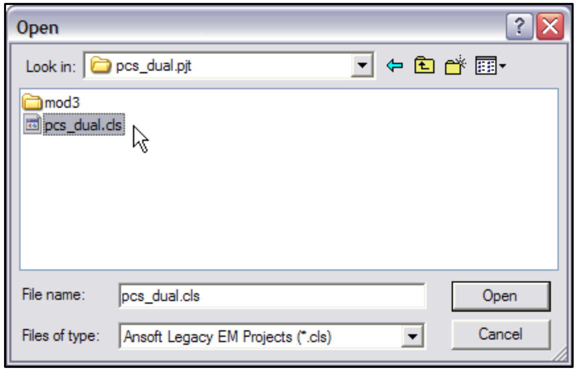


- *Tools > Options > Modeler Options > Display* tab to enable playback
- Must close and re-open Maxwell after making change for this setting, to activate
- Visualization is seen by clicking on primatives in the history tree (under subtract command, for instance)



Converting Older Maxwell Projects (pre-Maxwell v12) to Maxwell v12

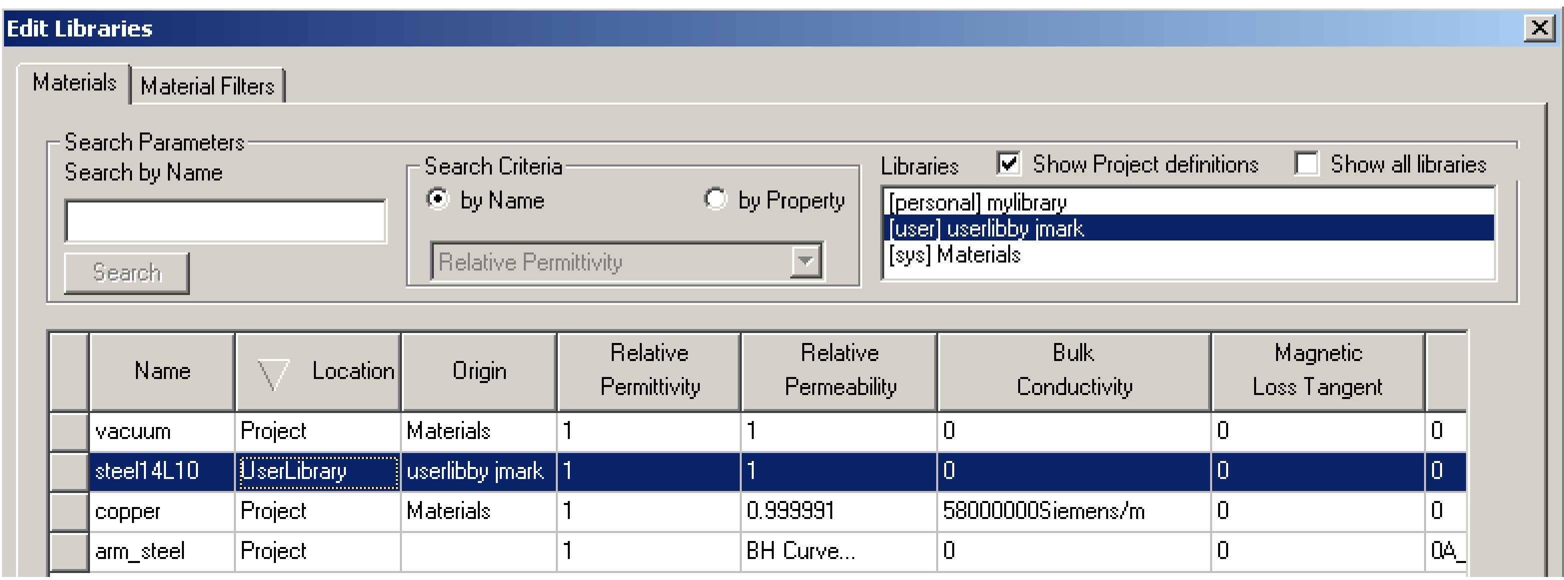
- From Maxwell v 11 and older,
  1. Select the menu item *File > Open*
  2. Open dialog
     1. Files of Type: **Ansoft Legacy EM Projects (.cls)**
     2. Browse to the existing project and select the .cls file
     3. Click the **Open** button



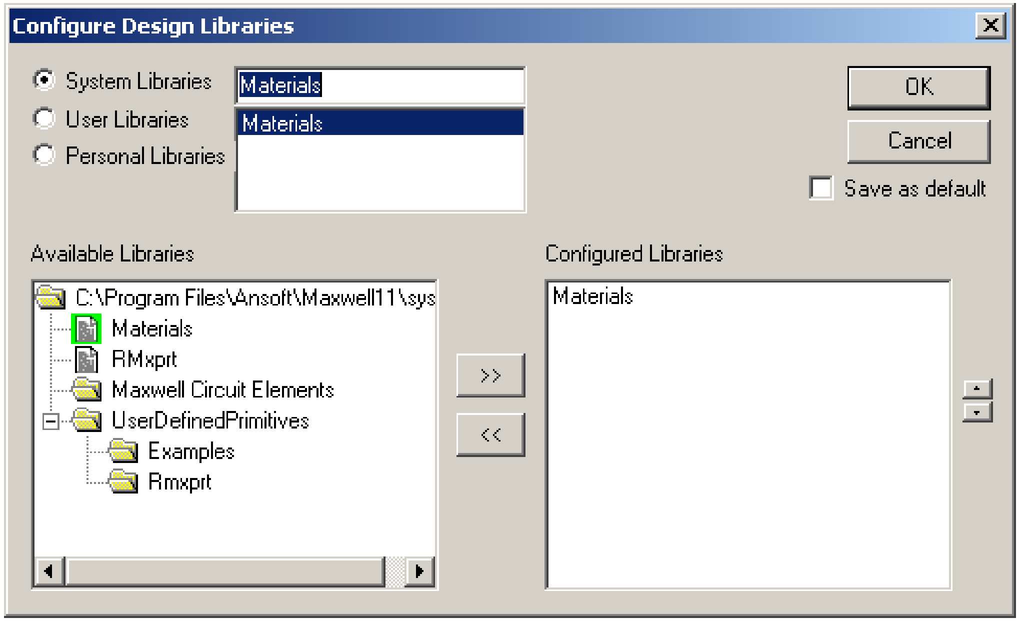
- What is Converted?
  - Converts Entire Model: Geometry, Materials, Boundaries, Sources and Setup
  - *Solutions, Optimetrics projects and Macros are not converted*

Material Setup - Libraries

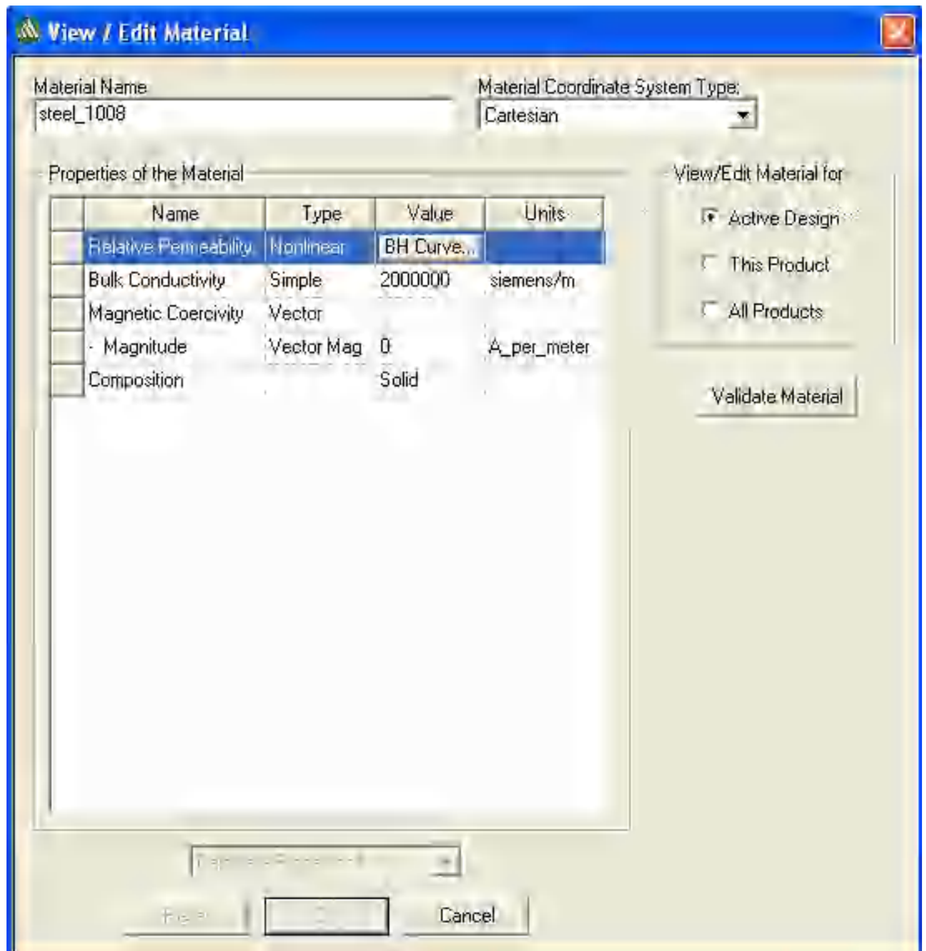
- *3-Tier library structure*
  - *System (global) level - predefined from Ansoft and ships with new upgrades, users cannot modify this*
  - *User Library - to be shared among several users at a company (can be encrypted)*
  - *Personal libraries - to be used only by single user (can be encrypted)*
- Add a new material: *Tools > Edit Configured Libraries > Materials*
- New Interface for Materials Setting shared with RMxprt



- Click “Add Material …”. The Material is only available in Project
- To add a material in the user or personal library: click on “Export Library” and save it in the desire library.
- In the main project window, click on *Tools > Configured Libraries*. Locate the library to have the material available for all the projects.
- Click on *Save as default* to automatically load library for any new project.

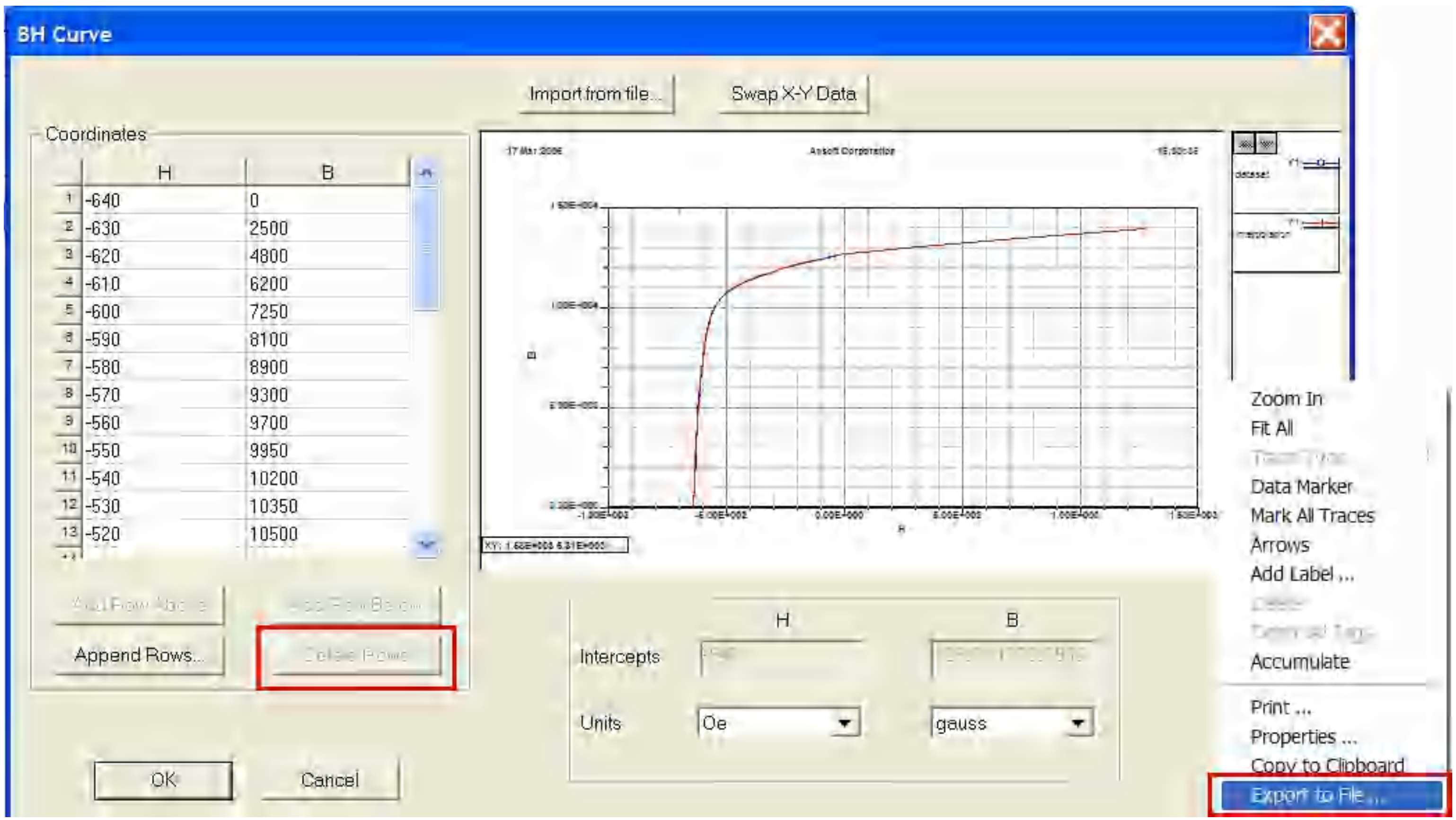


Materials Setup - Editing



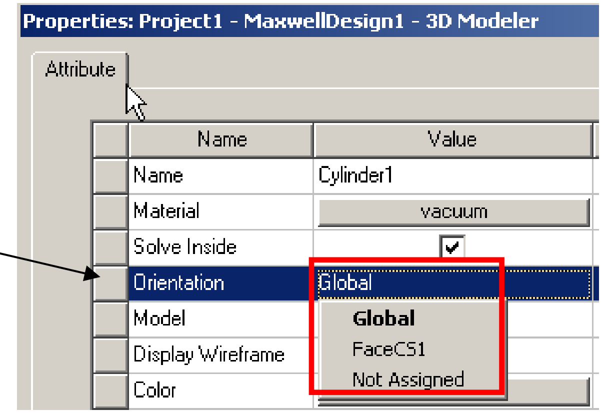
Material Setup – BH curve

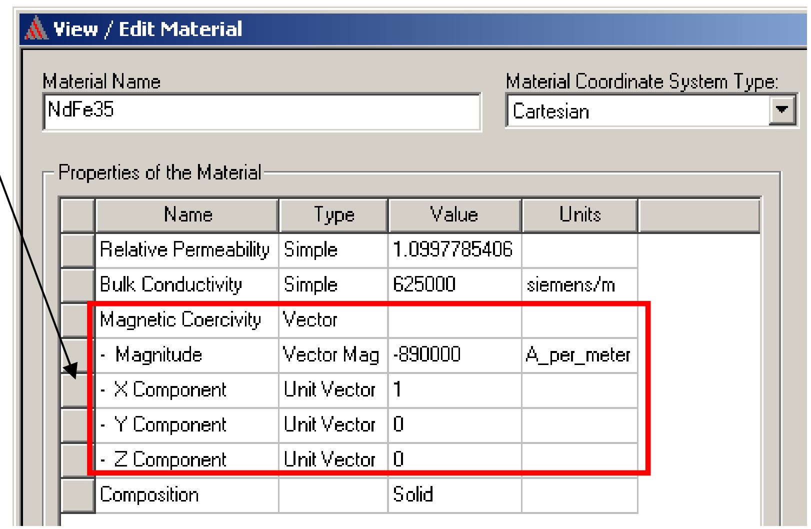
- Robust BH curve entry – can delete points if you make a mistake
- Can import data from a file
- To export BH curve for use in future, right-mouse-click on curve and select Export to File…

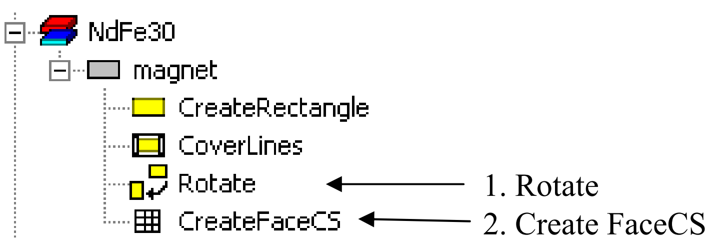


Material Setup - Permanent Magnets

- Direction of magnetization determined by material's object's **Orientation** and **Magnetic Coercivity Unit Vectors**.
- To modify the **Orientation**, open the Attribute for the object and change the coordinate system. The default **Orientation** for permanent magnets is Global CS.
- To modify the **Magnetic Coercivity Unit Vectors** for a permanent magnet material, enter the Materials Library and edit the material.
- The material coordinate system type can be described in Cartesian, Cylindrical, Spherical
- The magnetic coercivity has unit vectors corresponding to the chosen coordinate system: for instance X,Y,Z for cartesian.
- To rotate a magnet in a parametric simulation and the magnetization direction, you must first rotate the object and second assign the FaceCS, as shown below in the history tree







Material Setup - Anisotropic Material Properties

- ε_1, μ_1, and σ_1 are tensors in the X direction.
- ε_2, μ_2, and σ_2 are tensors in the Y direction.
- ε_3, μ_3, and σ3 are tensors in the Z direction.

Note: Nonlinear anisotropic permeability not allowed in Maxwell 2D.

$$[\varepsilon]=\begin{bmatrix}\varepsilon_1 & 0 & 0\\ 0 & \varepsilon_2 & 0\\ 0 & 0 & \varepsilon_3\end{bmatrix},\quad [\mu]=\begin{bmatrix}\mu_1 & 0 & 0\\ 0 & \mu_2 & 0\\ 0 & 0 & \mu_3\end{bmatrix},\quad [\sigma]=\begin{bmatrix}\sigma_1 & 0 & 0\\ 0 & \sigma_2 & 0\\ 0 & 0 & \sigma_3\end{bmatrix}$$

Solver	Anisotropic Permitivity	Anisotropic Permeability	Anisotropic Conductivity	Dielectric Loss Tangent	Magnetic Loss Tangent
Electrostatic	yes	no	no	no	no
DC Conduction	no	no	yes	no	no
AC Conduction	yes	no	yes	no	no
Magnetostatic	no	yes	no	no	no
Eddy Current	no	yes	no	no	no
Transient	no	yes	no	no	no

Electric Field Boundary Conditions (Electrostatic, DC Conduction, AC Conduction)

Boundary Type	E-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: - Natural boundaries – The normal component of D changes by the amount of surface charge density. No special conditions are imposed. - Neumann boundaries – E is tangential to the boundary. Flux cannot cross a Neumann boundary.	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: - Even Symmetry (Flux Tangential) – E is tangential to the boundary; its normal components are zero. - Odd Symmetry (Flux Normal) – E is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Balloon	Field behaves so that voltage can fringe	Ground at infinity
Matching (Master and Slave)	The E-field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field on the master boundary.	Planes of symmetry in periodic structures where E is oblique to the boundary.
Resistance (DC conduction solver only)	A resistance boundary models a very thin layer of resistive material (such as that caused by deposits, coatings or oxidation on a metallic surface) on a conductor at a known potential.	Use this boundary condition when the resistive layer's thickness is much smaller than the other dimensions of the model.

Magnetic Field Boundary Conditions (Magnetostatic, Eddy Current, Transient)

Boundary Type	H-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – H is continuous across the boundary. • Neumann boundaries – H is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Magnetic Vector Potential	Sets the magnetic vector potential on the boundary. Note: In the Magnetostatic solver, A is RMS while in the Eddy Current solver, A is peak.	Magnetically isolated structures.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) – H is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) – H is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Impedance (Eddy Current only)	Includes the effect of induced currents beyond the boundary surface.	Conductors with very small skin depths.
Balloon	Field behaves so that magnetic flux can fringe	No fringing at infinity
Matching (Master and Slave)	The H-field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the H-field on the master boundary.	Planes of symmetry in periodic structures where H is oblique to the boundary.

- Electric Field Sources (Electrostatic, DC Conduction, AC Conduction)

Source	Type of Excitation
Floating Conductor	Used to model conductors at unknown potentials.
Voltage	The DC voltage on a surface or object.
Charge	The total charge on a surface or object (either a conductor or dielectric).
Charge Density	The charge density in an object.
Notes: - In the Electrostatic solver, any conductor without a source condition will be assumed to be floating.	

Magnetic Field Sources (Magnetostatic)

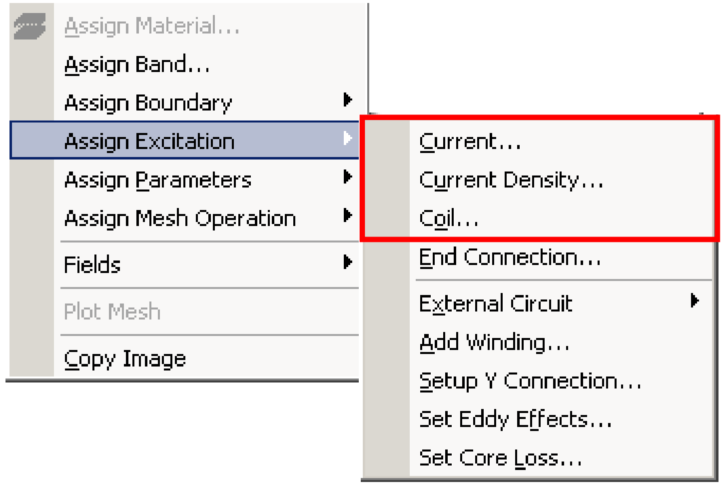
Source	Type of Excitation
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Notes: In the Magnetostatic solver, current is RMS ampturns. Permanent magnets will also act as a source in the Magnetostatic solver.	

Magnetic Field Sources (Eddy Current)

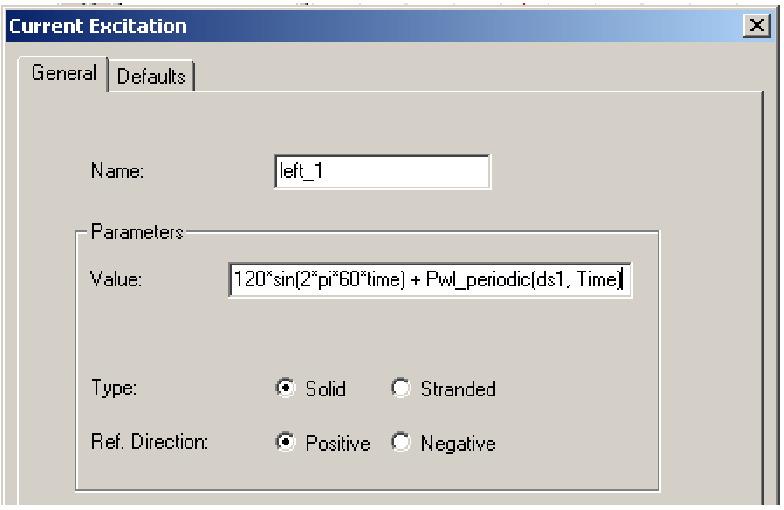
Source	Type of Excitation
Current	The total current in a conductor.
Parallel Current	The total current in a a group of parallel conductors.
Current Density	The current density in a conductor.
Notes: In the Eddy Current solver, current is peak amp-turns. Sources can be solid (with eddy effects) or stranded (without eddy effects).	

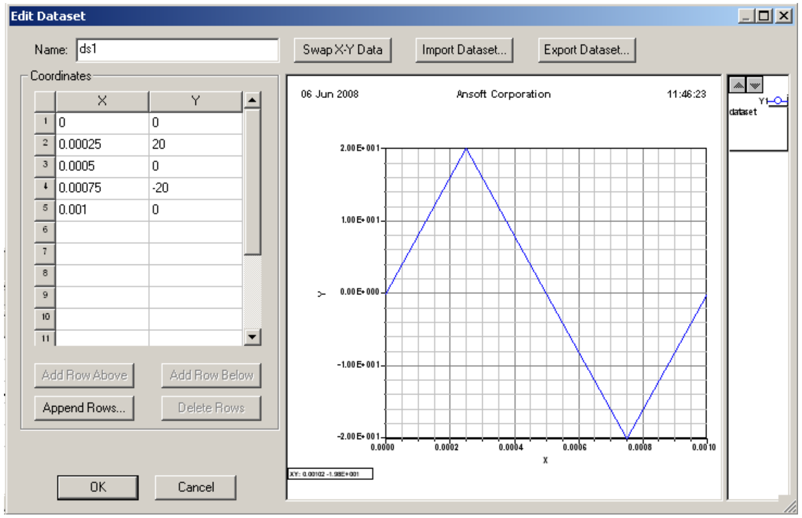
Magnetic Field Sources (Transient)

Source	Type of Excitation
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Coil	Current or voltage on a winding representing 1 or more turns
Permanent magnets will also act as a source in the Transient solver.	



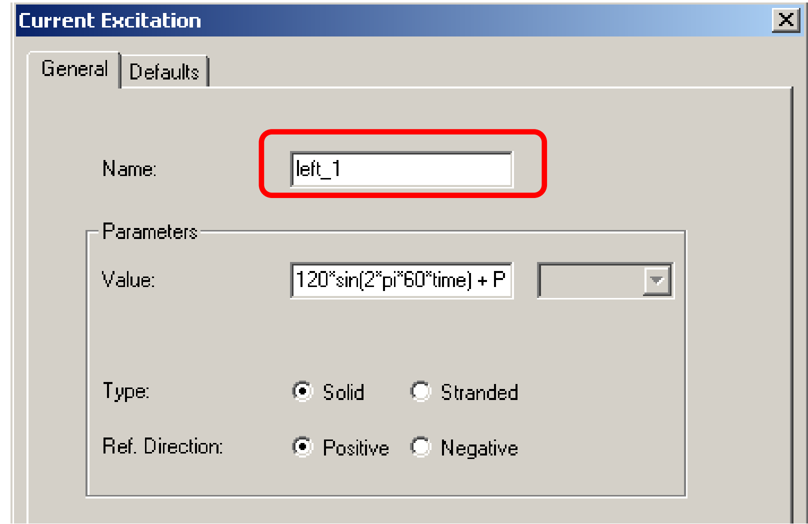
- Current and voltage sources (solid or stranded) can be constant or functions of intrinsic variables: speed (rpm or deg/sec), position (degrees), or time (seconds)
- Dataset function can be used for piecewise linear functions: Pwl_periodic (ds1, Time)

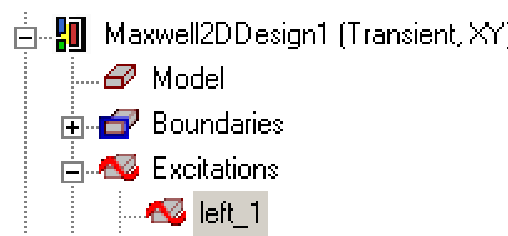




Magnetic Field Sources (Transient)

- Maxwell 2D > Excitation > Current
  - Value: applies current in amps
  - Type:
    - Solid
      - for windings having a single conductor/turn
      - eddy effects are considered
    - Stranded
      - for windings having many conductors/turns
      - eddy effects are not considered
  - Ref Direction:
    - Positive or Negative



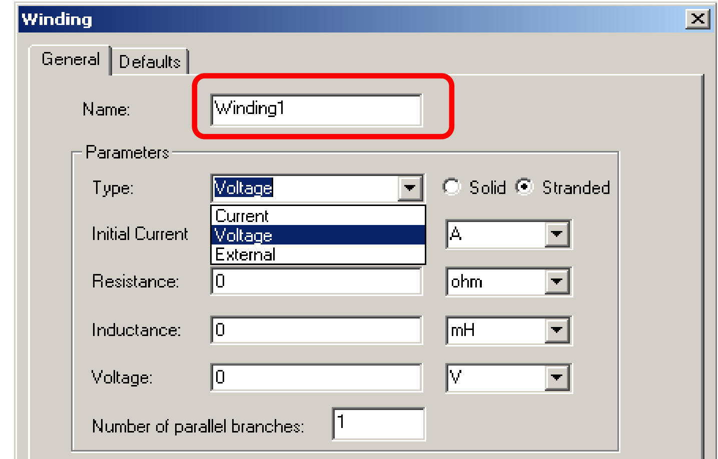


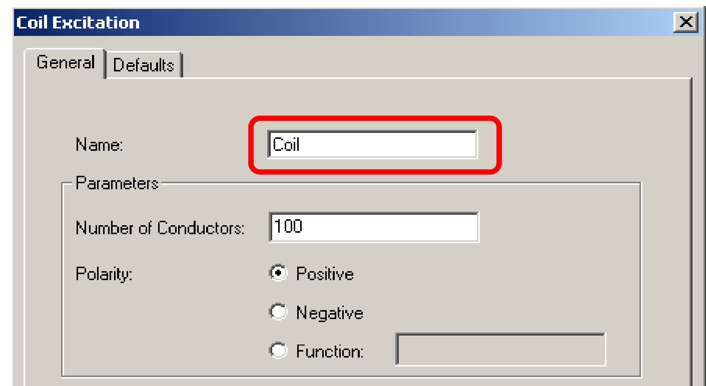
Magnetic Field Sources (Transient)

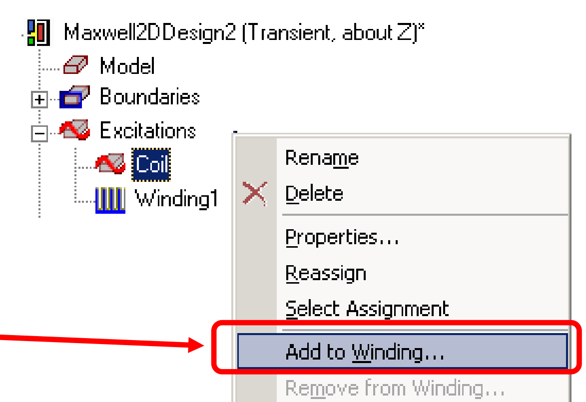
- ### Maxwell 2D > Excitation > Add Winding
  - Current - applies current in amps
    - Solid or Stranded
    - Input current and number of parallel branches as seen from terminal
  - Voltage - applies voltage (total voltage drop over the length of a solid conductor or the entire winding)
    - Solid or Stranded
    - Input initial current, winding resistance, extra series inductance not considered in FEA model, voltage, and number of parallel branches as seen from terminal
  - External - couples to Maxwell Circuit Editor
    - Solid or Stranded
    - Input initial current and number of parallel branches
- ### Maxwell 2D > Excitation > Assign > Coil

  Pick a conductor on the screen and then specify:
  - Name
  - Number of Conductors
  - Polarity: positive, negative, or functional winding direction

  Note: Windings in the XY solver will usually have 2 coils: one positive and one negative polarity. Both coils will be added to the appropriate winding by right-mouse clicking on **Coil** in the project tree and choosing **Add to Winding**

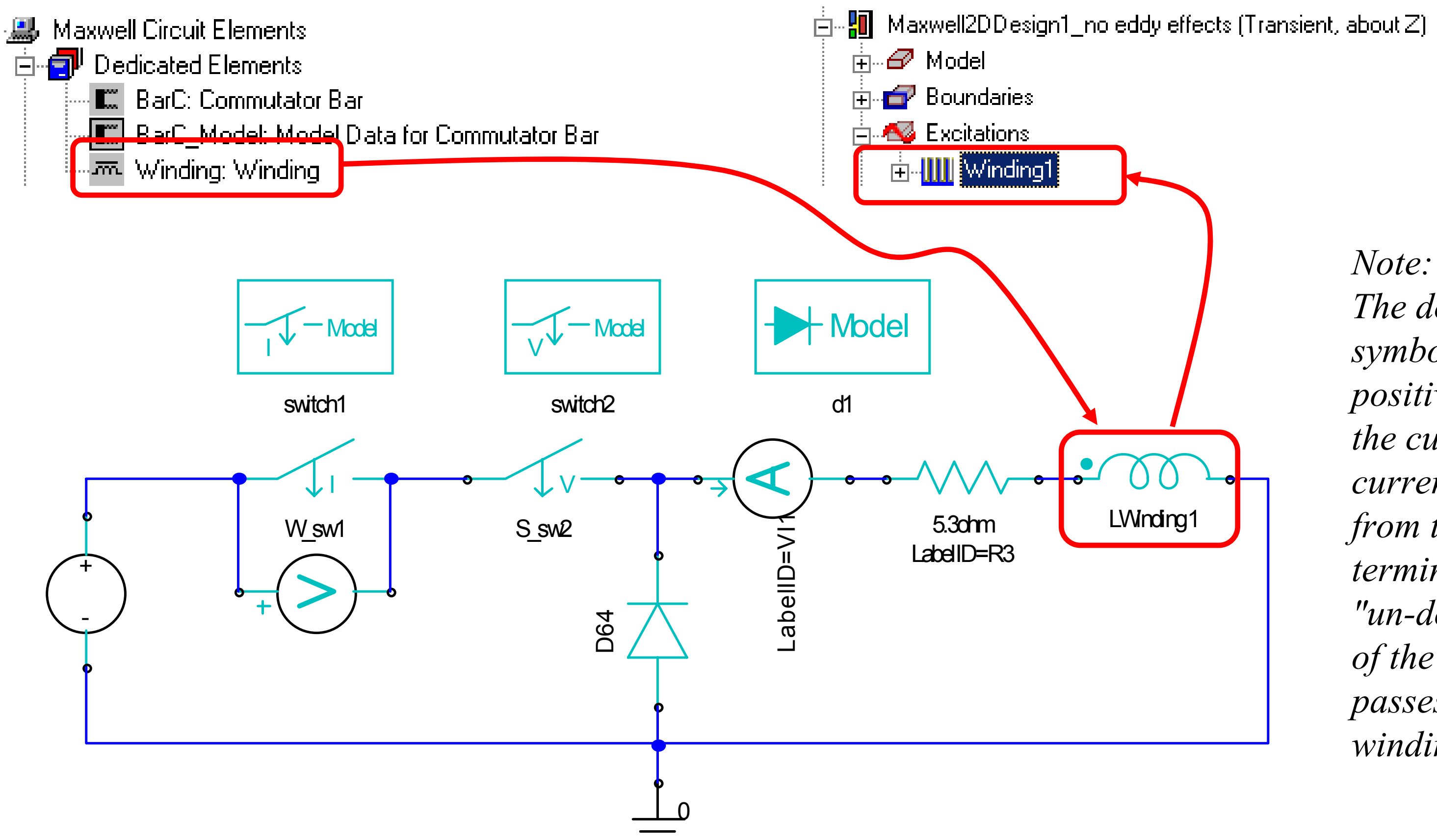






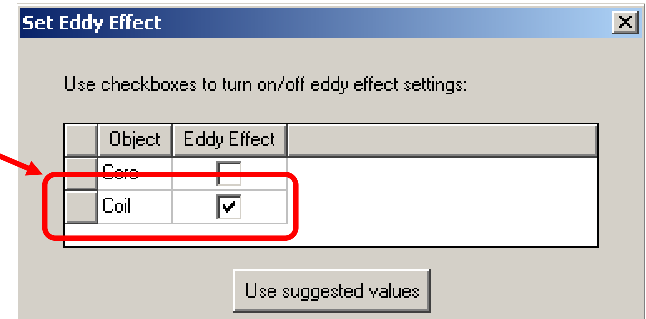
To Create an External Circuit

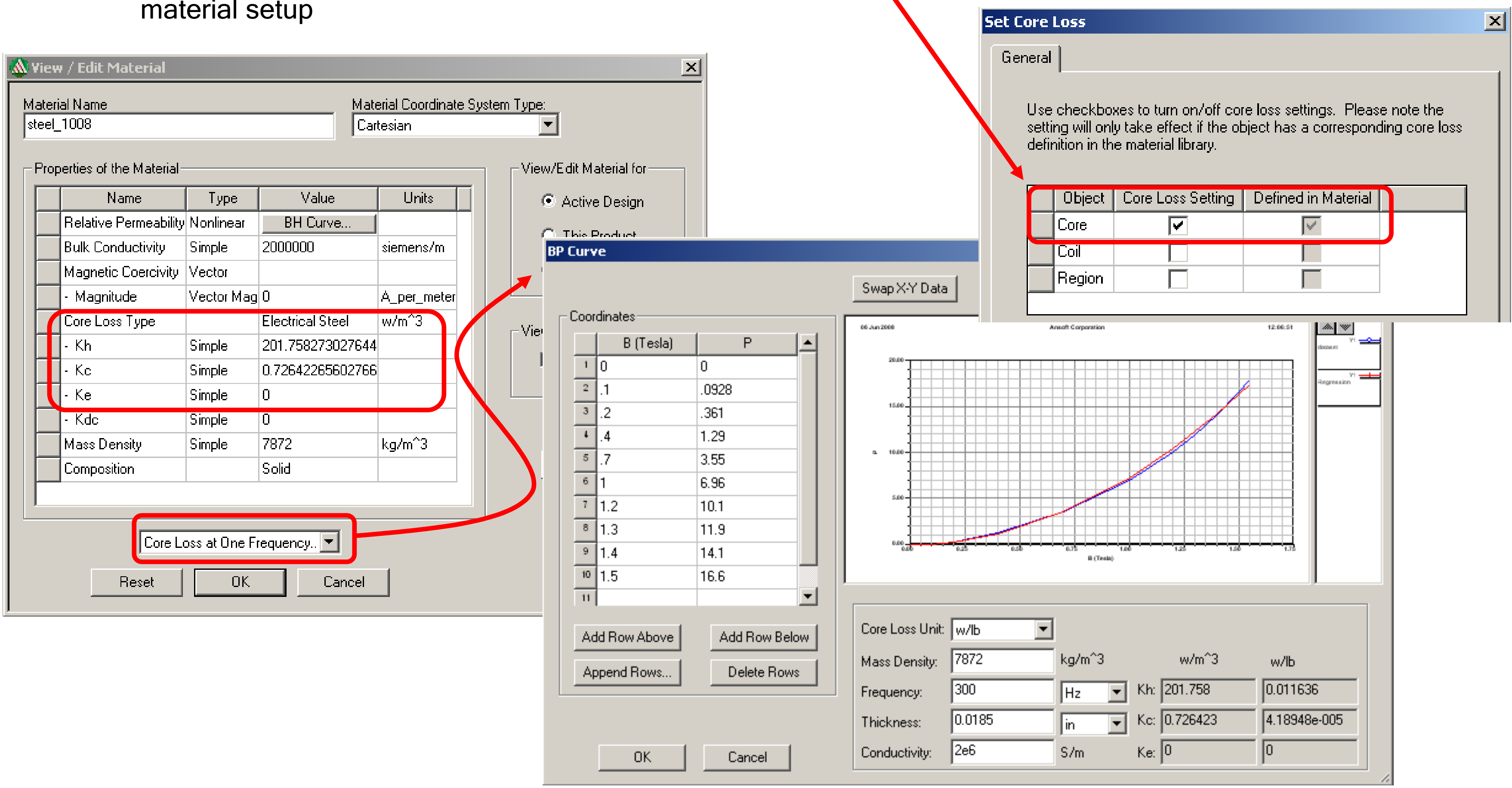
1. Select: *Maxwell2D > Excitations > External Circuit > Edit External Circuit > Import Circuit*
2. After circuit editor opens, add elements to construct the circuit. Note that the name of the Winding in the circuit (Winding1) must match the name of the Winding in Maxwell (Winding1)
3. Save circuit as *.amcp file and then *Maxwell Circuit > Export Netlist > *.sph* file.



Note:
The dot on the winding symbol is used as the positive reference for the current (positive current is oriented from the "dotted" terminal towards to "un-dotted" terminal of the winding as it passes through the winding).

- ## Maxwell 2D > Excitation > Set Eddy Effects
  - Need to enable the calculation of eddy effects in objects
- ## Maxwell 2D > Excitation > Set Core Loss
  - For objects with zero conductivity (such as a laminated core), you can calculate the core loss
  - Note that the core loss coefficients must be defined in the material setup





Core Loss Calculation Method

- The core loss for electrical steel is based on:

$$p = K_h B_{\max}^2 f + K_c \left(B_{\max} f\right)^2 + K_e \left(B_{\max} f\right)^{1.5}$$

where:

- Kh is the hysteresis coefficient.
- Kc is the classical eddy coefficient.
- Ke is the excess or anomalous eddy current coefficient due to magnetic domains.
- Bmax the maximum amplitude of the flux density.
- f is the frequency.

- The power ferrite core loss is based on:

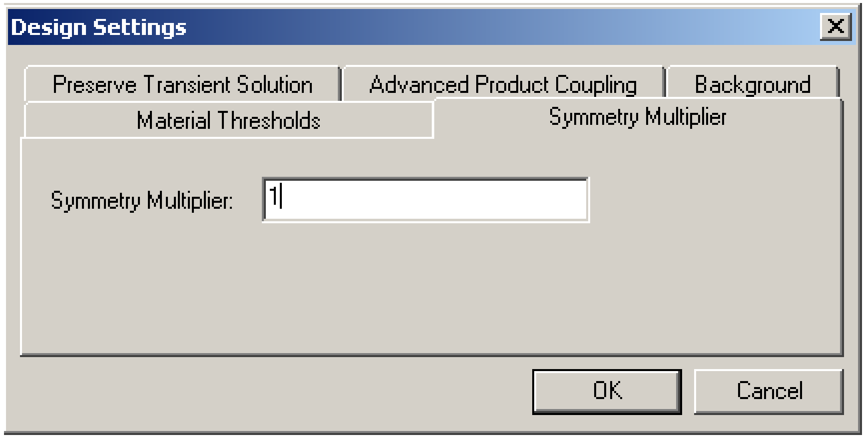
$$p = C_m f^x B_{\max}^y$$

where:

- Cm is constant value determined by experiment.
- fx is the frequency.
- Bymax is the maximum amplitude of the flux density

Maxwell 2D > Design Settings

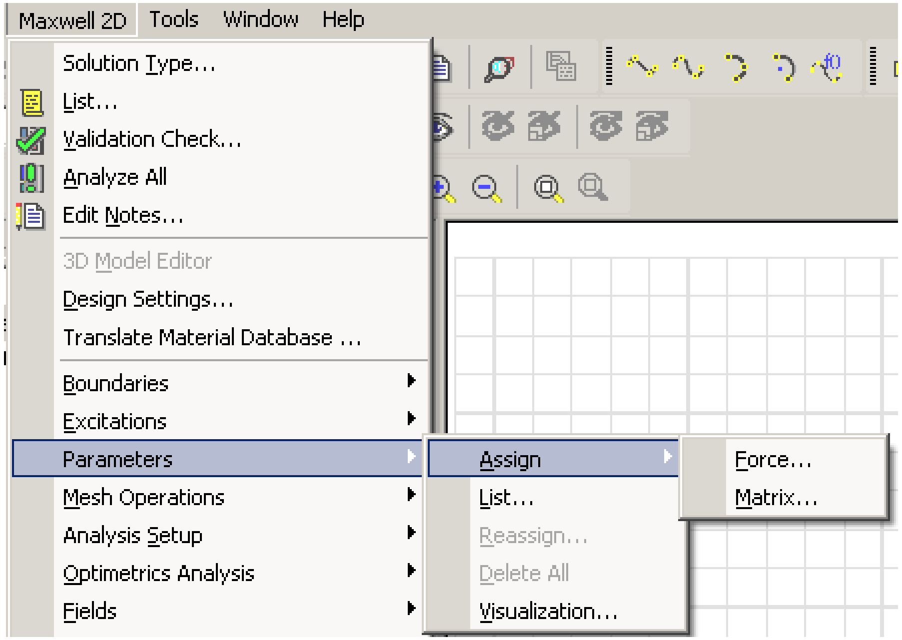
- The Design Settings window allows you to specify how the simulator will deal with some aspects of the design. Tabs vary by solver used (the panel below is for the transient solver)
- Set the Symmetry Multiplier (For Transient XY Solutions only).



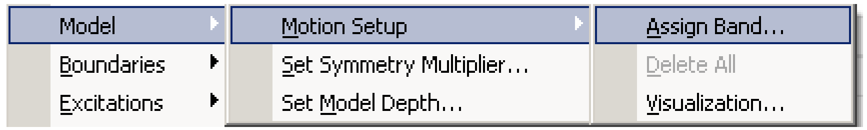
- Set the Material Threshold for treating materials as conductors vs. insulators.
- Set Preserve Transient Solution options (For Transient Solutions Only).
- Set transient coupling with Simplorer on the Advanced Product Coupling tab (For Transient Solutions Only)
- Set the Model Depth (Maxwell2D XY Transient Designs Only).
- Set the default Background material (Maxwell2D Designs Only).

Maxwell 2D > Parameters

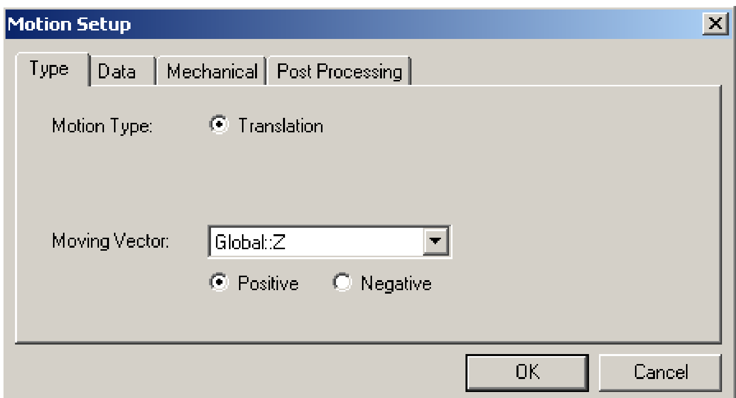
- Allows the automatic calculation of parameters following the field solution
- Includes: Force, Torque, Flux linkage, Core loss, and Matrix

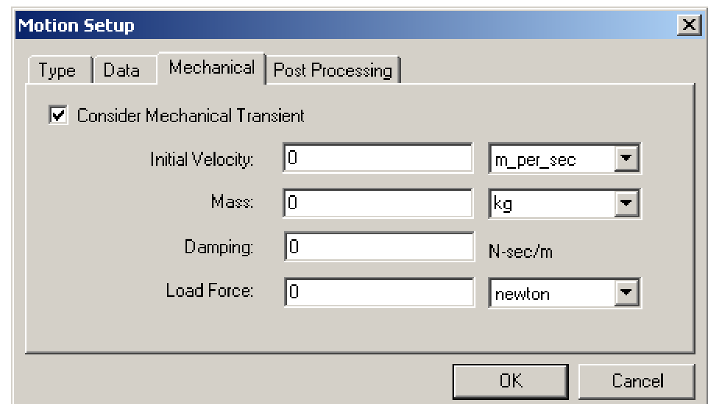


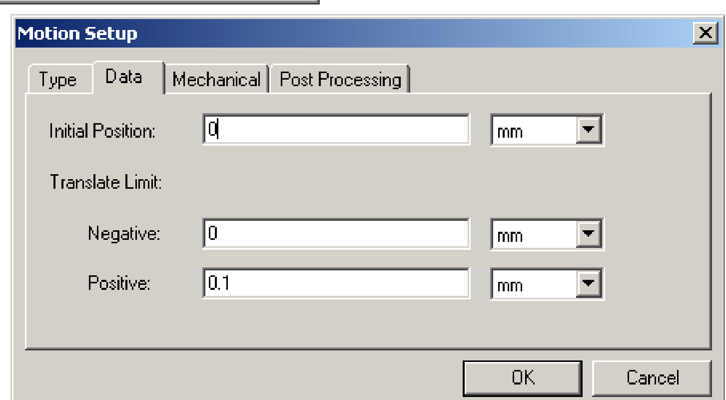
Maxwell 2D > Model > Motion Setup > Assign Band



1. Defines the direction and type of motion (translation or rotation)
2. Defines the mechanical parameters such as mass, damping, and load force
3. Defines limits of motion

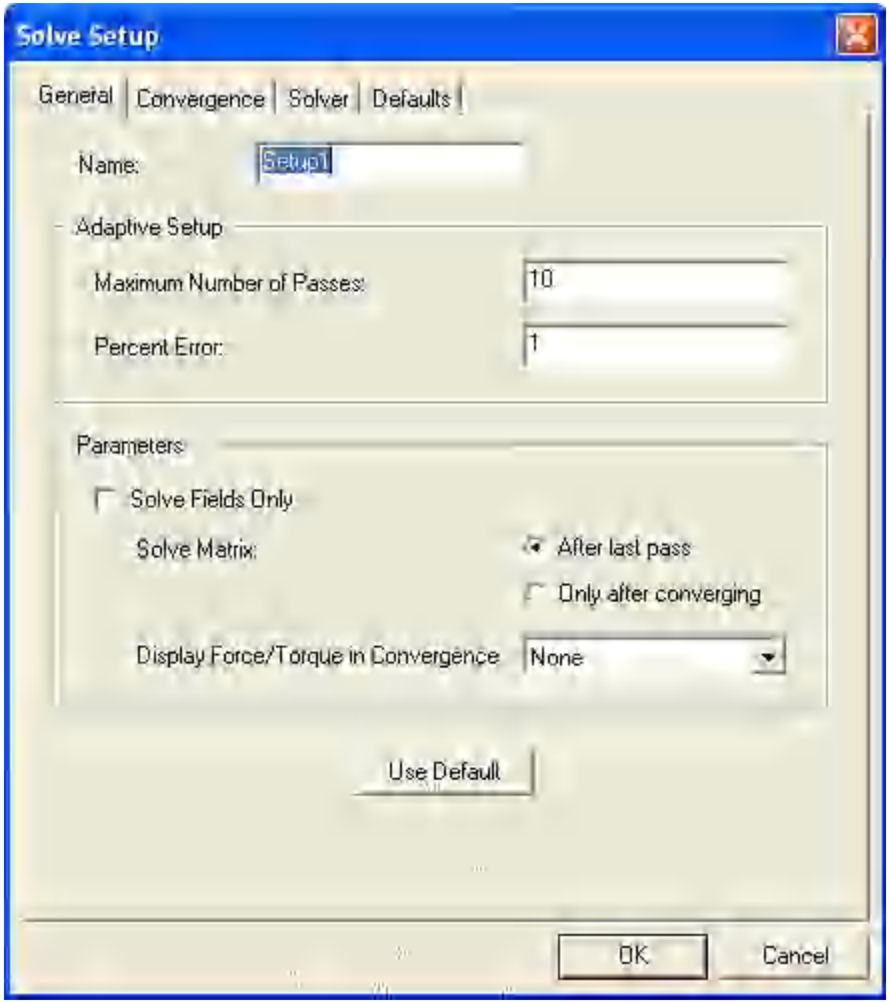


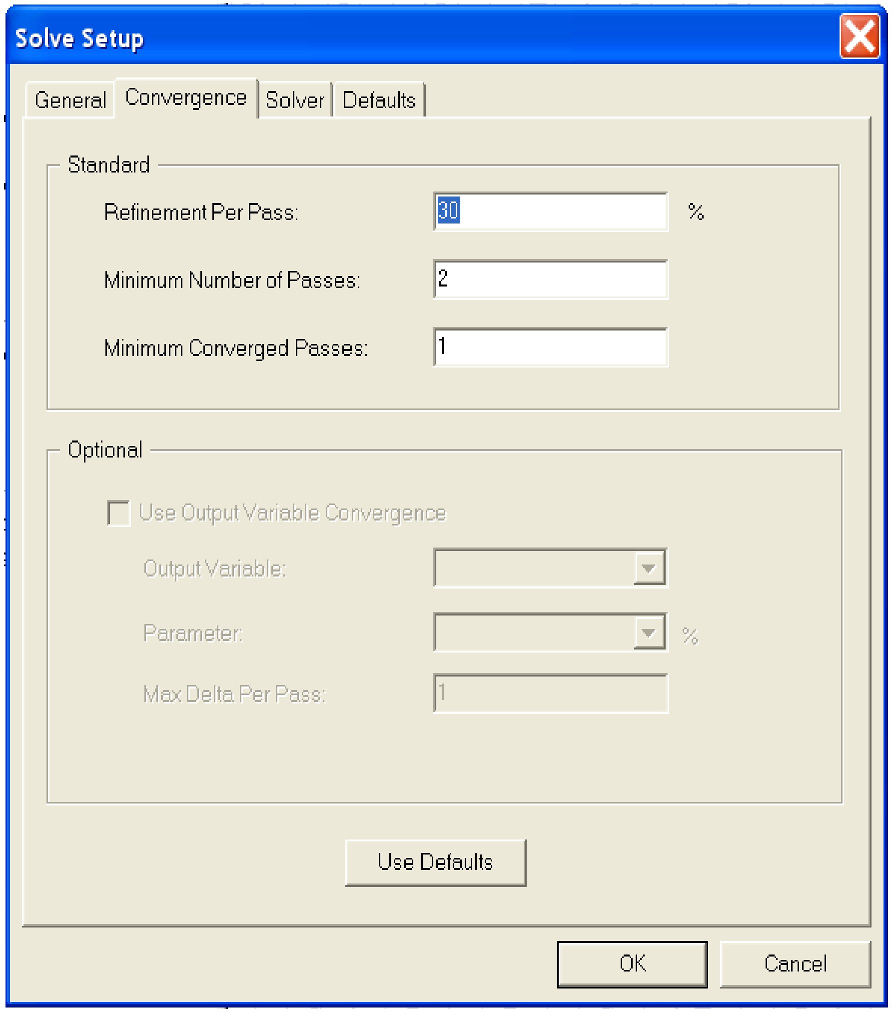




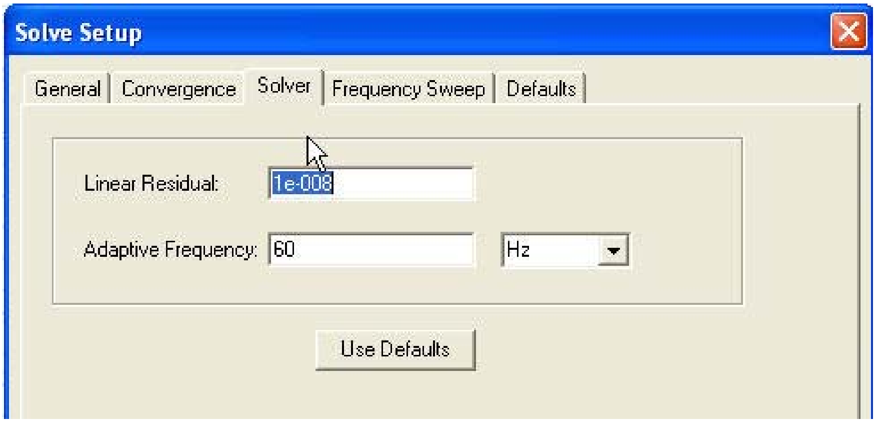
Magnetostatic and Electric Solution Setup

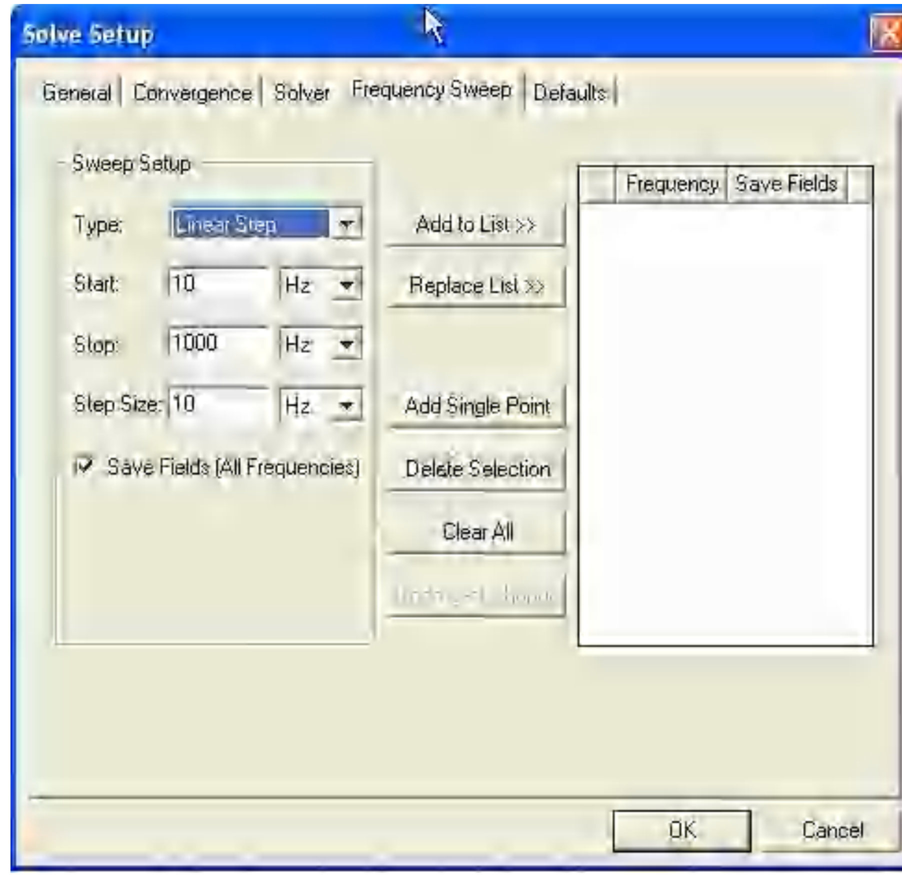
- Start the menu of solution setup by: *Maxwell > Analysis Setup > Add Solution Setup …*
- For Magnetostatic solver on Solver tab, suggest setting nonlinear residual = 0.001. On default tab choose Save Defaults to set this value for all future projects.



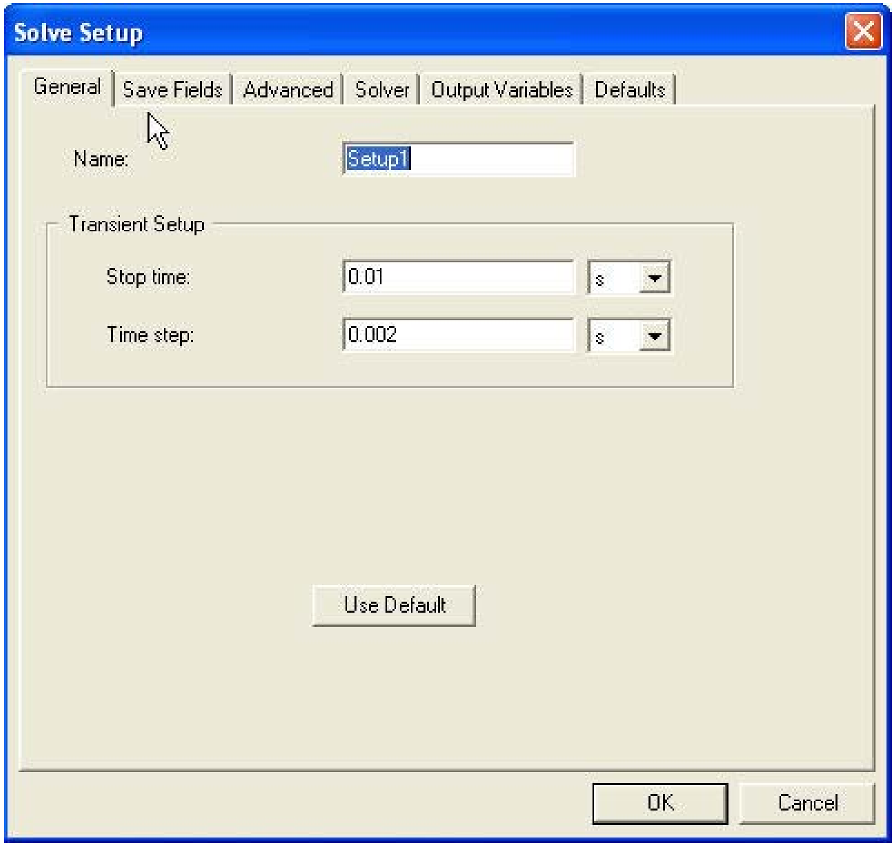


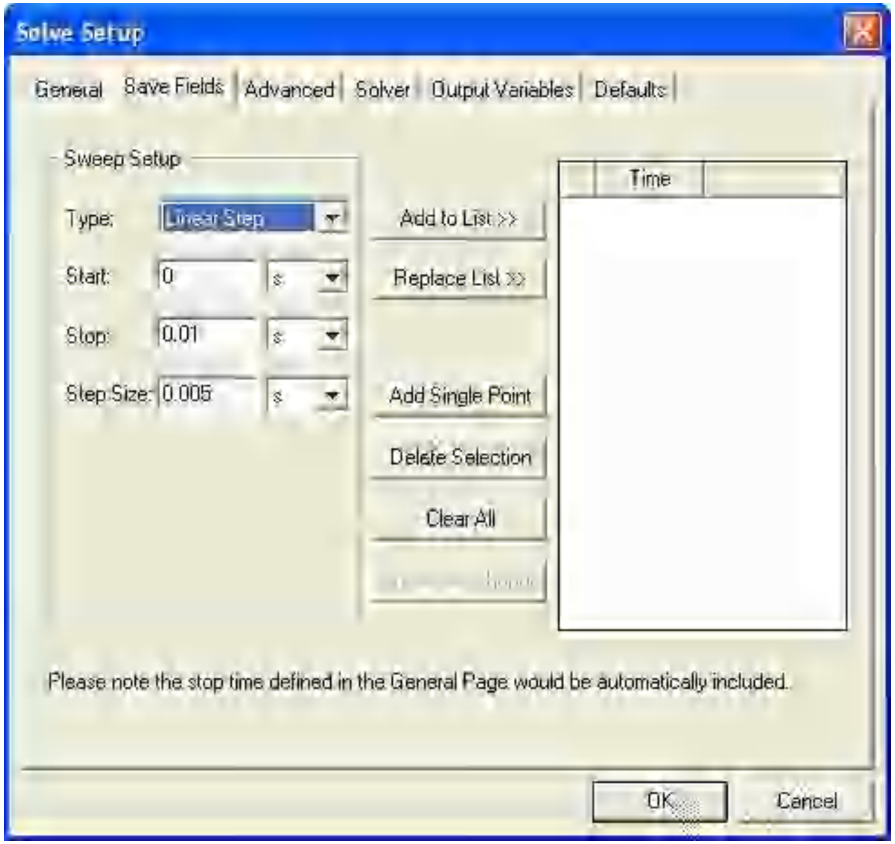
Eddy Current Solution Setup





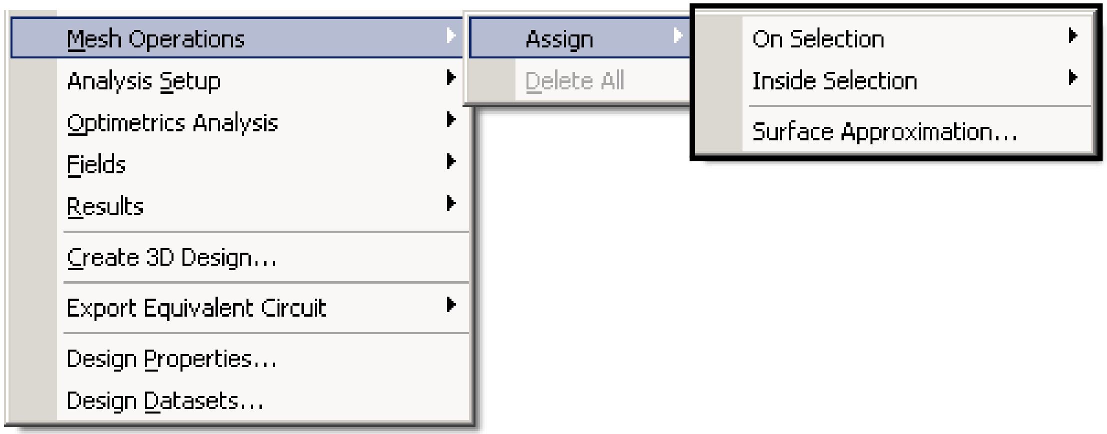
Transient Solution Setup





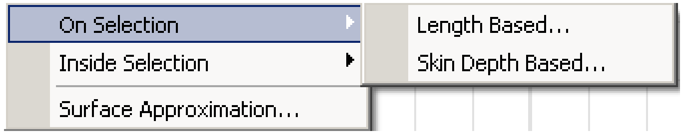
Mesh Operations

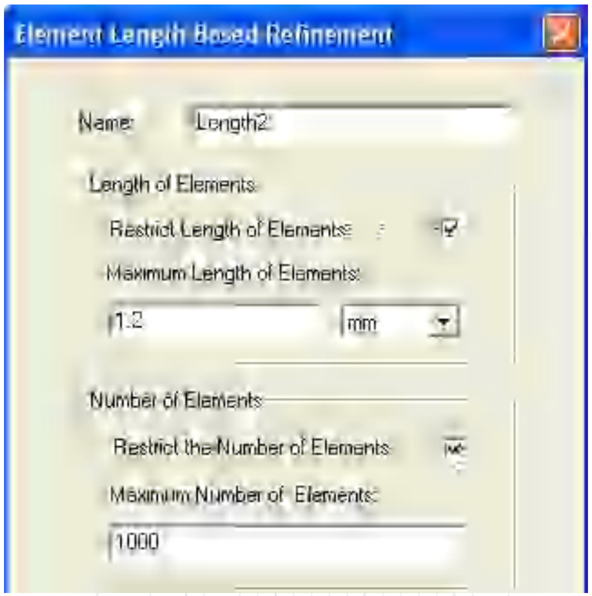
- To assign Mesh operations to Objects, select the Menu item: *Maxwell 2D > Assign Mesh Operations*
  1. *On Selection* is applied on the surface of the object
  2. *Inside Selection* is applied through the volume of the object
  3. *Surface approximation* is applied to set faceting guidelines for true surface objects

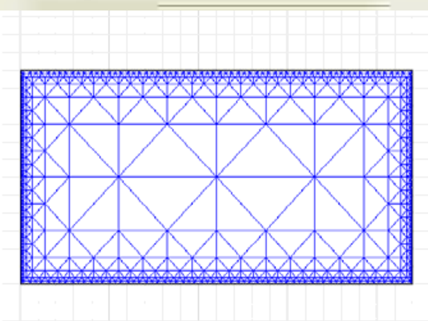


1. Mesh Operations “On selection” applied on the perimeter of the object

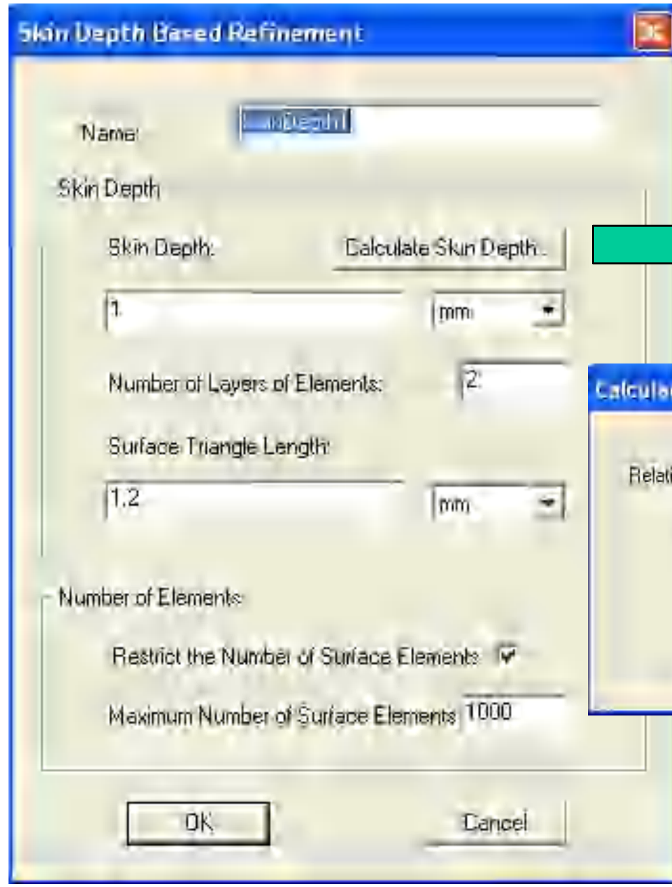
- Element length based refinement: *Length Based*
- Skin Depth based refinement: *Skin Depth Based*

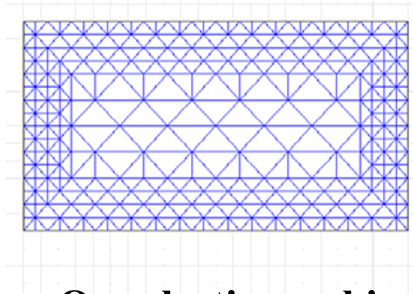




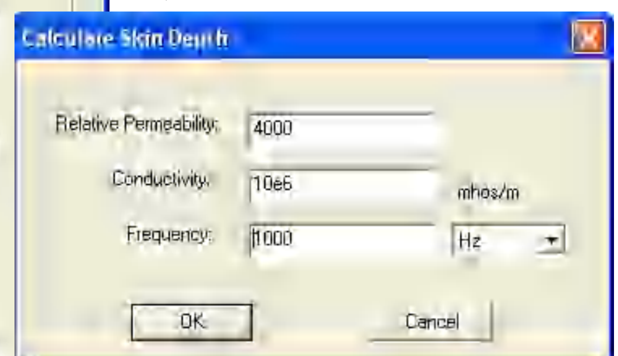


On selection – length based



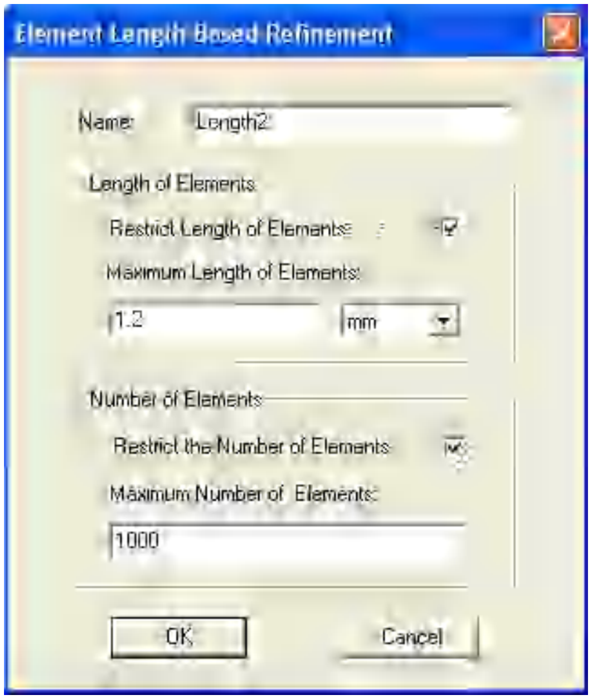


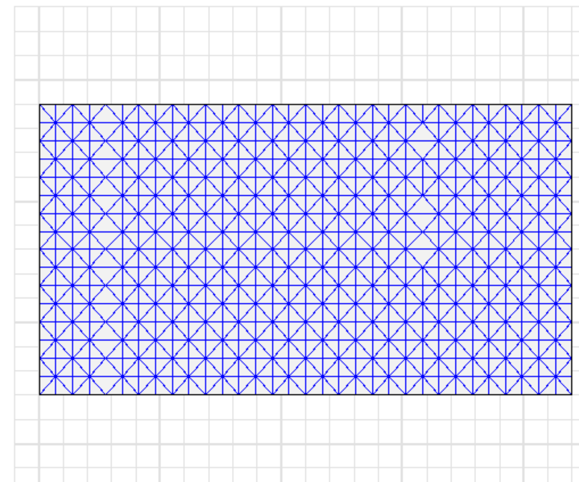
On selection – skin depth based (2 layers)



2. Mesh Operations “Inside selection” - applied throughout the volume of the object

- Element length based refinement: *Length Based*

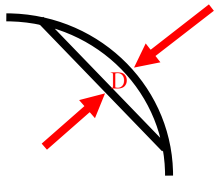




Inside selection – length based

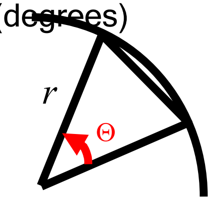
3. Mesh Operations “Surface Approximation”

- For true surfaces, perform faceting control on a face-by-face basis
- Select **Mesh operation > Assign > Surface approximation** and specify one or more settings:
  - Maximum surface deviation (length)



D = Maximum Surface Deviation

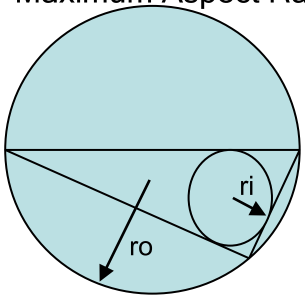
  - Maximum Surface Normal Deviation (degrees)



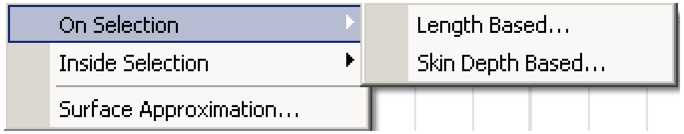
Θ = Maximum Surface Normal Deviation

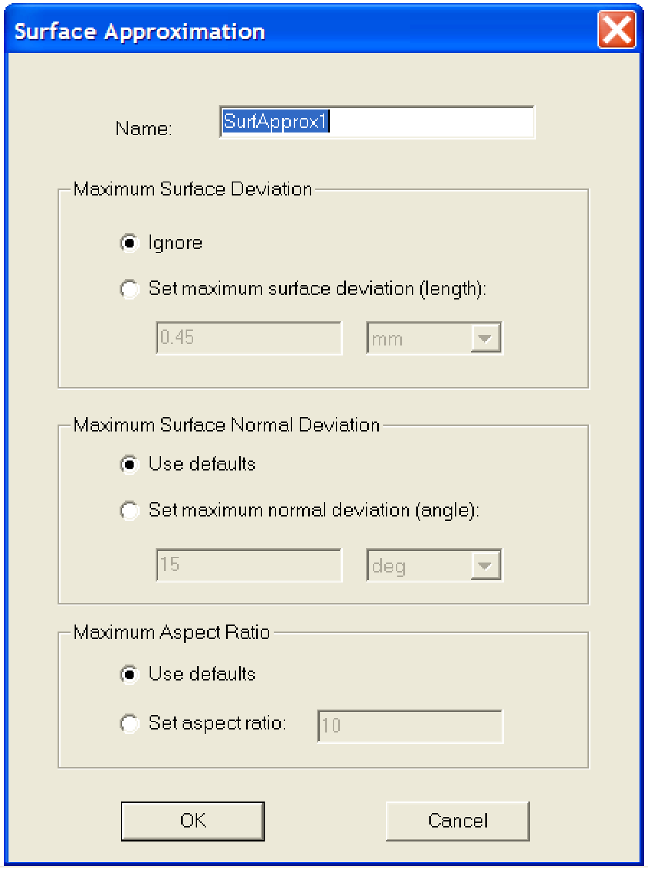
$$D = r(1 - \cos(\Theta/2))$$

  - Maximum Aspect Ratio



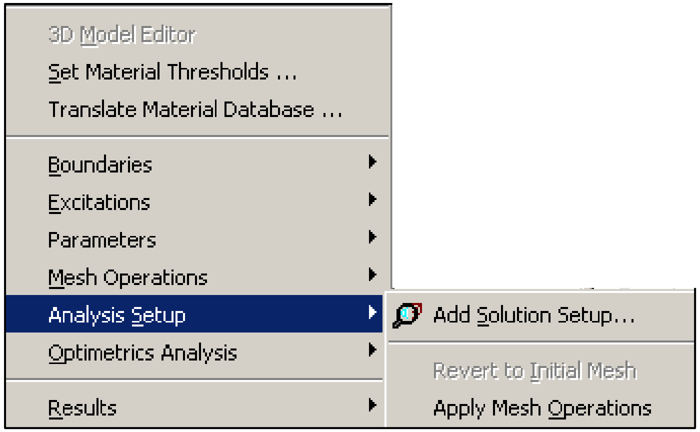
$$AspectRatio = \frac{ro}{2 * ri}$$





Manual mesh creation

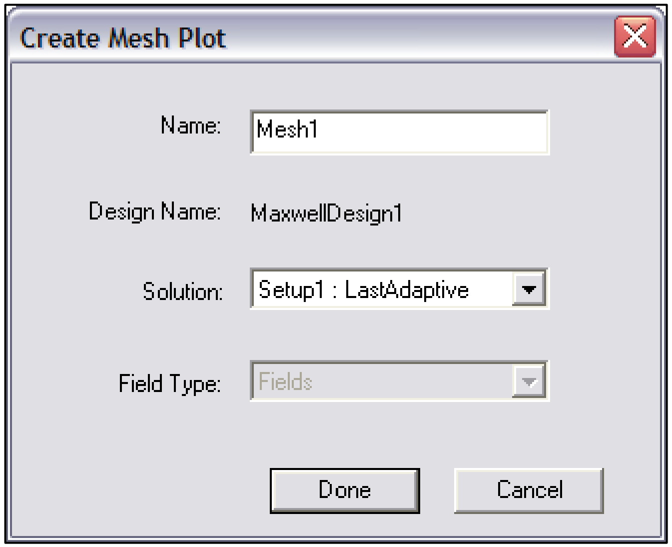
- To create the initial mesh: Click *Maxwell > Analysis Setup > Apply Mesh Operations*
- To refine the mesh without solving
  1. Define mesh operations as previously discussed
  2. Click *Maxwell > Analysis Setup > Apply Mesh Operations*
  3. Click *Maxwell > Analysis Setup > Revert to Initial Mesh* to restart to the initial mesh

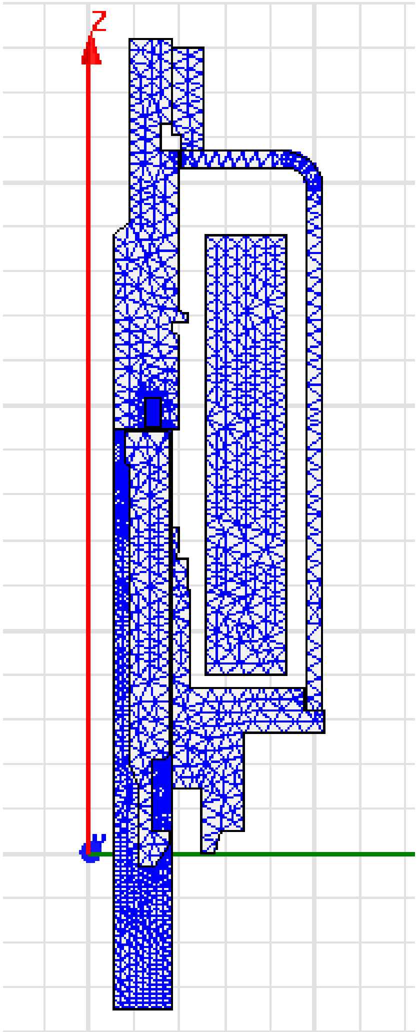


- To view mesh information: Click *Maxwell > Results > Solution Data* and click on the tab *Mesh Statistics*

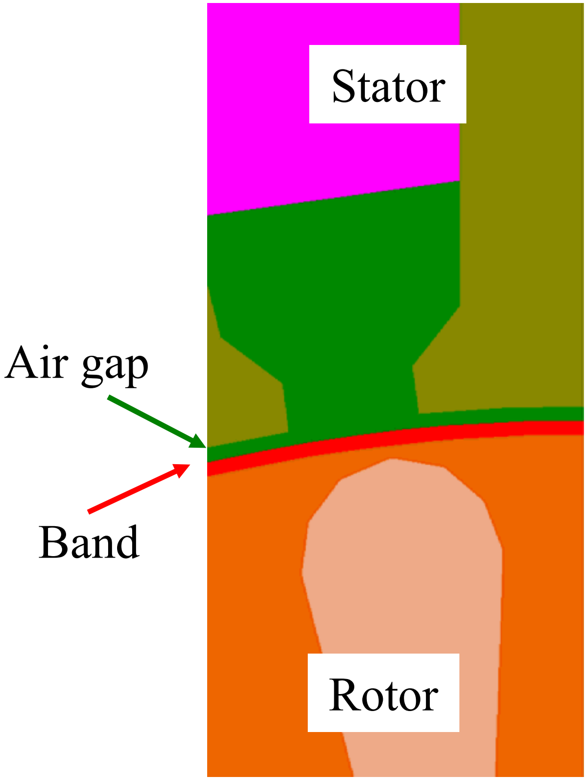
Mesh Display

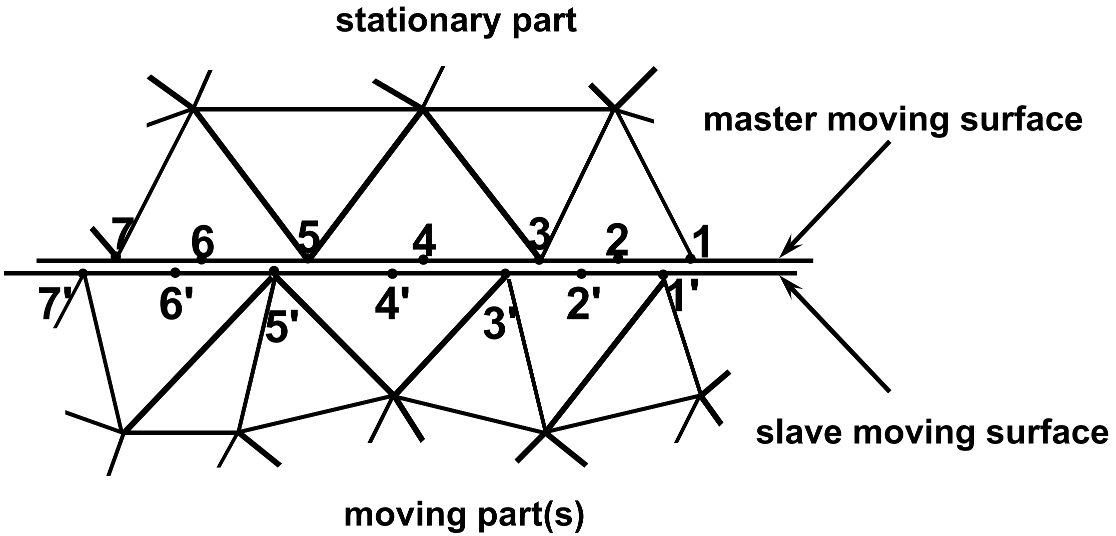
1. Select an object
2. Select the menu item *Maxwell 2D > Fields > Plot Mesh*



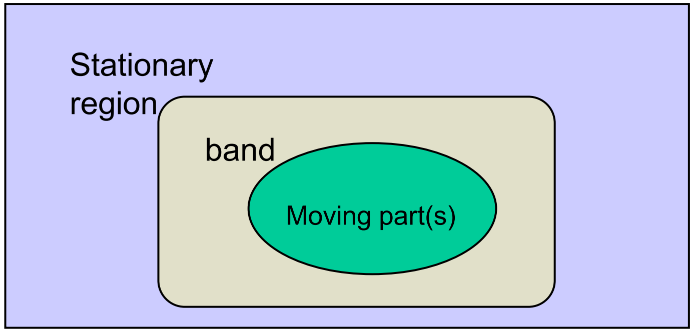


- 2D transient meshing for rotational models
- “Moving Surface” method used





- 2D transient meshing for translational models
- "Moving Band" method used
  - Adaptive meshing not used, so user must manually create the mesh or link to a solved MS or Eddy design
  - The band area is re-meshed at each time step
  - The stationary region and moving part(s) are not re-meshed
  - If you link the mesh to a solved MS or Eddy design:
    - The entire mesh from the linked design is transferred to the transient design.
    - The mesh in objects inside and outside of the band never changes as motion occurs.
    - If the starting transient position is the same as the linked MS or Eddy design, then the linked mesh in the band object is reused.
    - If the starting transient position is the different than the linked MS or Eddy design, then the linked mesh in the band object is completely deleted. The band is then re-meshed based only on mesh operations in the transient solver. Any mesh or mesh operation on the band in the linked MS or Eddy Design is ignored. <u>The key point is that mesh operations are always required on the band object (use inside selection) for Maxwell 2D transient designs.</u>
    - For subsequent positions as the object(s) move in the band, the mesh operations on the band in the transient design are re-applied at every timestep and a new mesh is created.

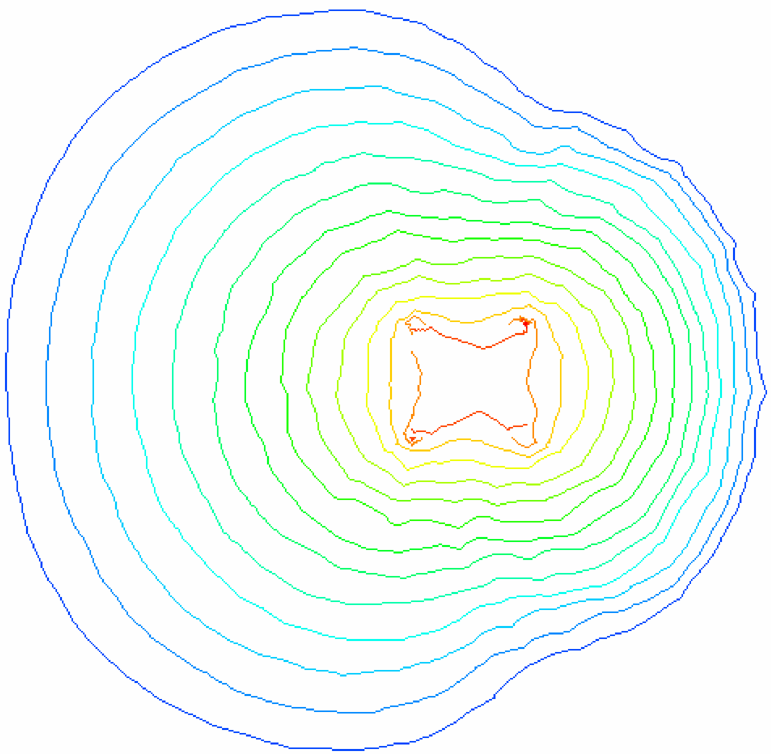


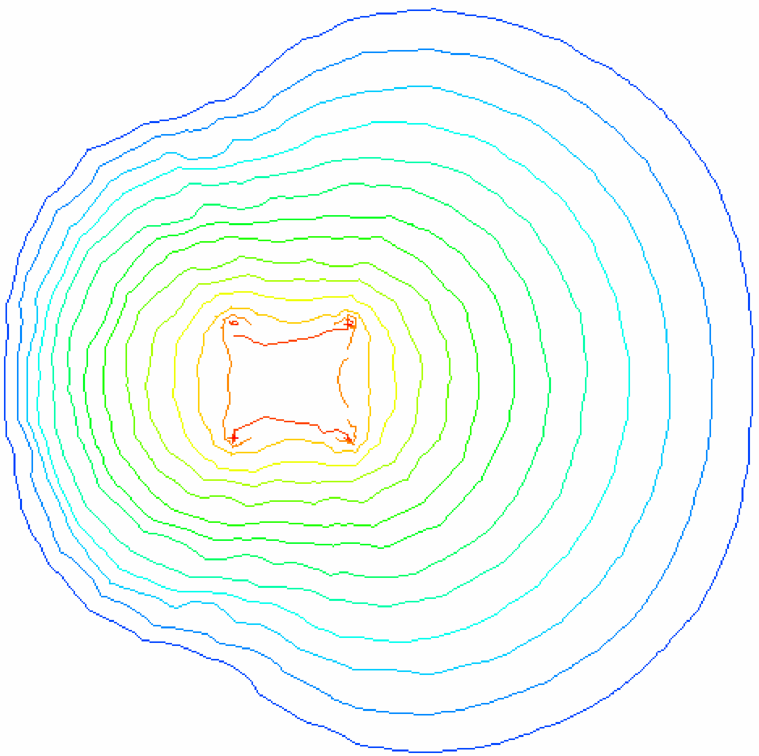
Post Processing

- Two Methods of Post Processing Solutions:
  - Viewing Plots
  - Manipulating Field Quantities in Calculator

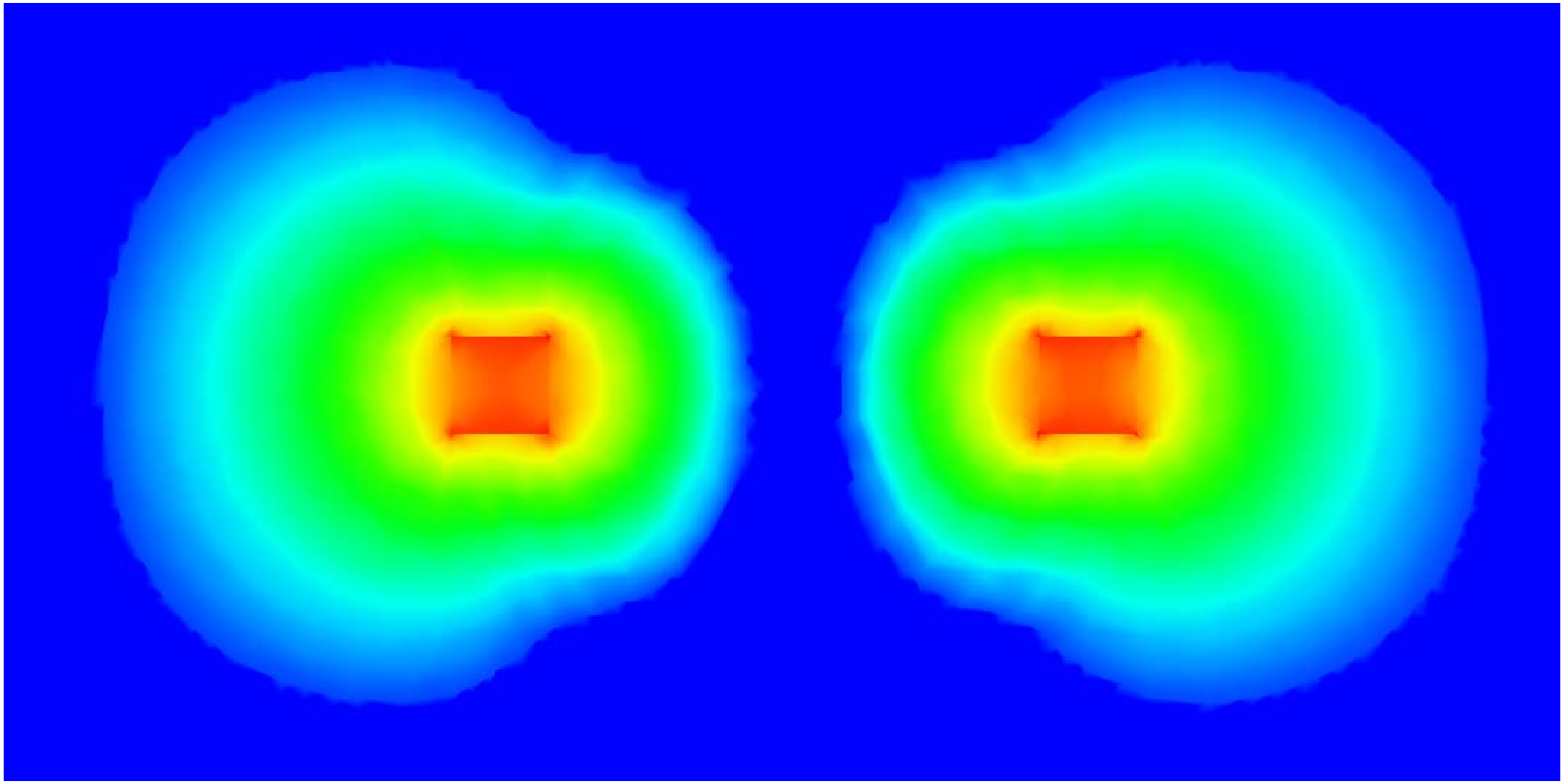
- Five Types of Plots:
  1. Contour plots (scalars): equipotential lines, ...
  2. Shade plots (scalars): Bmag, Hmag, Jmag, …
  3. Arrow plots (vectors): B vector, H vector, …
  4. Line plots (scalars): magnitude vs. distance along a predefined line
  5. Animation Plots

Contour plot

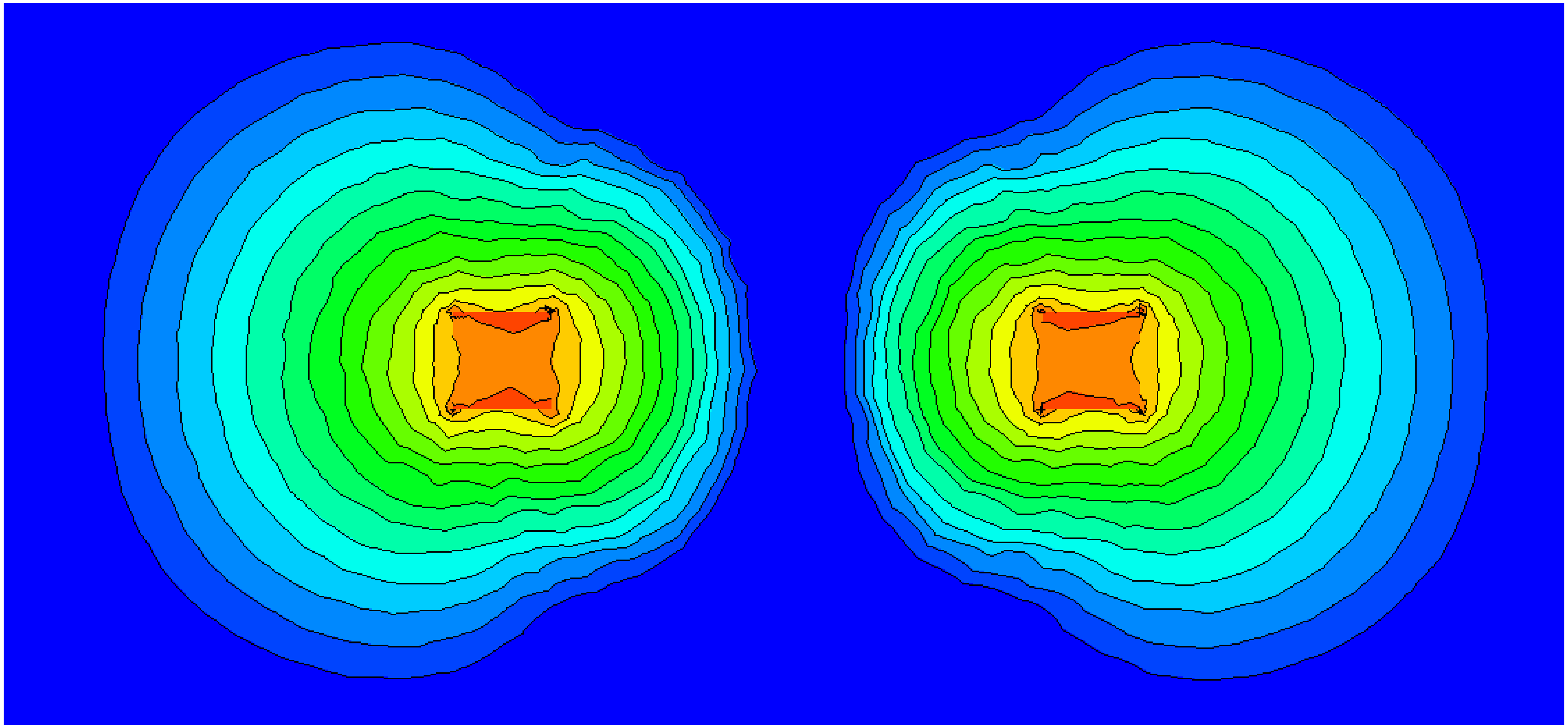




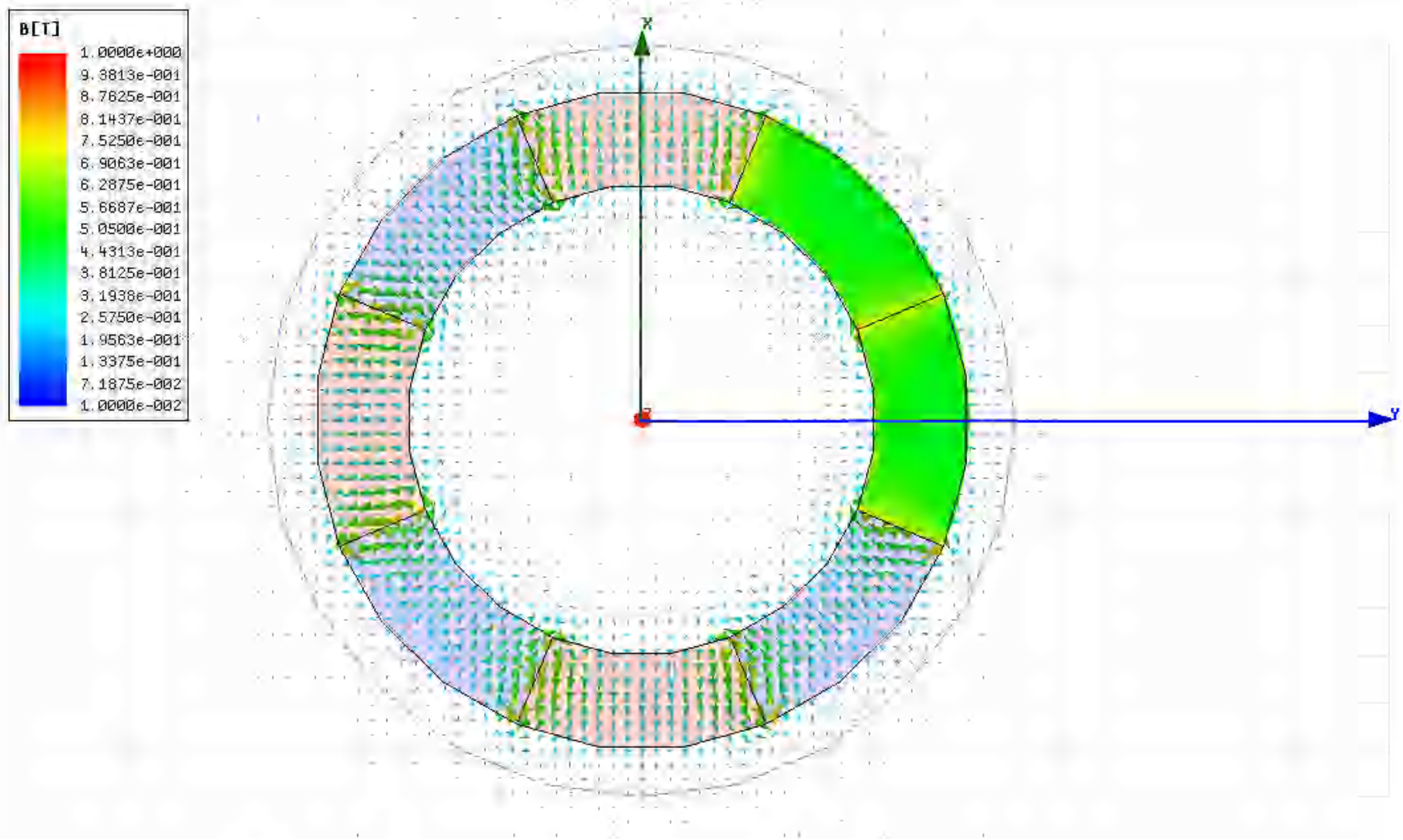
Shade plot (tone)



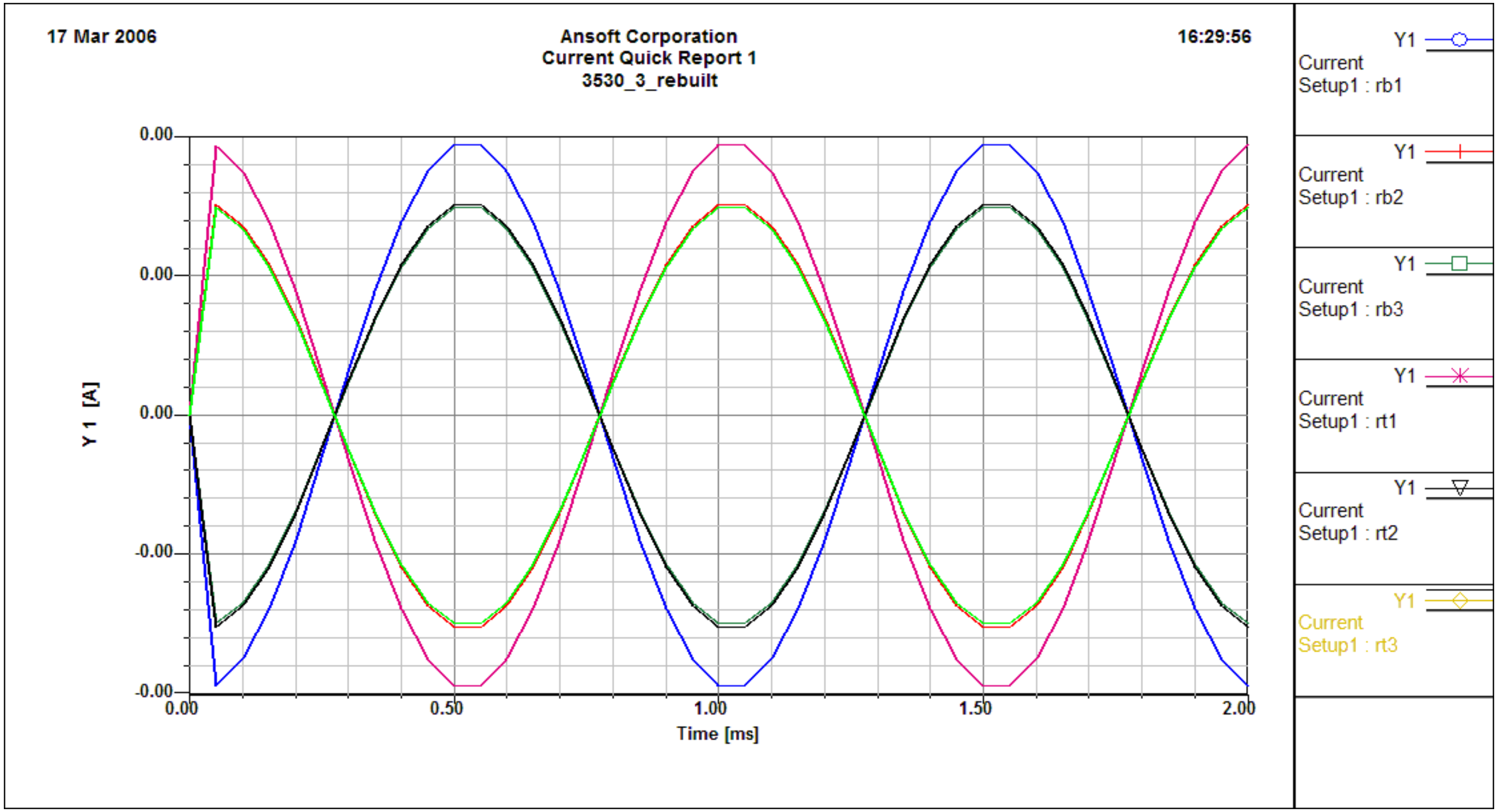
- Shade plot (fringe with outline)



Arrow plot


B[T]
1.0000e+000
9.3813e-001
8.7625e-001
8.1437e-001
7.5250e-001
6.9063e-001
6.2875e-001
5.6687e-001
5.0500e-001
4.4313e-001
3.8125e-001
3.1938e-001
2.5750e-001
1.9563e-001
1.3375e-001
7.1875e-002
1.0000e-002
X
Y

Line plot


17 Mar 2006
Ansoft Corporation
Current Quick Report 1
3530_3_rebuilt
16:29:56
0.00
0.00
0.00
-0.00
-0.00
Y 1 [A]
0.00
0.50
1.00
1.50
2.00
Time [ms]
Y1
Current
Setup1 : rb1
Y1
Current
Setup1 : rb2
Y1
Current
Setup1 : rb3
Y1
Current
Setup1 : rt1
Y1
Current
Setup1 : rt2
Y1
Current
Setup1 : rt3

- Multiple windows and multiple plots