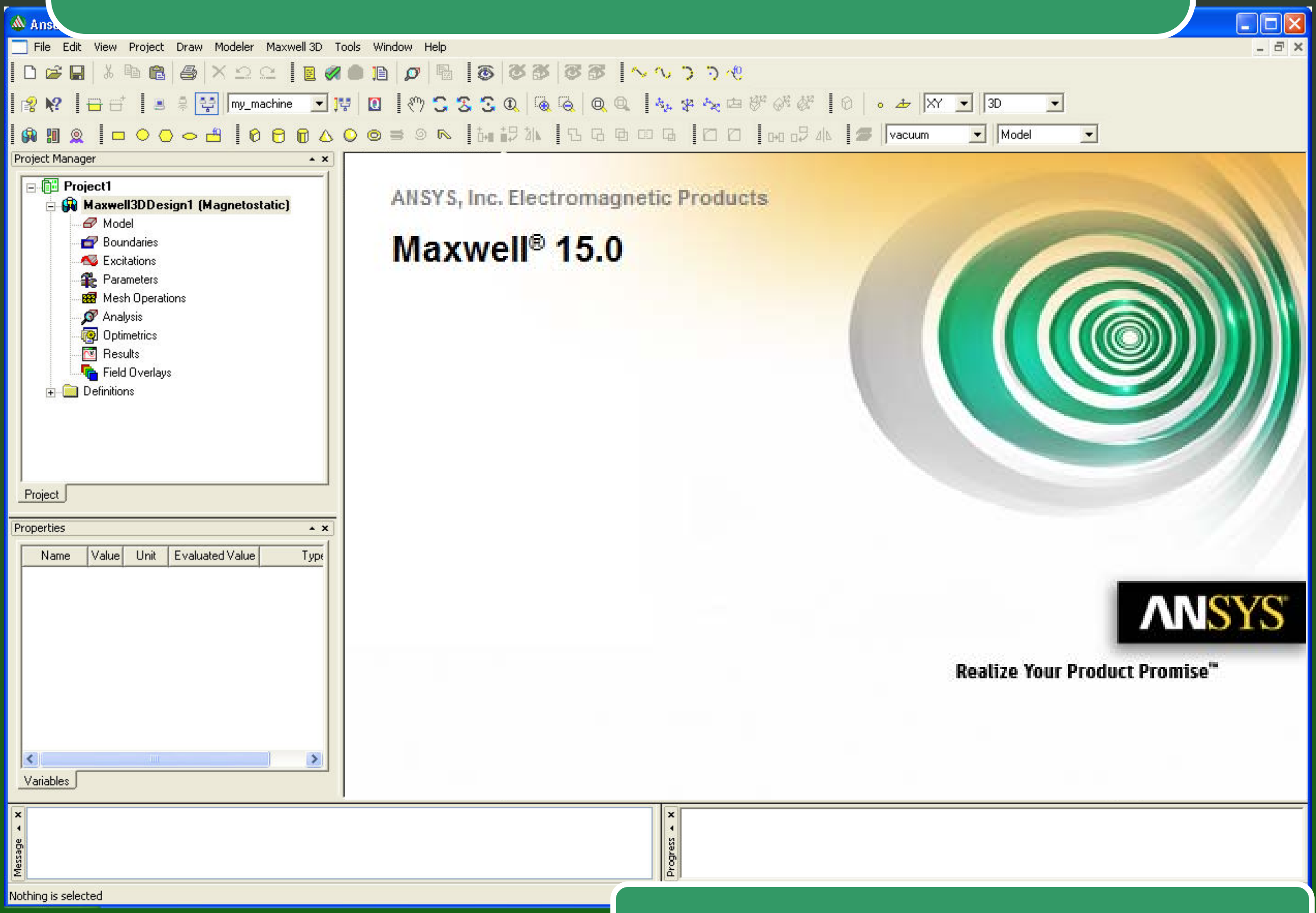
Maxwell 3D



electronic design automation software

user's guide – Maxwell 3D

ANSYS Inc • Southpointe • 275 Technology Drive • Canonsburg, PA 15317

The information contained in this document is subject to change without notice. ANSYS Inc. 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. ANSYS Inc. 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.

© 2010 ANSYS Inc. All rights reserved.

ANSYS Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304 (F) 724-514-9494

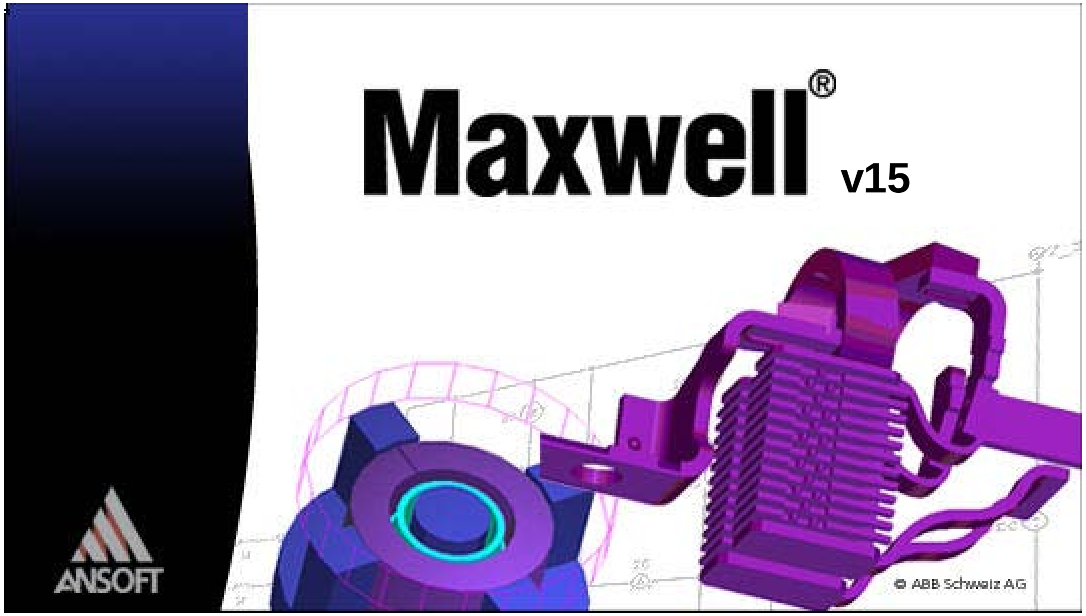
Ansoft Maxwell, Simplorer, RMxprt and Optimetrics and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or 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: REV6.0
Date: 13 March 2012
Software Version: 15

Contents

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



- Maxwell 3D is a high-performance interactive software package that uses finite element analysis (FEA) to solve electric, magnetostatic, eddy current, and transient problems.

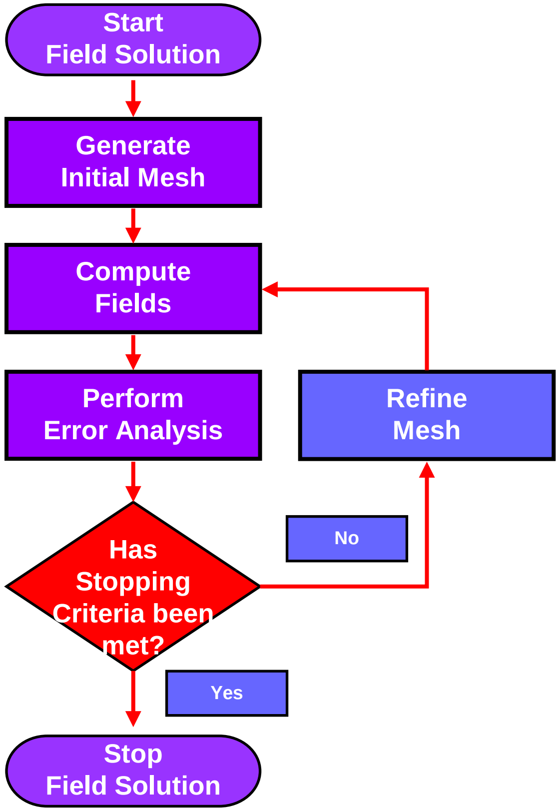
Maxwell® v15

- Maxwell solves the electromagnetic field problems by solving Maxwell's equations in a finite region of space with appropriate boundary conditions and — when necessary — with user-specified initial conditions in order to obtain a solution with guaranteed uniqueness.

- **Electric fields:**
  - Electrostatic fields in dielectrics
  - Electric fields in conductors
  - A combination of the first two with conduction solutions being used as boundary conditions for an electrostatic problem.
- **Magnetostatic fields**
- **Eddy current fields**
- **Transient fields**

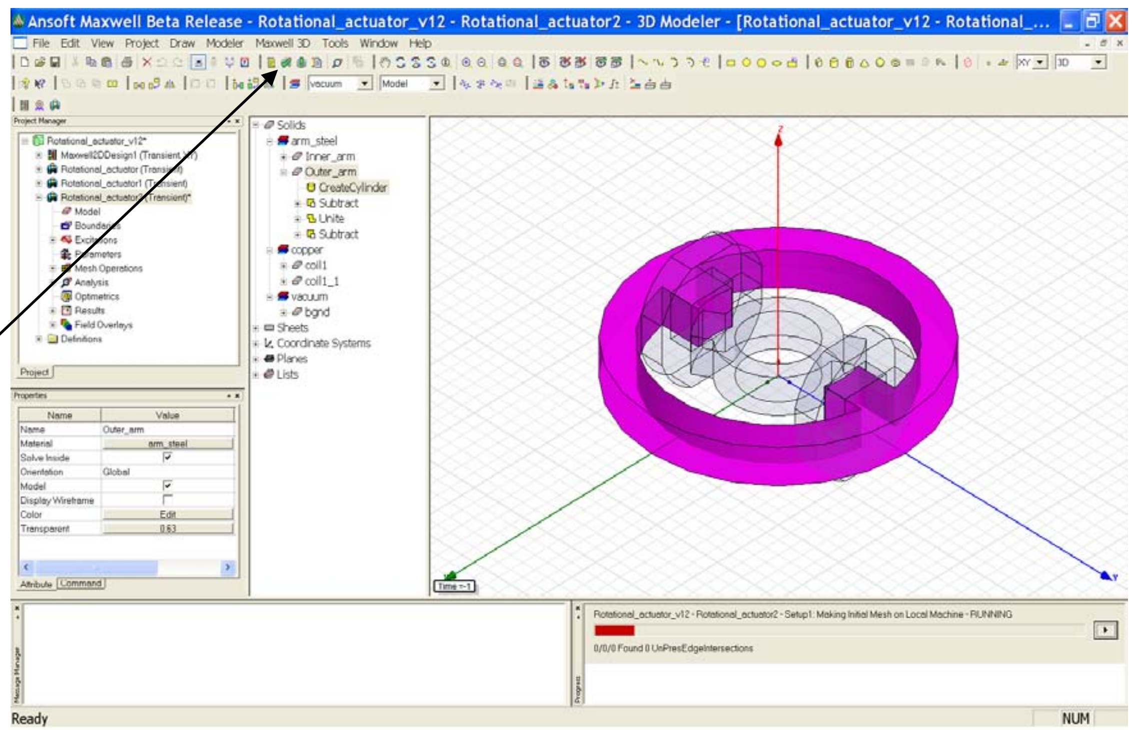
FEM and adaptive meshing

- In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into small elements (e.g., tetrahedra in 3D).
- All the model solids are meshed automatically by the mesher.
- The assembly of all tetrahedra is referred to as the finite element mesh of the model or simply the mesh.



GUI - Desktop

- The complex functionality built into the Maxwell solvers is accessed through the main user interface (called the desktop).
- Problem can be setup in a fairly arbitrary order (rather than following the steps in a precise order as was required in previous versions of Maxwell).
- A “validation check” insures that all required steps are completed.

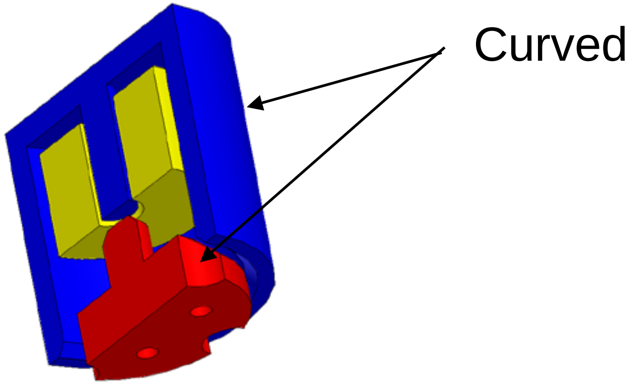


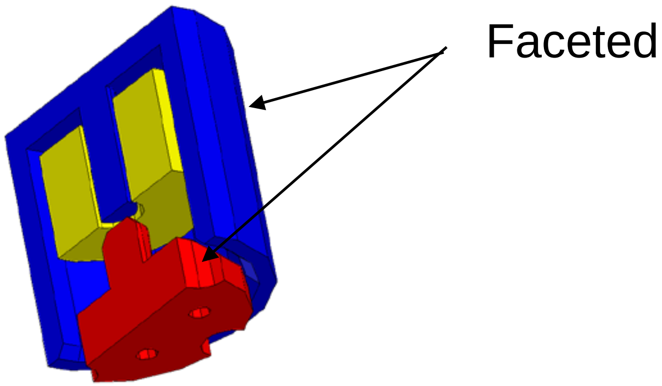
ACIS solid modeling kernel

- The underlying solid modeling technology used by Maxwell products is provided by ACIS geometric modeler. ACIS version 21 is used in Maxwell v15.
- 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 Maxwell products, translators are invoked that convert the models to an ACIS native format (sat format).
- Maxwell can also import CAD files (CATIA, Pro-E and Unigraphics) files directly. The import can be parametric or non-parametric
- Exports directly .sat, .dxf, .sm3, .sm2, .step, .iges

Curved vs. Faceted Surfaces

- The Maxwell 3D interface encourages the use of curved surfaces
- Many CAD programs use curved surfaces
- Curved surfaces do not directly translate into a better solution, due to the nature of the finite tetrahedral mesh elements – there is always some deviation from the curved surface and the meshed surface
- Using faceted surfaces can be more robust in many cases
- Faceted surfaces may converge in fewer iterations and may use fewer mesh elements – therefore, there is a possible large speed improvement!





v15 Supported platforms

- Windows XP 32-bit Service Pack 2
- Windows Server 2003 32-bit Service Pack 1
- Windows XP 64-bit Service Pack 2
- Windows Server 2003 64-bit Service Pack 1
- Windows HPC Server 2008
- Windows 7 Business Editions (32-bit and 64-bit versions)
- Red Hat (32 and 64 bit) v.4, v.5
- SuSE (32 and 64 bit) v10, v11

System requirements (Windows)

32-Bit System Requirements

Minimum System Requirements:

Processor: All fully compatible 686 (or later) instruction set processors, 500 MHz

Hard Drive Space (for Maxwell software): 200 MB, RAM: 512 MB

Recommended Minimum Configuration (for Optimal Performance):

Processor: All fully compatible 786 (or later) instruction set processors, 1 GHz

Video card: 128-bit SVGA or PCI Express video card

Hard Drive Space (for Maxwell software and temporary files): 500 MB, RAM: 2 GB

64-bit System Requirements

Minimum System Requirements:

Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support

Hard Drive Space (for Maxwell software): 200 MB, RAM: 2 GB

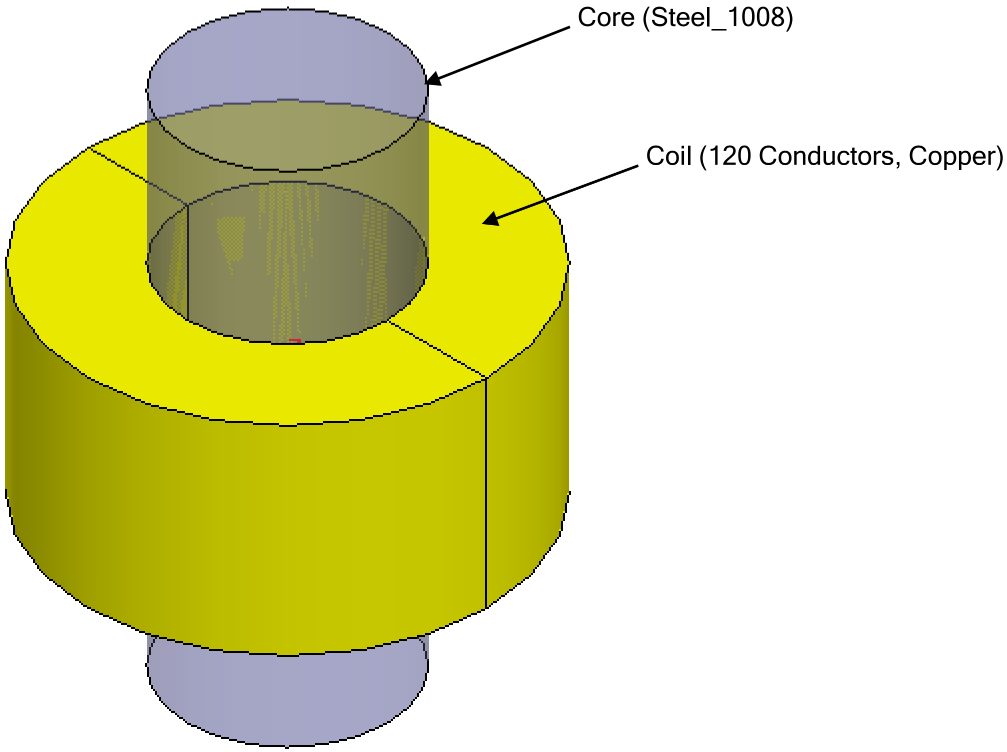
Recommended Minimum Configuration (for Optimal Performance):

Supported processors: AMD Athlon 64, AMD Opteron, Intel Xeon with Intel EM64T support, Intel Pentium 4 with Intel EM64T support

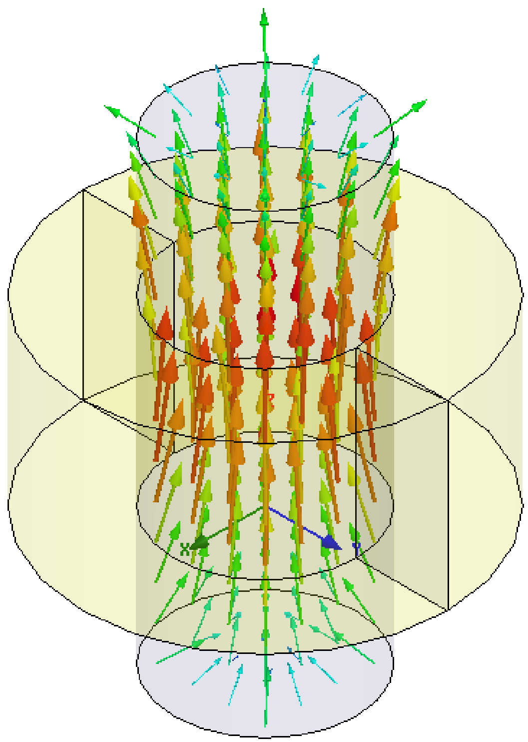
Video card: 128-bit SVGA or PCI Express video card

Hard Drive Space (for Maxwell software and temporary files): 700 MB, RAM: 8 GB

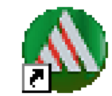
- Quick Example - Coil



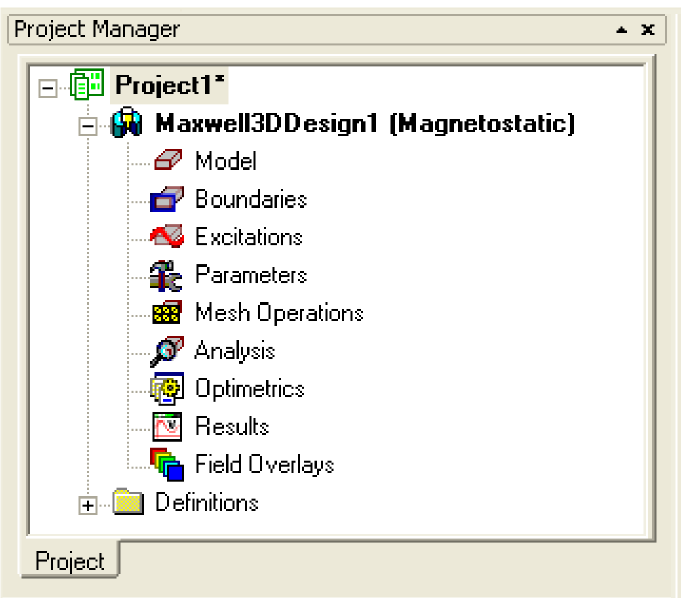
- Maxwell 3D
  - 3D field solver
  - Solves for the fields in an arbitrary volume

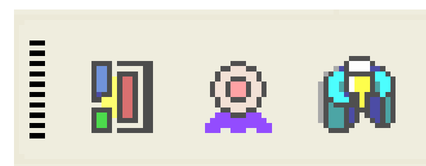


- # Starting Maxwell
  - Click the Microsoft **Start** button, select **Programs**, and select the ***Ansoft > Maxwell 15 > Windows 64-bit > Maxwell 15***
  - **Or** Double click on the Maxwell 15 icon on the Windows Desktop



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





Toolbar:
Insert Maxwell 2D Design

Insert Maxwell 3D Design

Insert RMxprt Design

- Maxwell Desktop

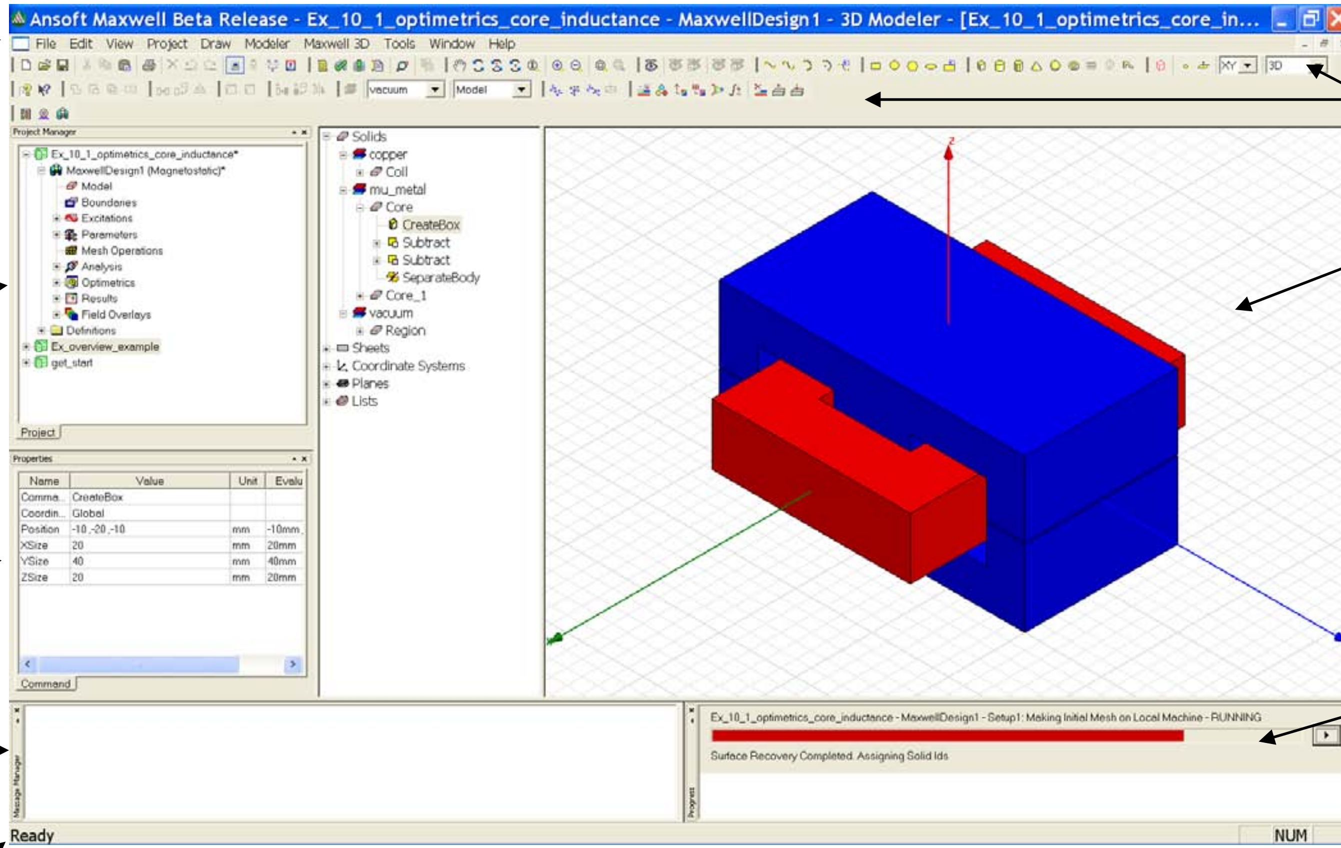
Menu bar

Toolbars

Project Manager with project tree

3D Modeler Window

Property Window

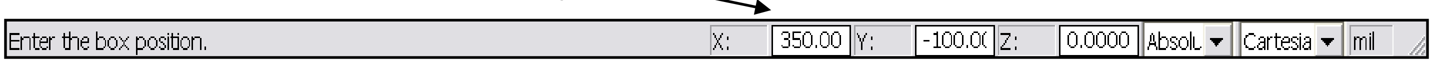


Message Manager

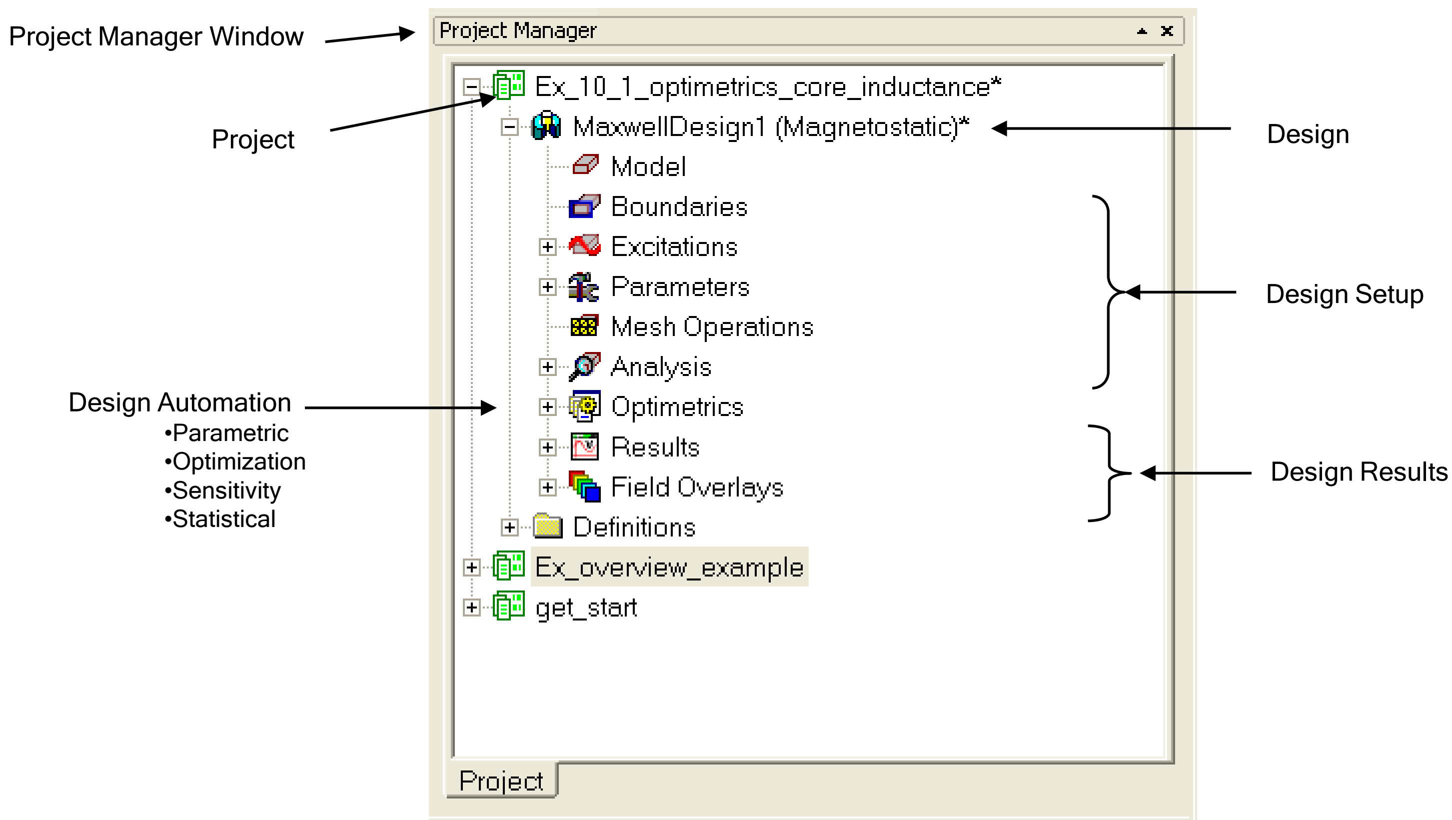
Progress Window

Status bar

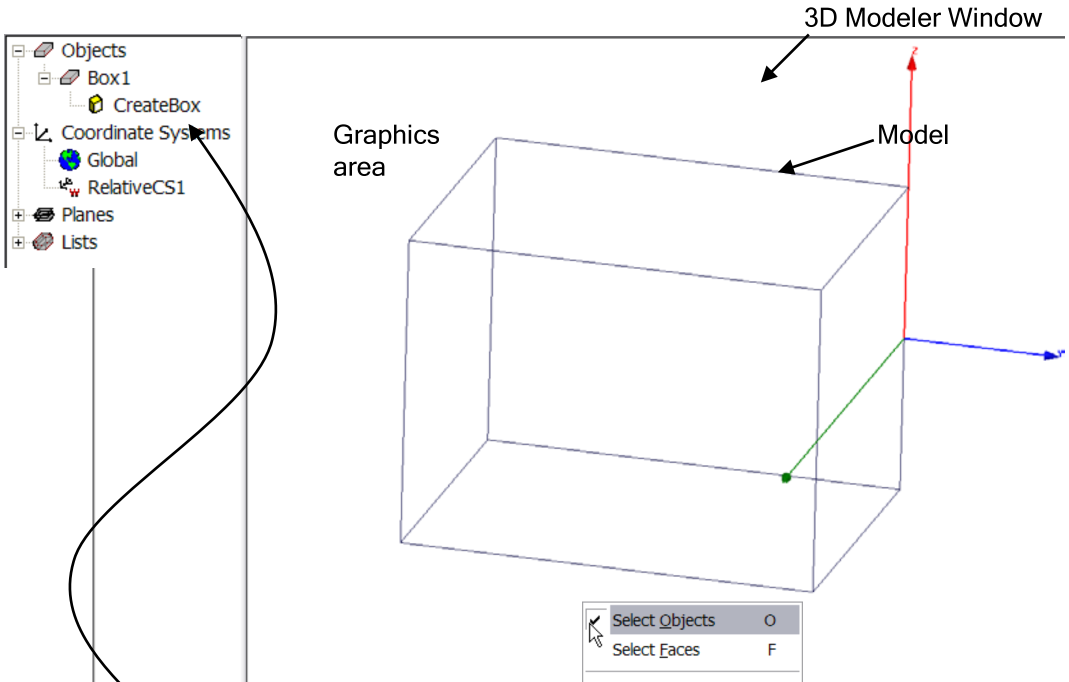
Coordinate Entry Fields



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

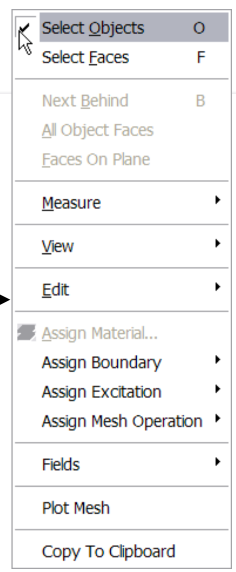


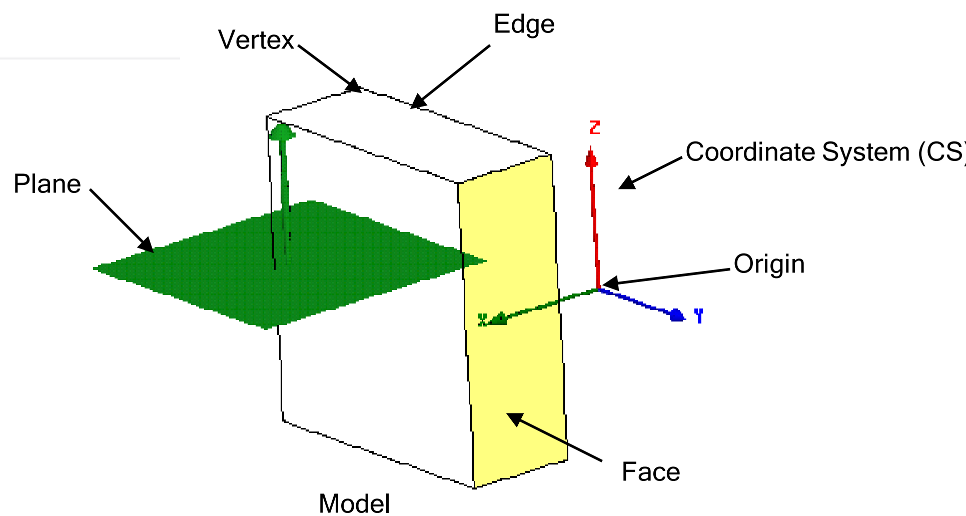
- Maxwell Desktop - 3D Modeler



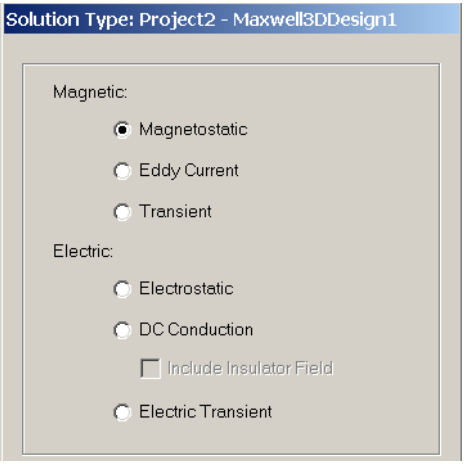
3D Modeler design tree (history)

Context menu (right mouse click on 3D modeler window)



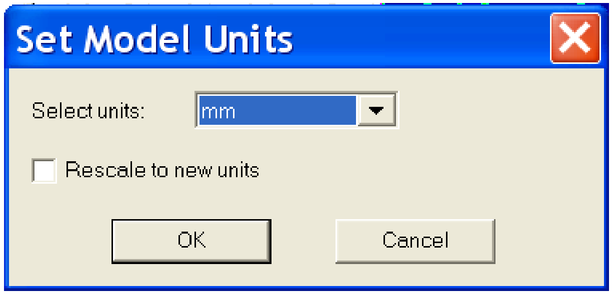


- **Set Solution Type**
  - **To set the solution type:**
    1. Select the menu item ***Maxwell 3D> Solution Type***
    2. Solution Type Window:
       1. Choose **Magnetostatic**
       2. Click the **OK** button
  - **To edit notes: Maxwell > Edit Notes**

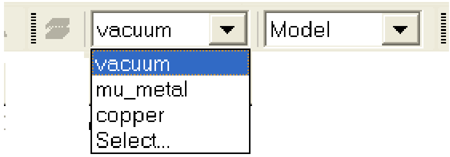


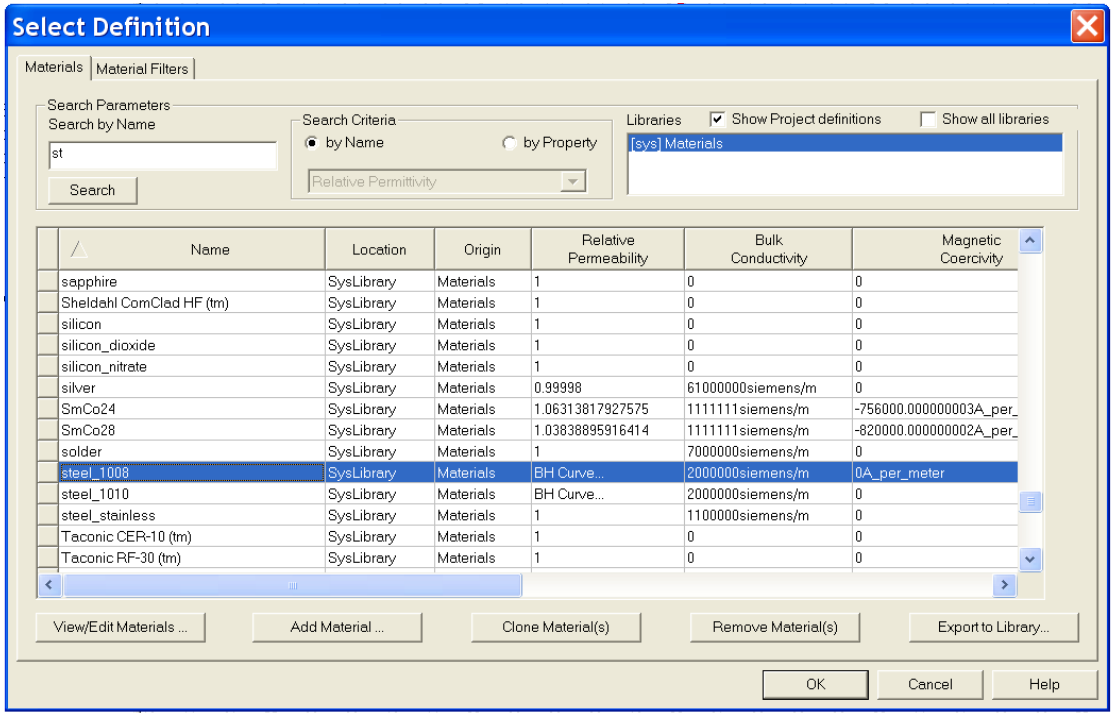
- **Maxwell - Solution Types**
  - **Magnetostatic -** Static magnetic fields, forces, torques, and inductances caused by DC currents, static external magnetic fields, and permanent magnets. Linear or nonlinear materials.
  - **Eddy Current -** Sinusoidally-varying magnetic fields, forces, torques, and impedances caused by AC currents and oscillating external magnetic fields. Linear materials only. Full wave solver considers displacement currents. Induced fields such as skin and proximity effects considered.
  - **Transient Magnetic** - Transient magnetic fields caused by time-varying or moving electrical sources and permanent magnets. Linear or nonlinear materials. Induced fields such as skin and proximity effects considered. Sources can be DC, sinusoidal, or transient voltages or currents. Can use external schematic circuit or link to Simplorer.
  - **Electrostatic** - Static electric fields, forces, torques, and capacitances caused by voltage distributions and charges. Linear materials only.
  - **DC Conduction** - Voltage, electric field, and the current density calculated from the potential. The resistance matrix can be derived quantity. Insulators surrounding the conductors can also be added to the simulation to calculate the electric field everywhere including the insulators
  - **Transient Electric** - Transient electric fields caused by time-varying voltages, charge distributions, or current excitations in inhomogeneous materials. The transient electric field simulator computes time-varying electric fields. Electric potential is the solution quantity.

- ## 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

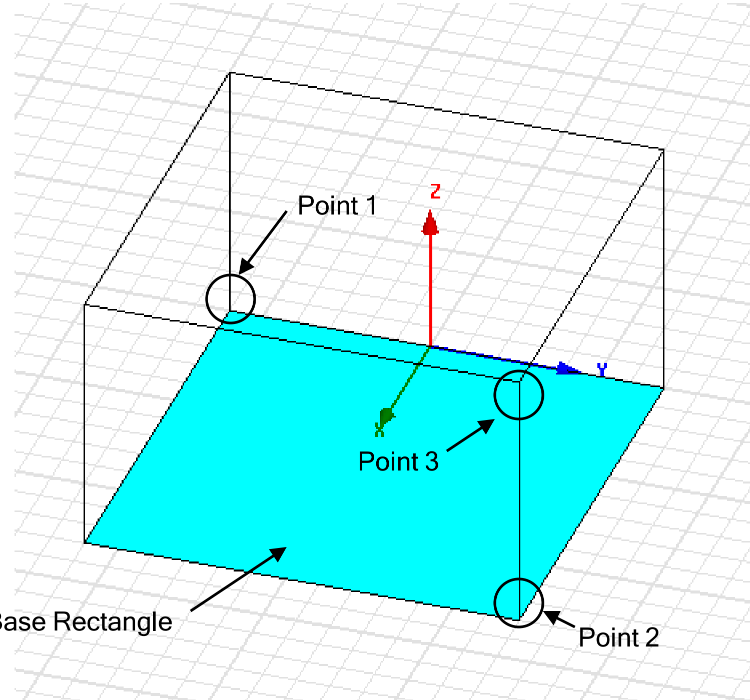


- ## Set Default Material
  - **To set the default material:**
    1. Using the Modeler Materials toolbar, choose **Select**
    2. Select Definition Window:
       1. Type **steel_1008** in the **Search by Name** field
       2. Click the **OK** button



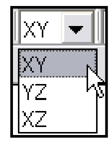


- ## Modeler - Create a Primitive

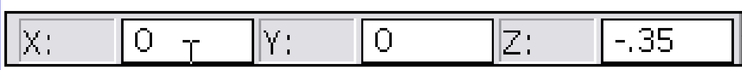


Base Rectangle

Grid Plane



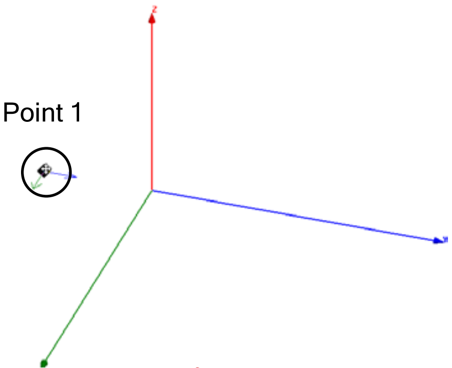
Note: Point 1 and Point 2 are drawn in the grid plane, while Point 3 is drawn in the 3rd direction

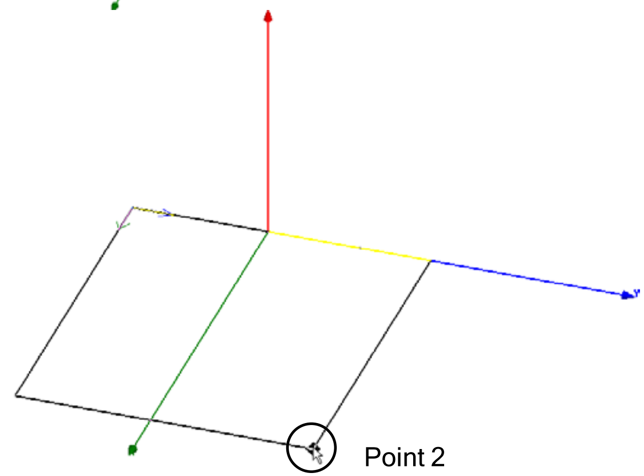


Coordinate Entry Fields

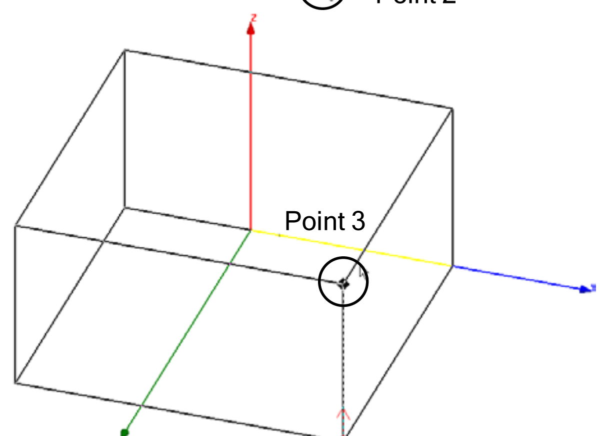
- The Coordinate Entry fields allow equations to be entered for position values.
  - Examples: 2*5, 2+6+8, 2*cos(10*(pi/180)).
- Variables are not allowed in the Coordinate Entry Field
- **Note**: Trig functions are in radians. Need to specify extension deg if specified in degrees. Ex. Sin(90deg)

Point 1





Point 2



- ## Modeler - Object Properties

Commands

(dimensions and history)

Properties: Project4 - MaxwellDesign1 - 3D Modeler

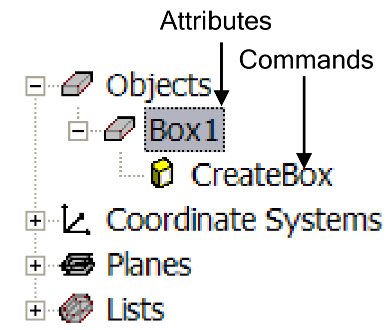
Command | Attribute

Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate System	Global		
Center Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
Axis	Z		
Radius	0.447213595499958	mm	0.447213595499958mm
Height	0.6	mm	0.6mm

Show Hidden

OK Cancel

In History Tree:



Attributes

(properties of the object)

Properties: Project4 - MaxwellDesign1 - 3D Modeler

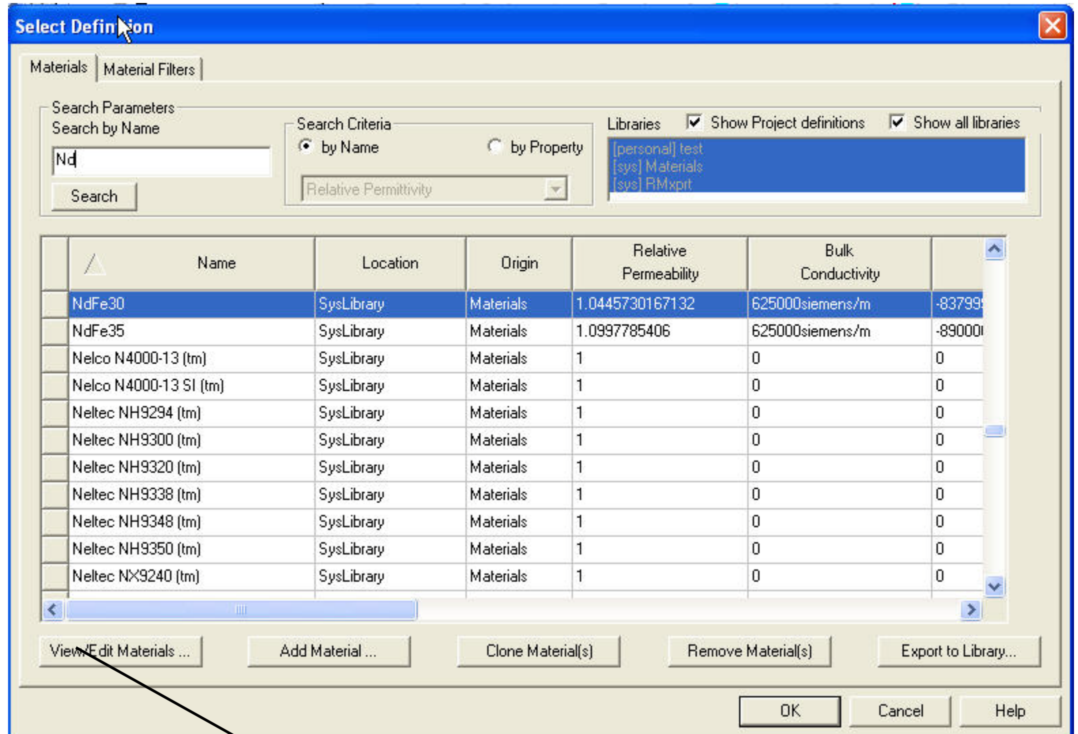
Command | Attribute

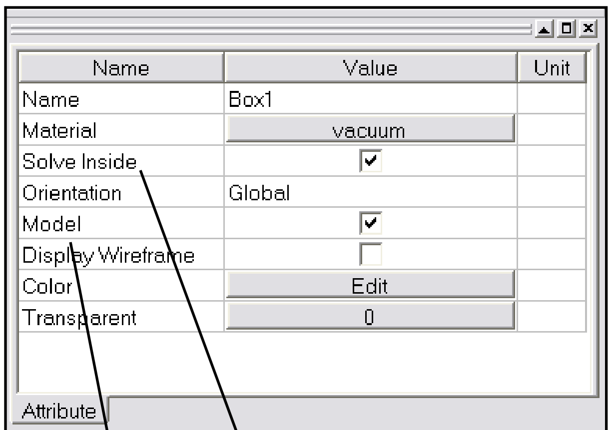
Name	Value	Unit	Evaluated Value	Description	Read-only
Name	Cylinder1				☐
Material	vacuum				☐
Solve Inside	☑				☐
Orientation	Global				☐
Model	☑				☐
Display Wireframe	☐				☐
Color	Edit				☐
Transparent	0				☐

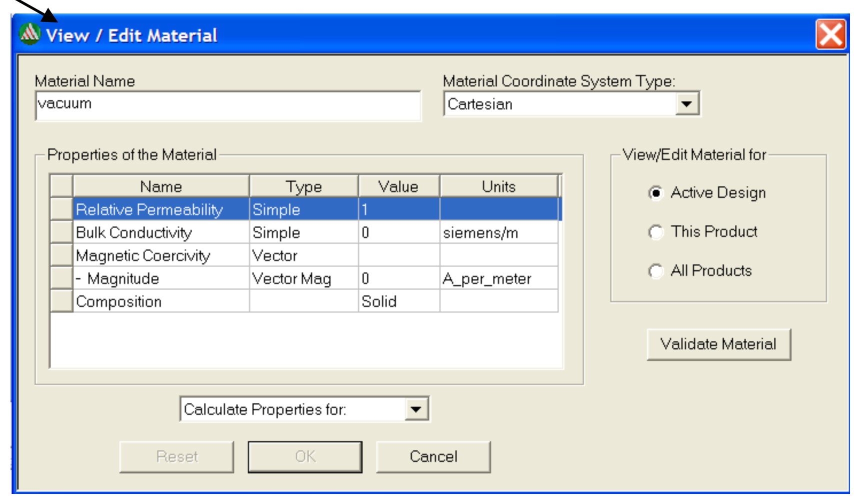
Show Hidden

OK Cancel

- Modeler - Attributes





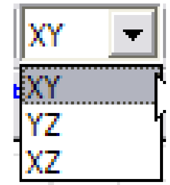


Solve Inside - if unchecked meshes but no solution inside (like the old exclude feature in material manager)

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

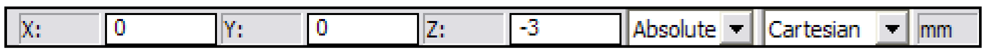
Set Grid Plane

- **To set the Grid Plane:**
  - Select the menu item ***Modeler > Grid Plane > XY***



Create Core

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

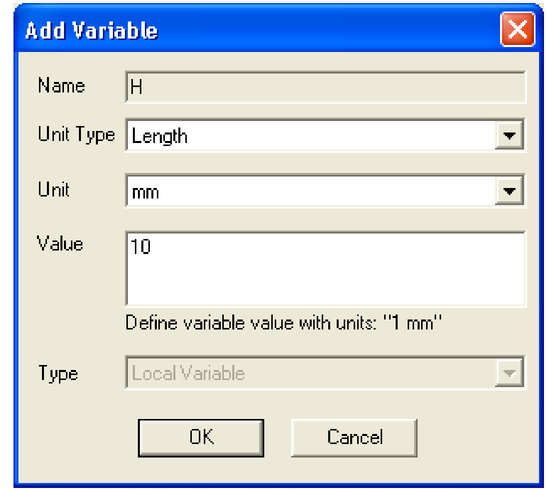


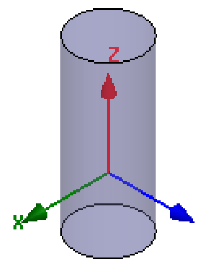
  4. Using the coordinate entry fields, enter the radius of the cylinder
     - dX**: 0.0**, dY**: 2.0**, dZ**: 0.0**, Press the **Enter** key

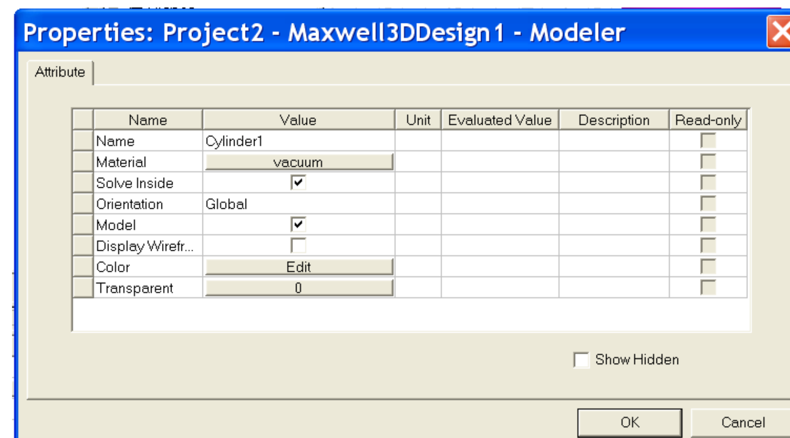
  5. Using the coordinate entry fields, enter the height of the cylinder
     - dX**: 0.0**, dY**: 0.0** dZ**: 10.0**, Press the **Enter** key

- **Continued on Next Page**

- **Create Core (Continued)**
  - **To Parameterize the Height**
    1. Select the **Command** tab from the **Properties** window
    2. Height: **H**
    3. Press the **Tab** key
    4. Add Variable Window
       1. Value: **10**
       2. Unit: **mm**
       3. 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 > Visibility > Fit All > Active View***

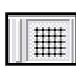




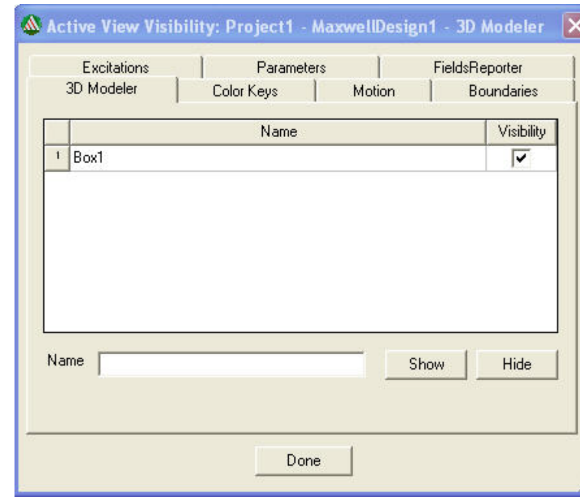


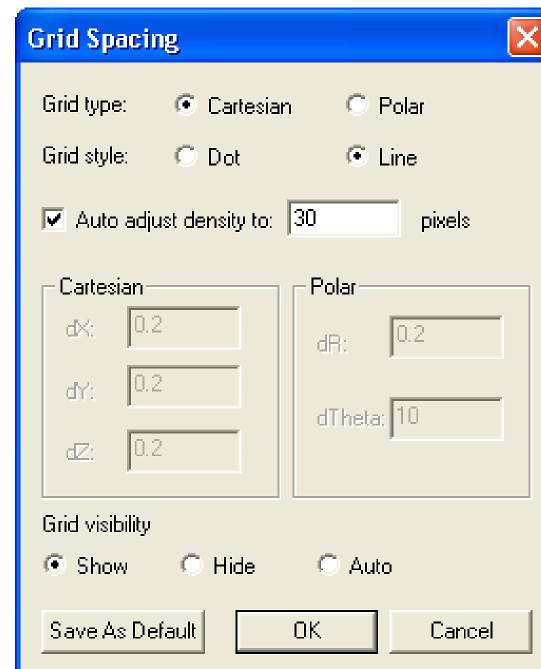
Modeler - Views

- *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 > Visibility > 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

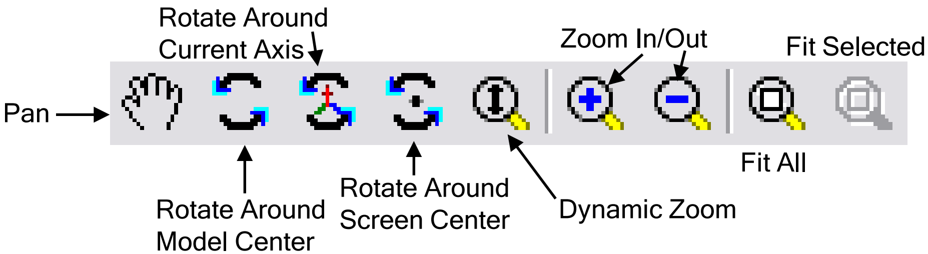


Toolbar: Toggle Grid Visibility

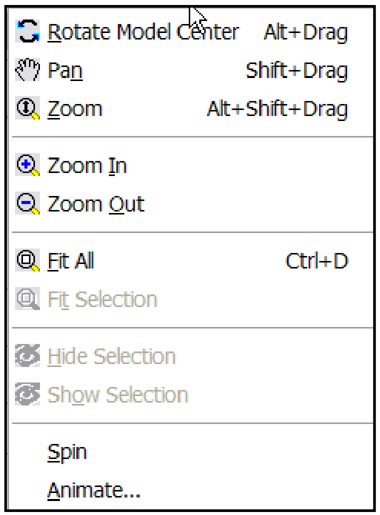




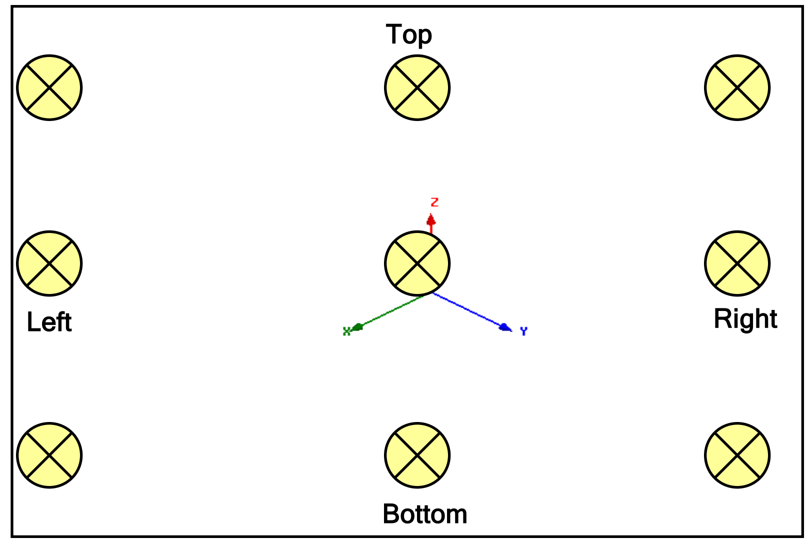
- **Changing the View**
  - **Toolbar**



  - **Context Menu**



Predefined View Angles



  - **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 3D Keyboard Shortcuts

General Shortcuts

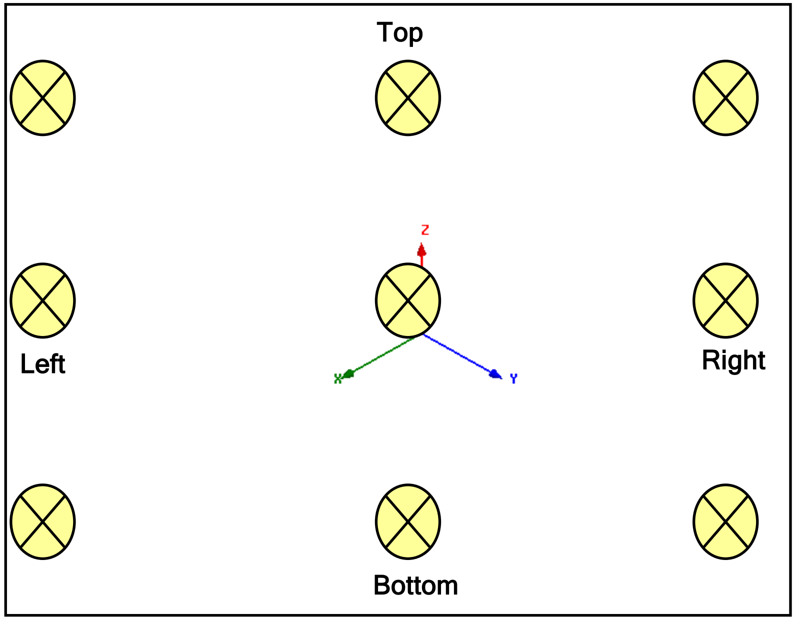
- F1: Help
- Shift + F1: Context help
- CTRL + F4: Close program
- CTRL + C: Copy
- CTRL + N: New project
- CTRL + O: Open...
- CTRL + S: Save
- CTRL + P: Print...
- CTRL + V: Paste
- CTRL + X: Cut
- CTRL + Y: Redo
- CTRL + Z: Undo
- CTRL + 0: Cascade windows
- CTRL + 1: Tile windows horizontally
- CTRL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

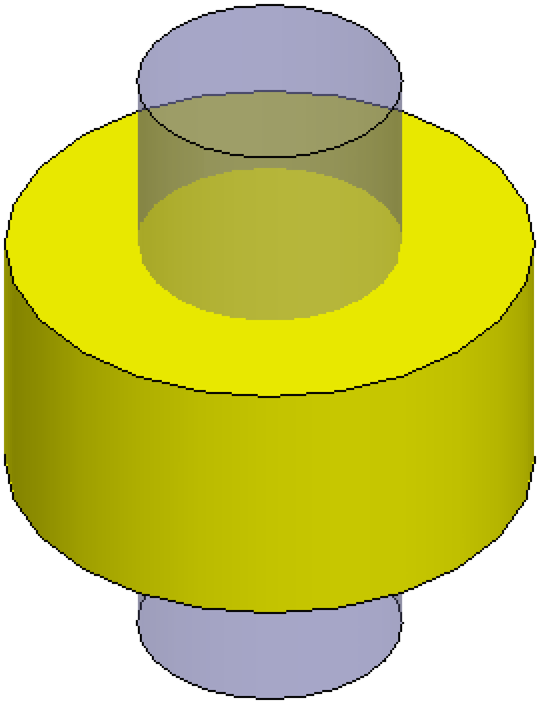
- Alt + Double Click Left Mouse Button at points on screen: Sets model projection to standard isometric projections (see diagram below).
- ALT + Right Mouse Button + Double Click Left Mouse Button at points on screen: give the nine opposite projections.

Predefined View Angles



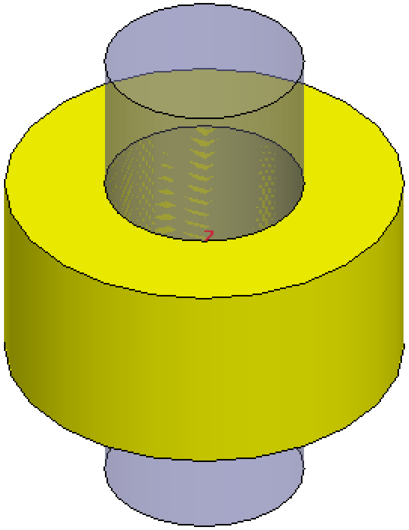
- ## Set Default Material
  - **To set the default material:**
    1. Using the 3D Modeler Materials toolbar, choose **Select**
    2. Select Definition Window:
       1. Type **copper** in the **Search by Name** field
       2. Click the **OK** button

- ## Create Coil
  - **To create the coil for the current to flow:**
    1. Select the menu item ***Draw > Cylinder***
    2. Using the coordinate entry fields, enter the center position
       - X**: 0.0**, Y**: 0.0**, Z**: 0.0,** Press the **Enter** key
    4. Using the coordinate entry fields, enter the radius of the cylinder
       - dX**: 0.0**, dY**: 4.0**, dZ**: 0.0**, Press the **Enter** key
    5. Using the coordinate entry fields, enter the height of the cylinder
       - dX**: 0.0**, dY**: 0.0** dZ**: 4.0**, Press the **Enter** key
  - **To set the name:**
    1. Select the **Attribute** tab from the **Properties** window.
    2. For the **Value** of **Name** type: **Coil**
    3. Click the **OK** button
  - **To fit the view:**
    1. Select the menu item ***View > Fit All > Active View***

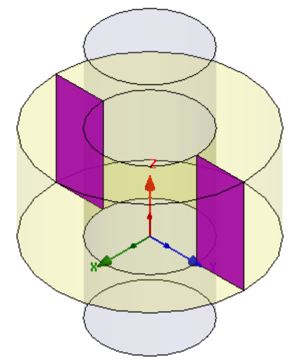


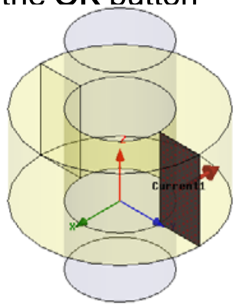
- ## Overlapping Objects
  - **About Overlapping Objects**
    - When the volume of a 3D object occupies the same space as two or more objects you will receive an overlap error during the validation process. This occurs because the solver can not determine which material properties to apply in the area of overlap. To correct this problem, Boolean operations can be used to subtract one object from the other or the overlapping object can be split into smaller pieces that are completely enclosed within the volume of another object. When an object is completely enclosed there will be no overlap errors. In this case, the material of the interior object is used in the area of overlap.

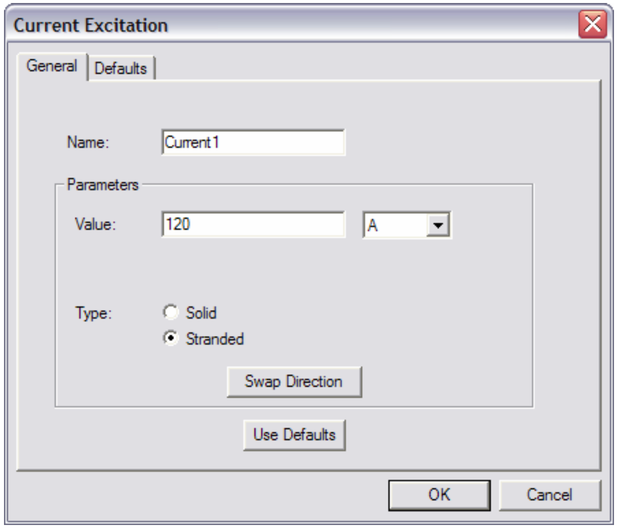
- ## Complete the Coil
  - **To select the objects Core and Coil:**
    1. Select the menu item ***Edit > Select All***
  - **To complete the Coil:**
    1. Select the menu item ***Modeler > Boolean > Subtract***
    2. Subtract Window
       - Blank Parts: **Coil**
       - Tool Parts: **Core**
       - Clone tool objects before subtract: ☑ **Checked**
       - Click the **OK** button

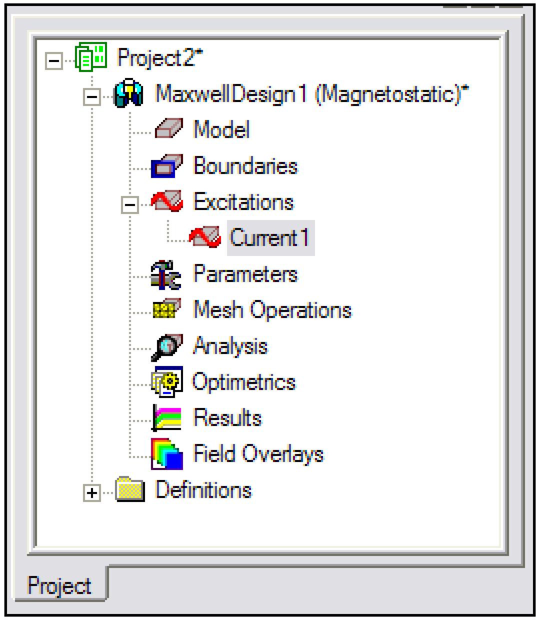


- ## Create Excitation
  - **Object Selection**
    1. Select the menu item ***Edit > Select > By Name***
    2. Select Object Dialog,
       1. Select the objects named: **Coil**
       2. Click the **OK** button
  - **Section Object**
    1. Select the menu item ***Modeler > Surface > Section***
       1. Section Plane: ***YZ***
       2. Click the **OK** button
  - **Separate Bodies**
    1. Select the menu item ***Modeler > Boolean > Separate Bodies***
    2. Delete the extra sheet which is not needed
  - **Assign Excitation**
    1. Select the menu item ***Maxwell 3D> Excitations > Assign > Current***
    2. Current Excitation : General
       1. Name: **Current1**
       2. Value: **120 A**
       3. Type: **Stranded**
    3. Click the **OK** button

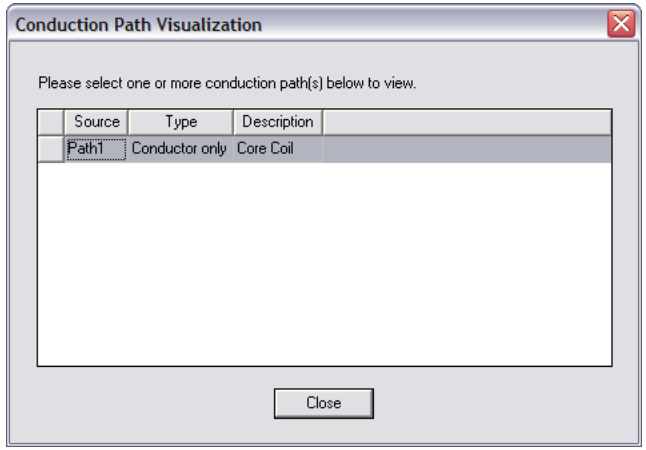


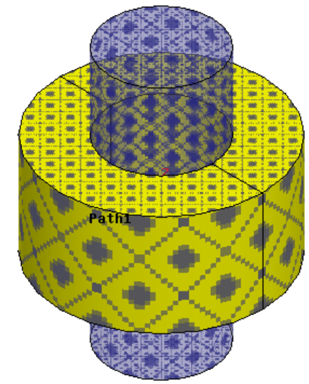






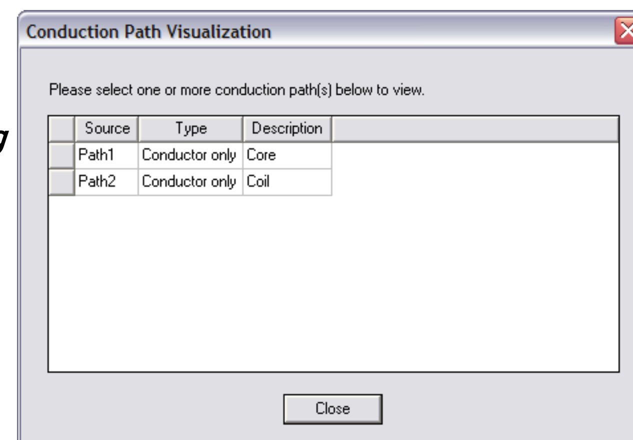
- **Show Conduction Path**
  - **Show Conduction Path**
    1. Select the menu item ***Maxwell 3D> Excitations > Conduction Path > Show Conduction Path***
    2. From the Conduction Path Visualization dialog, select the row1 to visualize the conduction path in on the 3D Model.
    3. Click the **Close** button



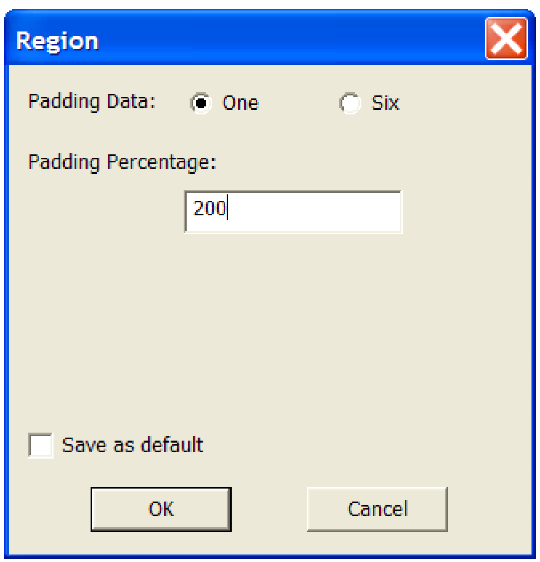


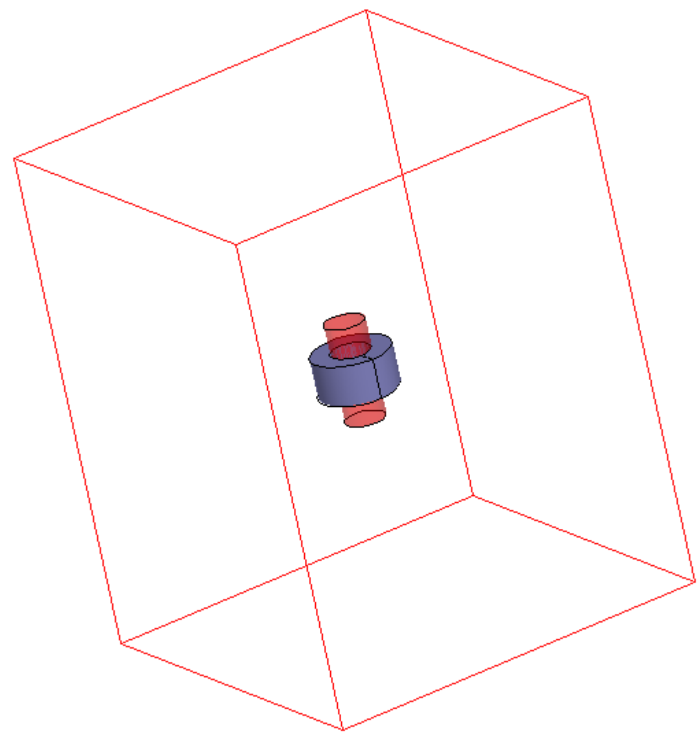
  - **Fix Conduction Path**
    - To solve this isolation problem an insulating boundary condition will be used.

    1. Select the outer face of the core by typing **f** on the keyboard and select the outer face of the core.
    2. Select the menu item ***Maxwell 3D> Boundaries > Assign > Insulating***
    3. Insulating Boundary
       1. Name: **Insulating1**
       2. Click the **OK** button
    4. Follow the instructions from above to redisplay the Conduction Path



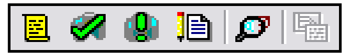
- **Define a Region**
- Before solving a project a region has to be defined. A region is basically an outermost object that contains all other objects. The region can be defined by a special object in ***Draw > Region***. This special region object will be resized automatically if your model changes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



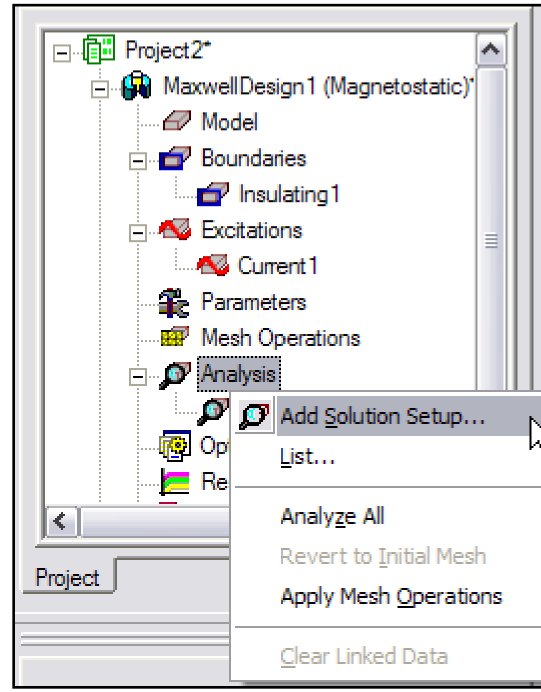


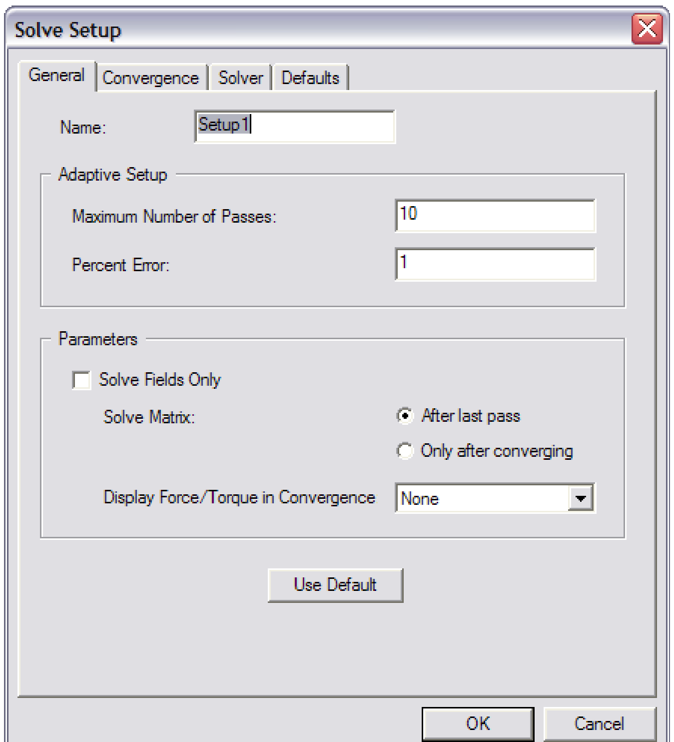
Note: Since there will be considerable fringing in this device, a padding percentage of at least 2 times, or 200% is recommended

- Maxwell - Solution Setup
- **Creating an Analysis Setup**
  - **To create an analysis setup:**
    1. Select the menu item ***Maxwell 3D> 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

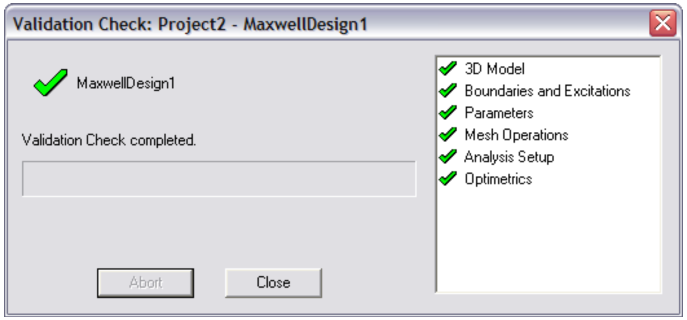




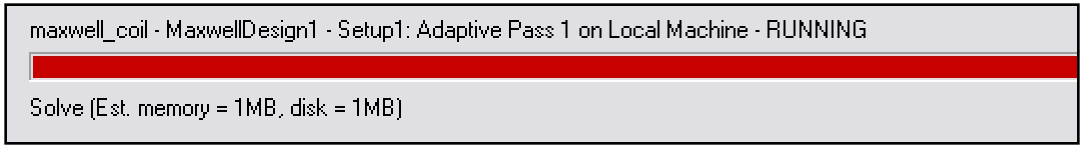
- ## Save Project
  - **To save the project:**
    1. In an Maxwell window, select the menu item ***File > Save As***.
    2. From the **Save As** window, type the Filename: **maxwell_coil**
    3. Click the **Save** button

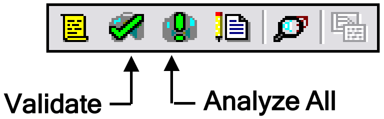
- ## Analyze

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



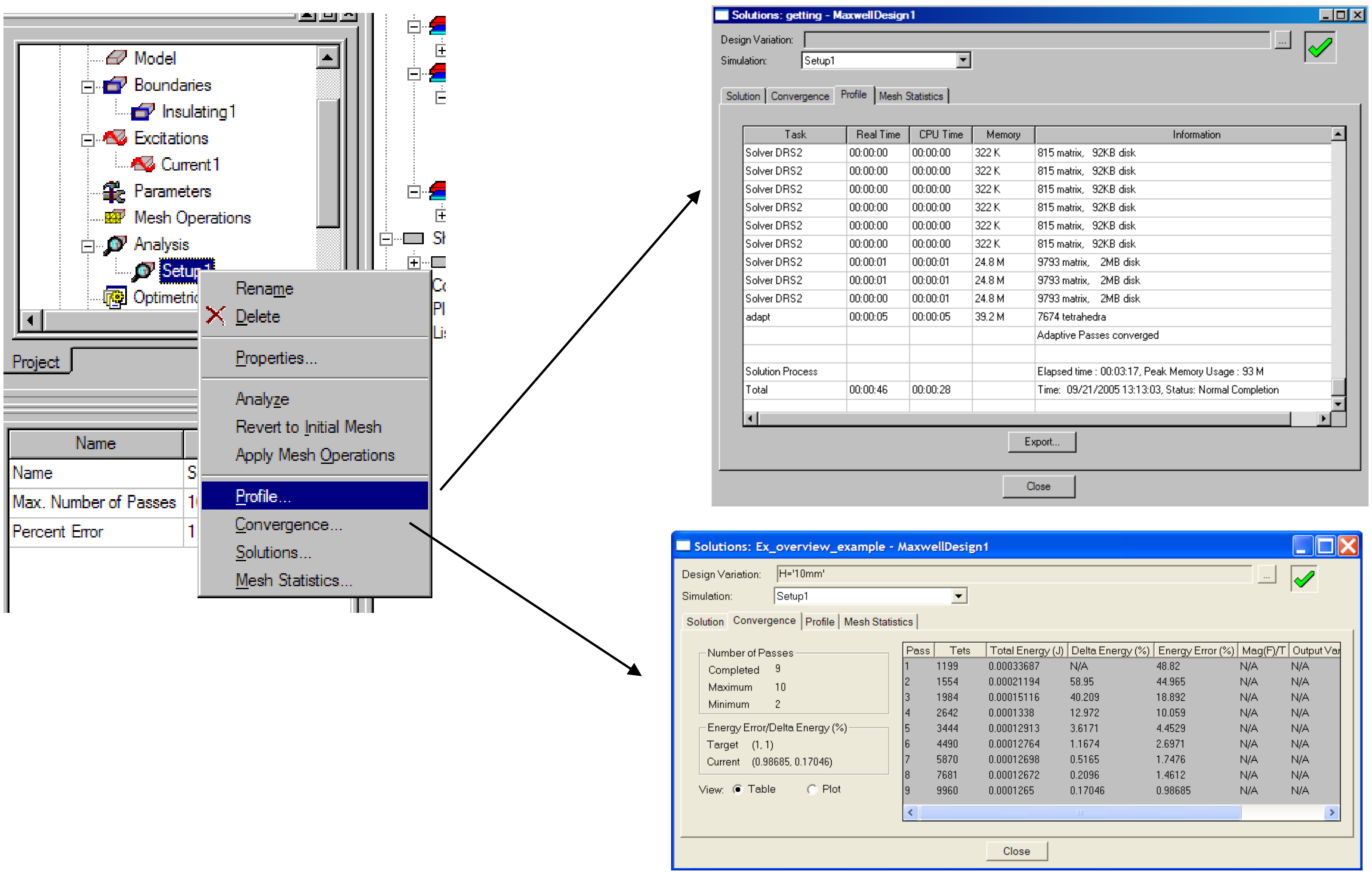
- ## Analyze
  - **To start the solution process:**
    1. Select the menu item ***Maxwell 3D> Analyze All***



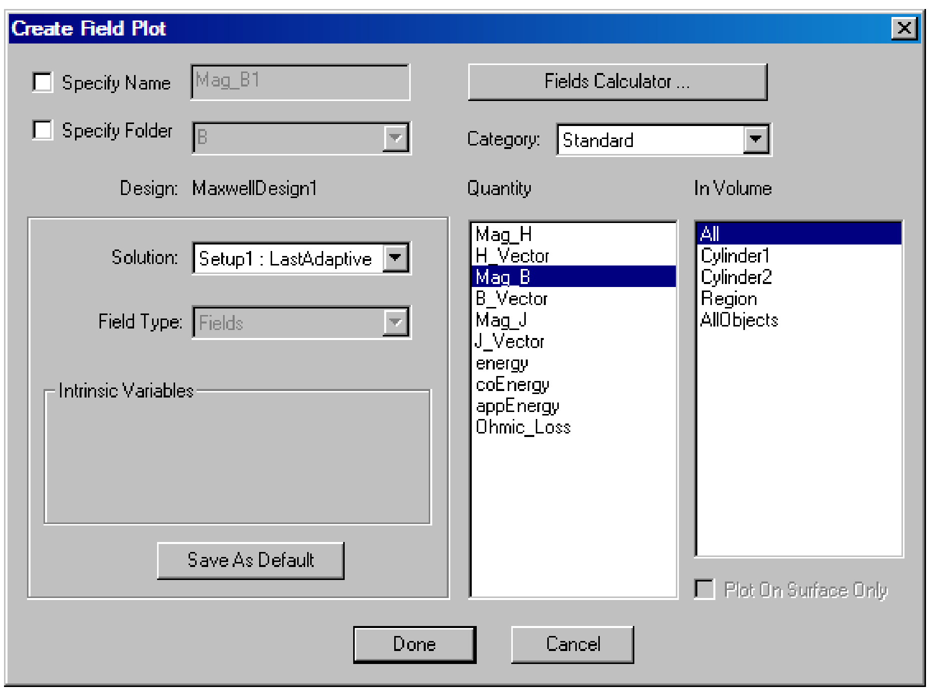


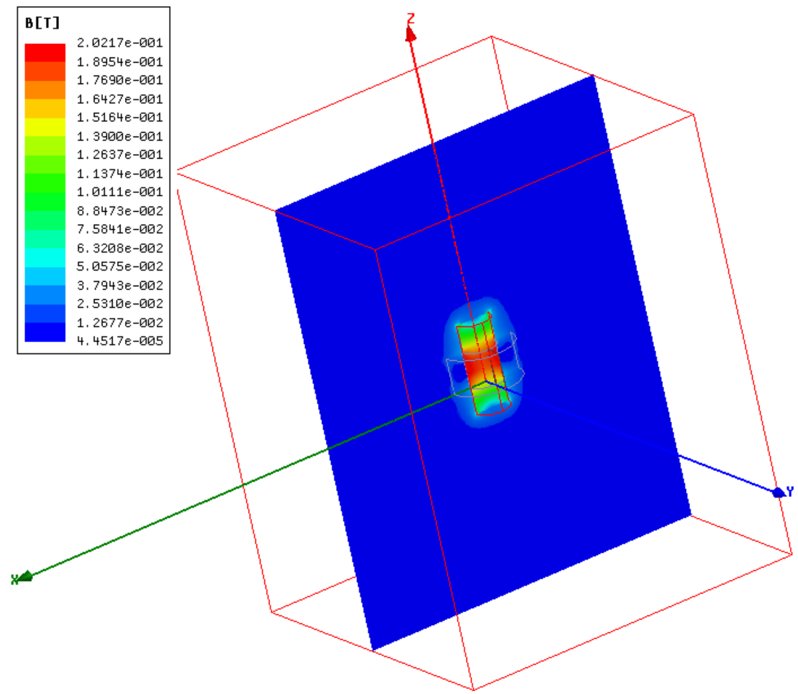
- **View detailed information about the progress**

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

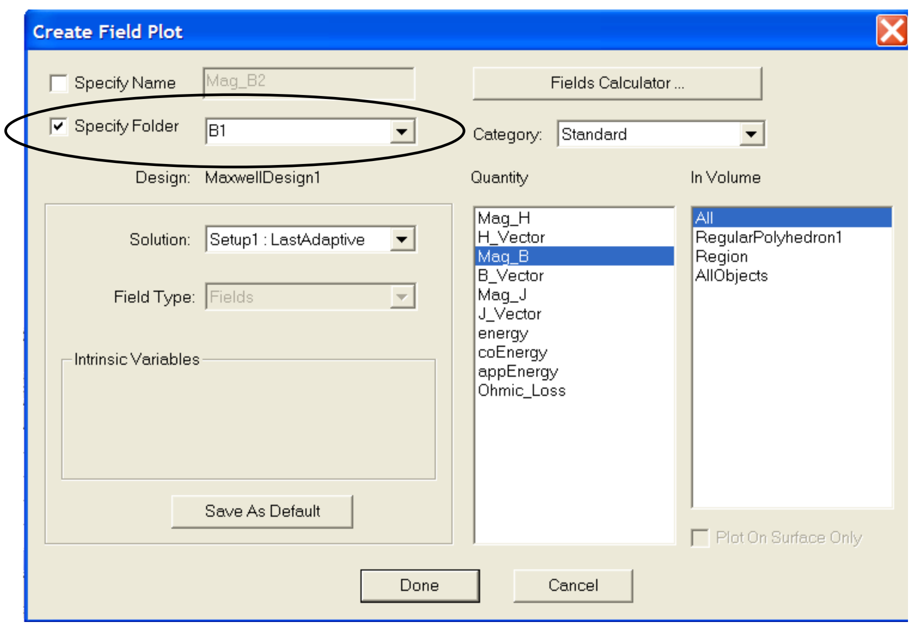


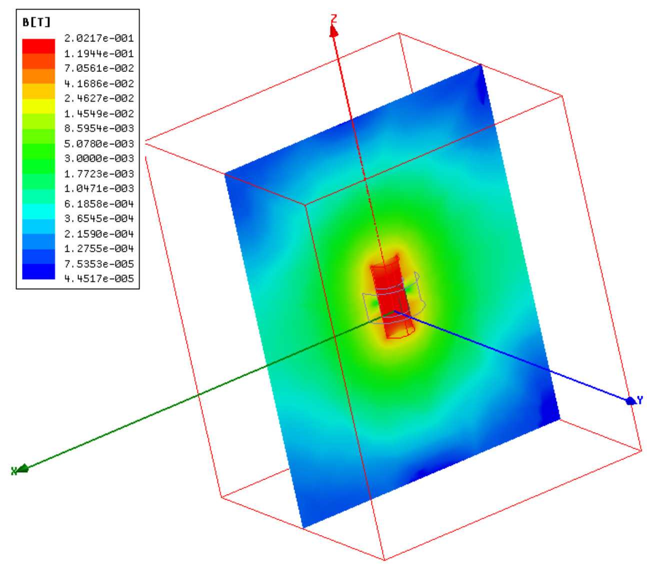
- **Field Overlays**
  - **To create a field plot:**
    1. Select the Global XZ Plane
       1. Using the Model Tree, expand **Planes**
       2. Select **Global:XZ**
    2. Select the menu item ***Maxwell > Fields > Fields > B > Mag_B***
    3. Create Field Plot Window
       1. Solution: **Setup1 : LastAdaptive**
       2. Quantity: **Mag_B**
       3. In Volume: **All**
       4. Click the **Done** button



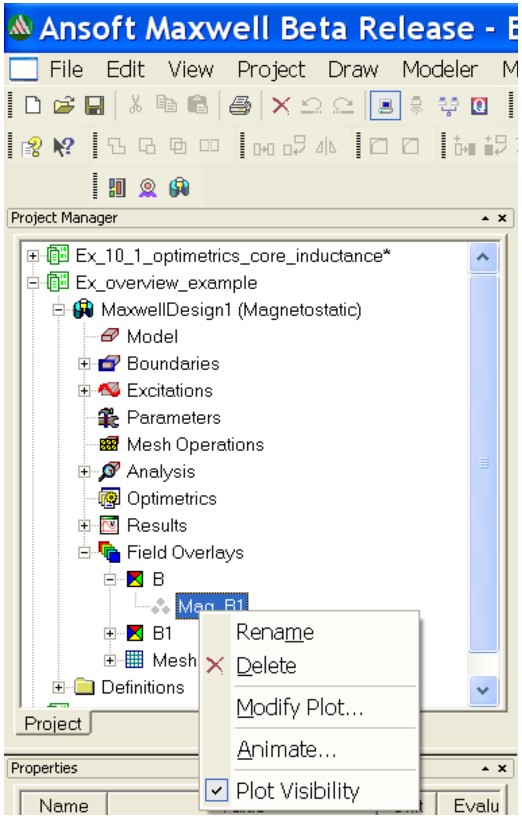


- **Field Overlays**
  - **To create a second field plot of same quantity, but different scale:**
    1. Select the Global XZ Plane
        1. Using the Model Tree, expand **Planes**
        2. Select **Global:XZ**
    2. Select the menu item ***Maxwell 3D> Fields > Fields > B > Mag_B***
    3. Create Field Plot Window
        1. Specify Folder: **B1**
        2. Solution: **Setup1 : LastAdaptive**
        3. Quantity: **Mag_B**
        4. In Volume: **All**
        5. Click the **Done** button

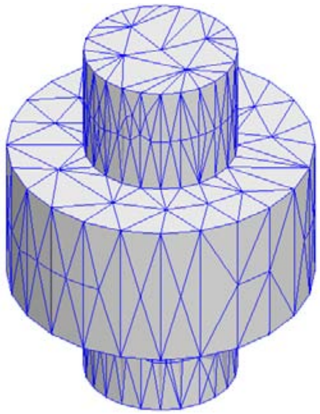




- **To modify a Magnitude field plot:**
  1. Select the menu item ***Maxwell 3D> Fields > Modify Plot Attributes***
  2. Select Plot Folder Window:
     1. Select: **B**
     2. Click the **OK** button
  3. B-Field Window:
     1. Click the **Scale** tab
        1. Scale: **Log**
     2. Click the **Close** button
  4. Hide the plots by clicking in the project tree and unchecking Plot Visibility



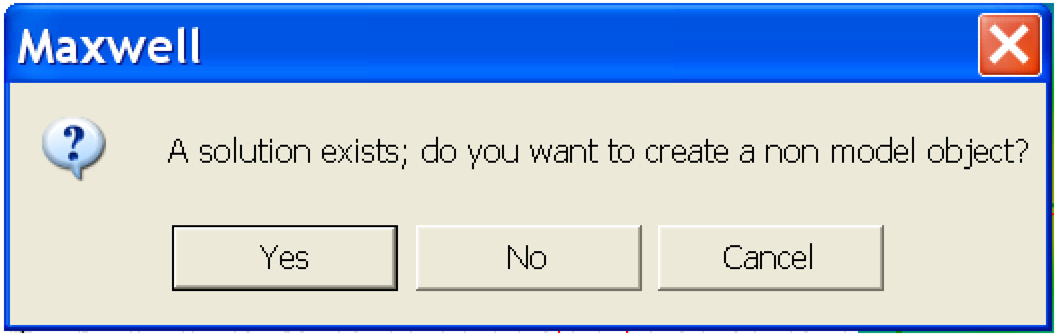
- **Mesh Overlay**
  - **To select the objects Core and Coil:**
    1. Select the menu item ***Edit > Select > By Name***
    2. Press and hold the CTRL key and select Core and Coil from the list
    3. Click the **OK** button
  - **To create a mesh plot:**
    1. Select the menu item ***Maxwell 3D> Fields > Plot Mesh***
    2. Create Mesh Window:
       1. Click the **Done** button



- ## Plot the z-component of B Field along the line

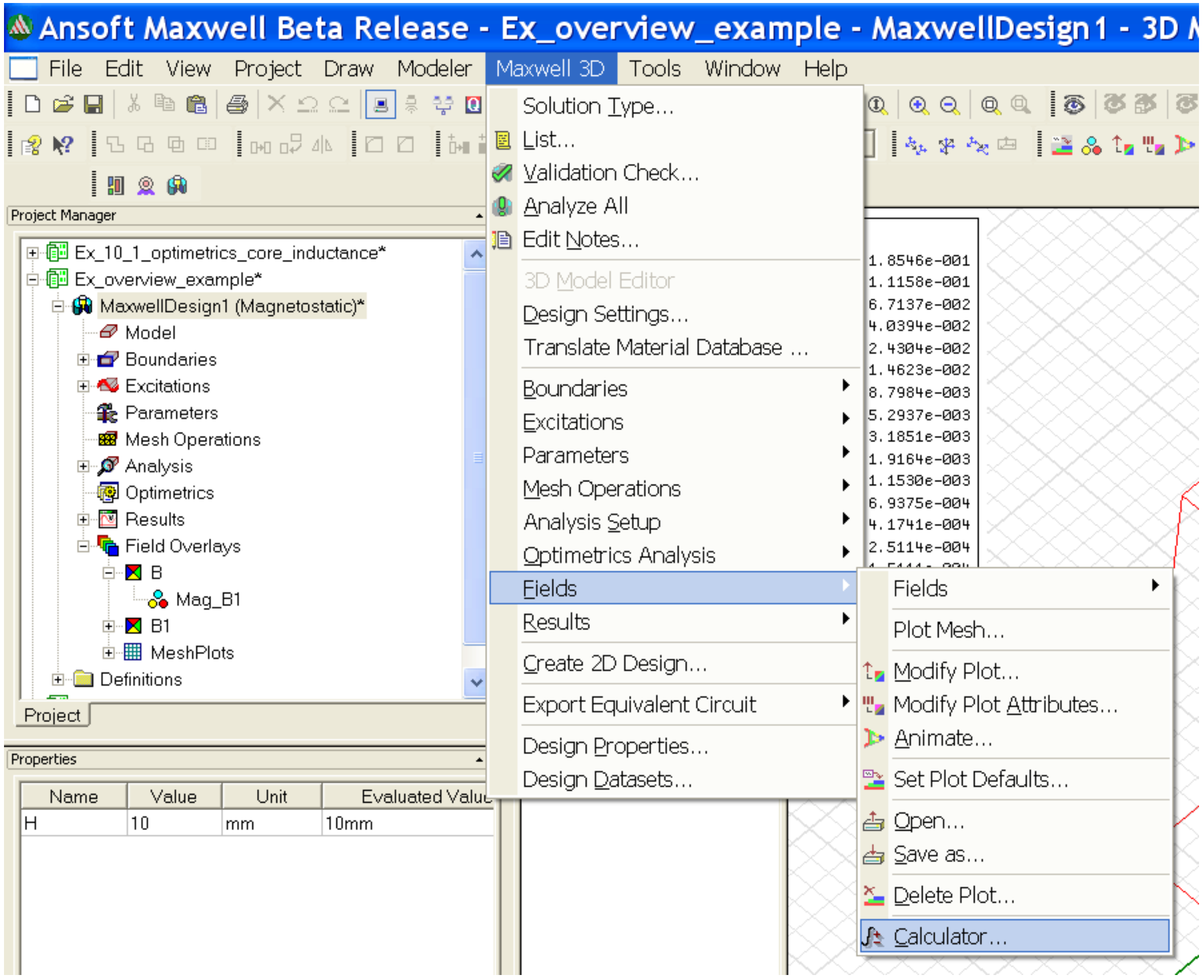
1. Create a line
2. Calculate the z-component of B using the **Calculator tool**
3. Create report – plot the desired graph

- **To create a line**
  1. Select the menu item ***Draw > Line.*** The following window opens:

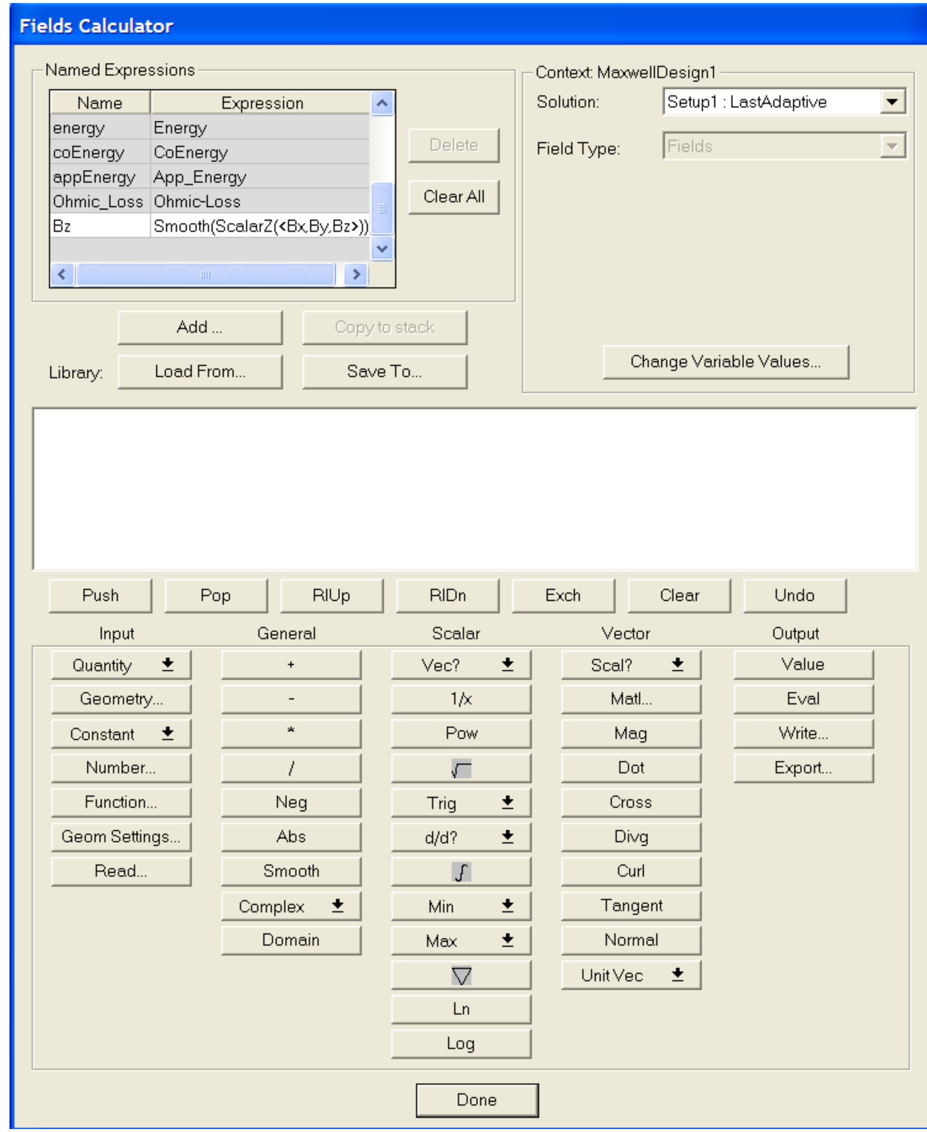


  2. Answer **YES** to create a non model object and not to destroy the solution.
  3. Place the cursor at the top face of the core and let the cursor snap to the center.
  4. Click the left mouse button and hold the z button on the keyboard while moving the cursor up. Left click at some distance from the core and double click left to end the line.
  5. Leave the values in the upcoming dialog box and close the dialog by pressing OK.

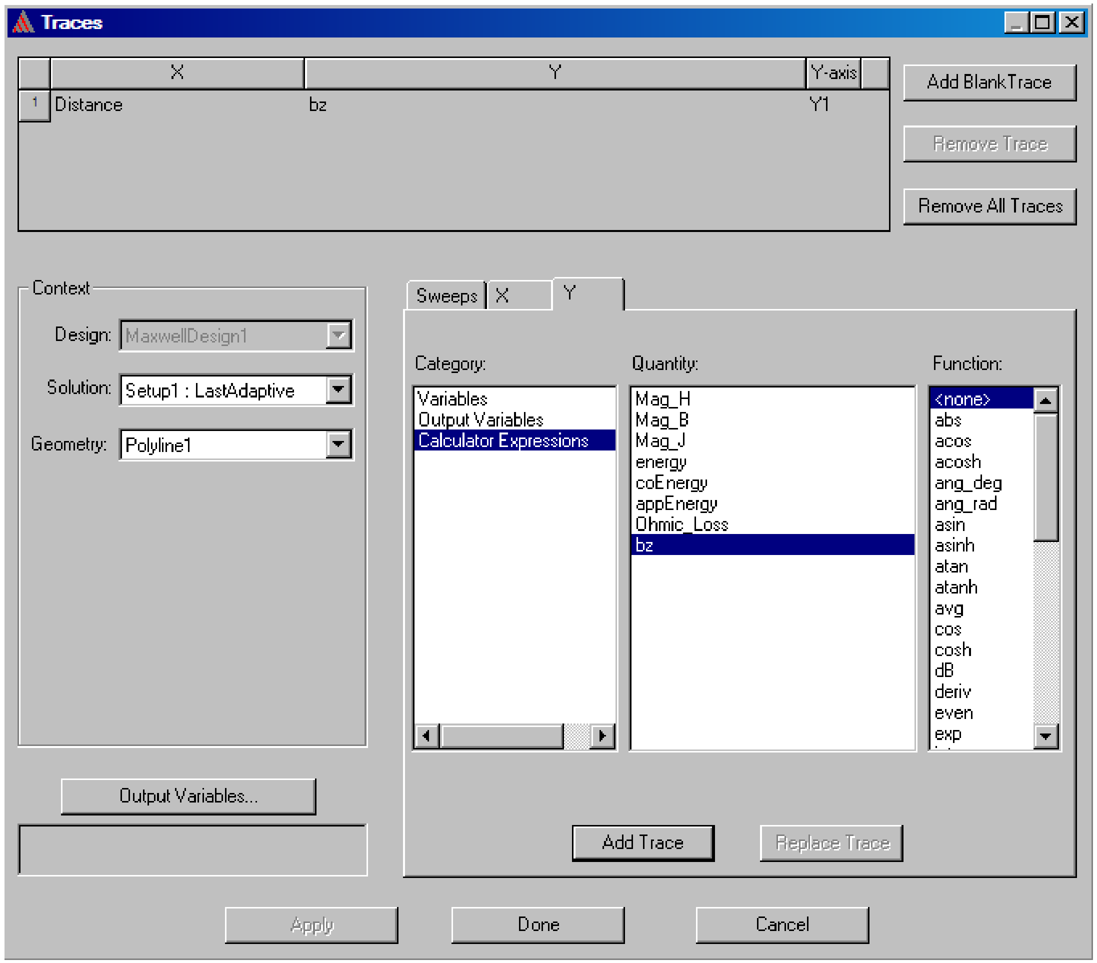
- **To bring up the Calculator tool**
  1. Select the menu item ***Maxwell 3D> Fields- > Calculator***

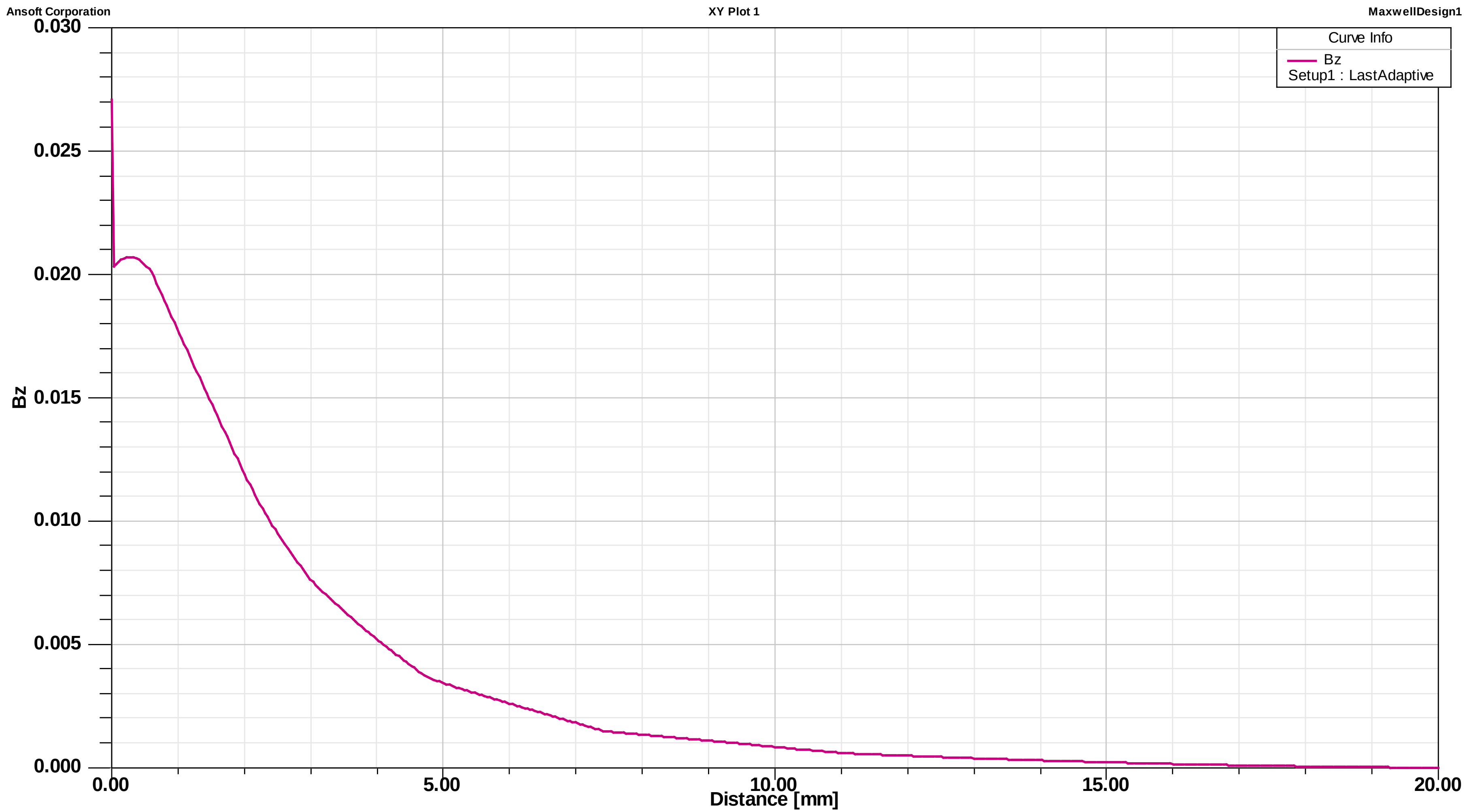


- **To calculate the z-component of B field**
  1. In the Input Column select the **Quantity B**
  2. In the Vector Column select the **ScalarZ** component
  3. In the General Column select **Smooth**
  4. Press the **Add** button in the upper part of the calculator and enter a name that describes the calculation
     - enter **Bz** for B field in z-direction
  5. Click **Done** to close the calculator

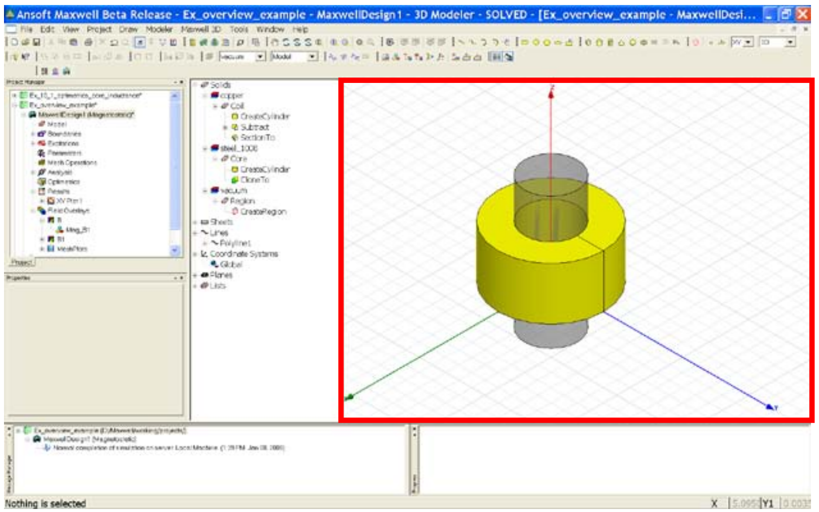


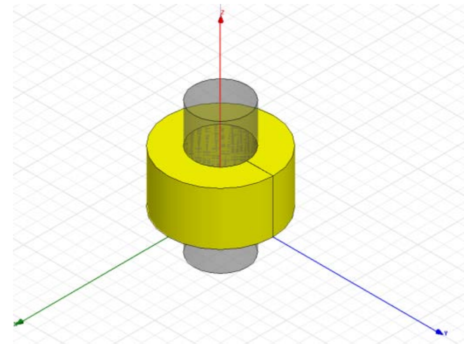
- **To create report (plot the desired graph)**
  1. Select the menu item ***Maxwell 3D> Result > Create Report***
  2. Enter Fields and Rectangular Plot and click the **OK** button
  3. Select the following: Geometry – Polyline1; Category – Calculator Expressions; Quantity – Bz
  4. Press the **Add Trace** to copy your selection to the upper part of the window and click on the **Done** button

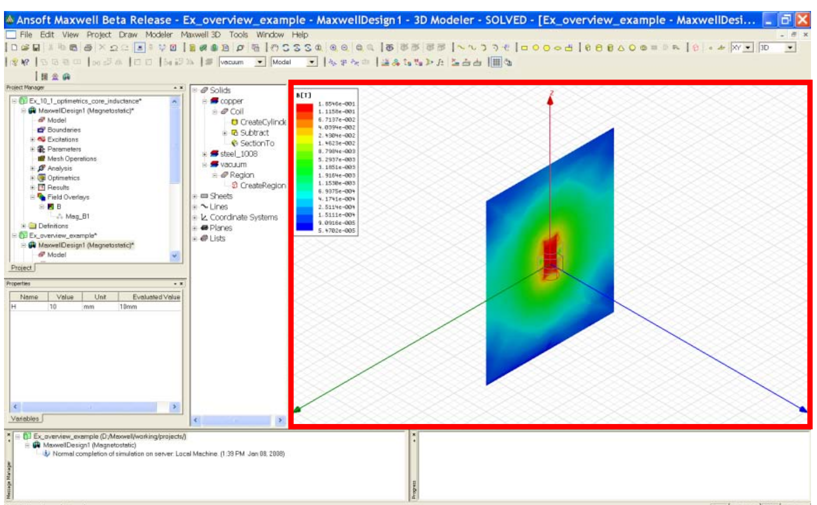


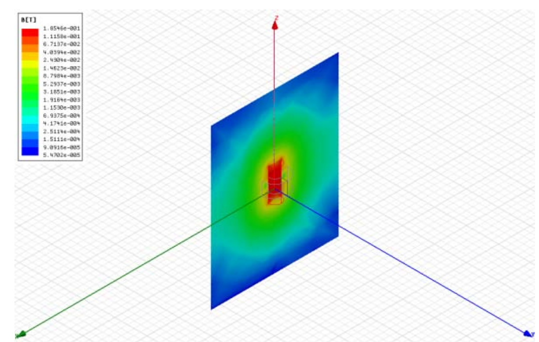

Ansoft Corporation
XY Plot 1
MaxwellDesign1
Curve Info
Bz
Setup1 : LastAdaptive
Bz
0.030
0.025
0.020
0.015
0.010
0.005
0.000
0.00
5.00
10.00
15.00
20.00
Distance [mm]

- **To save drawing Window or a plot to clipboard**
  1. Select the menu item ***Edit > Copy Image***

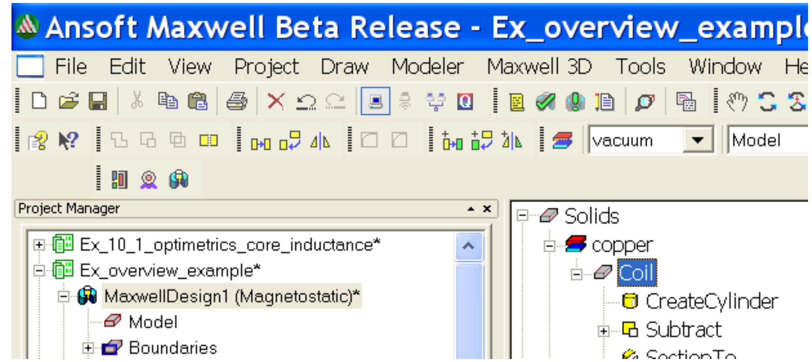


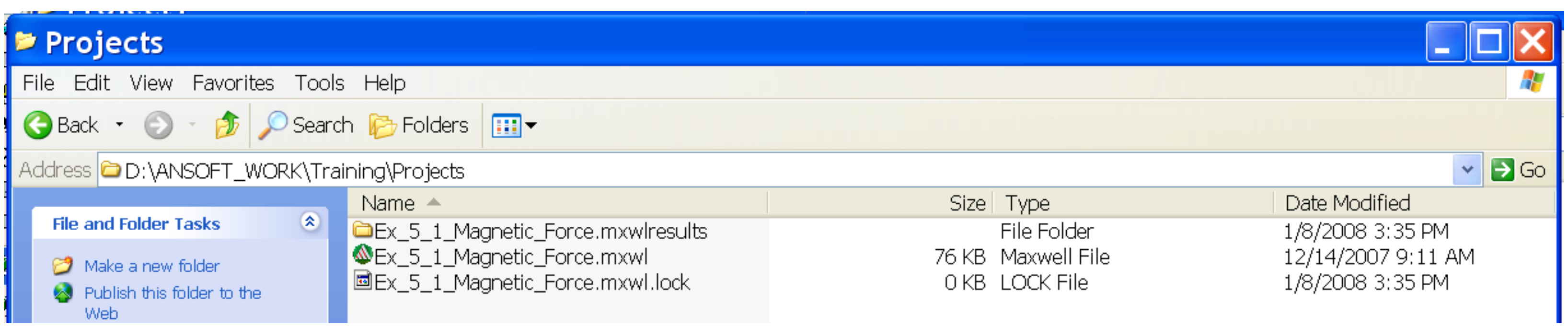




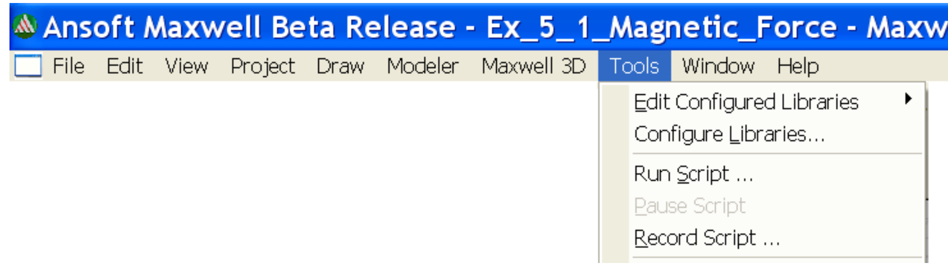


- File Structure
  - Everything regarding the project is stored in an ascii file
    - File: **<project_name>.mxwl**
    - Double click from Windows Explorer will open and launch Maxwell 3D
  - Results and Mesh are stored in a folder named **<project_name>.mxwlresults**
  - Lock file: **<project_name>.lock.mxwl**
    - 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)

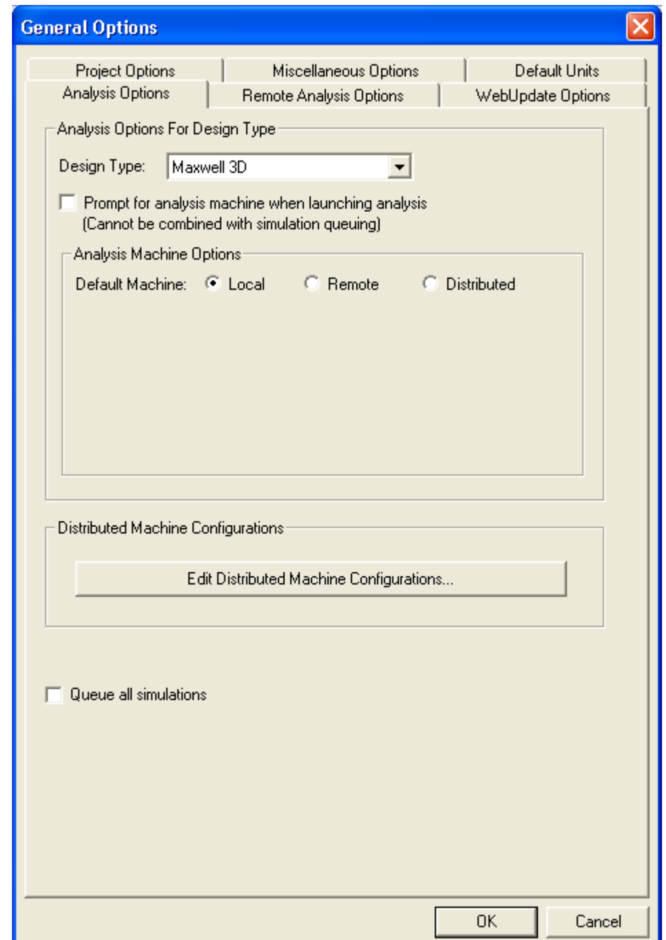




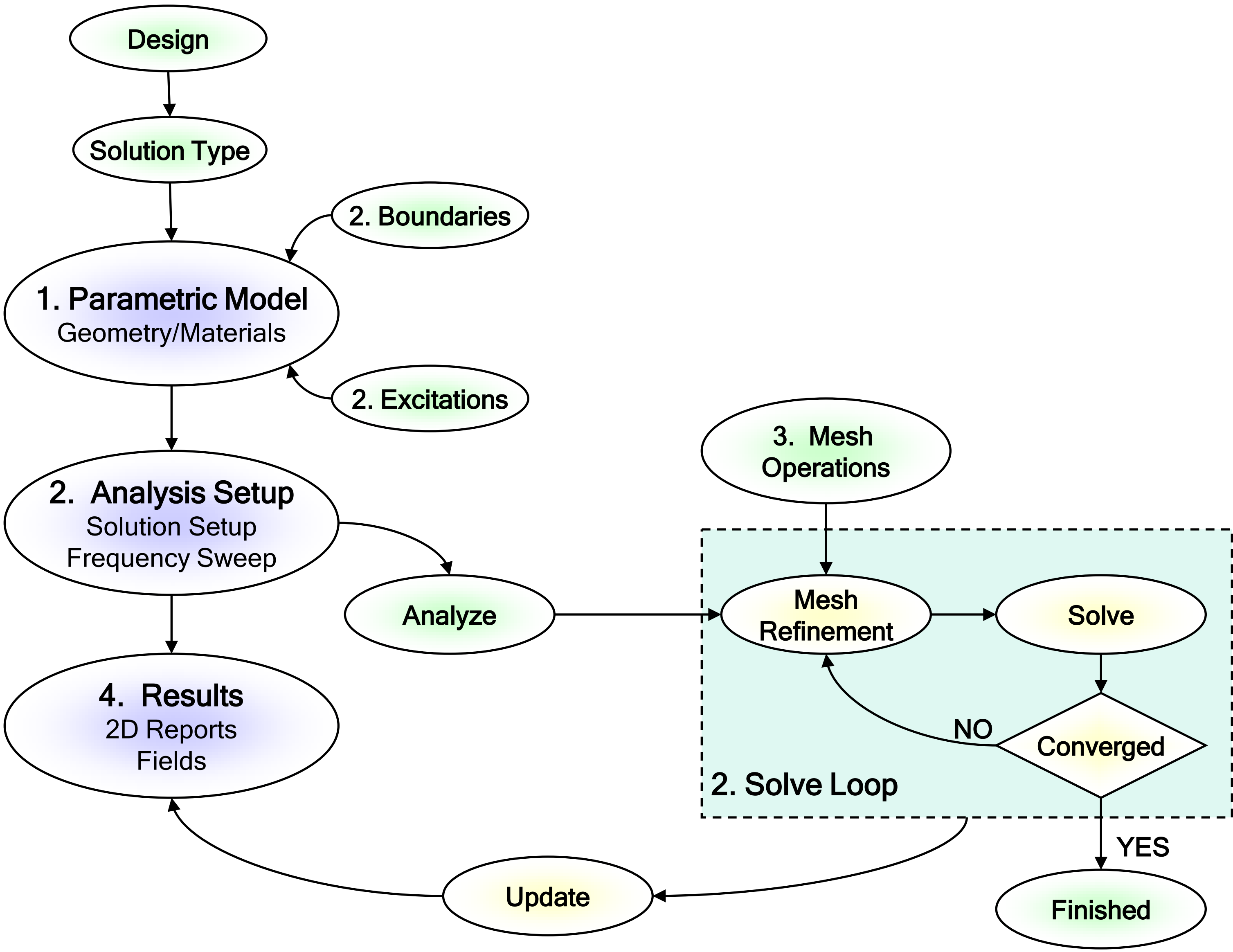
- **Scripts**
  - Default Script recorded in Maxwell
    - Visual Basic Script



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

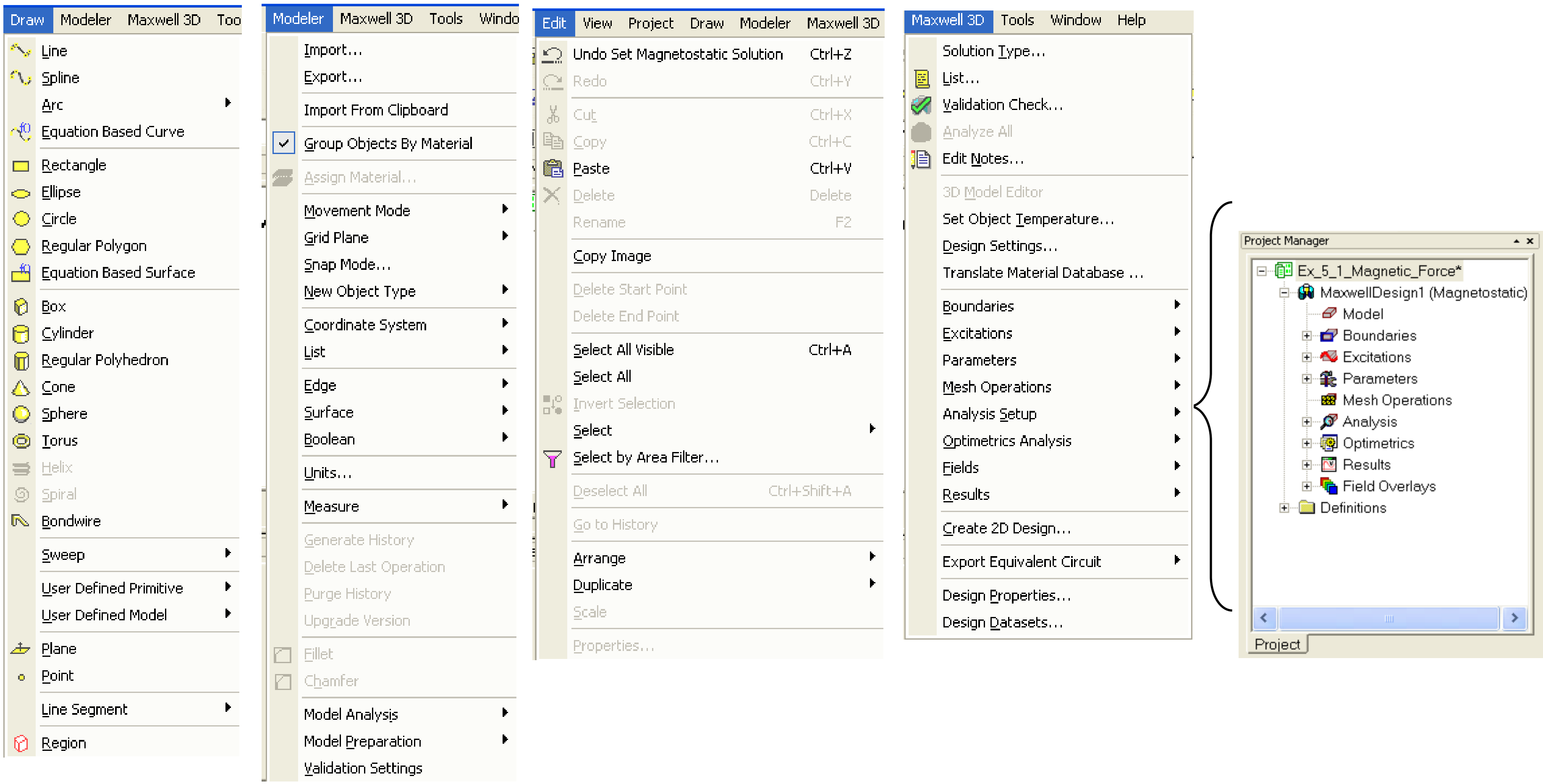


- The Process

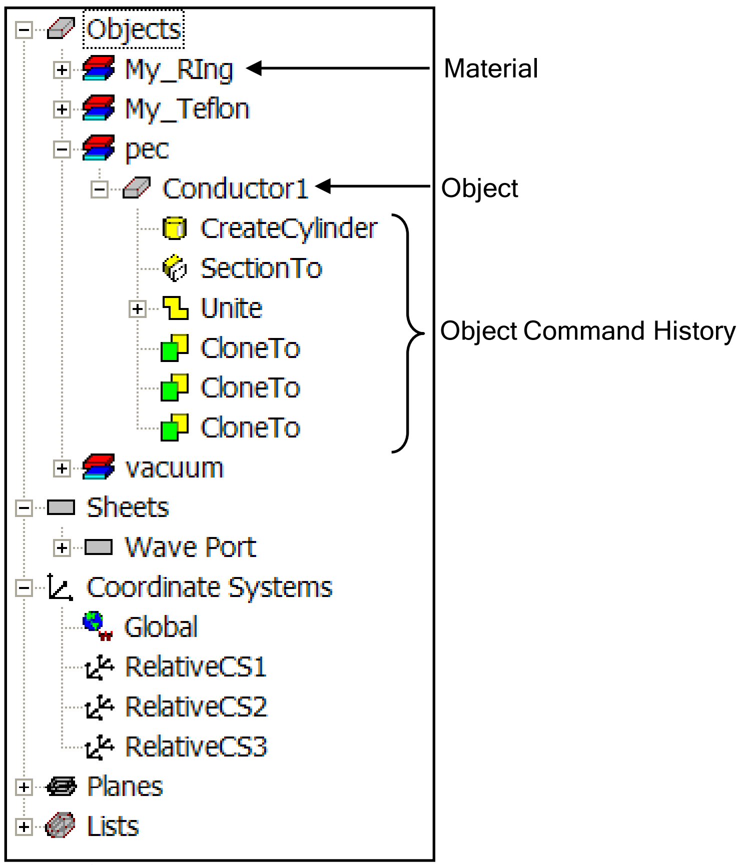

Design
Solution Type
2. Boundaries
1. Parametric Model
Geometry/Materials
2. Excitations
3. Mesh Operations
2. Analysis Setup
Solution Setup
Frequency Sweep
Analyze
Mesh Refinement
Solve
Converged
NO
YES
2. Solve Loop
4. Results
2D Reports
Fields
Update
Finished

- ## Menu Structure
  - **Draw** - Primitives
  - **Modeler** - Settings and Boolean Operations
  - **Edit** - Copy/Paste, Arrange, Duplicate
  - **Maxwell 3D**- Boundaries, Excitations, Mesh Operations, Analysis Setup, Results

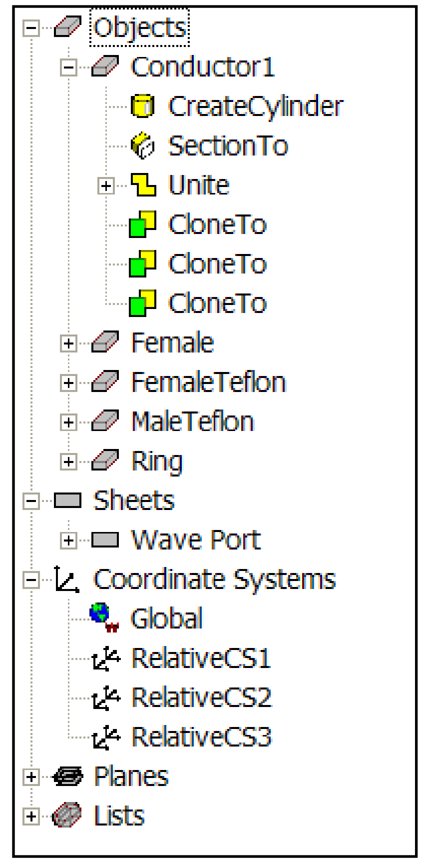
Note:
- Edit Copy/Paste is a dumb copy for only dimensions and materials, but not the history
- Edit > Duplicate clones objects include dimensions, materials and the creation history with all variables



- ## Modeler - Model Tree
  - Select menu item ***Modeler > Group by Material***

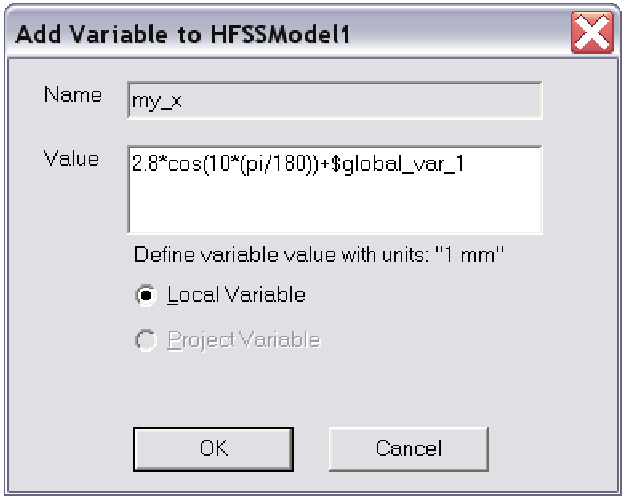


Grouped by Material



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



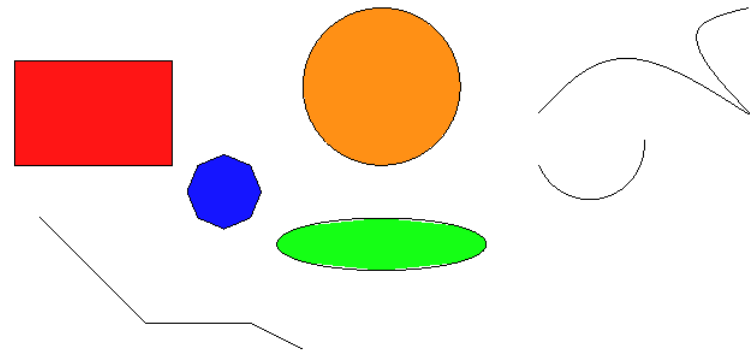
Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	-1 , -1.6 , 0	mm
XSize	2.6	mm
YSize	2.8	mm
ZSize	1	mm

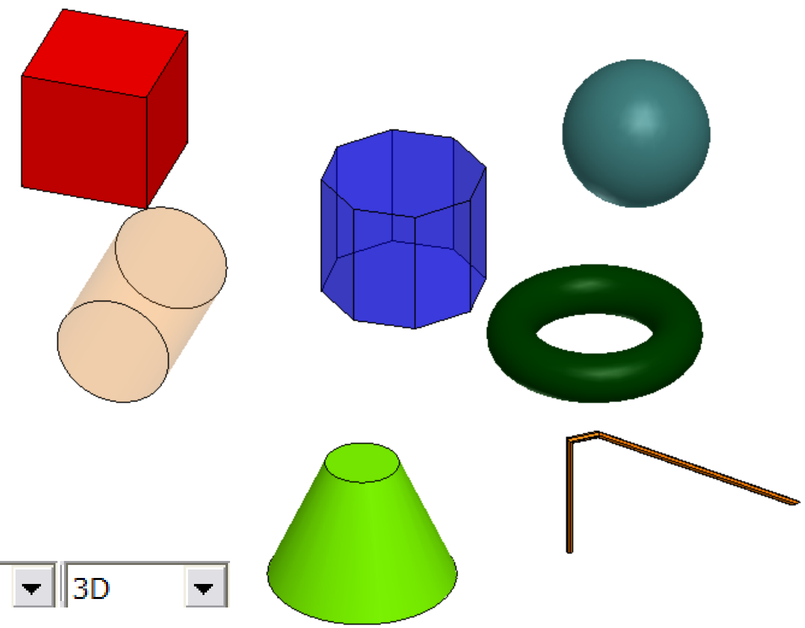
Command

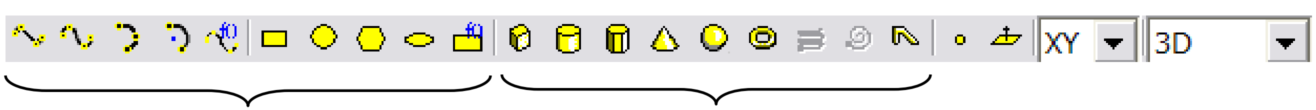
Name	Value	Unit
Comma...	CreateBox	
Coordin...	Global	
Position	-1 , -1.6 , 0	mm
XSize	my_x	
YSize	2.8	mm
ZSize	1	mm

Command

- **Modeler - Primitives**
  - **2D Draw Objects**
    - The following 2D Draw objects are available:
      - Line, Spline, Arc, Equation Based Curve, Rectangle, Ellipse, Circle, Regular Polygon,
        Equation Based Surface
  - **3D Draw Objects**
    - The following 3D Draw objects are available:
      - Box, Cylinder, Regular Polyhedron
        Cone, Sphere, Torus, Helix, Spiral, Bond Wire
  - **True Surfaces**
    - Circles, Cylinders, Spheres, etc are represented as true surfaces. In versions prior to release 11 these primitives would be represented as faceted objects. If you wish to use the faceted primitives, select the Regular Polyhedron or Regular Polygon.





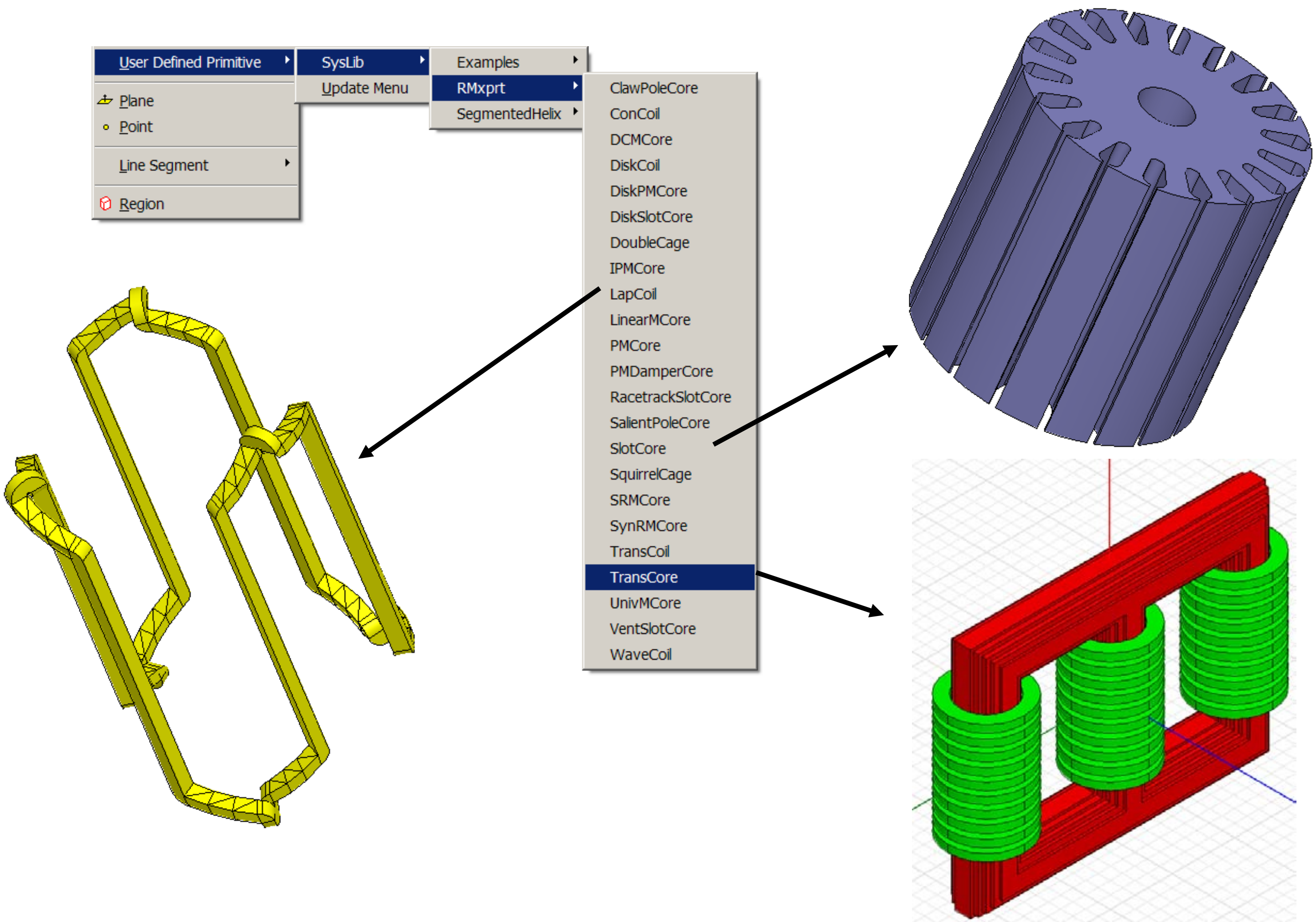


Toolbar: 2D Objects

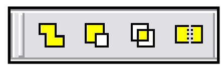
Toolbar: 3D Objects

User Defined Primitives

- Allow automated creation and parameterization of complicated geometrical structures
- ***Draw > User Defined Primatives > SysLib***

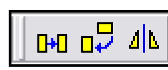


- Modeler - Boolean Operations/Transformations
  - *Modeler > Boolean >*
    - **Unite** - combine multiple primitives
      - Unite disjoint objects (**Separate Bodies** to separate)
    - **Subtract** - remove part of a primitive from another
    - **Intersect**- keep only the parts of primitives that 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



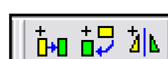
Toolbar: Boolean

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



Toolbar: Arrange

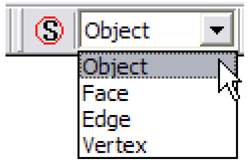
  - *Edit > Duplicate >*
    - **Along Line -** Create multiple copies of an object along a vector
    - **Around Axis -** Create multiple copies of an object rotated by a fixed angle around the x, y, or z axis
    - **Mirror -** Mirrors the shape around a specified plane and creates a duplicate

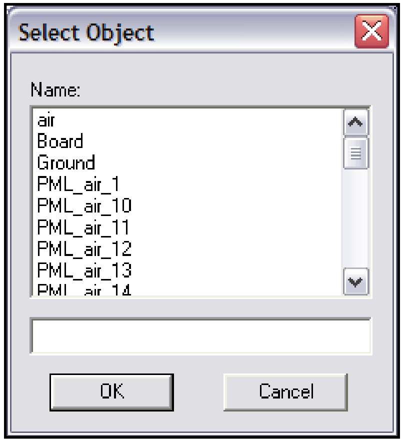


Toolbar: Duplicate

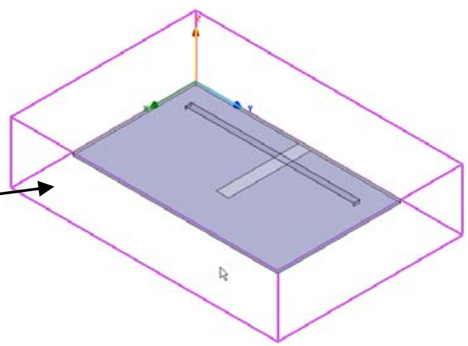
  - *Edit > Scale* - Allows non-uniform scaling in the x, y, or z direction

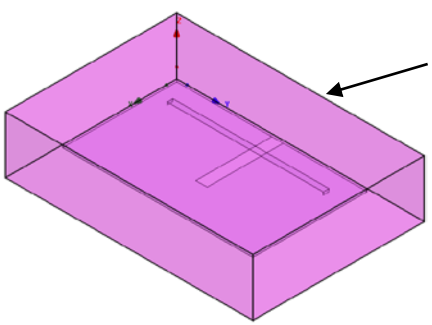
- **Modeler - Selection**
  - **Selection Types**
    - **Object (Default)**
    - **Face**
    - **Edge**
    - **Vertex**
  - **Selection Modes**
    - **All Objects**
    - **All Visible Object**
    - **By Name**
  - **Highlight Selection Dynamically** - By default, moving the mouse pointer over 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**





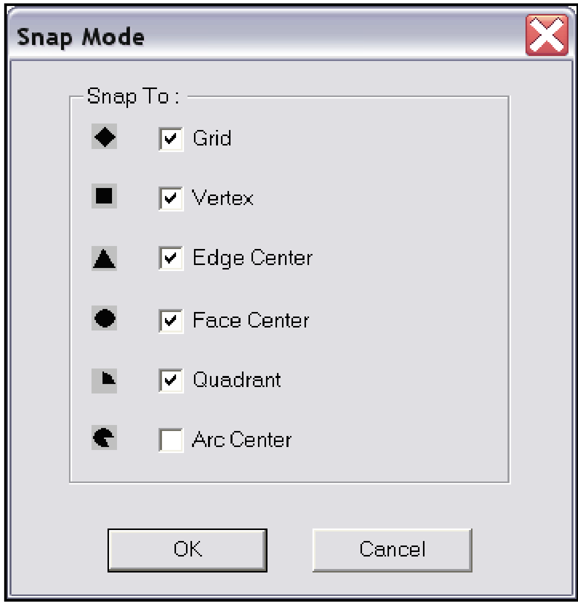
Dynamically Highlighted
(Only frame of object)

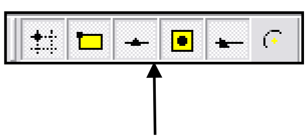




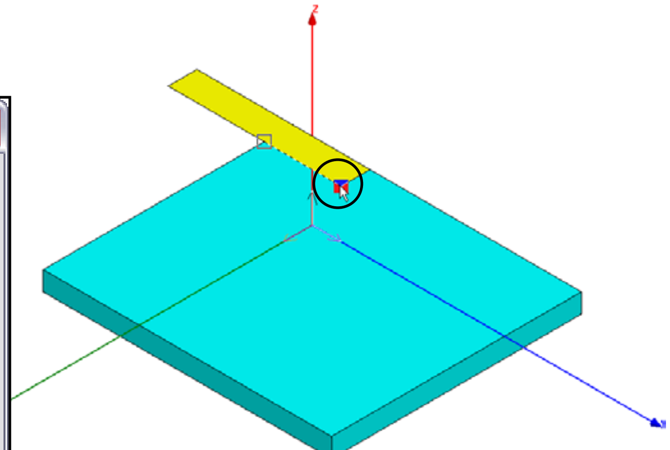
Selected

- Modeler - Moving Around

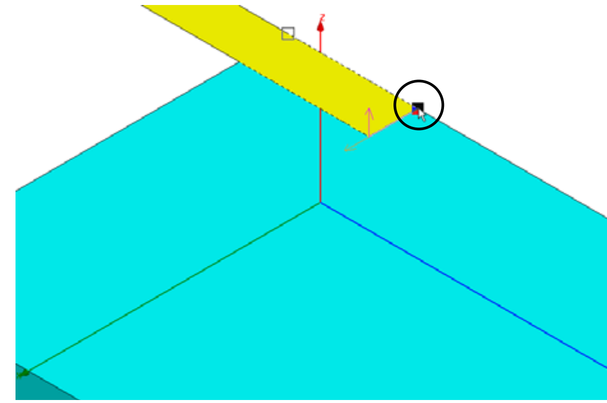




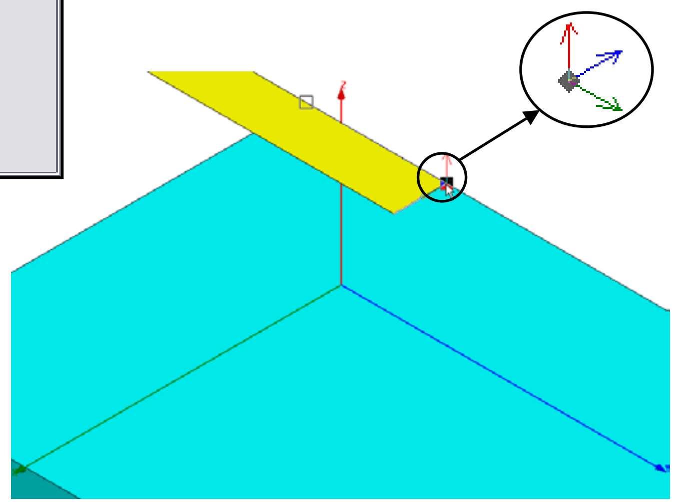
Toolbar: Snap Mode



Step 1: Start Point

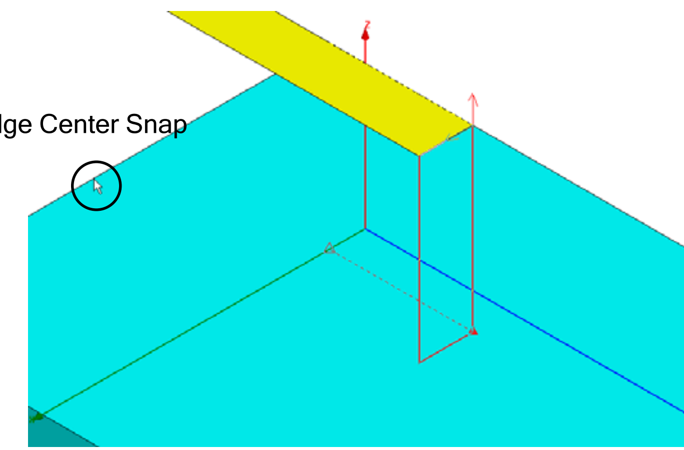


Step 2: Hold X key (to only allow motion in X-direction) and select vertex point



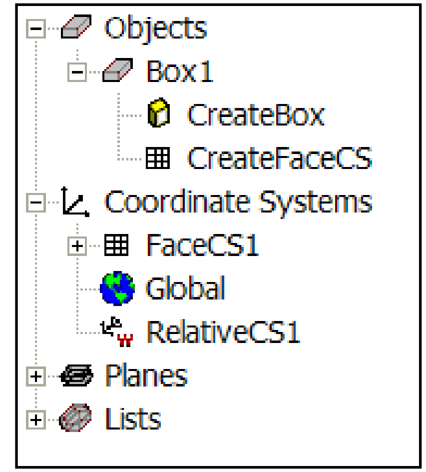
Step 3: CTRL+Enter Keys (sets a temporary local reference)

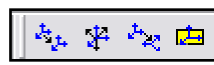
Edge Center Snap



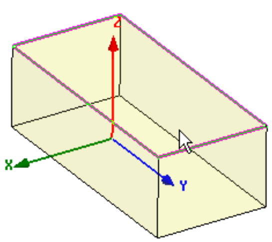
Step 4: Hold Z key and set height

- **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)

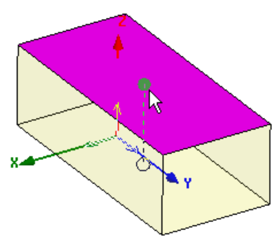




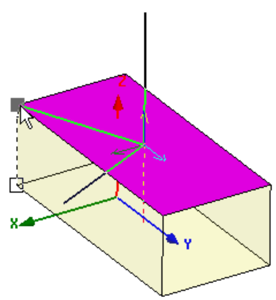
Toolbar: C oordinate System



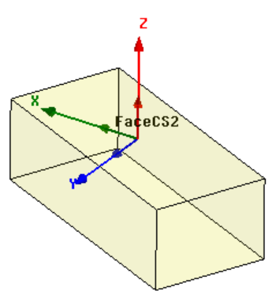
Step 1: Select Face



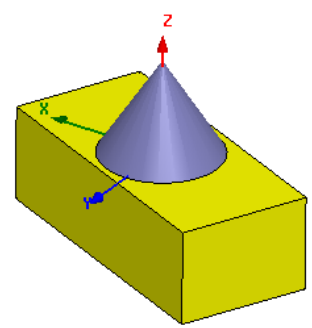
Step 2: Select Origin



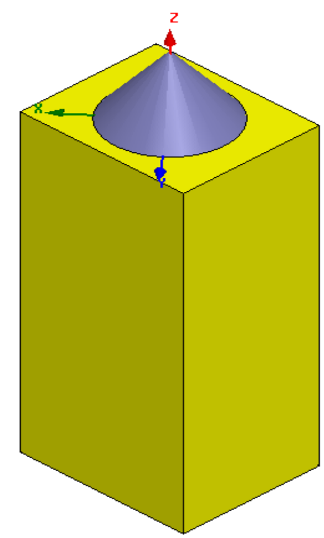
Step 3: Set X-Axis



New Working CS

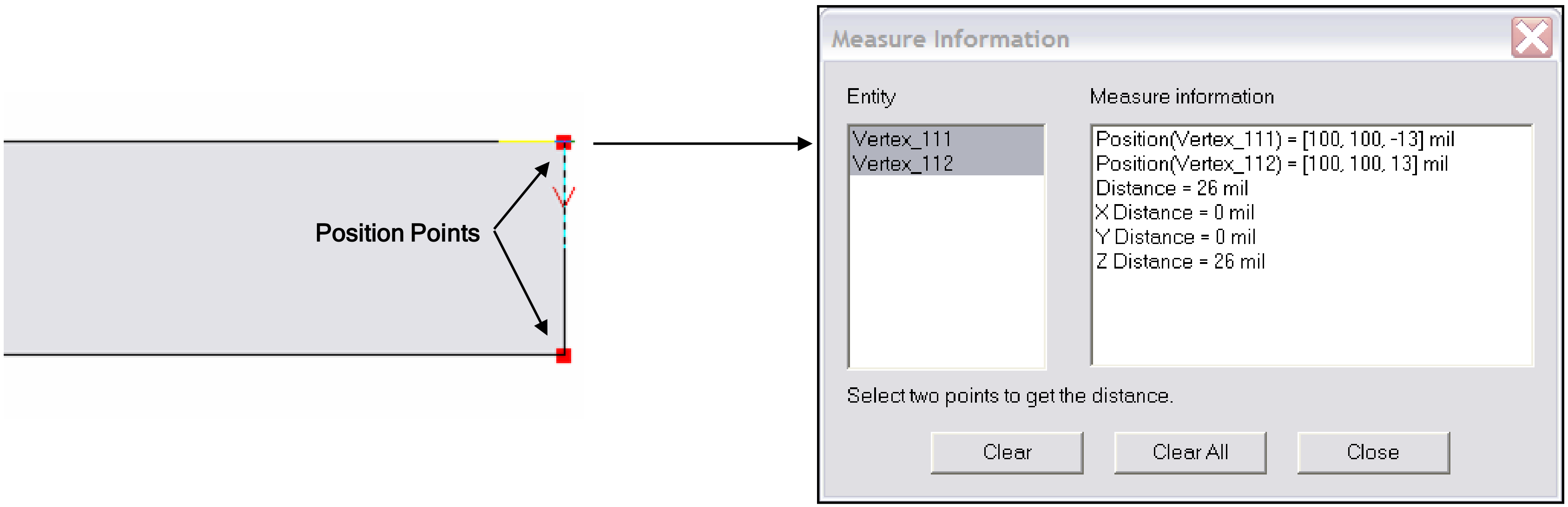


Cone created with Face CS

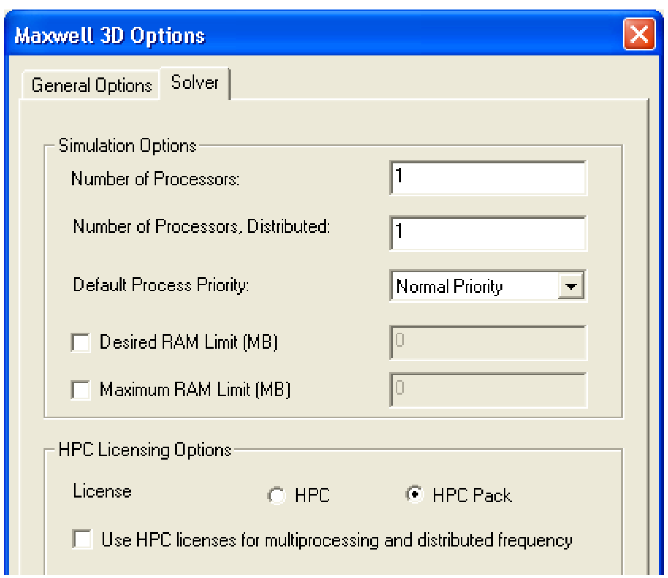


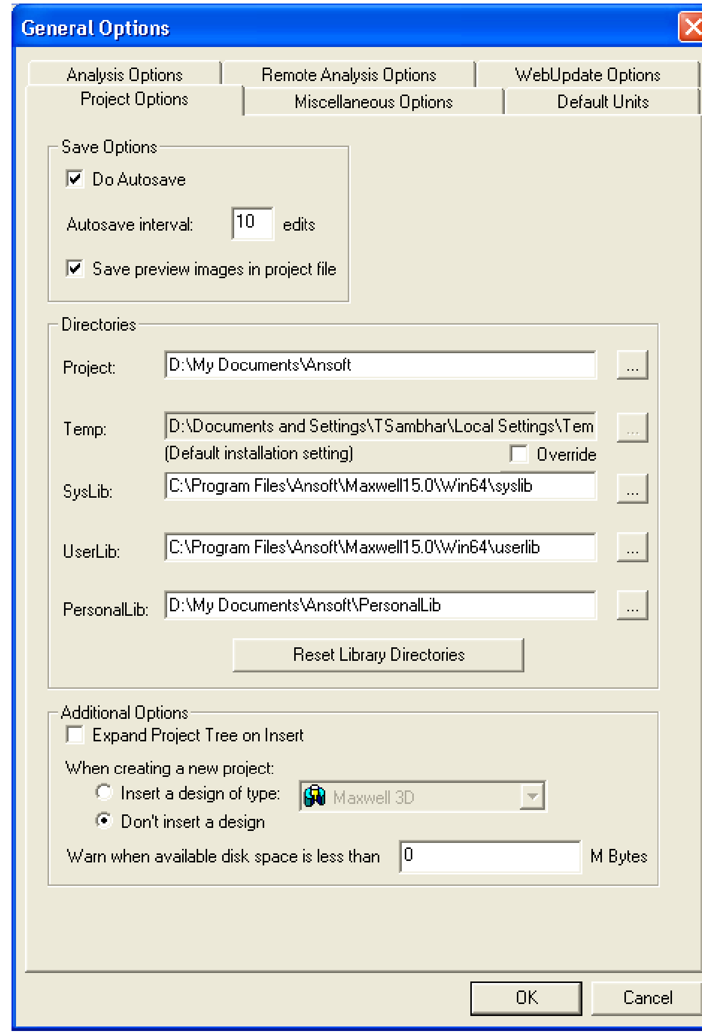
Change Box Size and Cone is automatically positioned with the top face of the box

- ## Measure
  - ***Modeler > Measure >***
    - ***Position*** - Points and Distance
    - ***Length*** - Edge Length
    - ***Area*** - Surface Area
    - ***Volume*** - Object Volume



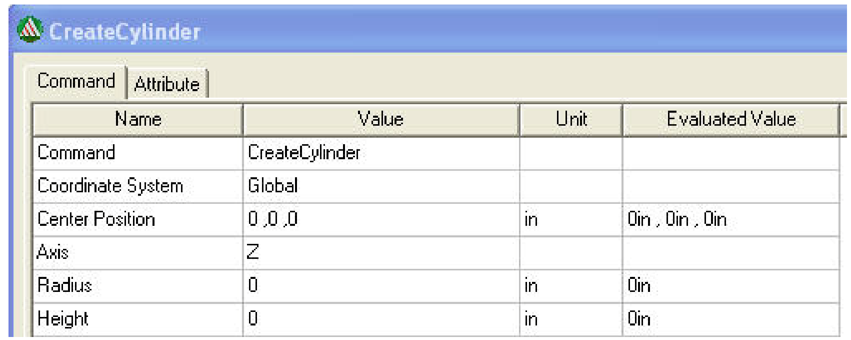
- **Options - General**
  - ***Tools > Options > General Options > Project Options***
    - **Temp Directory** – Location used during solution process
      - Make sure it is at least 512MB free disk.
- **Options - Maxwell**
  - ***Tools > Options > Maxwell 3D Options > Solver***
    - Set Number of Processors = 2 for 1 dual-core processor or two single-core processors. Requires additional license
    - Default Process Priority – set the simulation priority from Critical
      (highest) to Idle (lowest)
    - Desired RAM Limit – determines when solver will use hard disk off core (leave it unchecked for auto-detect)
    - Maximum RAM Limit – determines when swapping will occur with hard drive (leave it unchecked for auto-detect)





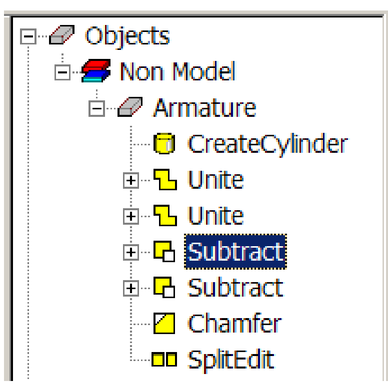
- Options - 3D Modeler Options
  - *Tools > Options > Modeler Options > Drawing* for Point and Dialog Entry Modes
  - Can enter in new dimensions using either Point (mouse) or Dialog entry mode

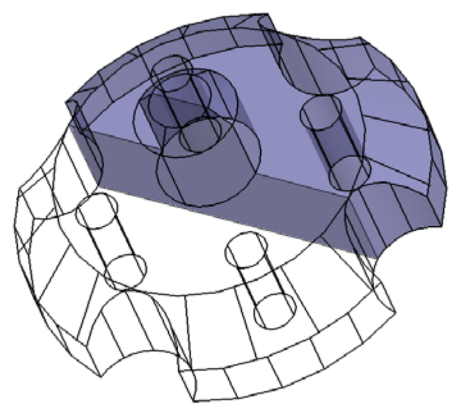
Typical "Dialog" entry mode window



Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate System	Global		
Center Position	0 ,0 ,0	in	0in , 0in , 0in
Axis	Z		
Radius	0	in	0in
Height	0	in	0in

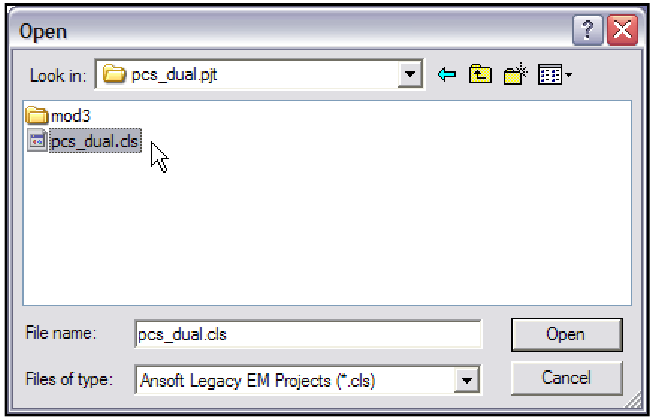
  - *Tools > Options > Modeler Options > Display* tab to Visualize history of objects
  - Visualization is seen by clicking on primatives in the history tree





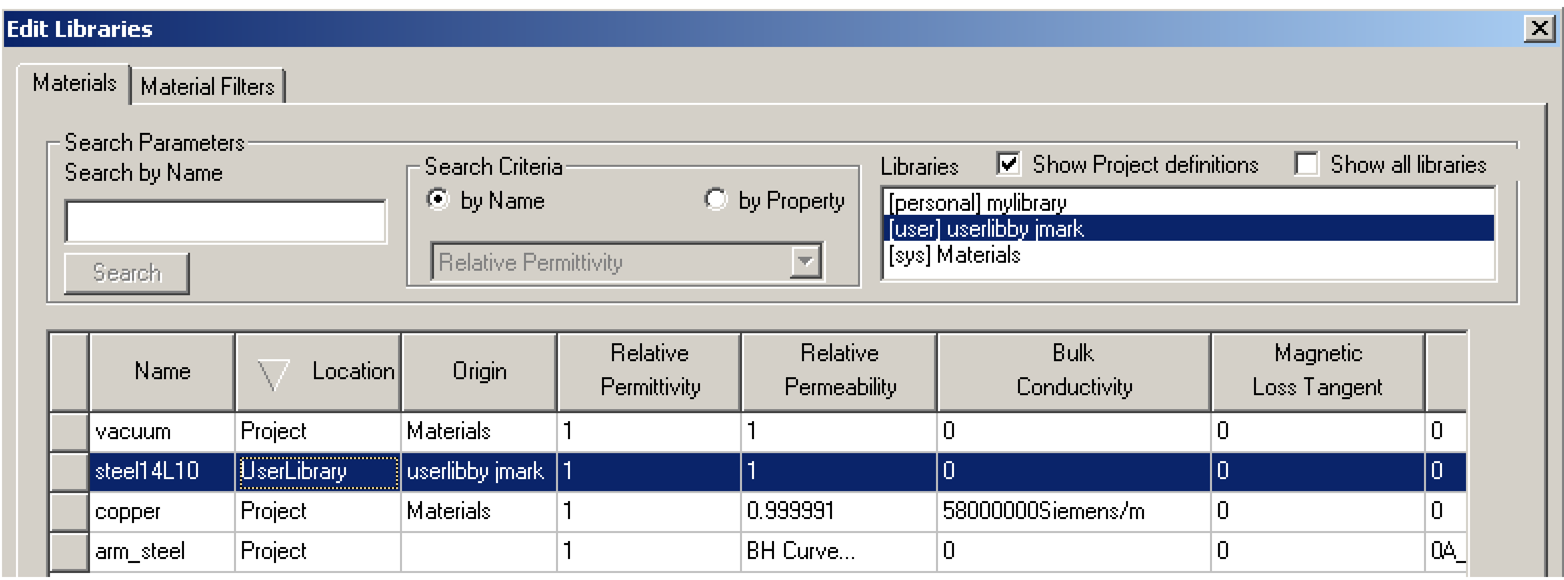
Converting Older Maxwell Projects (pre-Maxwell v11) to Maxwell v15

- **From Maxwell**
  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

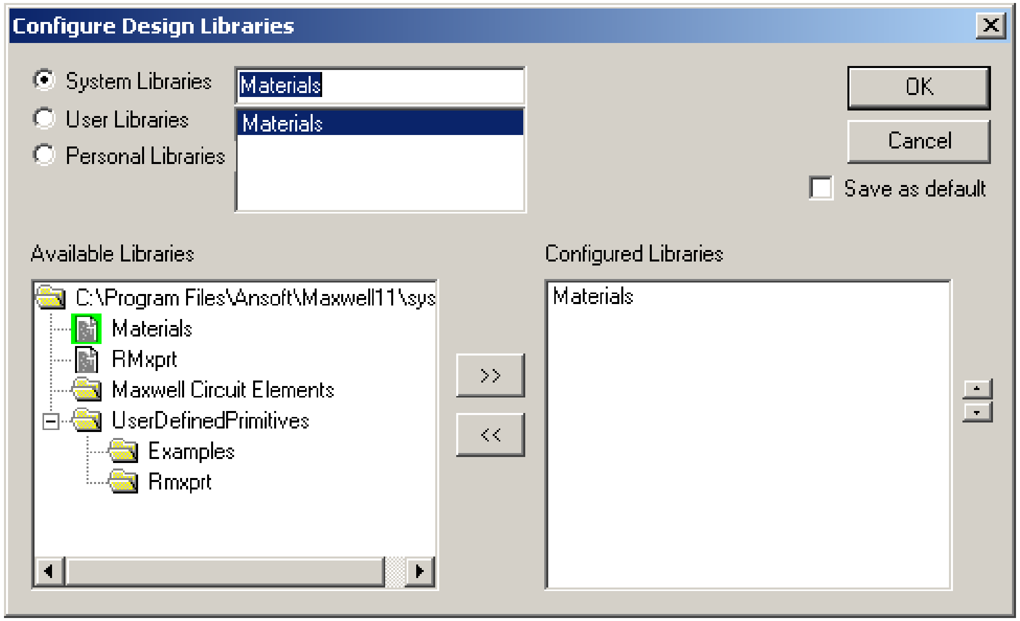


- **What is Converted?**
  - Converts Entire Model: Geometry, Materials, Boundaries, Sources and Setup
  - *Solutions, Optimetrics projects and Macros are not converted*

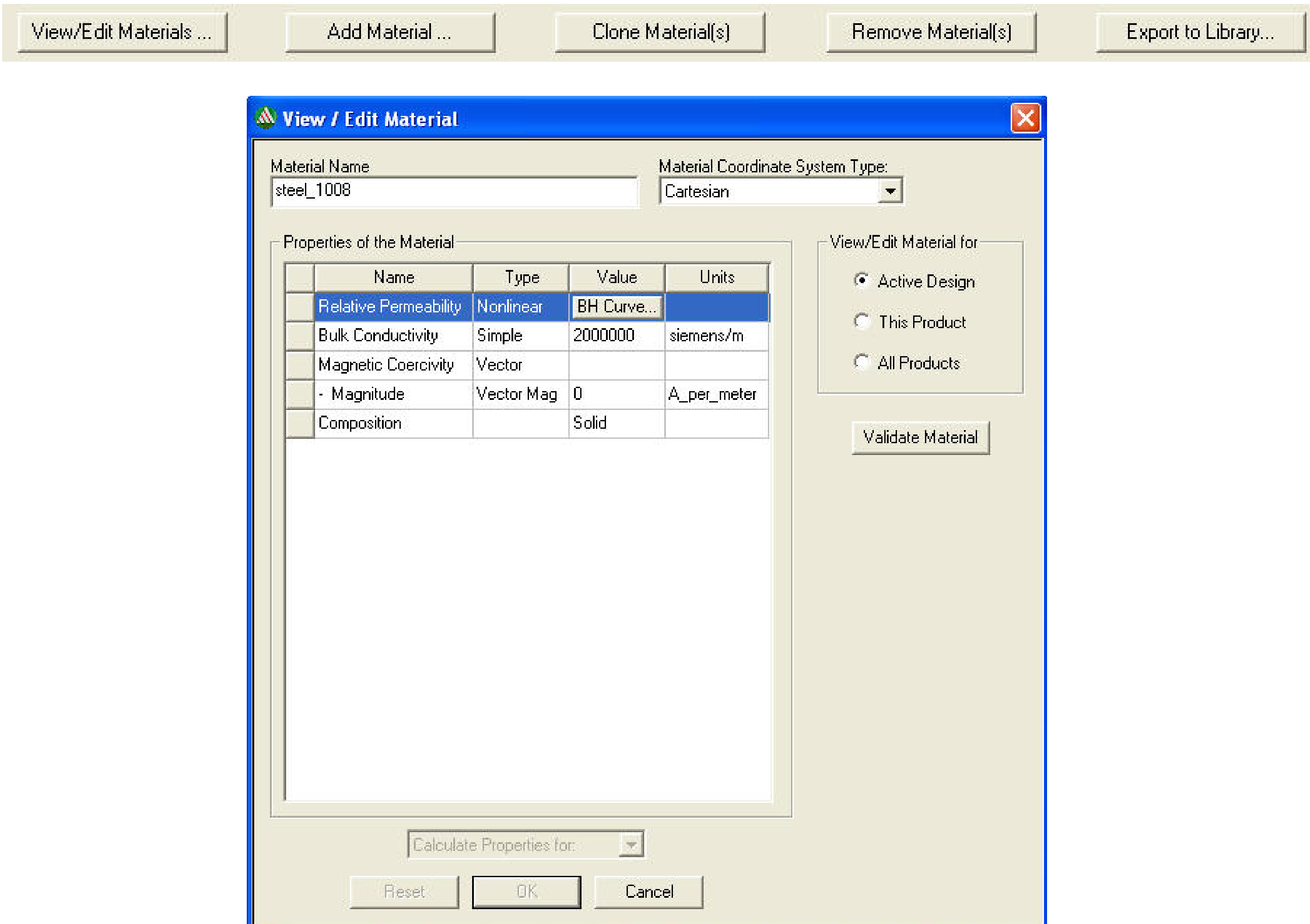
- Material Setup - Libraries
  - ***3-Tier library structure***
    - *System (global) level – predefined from ANSYS and ships with new upgrades, users cannot modify this*
    - *User Library – to be shared among several users at a company (can be encrypted)*
    - *Personal libraries - to be used only 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.

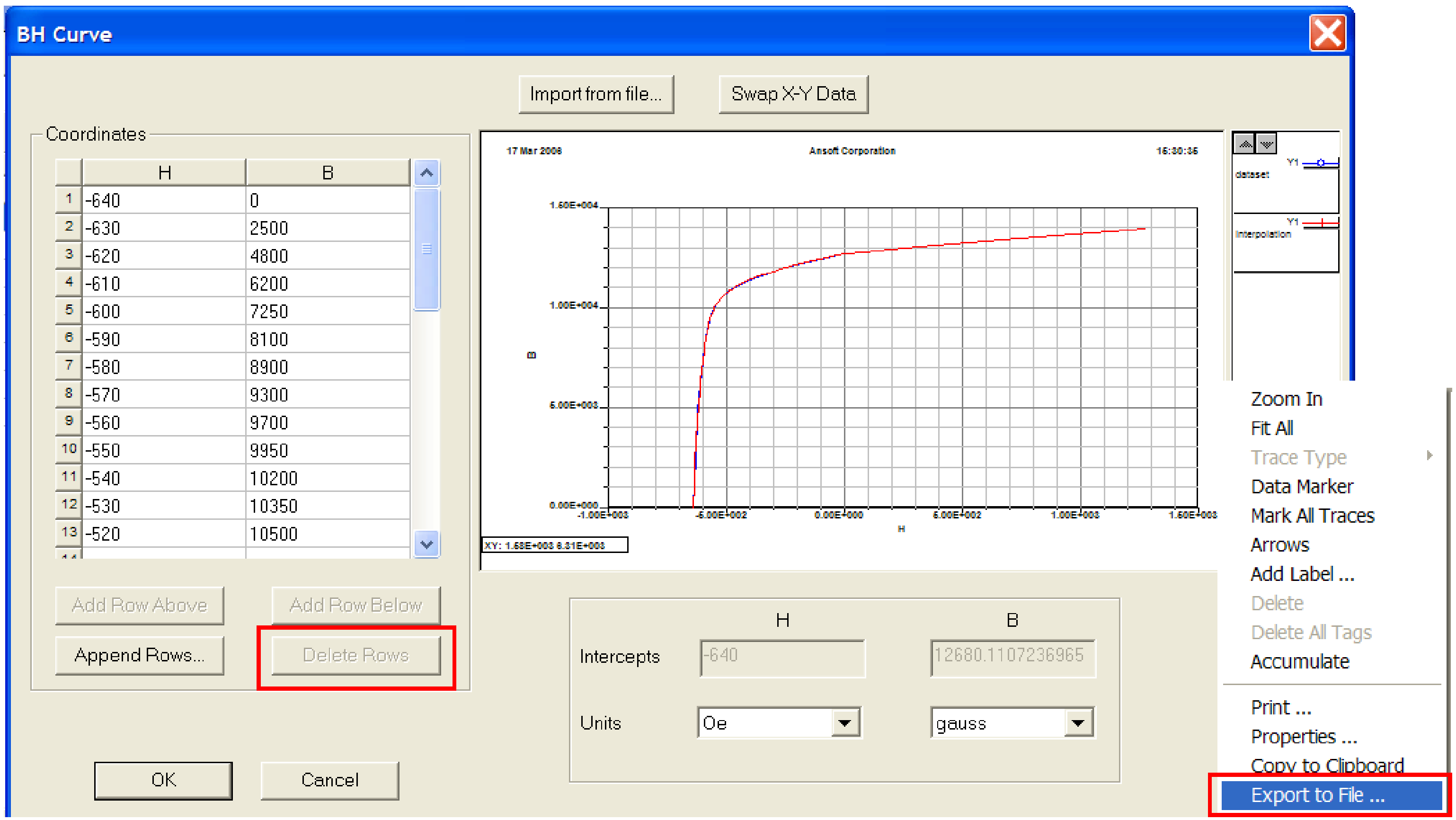


- Materials Setup - Editing



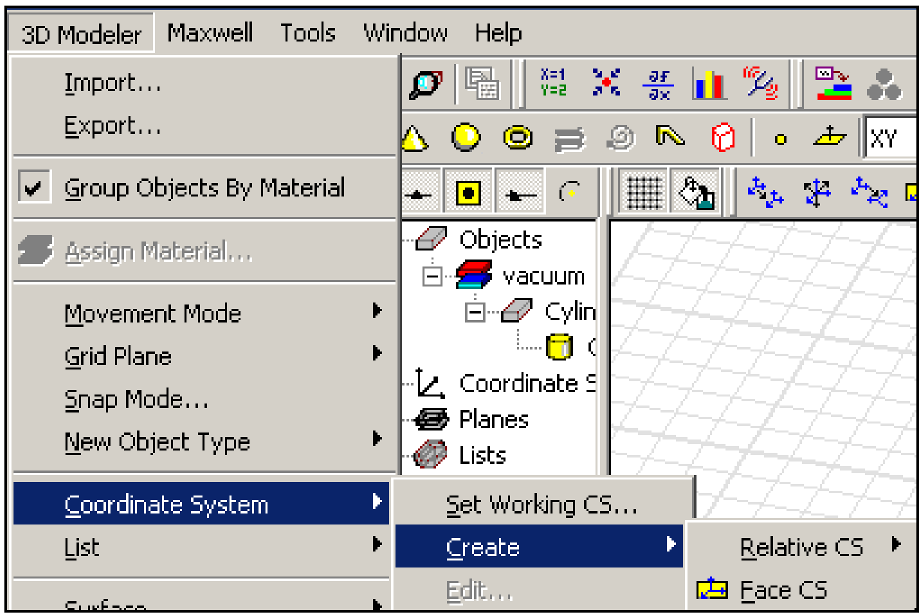
Material Setup - BH curve

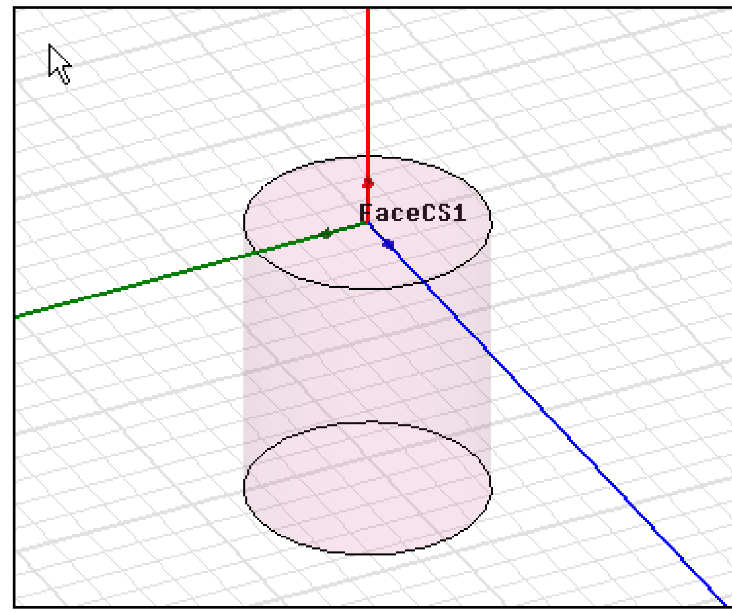
- Lamination model to account for stacking factor (modifies permeability, not conductivity)
- Robust BH curve entry - can delete points Create the Object
- To export BH curve for use in future, right-mouse-click on curve and select Export to File…



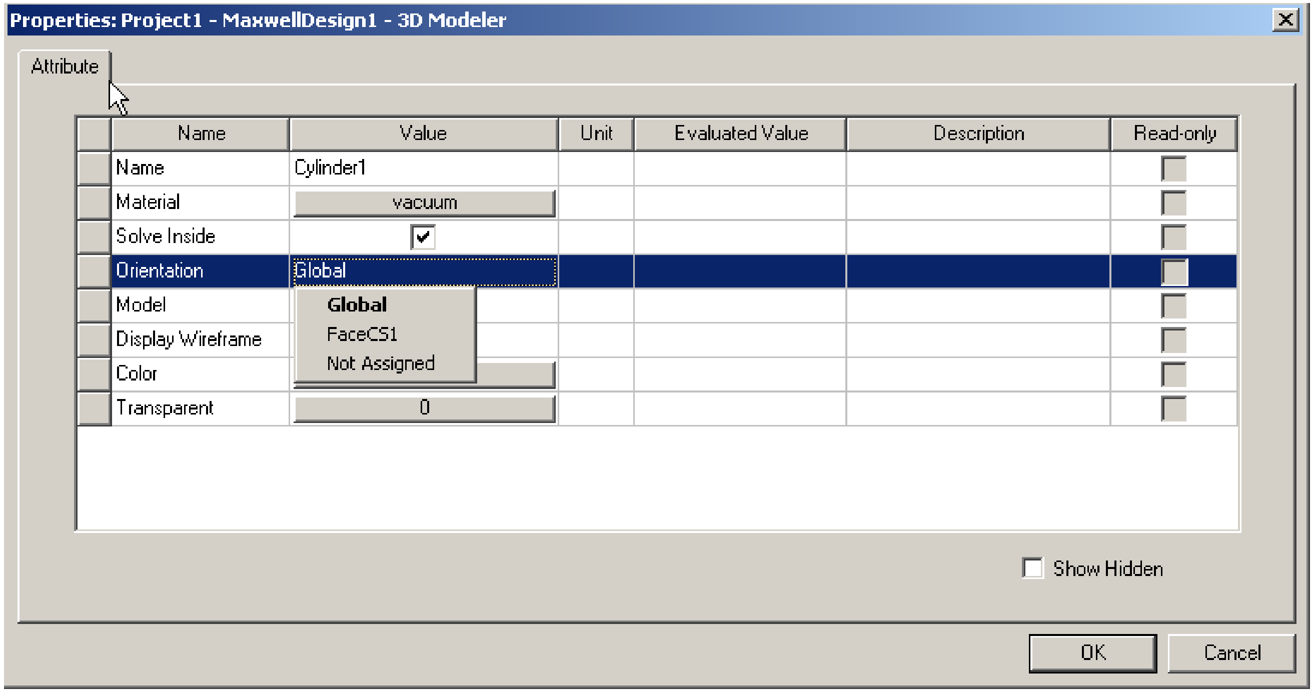
Material Setup - Permanent Magnets:

- Create the Object
- Create a Face Coordinate System (Face CS)

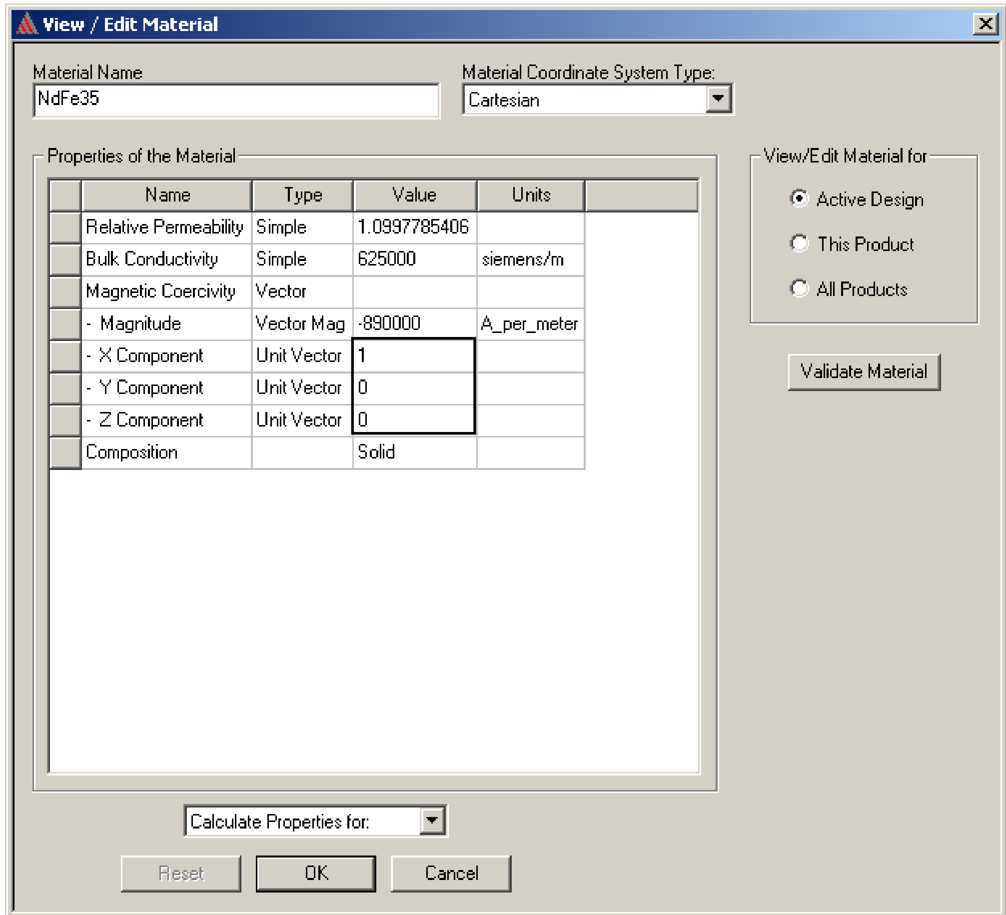




- Enter the Materials Library and choose the Desire Material
- Permanent Magnet Orientation refers by default to the global CS. To change the reference CS for orientation, open the property window and change the CS reference



- In the Material Window, you can assign orientation
- The Orientation can be described in Cartesian, Cylindrical, Spherical



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 $\sigma3$ are tensors in the Z direction.
- Anisotropic permeability definitions can be either **LINEAR** or **NONLINEAR**.

Electrostatic & Eddy Current — Magnetostatic, Eddy Current, & Transient — Eddy Current, & Transient

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

Electric Boundary Conditions

Boundary Type	E-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – The normal component of **D** changes by the amount of surface charge density. No special conditions are imposed. • Neumann boundaries – **E** is tangential to the boundary. Flux cannot cross a Neumann boundary.	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) – **E** is tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) – **E** is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Matching (Master and Slave)	The **E**-field on the slave boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field on the master boundary.	Planes of symmetry in periodic structures where **E** is oblique to the boundary.
Note: No Balloon Boundary available		

Electric Sources

Source	Type of Excitation
Floating Conductor	Used to model conductors at unknown potentials.
Voltage	The DC voltage on a surface or object.
Charge	The total charge on a surface or object (either a conductor or dielectric).
Charge Density	The charge density in an object.
Note: DC conduction solution (voltage) can also be used as input for an electric field solution	

Magnetostatic Boundaries

Boundary Type	H-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: - Natural boundaries – **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 Field (H-Field)	The tangential components of **H** are set to pre-defined values. Flux is perpendicular.	External magnetic fields.
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.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
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.
Note: No Balloon Boundary available		

Magnetostatic Sources

Source	Type of Excitation
Voltage	The DC voltage on a surface or object.
Voltage Drop	The voltage drop across a sheet object.
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Current Density Terminal	The terminal source current.

Notes:
- Current and H-field are RMS (or DC) values
- Current sources require one or more 2D sheet objects (terminal)
- Permanent Magnets can also be a source

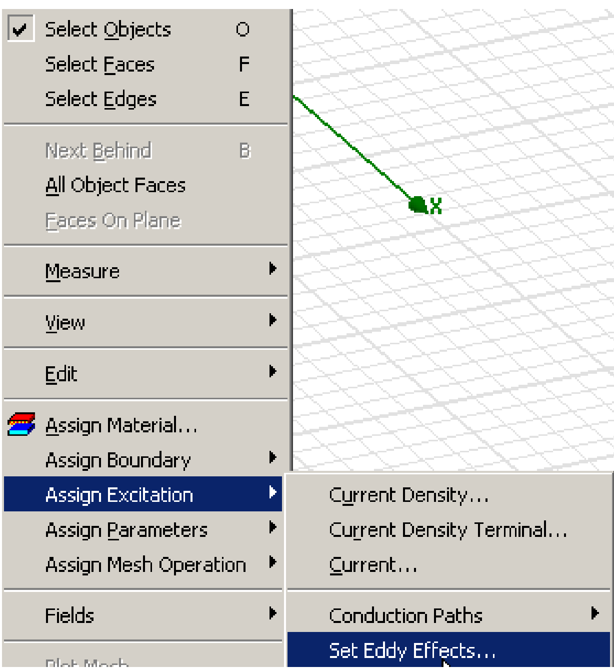
Eddy-current boundaries

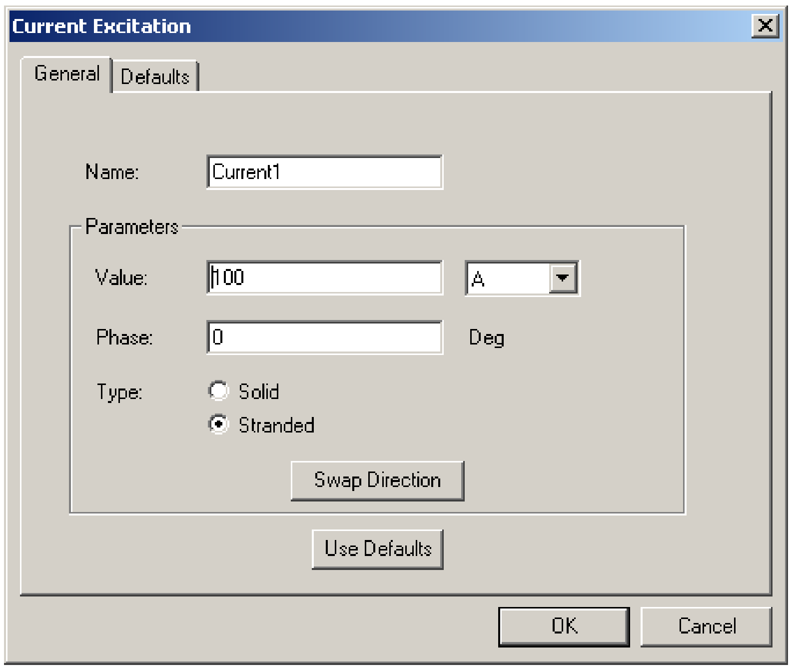
Boundary Type	H-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – **H** is continuous across the boundary. • Neumann boundaries – **H** is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Magnetic Field (H-Field)	The tangential components of **H** are set to pre-defined values. Flux is perpendicular.	External AC magnetic fields.
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	Includes the effect of induced currents beyond the boundary surface.	Conductors with very small skin depths.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Radiation	No restrictions on the field behavior.	Unbounded eddy currents.
Matching (Master and Slave)	The **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.
Note: Radiation (Balloon) Boundary available		

Eddy-current Sources

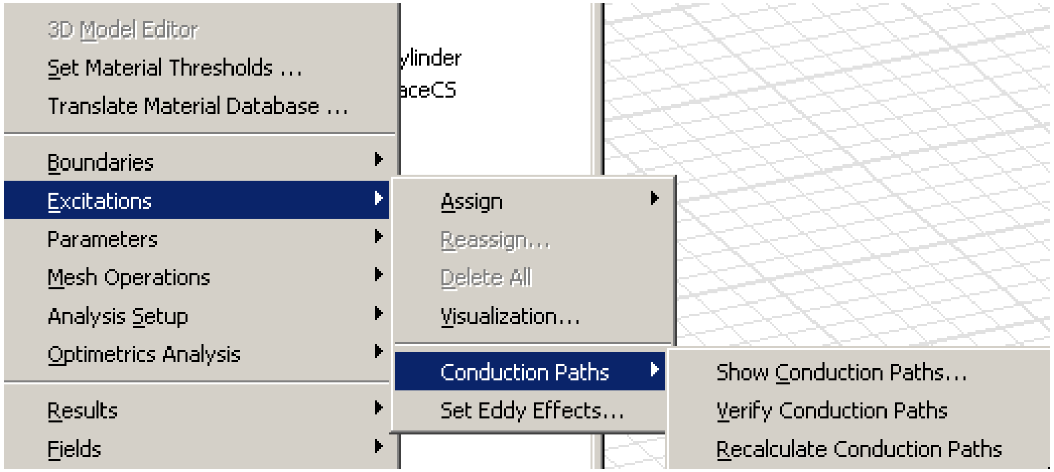
Source	Type of Excitation
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Current Density Terminal	The current density terminals in a conductor.
Notes: • Current and H-field are Peak (Not RMS) values • Current sources requires a 2D sheet objects (terminal) which lie in the cross-section of the conductor	

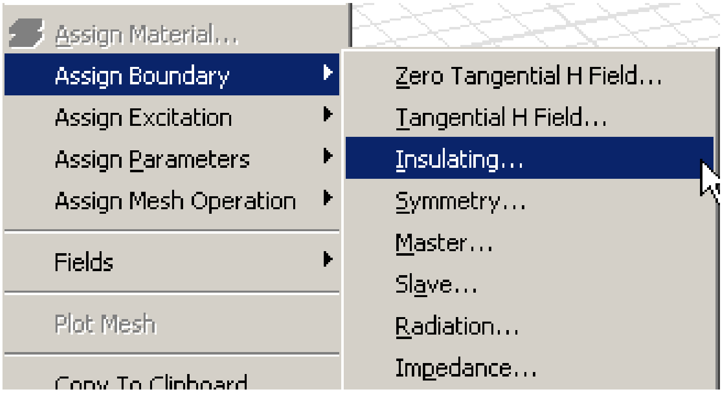
- ## Setup Boundaries/Sources
  - Select Object (for solid sources and current terminals)
  - Select Face (for insulating boundaries)
  - Select item ***Maxwell > Excitations > Assign*** for sources
  - Select item ***Maxwell > Boundaries > Assign*** for boundary conditions
  - Turn on eddy or displacement current calculation in eddy current solver for materials of interest: ***Maxwell > Excitations > Set Eddy Effects …***
  - Stranded coil option available in Magnetostatic and Eddy Current solvers



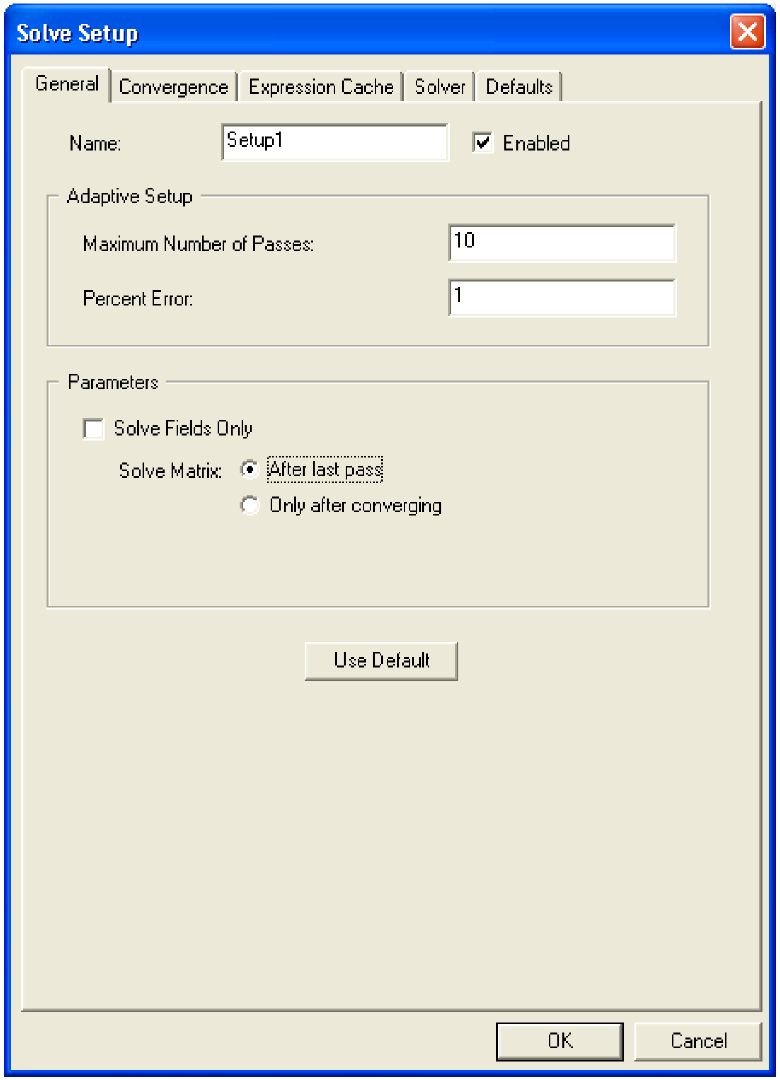


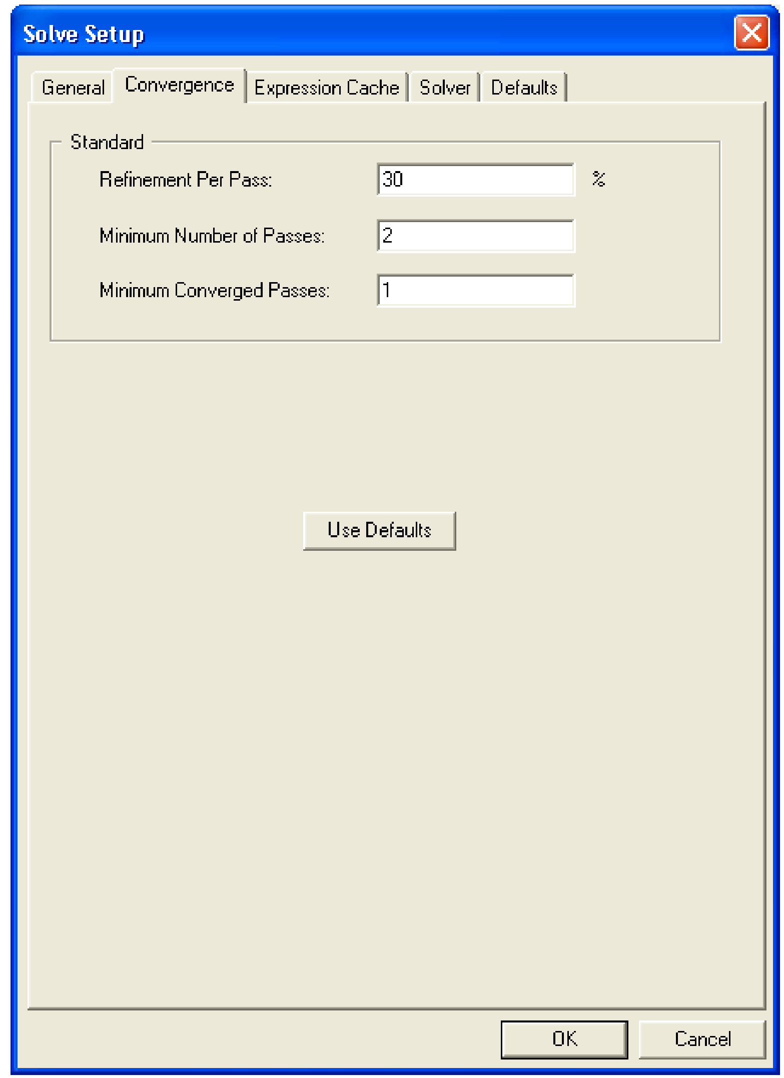
- View conduction paths by selecting: ***Maxwell > Excitations > Conduction Paths > Show Conduction Paths***
- Assign insulating boundaries on faces between touching conductors if necessary



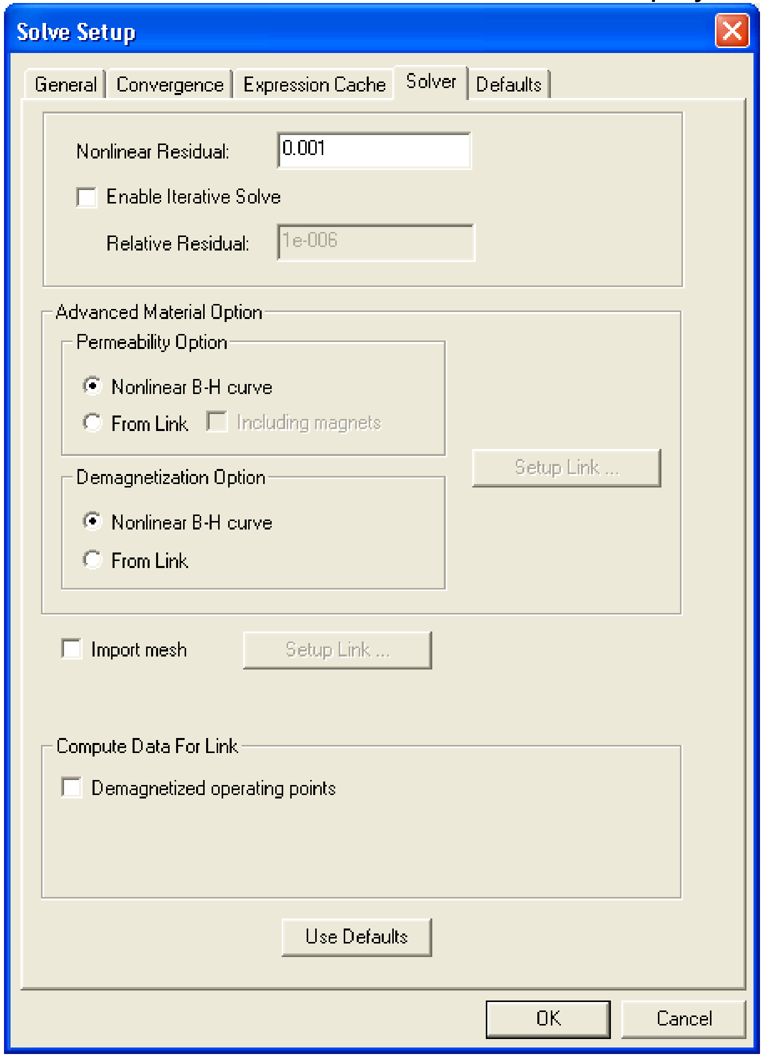


- ## Magnetostatic and Electric Solution Setup
  - Start the menu of solution setup by: ***Maxwell > Analysis Setup > Add Solution Setup …***

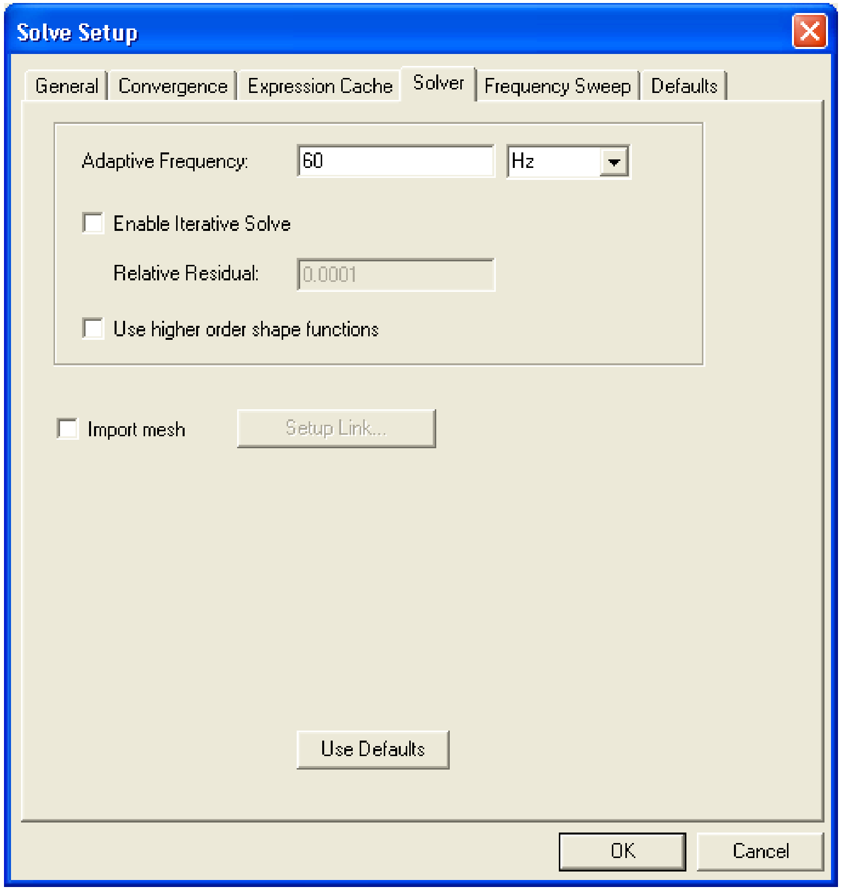


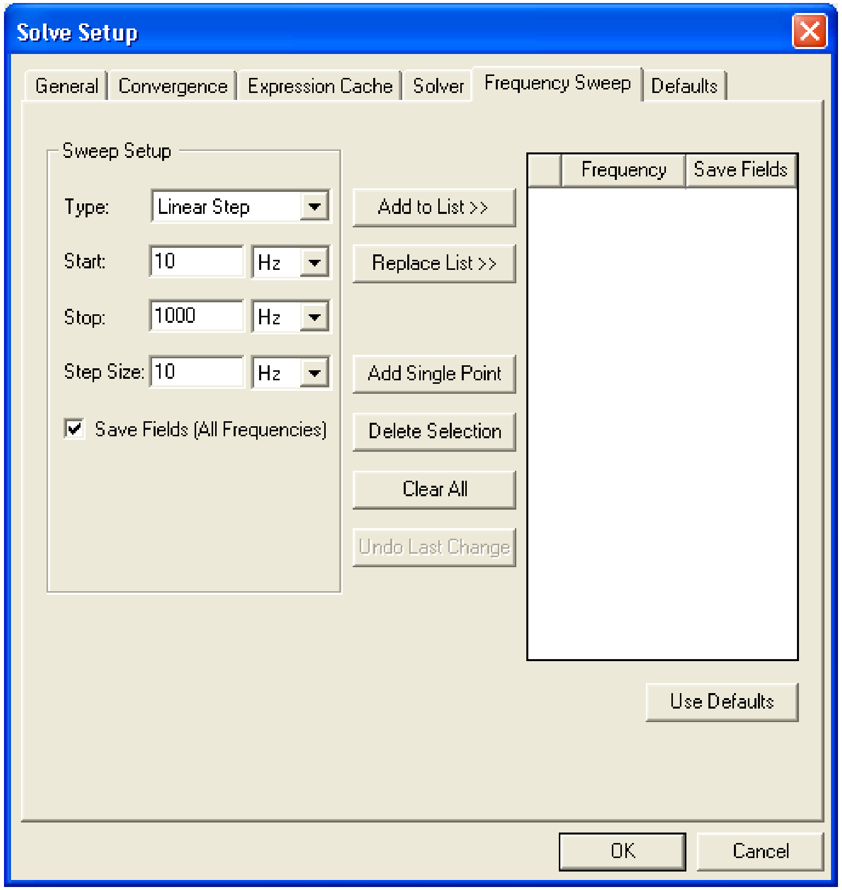


- # Magnetostatic Setup
  - For Magnetostatic solver on Solver tab, suggest setting nonlinear residual = 0.001.
  - Expression Cache tab to set a calculator expression computation at each pass and use it as convergence criteria
  - On default tab choose Save Defaults to set this value for all future projects.

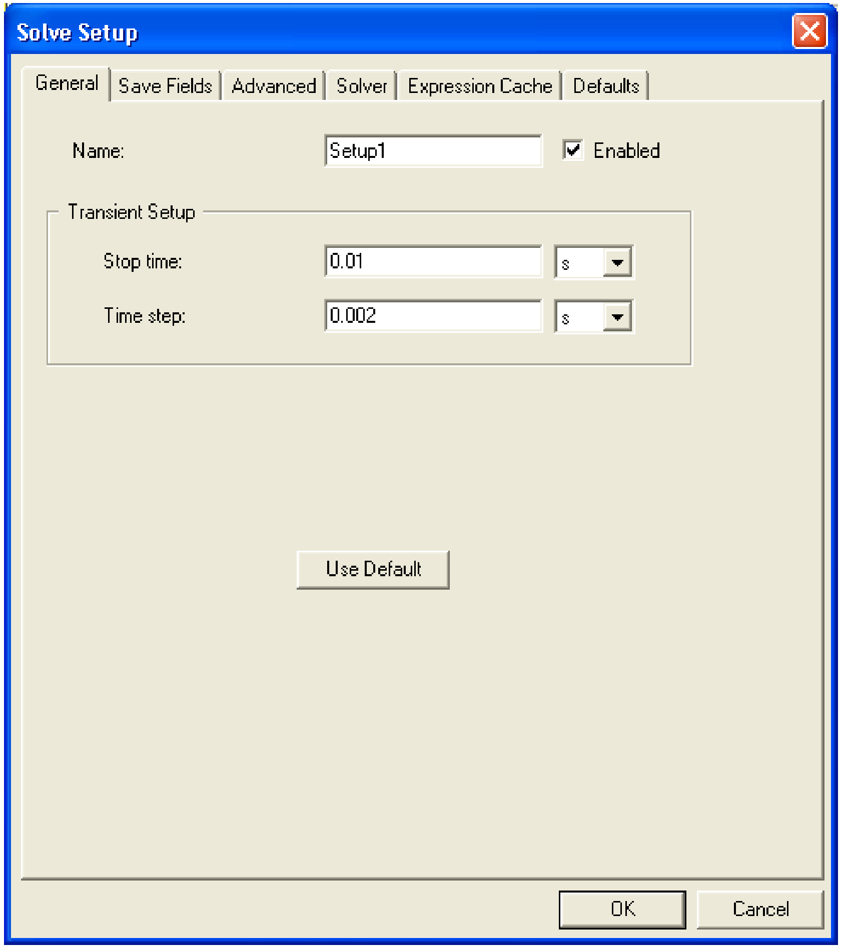


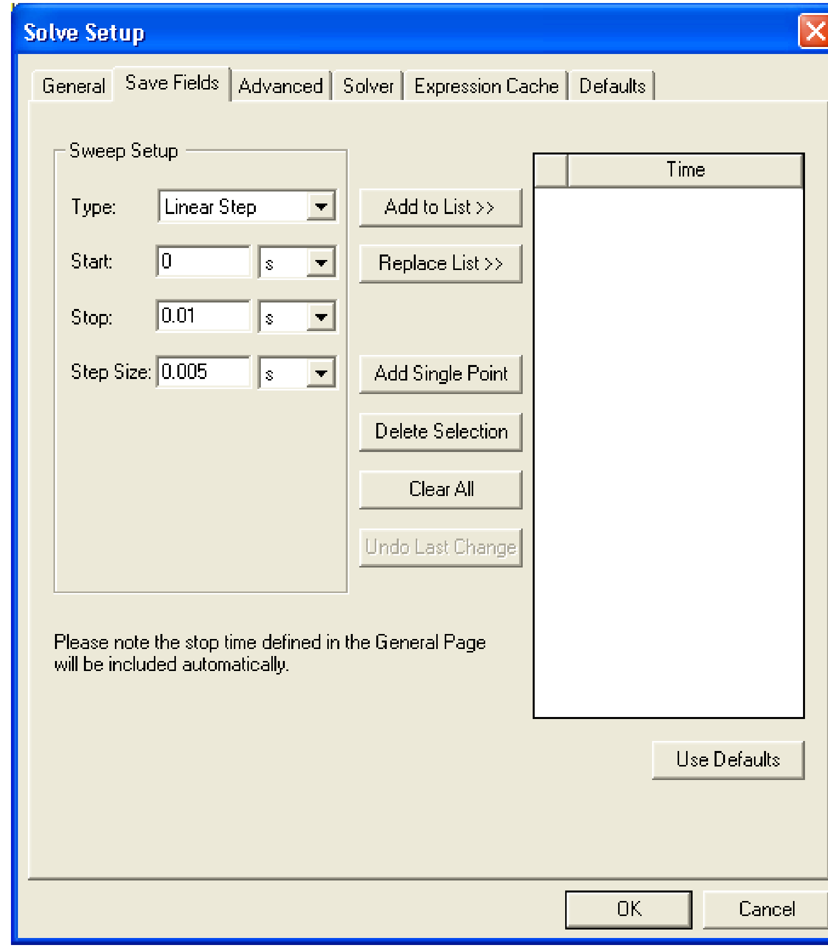
- Eddy Current Solution Setup



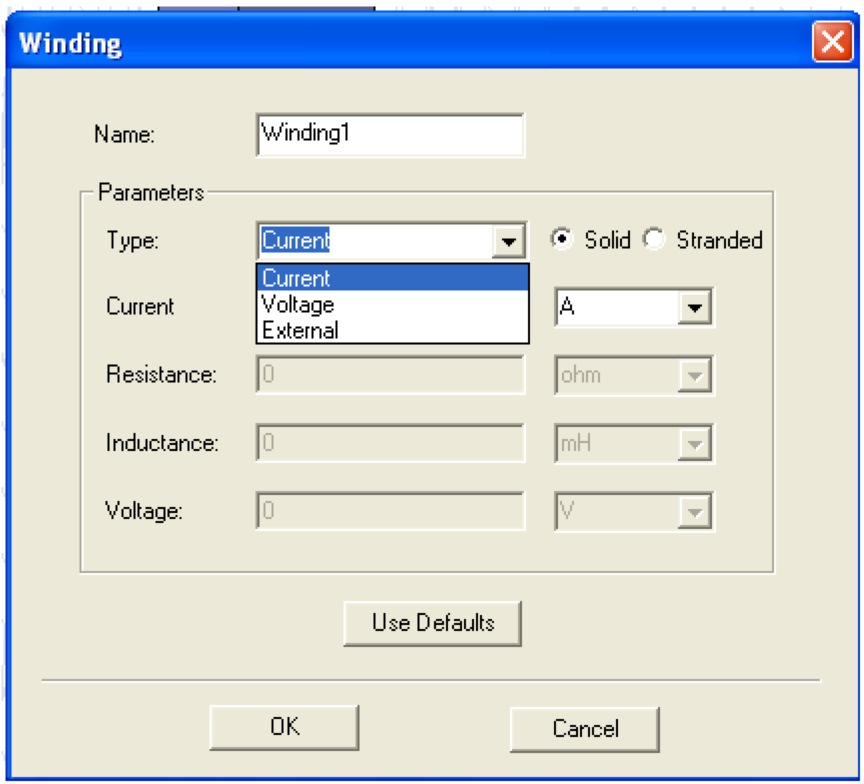


- Transient Solution Setup



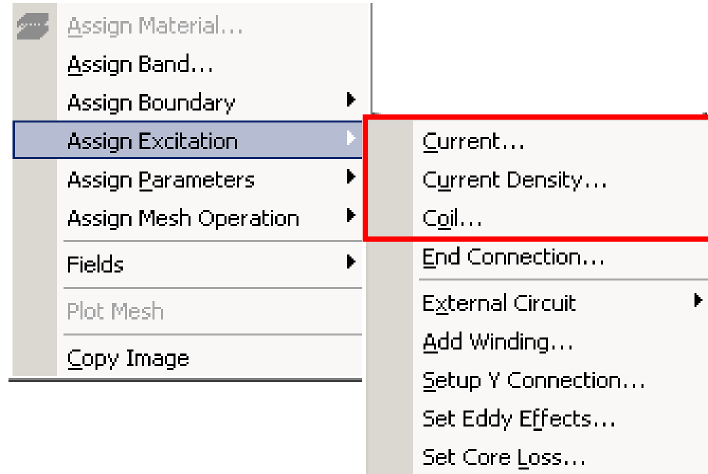


- Transient Sources
  - Coil Terminal - used to define one or more model windings
  - Winding With Current - stranded and solid conductor
  - Winding With Voltage - stranded and solid conductor
  - Winding With External Circuit Connection - stranded and solid conductor
  - In addition, permanent magnets serve as sources of magnetic fields

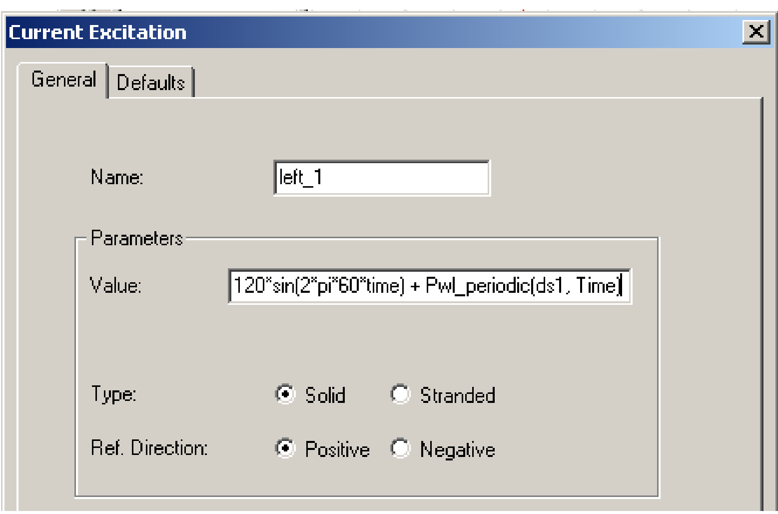


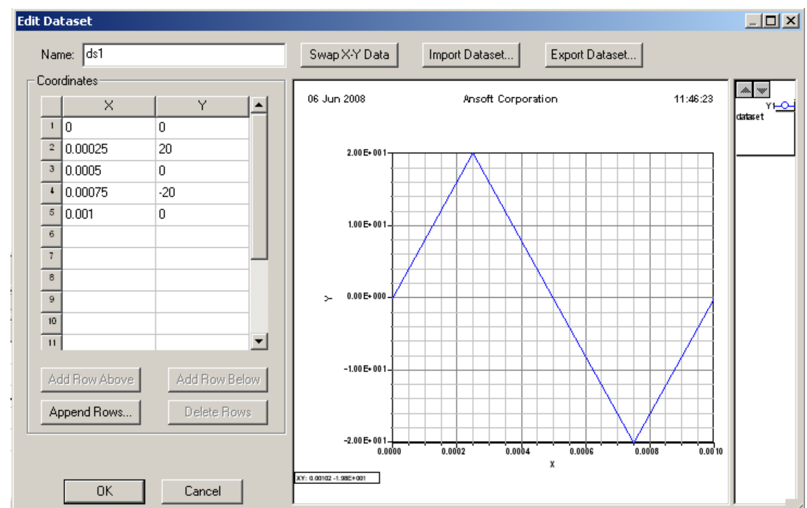
- Magnetic Field Sources (Transient)

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.	



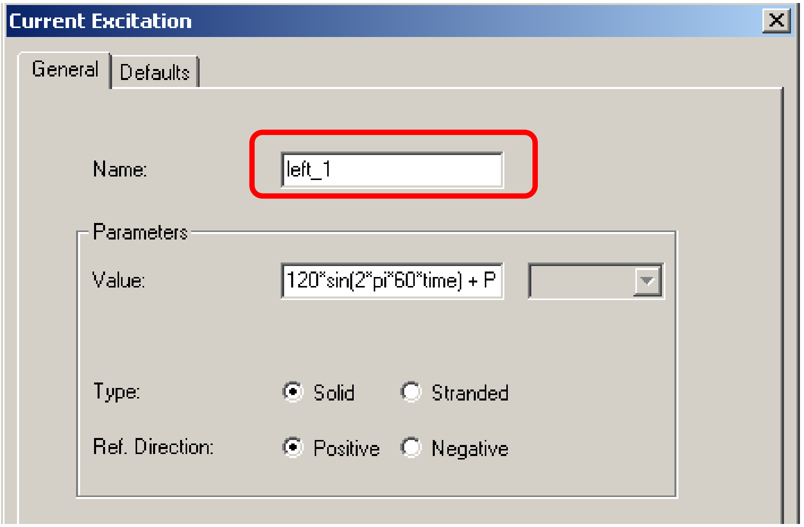
- Current and voltage sources (solid or stranded) can be constant or functions of intrinsic variables: speed (rpm or deg/sec), position (degrees), or time (seconds)
- Dataset function can be used for piecewise linear functions: Pwl_periodic (ds1, Time)





Magnetic Field Sources (Transient)

- Maxwell 3D > 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 <u>not</u> considered
  - Ref Direction:
    - Positive or Negative

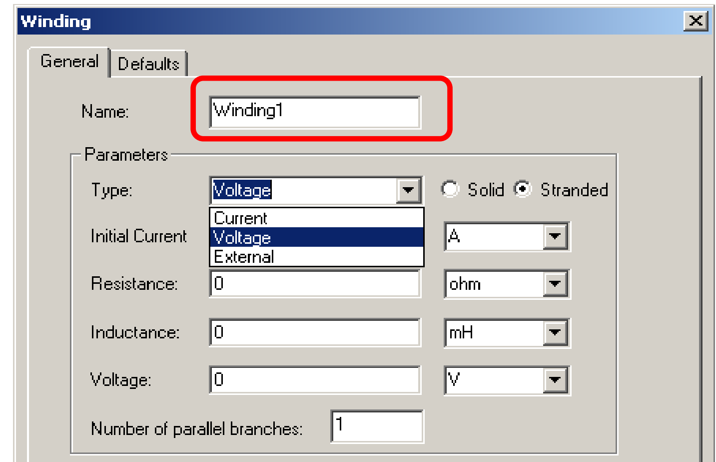


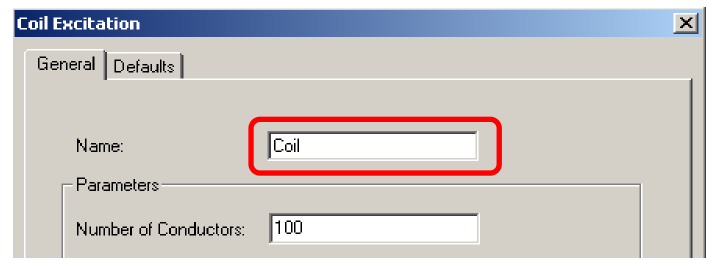
Magnetic Field Sources (Transient)

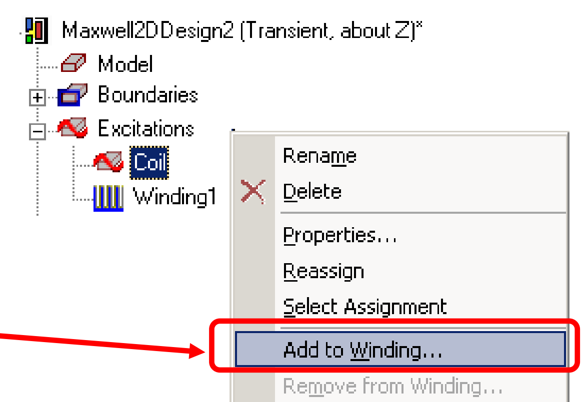
- **Maxwell 3D > Excitation > Add Winding**
  - Current - applies current in amps
    - Solid or Stranded
    - Input current and number of parallel branches as seen from terminal
  - Voltage - applies voltage (total voltage drop over the length of a solid conductor or the entire winding)
    - Solid 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 3D > Excitation > Assign > Coil Terminal**

  Pick a conductor on the screen and then specify:
  - Name
  - Number of Conductors

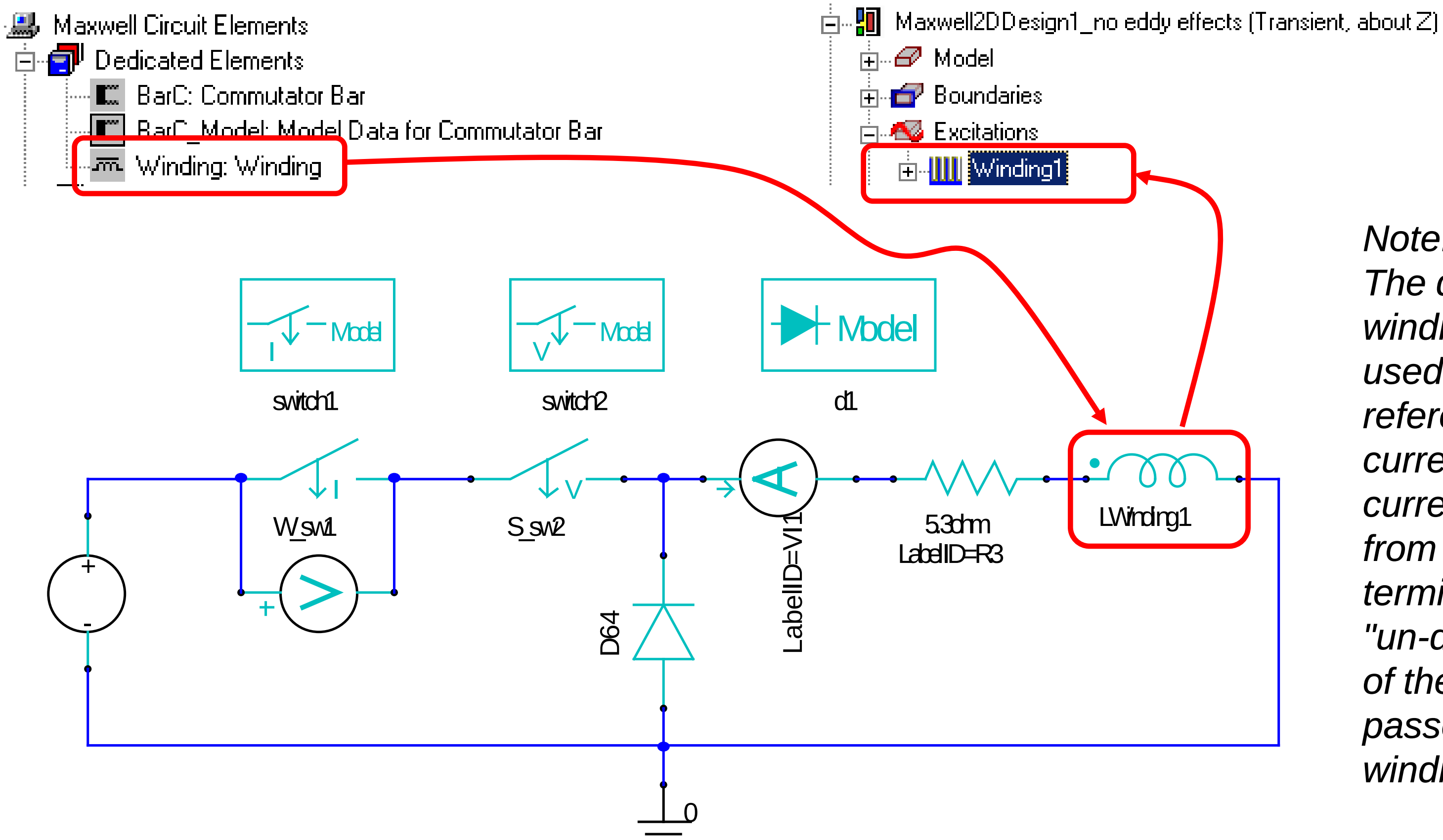
  Note: Windings in the XY solver will usually have 2 coils: one positive and one negative polarity. Both coils will be added to the appropriate winding by right-mouse clicking on **Coil** in the project tree and choosing **Add to Winding**





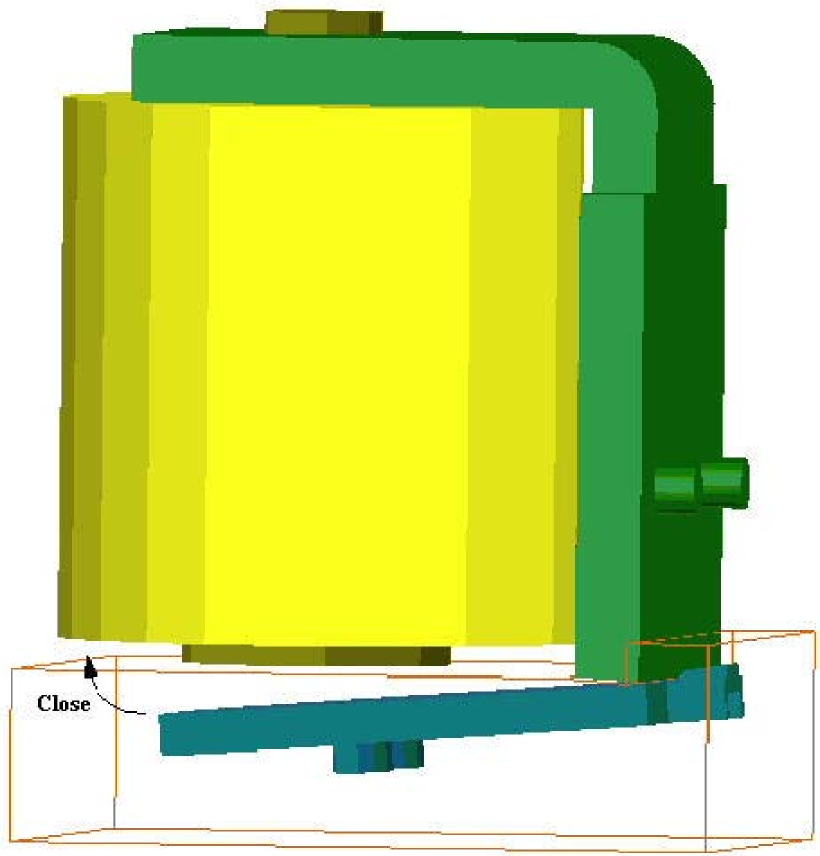


- To Create an External Circuit
  1. Select: ***Maxwell3D > Excitations > External Circuit > Edit External Circuit > Import Circuit***
  2. After circuit editor opens, add elements to construct the circuit. Note that the name of the Winding in the circuit (Winding1) must match the name of the Winding 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).

- **Transient Motion**
  - Translational motion (motion along a user specified, linear direction).
  - Rotational motion (non-cylindrical such as the pivoting rotation around an axis encountered in the armature of a relay).
  - Rotational motion (cylindrical, such as the type of rotation encountered in an electric machine type of application).



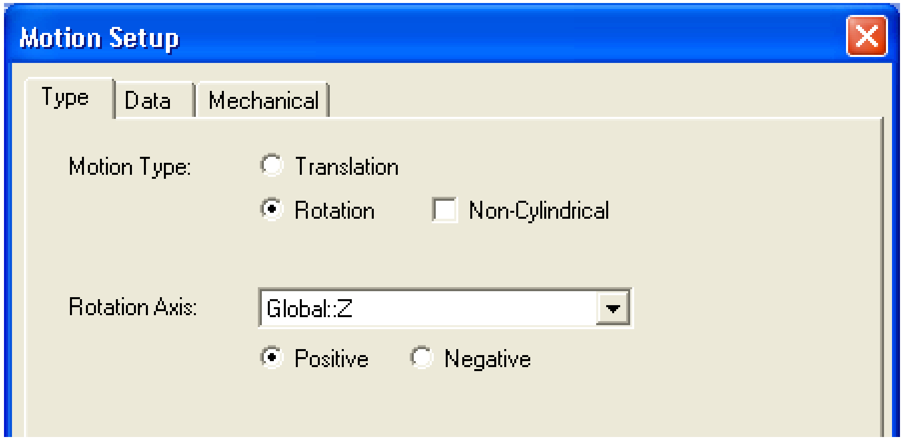
Ex: Pivoting Rotation

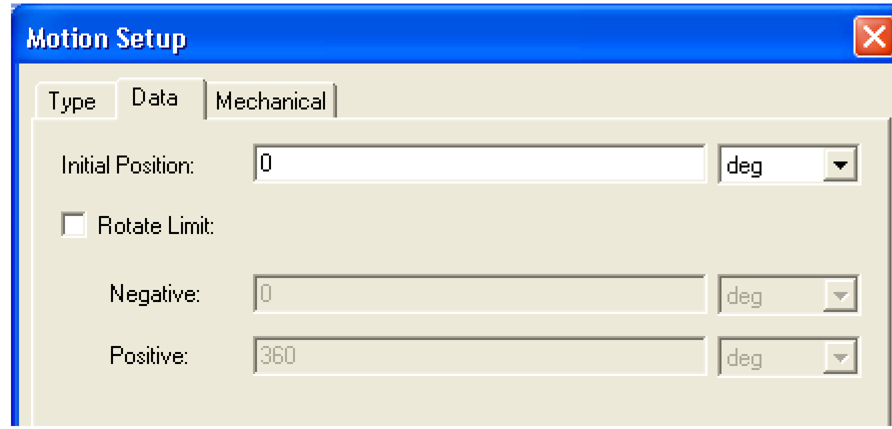
- Band
  - Regardless of the particular type of motion involved, all types of motion applications require a band object that must contain the moving part(s).
  - If there are multiple moving objects, all of the moving parts must be included in one all-inclusive object – this is because they all must be moving as one rigid body, with a single force acting on the assembly.
  - Regardless of the type of motion, the user must create a mesh density capable of capturing the physical effects characteristic for the specific application, such as field gradients, skin and proximity effects, etc.

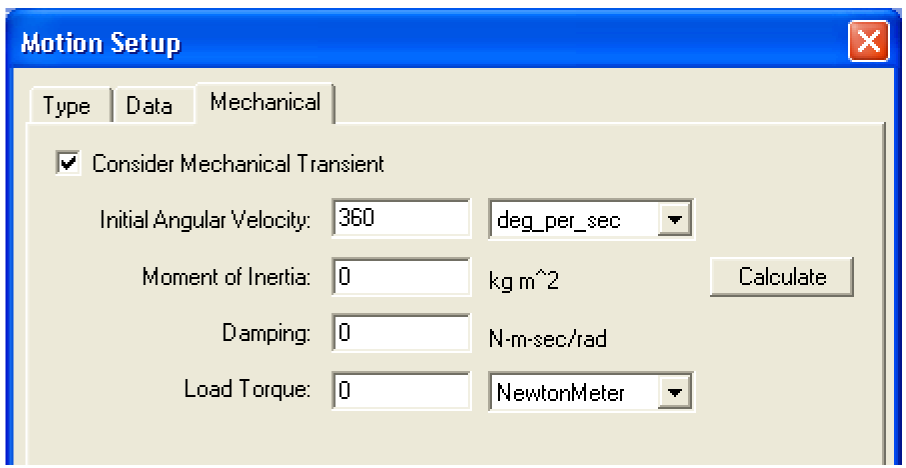
- Translational and non-cylindrical rotation types of motion:
  - The band object can touch the symmetry plane if any exists.
  - The moving object cannot touch a stationary object during motion (the gap between the moving object and the band can never become zero during analysis).
  - The band cannot have true surface faces; all faces must be segmented (for example created with the regular polyhedron primitive).
  - The band object cannot separate the stationary part into unconnected sub-regions.
  - Hollow objects cannot be used for band objects.

- For the cylindrical type of rotational motion applications:
  - Always use a facetted (regular polyhedron) type of cylindrical object or a wedge object <u>if symmetry is used</u>. The angular aperture of each facet depends on the problem; however, an opening of 2-3 degrees per facet is usually sufficient.
  - Hollow cylinders cannot be used for band objects.
  - The band object can separate the stationary part into unconnected sub-regions (a rotor sandwiched between two stators is allowed).
  - Non-cylindrical band objects are allowed.

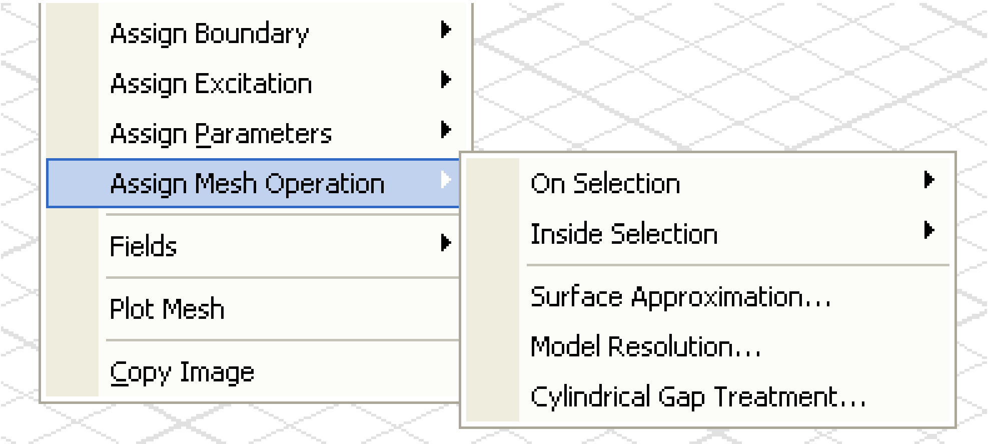
- **Motion Setup**
  - To assign a band: ***Maxwell 3D > Model > Motion Setup > Assign Band***





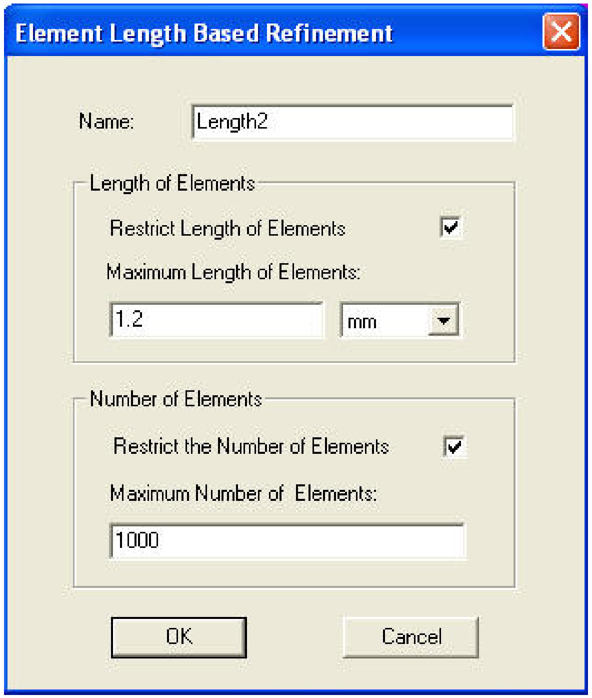


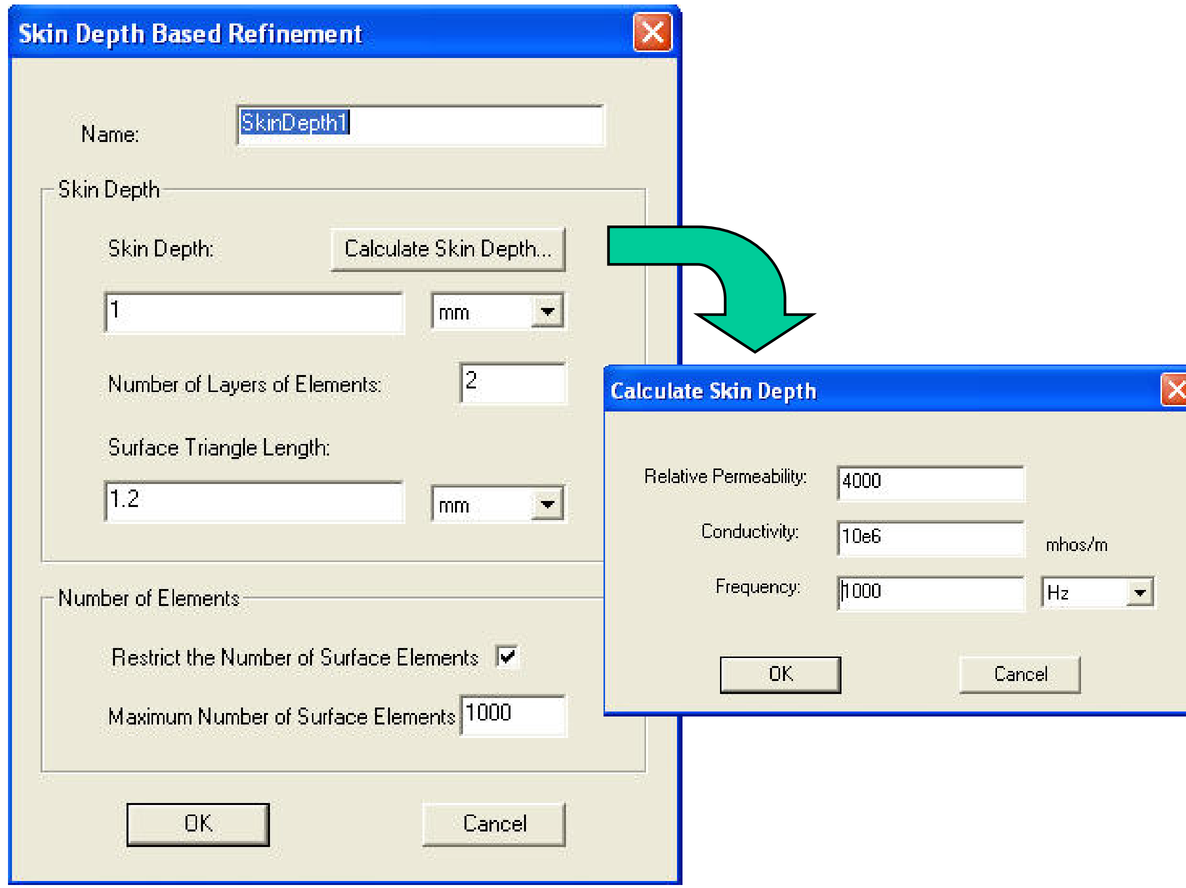
- ## Mesh Operations
  - To assign Mesh operations to Objects, select the Menu item: ***Maxwell 3D > Assign Mesh Operations***
    1. ***On Selection*** is applied on the surface of the object
    2. ***Inside Selection*** is applied through the volume of the object
    3. ***Surface approximation*** is applied to set faceting guidelines for true surface objects
    4. ***Model Resolution*** is applied to ignore small details of a geometry
    5. ***Cylindrical Gap Treatment*** is applied to achieve good mesh in narrow sections



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

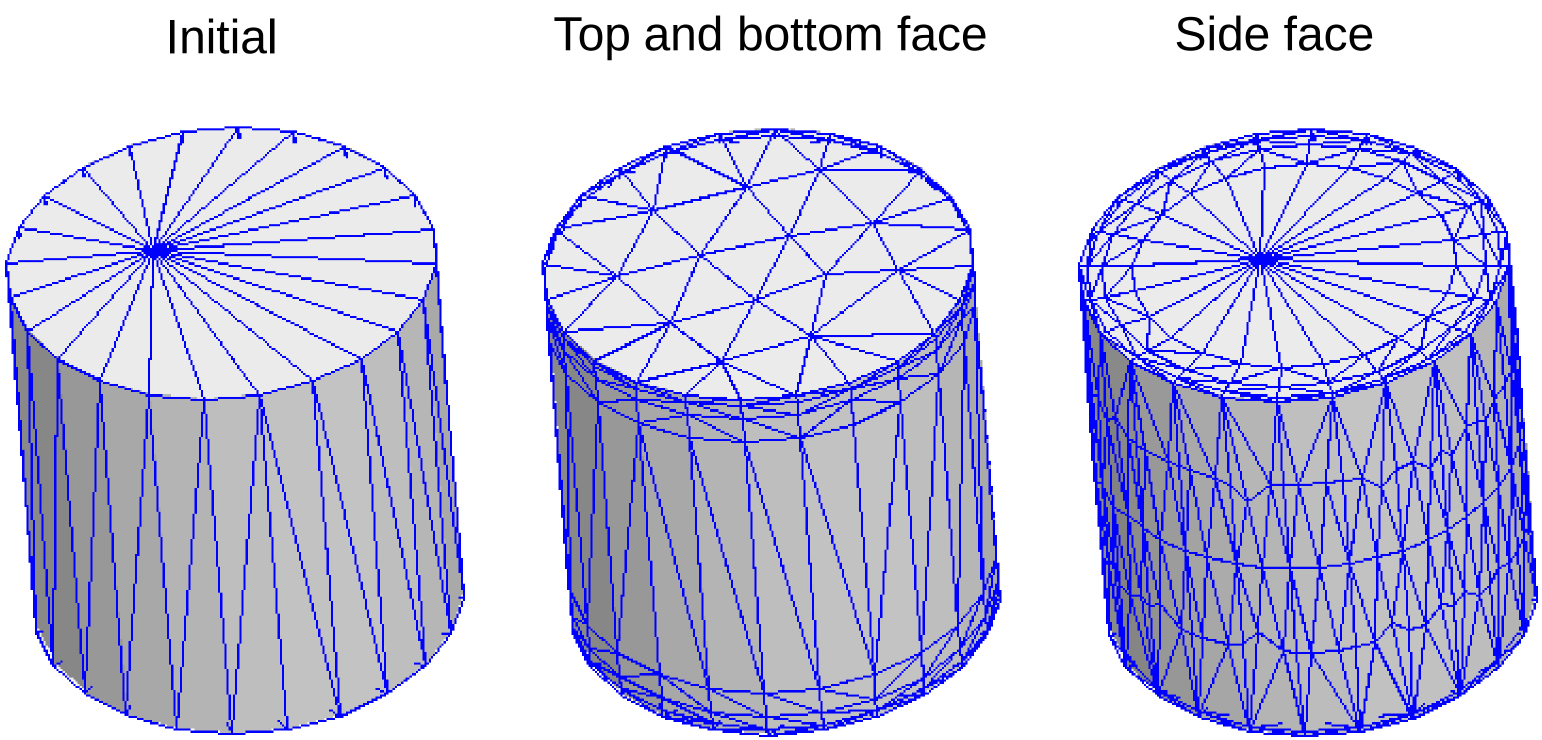
- Element length based refinement: ***Length Based***
- Skin Depth based refinement: ***Skin Depth Based***





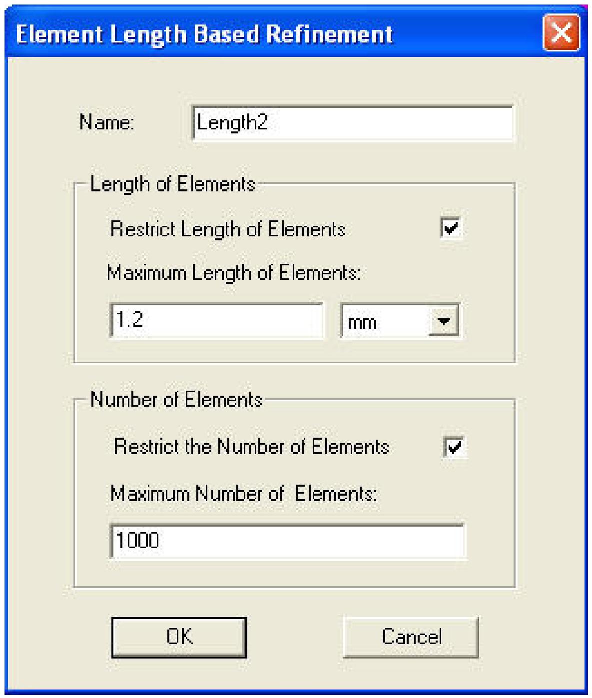
“On selection” Skin Depth Based

Note: Adaptive refinement will be applied to objects which have a skin depth mesh operation. In some cases, the skin depth mesh may be partially removed during the adaptive refinement process.



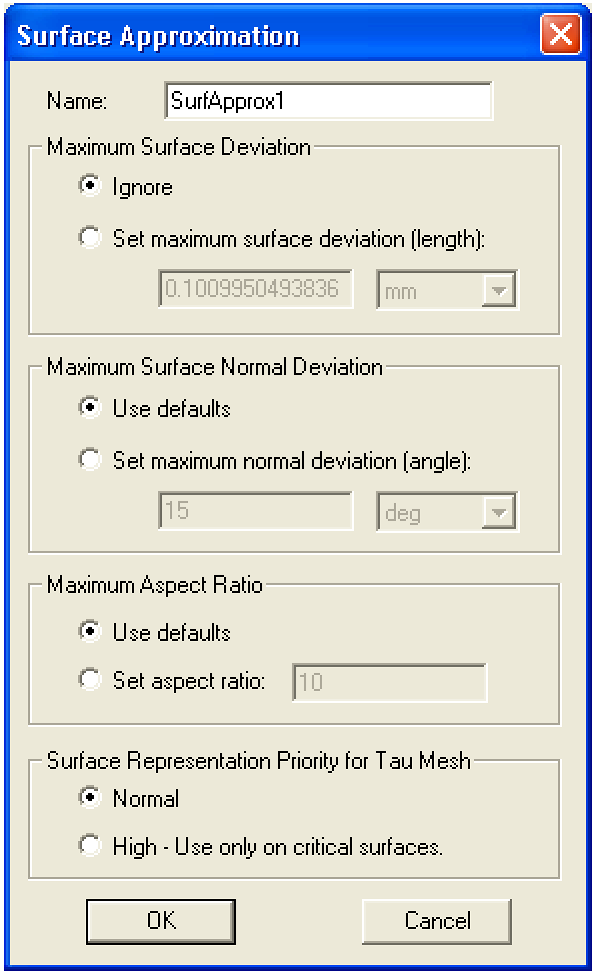
2. Mesh Operations “Inside selection” - applied throughout the volume of the object

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



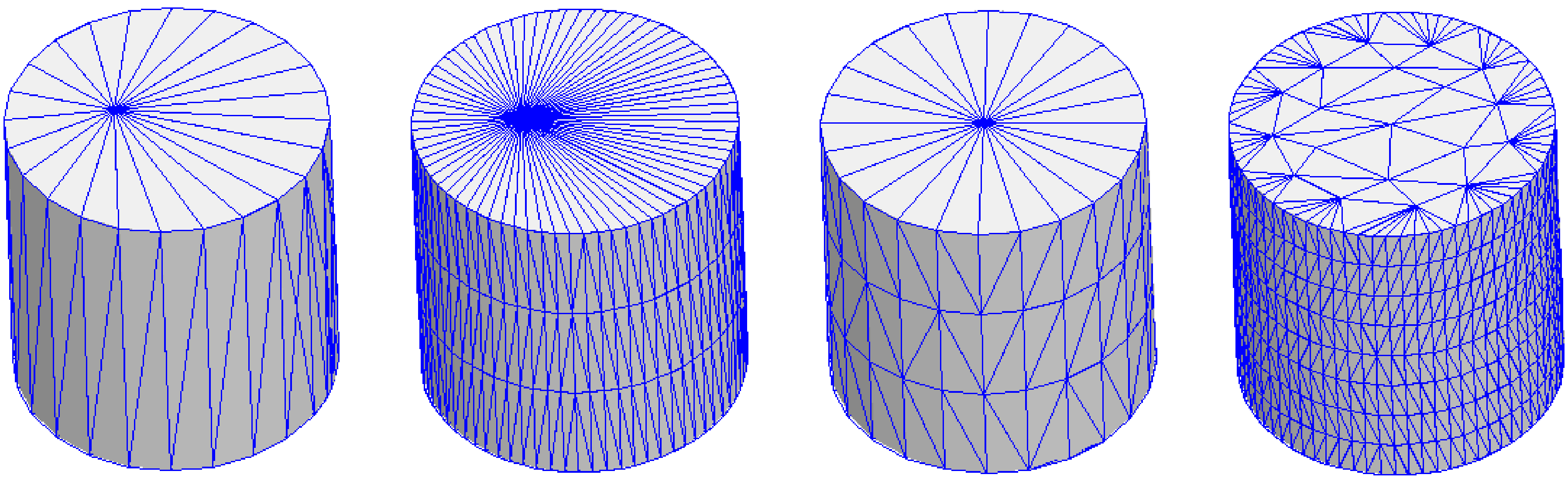
3. Mesh Operations "Surface Approximation"

- 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) which is the distance between the true surface and the approximating tetrahedral mesh. Comparing a true surface cylinder to a faceted cylinder will illustrate the surface deviation.
  - Maximum Surface Normal Deviation (degrees) which allows you to limit the maximum angle between normal vectors to adjacent tetrahedrons which break up a true surface. This can force more tetrahedron to break up a curved true surface like a cylinder.
  - Maximum Aspect Ratio is the ratio between the largest and shortest edges of triangles on a given face. A ratio = 1 would attempt to create equilateral triangles.



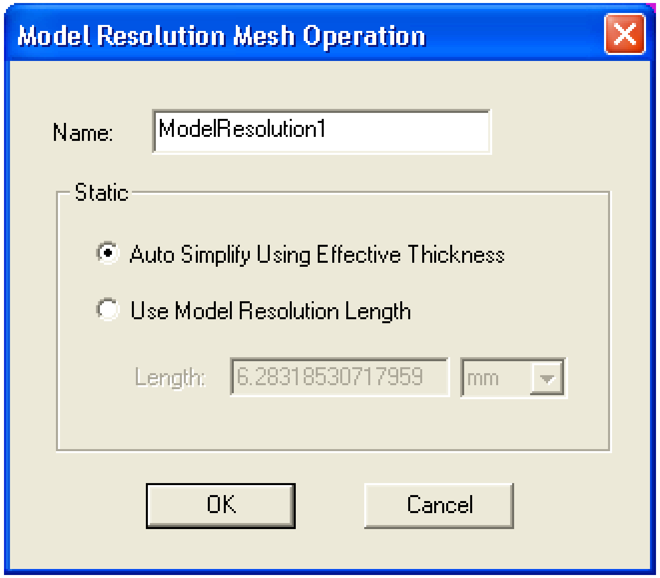
3. Mesh Operations “Surface Approximation”

Initial	Surface Normal Deviation – 5 deg (default 15 deg)	Maximum Aspect Ratio – 3 (default 10)	Both

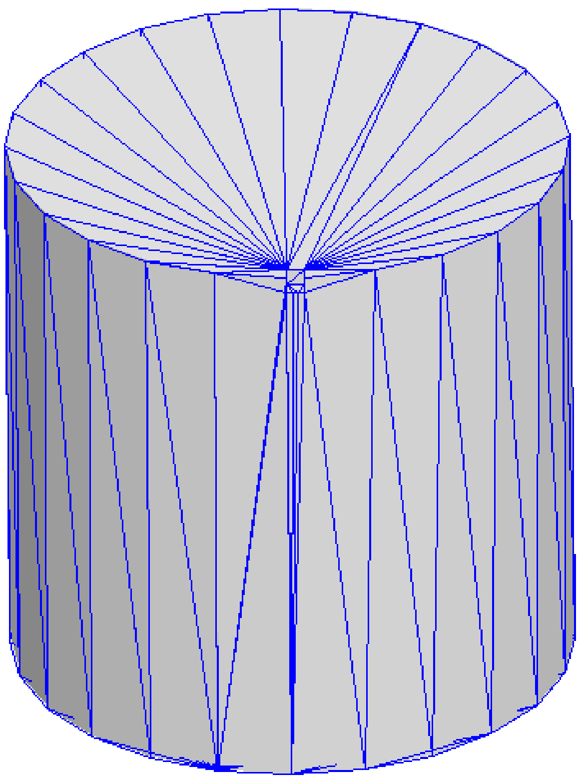


4. Mesh Operations > Model Resolution

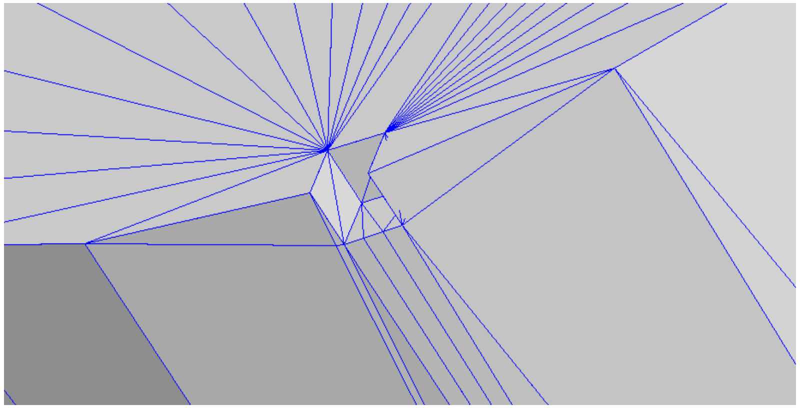
- To de-feature the mesh:
  1. Specify the maximum dimension to be used in the mesh under: **Mesh Operations > Assign > Model Resolution**
  2. Improves mesh quality and reduces mesh size for complex models
  3. Increases the mesh speed
  4. Reduces the load on the solver with reductions in mesh size



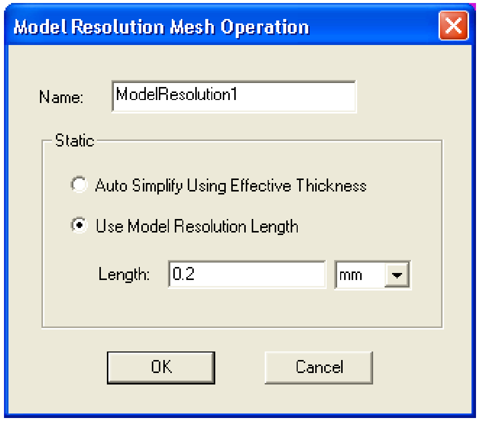
4. Mesh Operations > Model Resolution



Small box subtracted from the cylinder

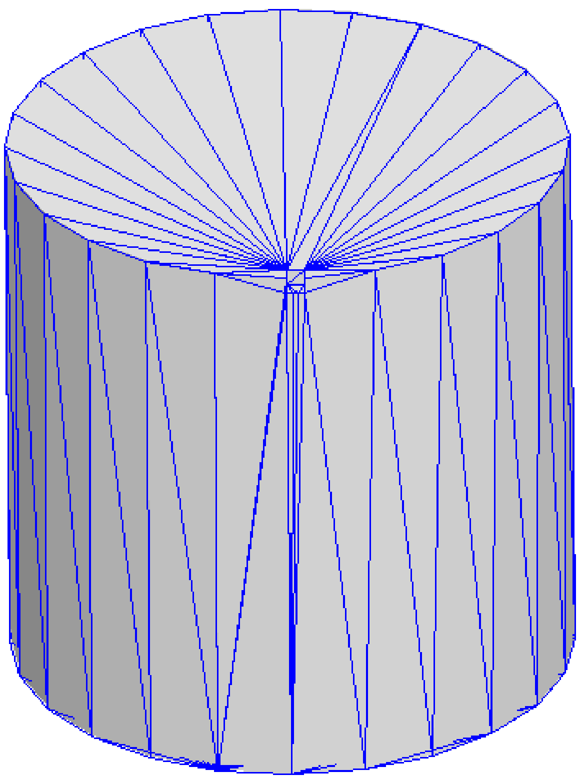


Select the object and specify the size of the small feature that needs to be ignored

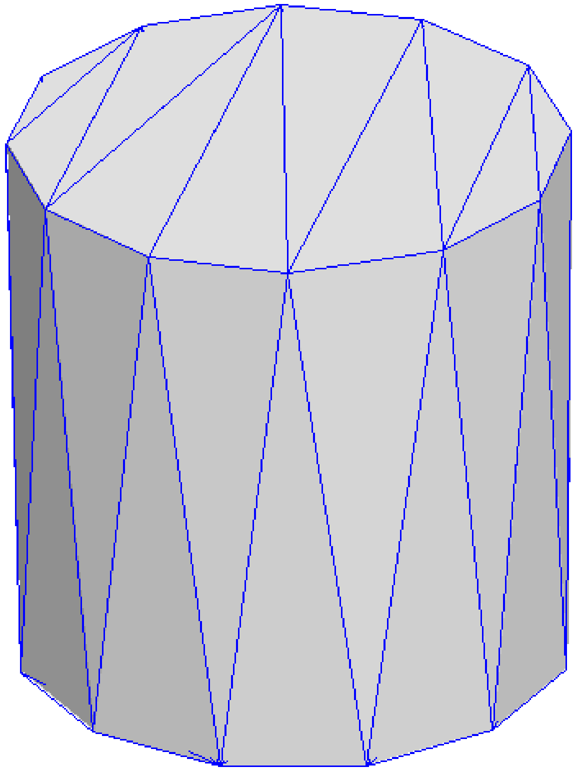


4. Mesh Operations > Model Resolution

No model resolution

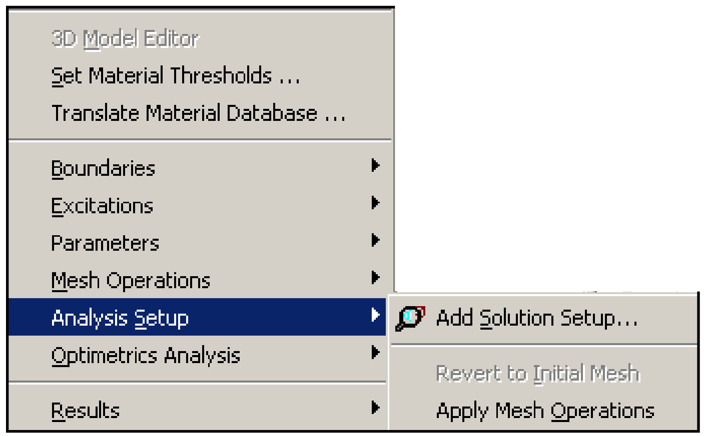


Model resolution larger that the small box edge length



Manual mesh creation

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

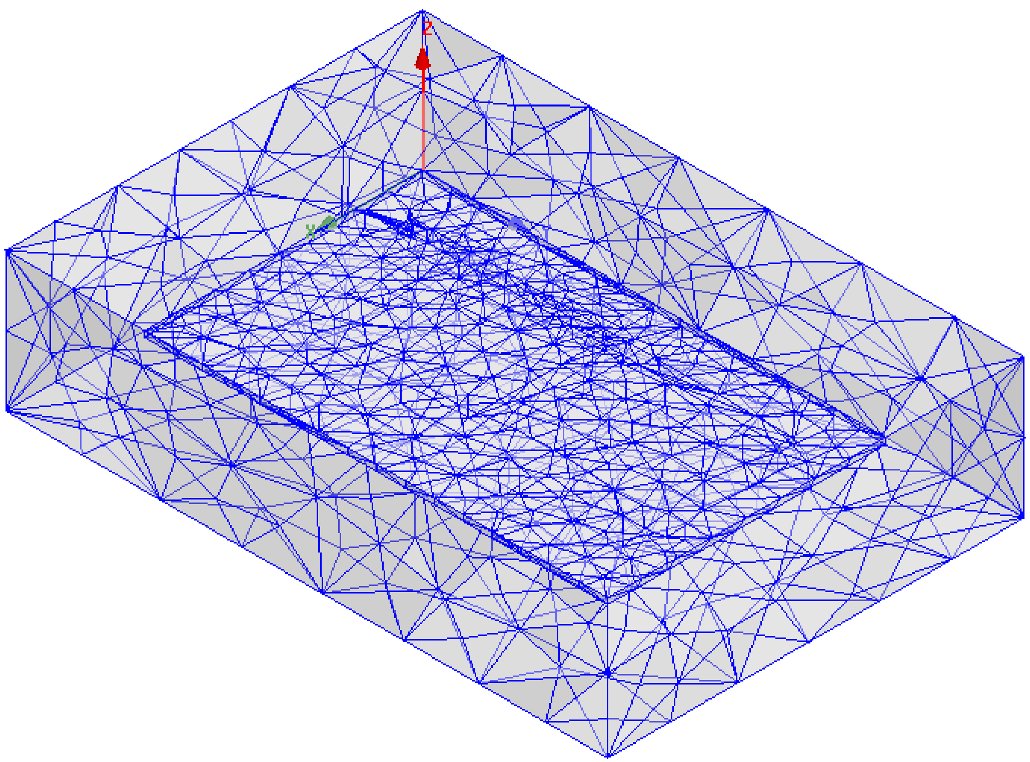


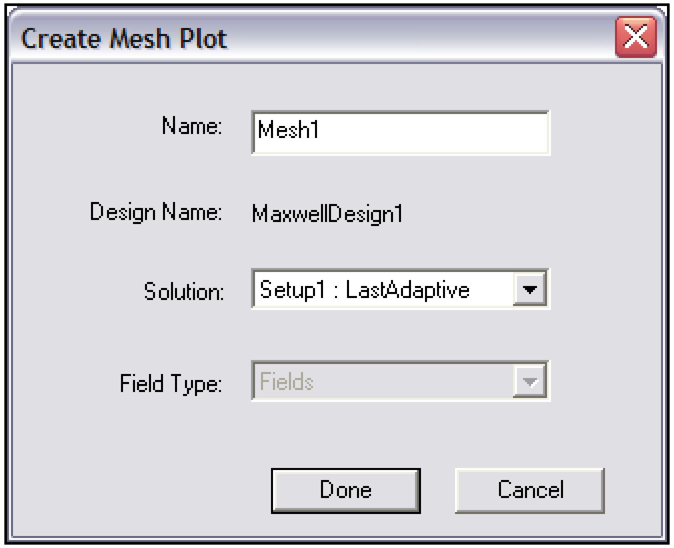
- If a mesh has been previously generated, Maxwell refines it using the defined mesh operations.
- If an initial mesh has not been generated, Maxwell generates it and applies the mesh operations to the initial mesh.
- If the defined mesh operations have been previously applied to the selected face or object, the current mesh is not altered.

  - Hint: When modifying an existing mesh operation, you should revert to initial mesh and then re-apply mesh operations.

- To view mesh information: Click ***Maxwell 3D > Results > Solution Data*** and click on the tab ***Mesh Statistics***

- ## Mesh Display
  1. Select an object
  2. Select the menu item ***Maxwell 3D > Fields > Plot Mesh***



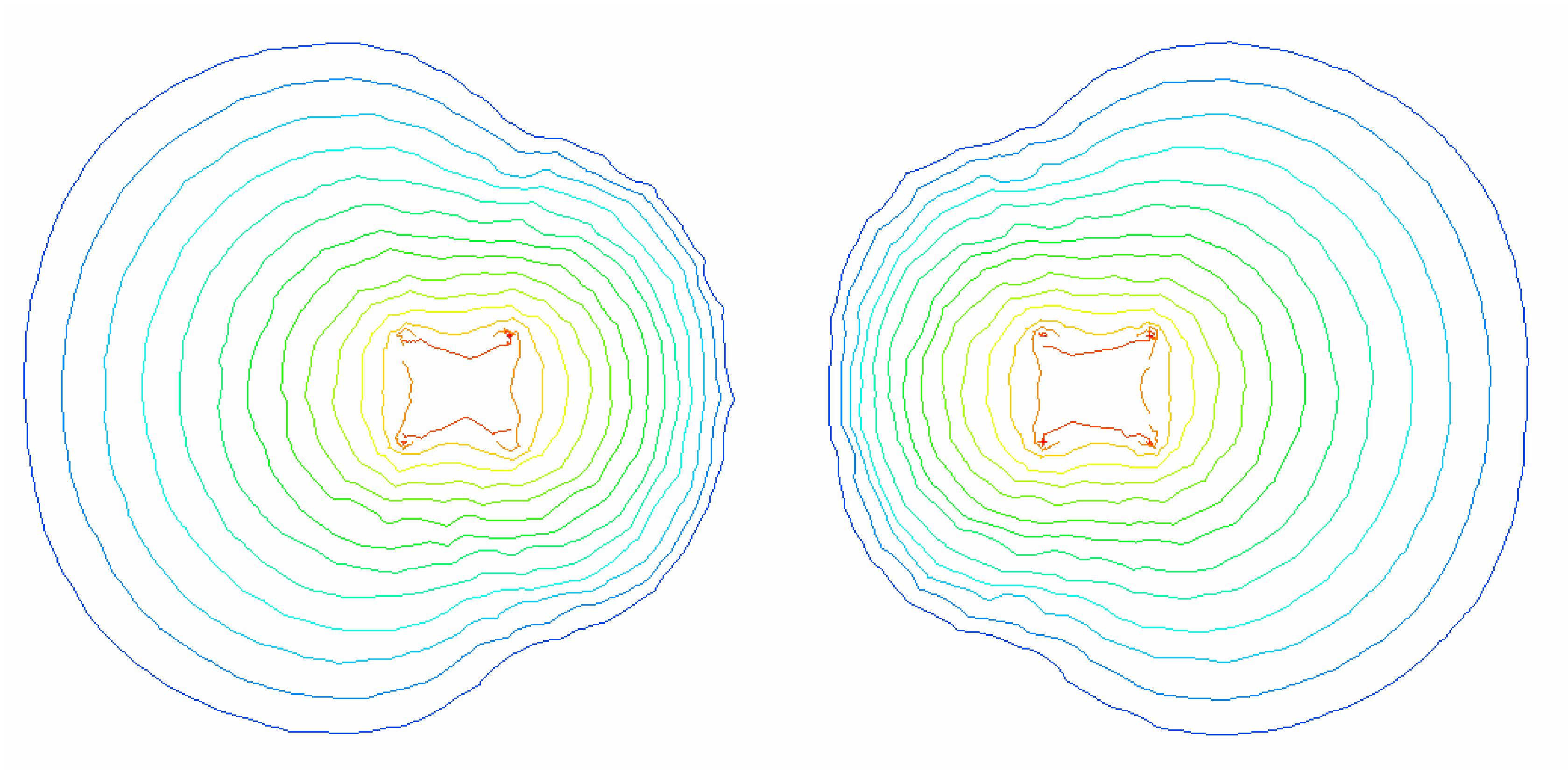


Post Processing

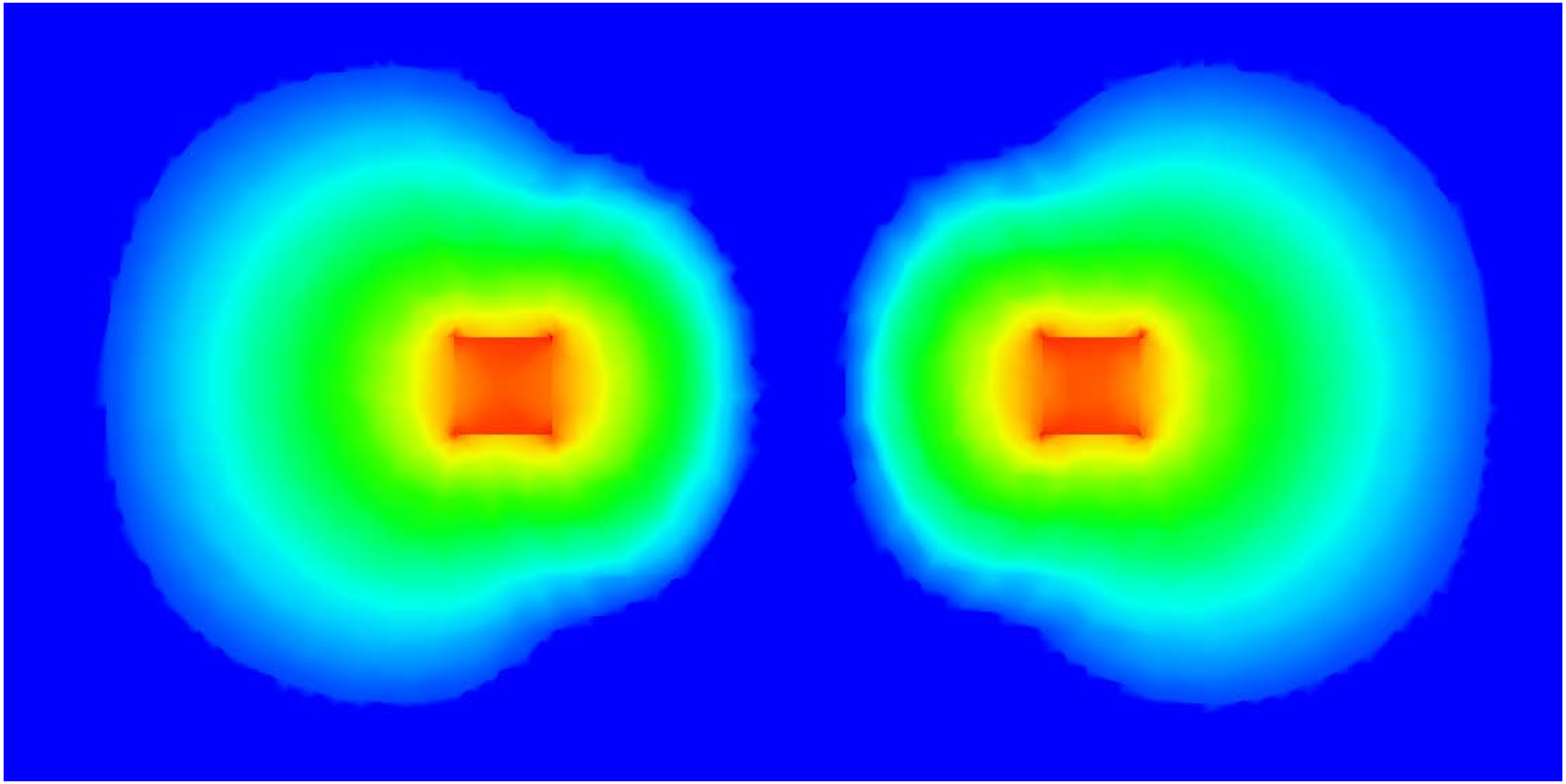
- Two Methods of Post Processing Solutions:
  - Viewing Plots
  - Manipulating Field Quantities in Calculator

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

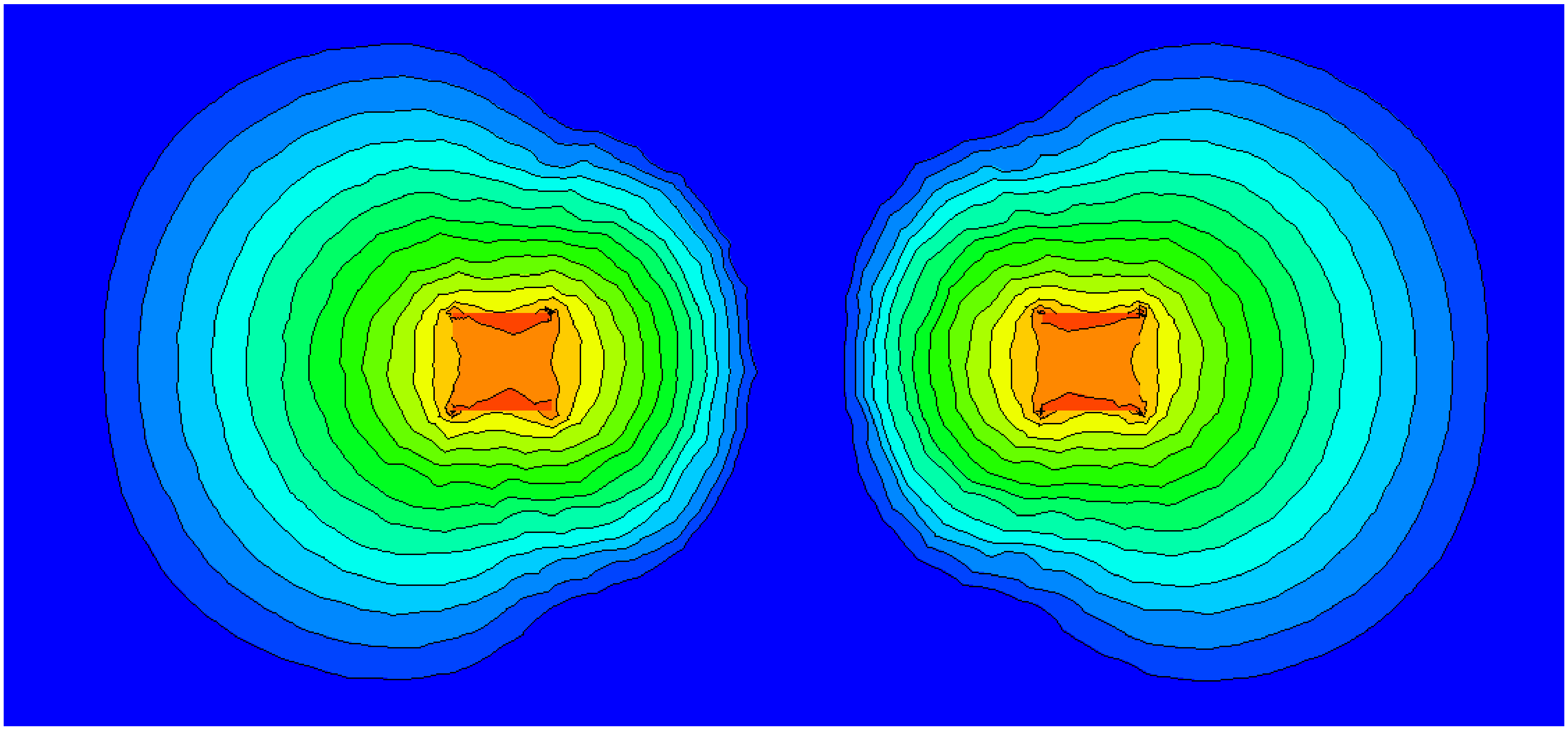
Contour plot



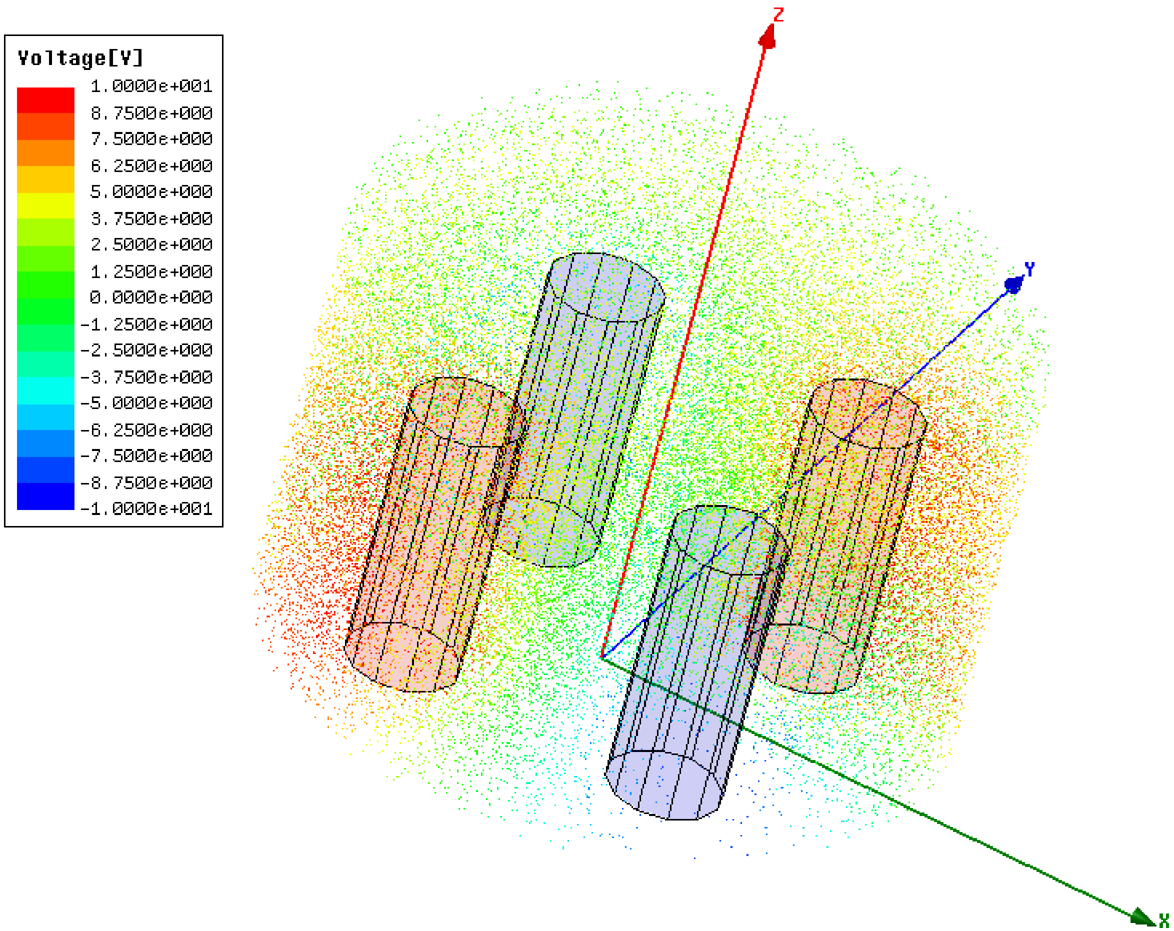
- Shade plot (tone)



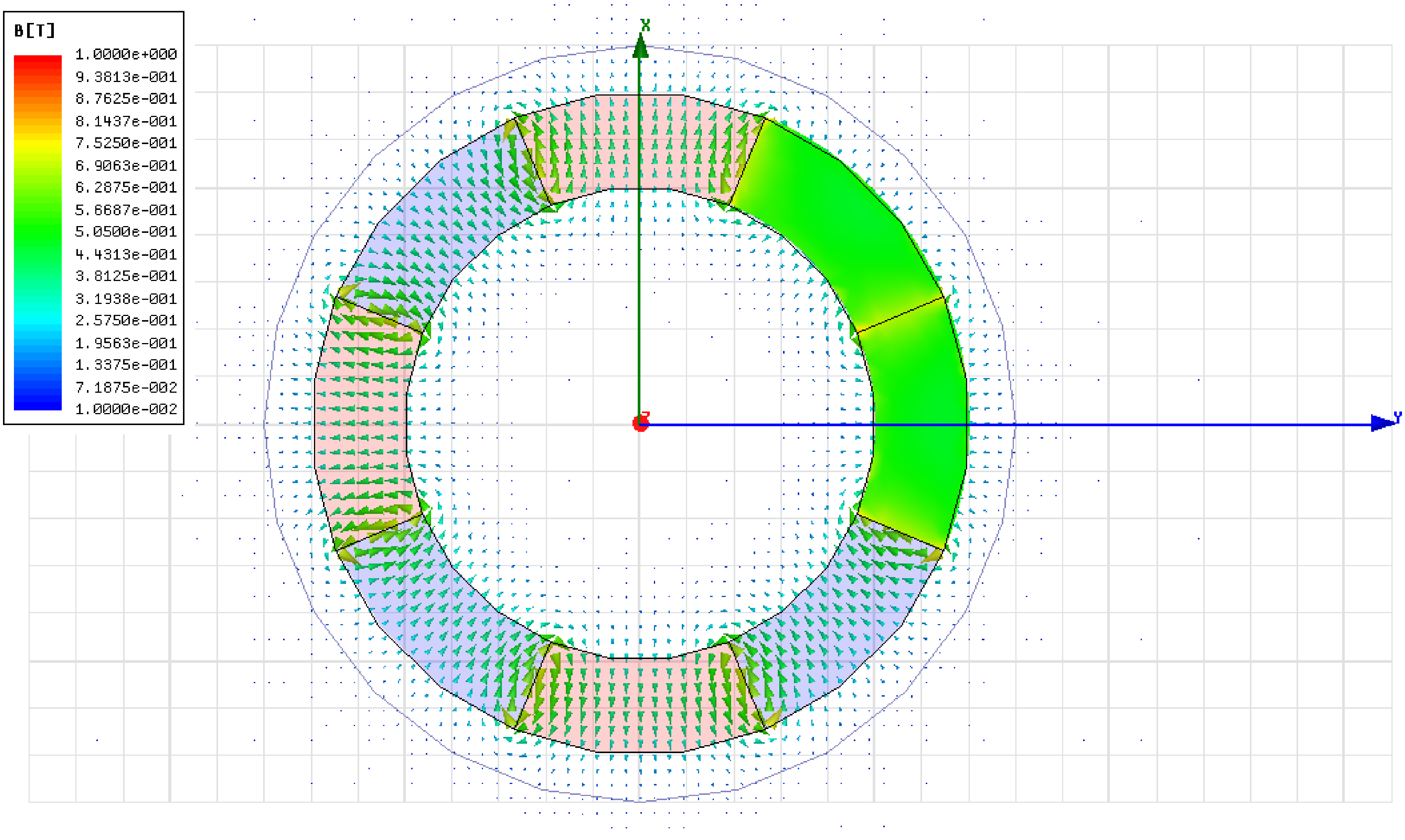
Shade plot (fringe with outline)



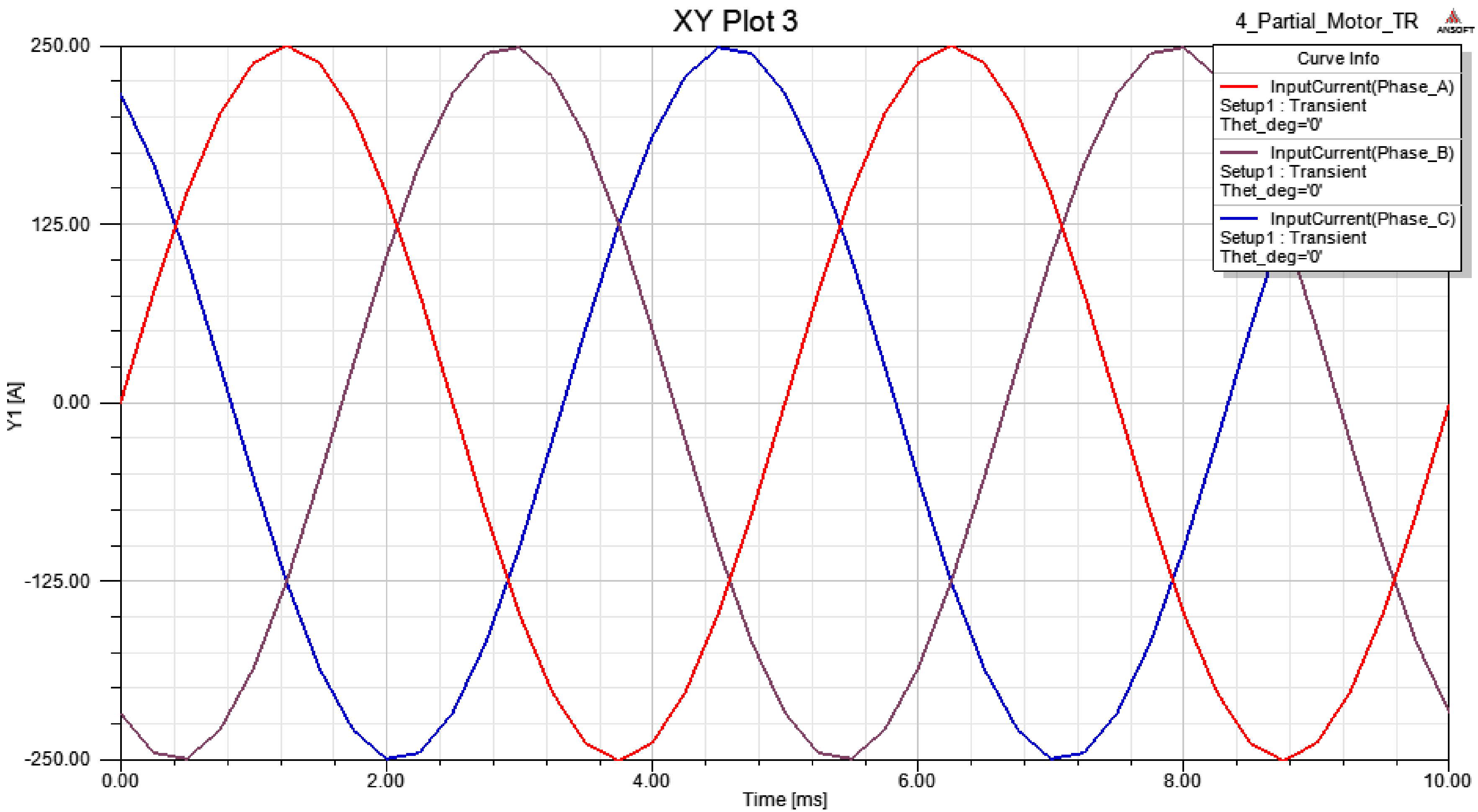
Scatter plot



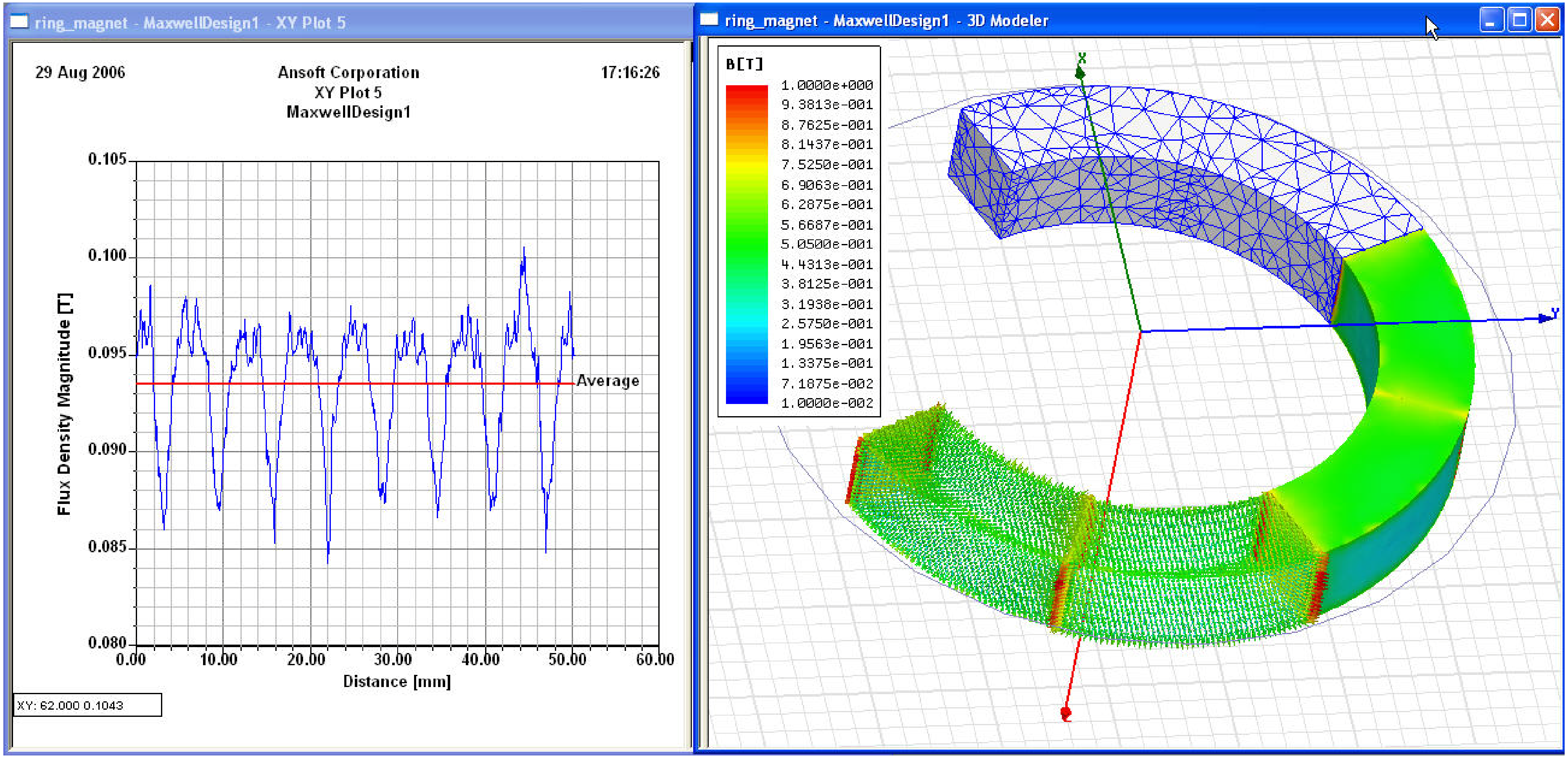
Arrow plot


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

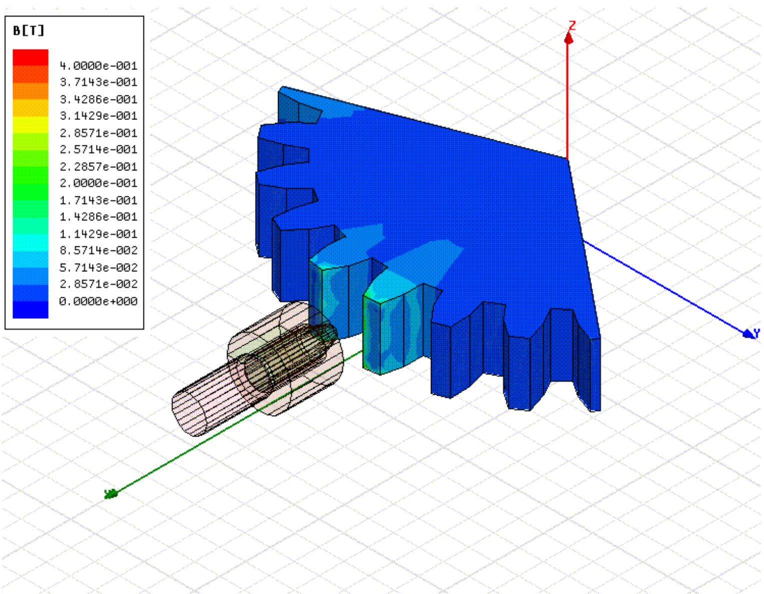
Line plot

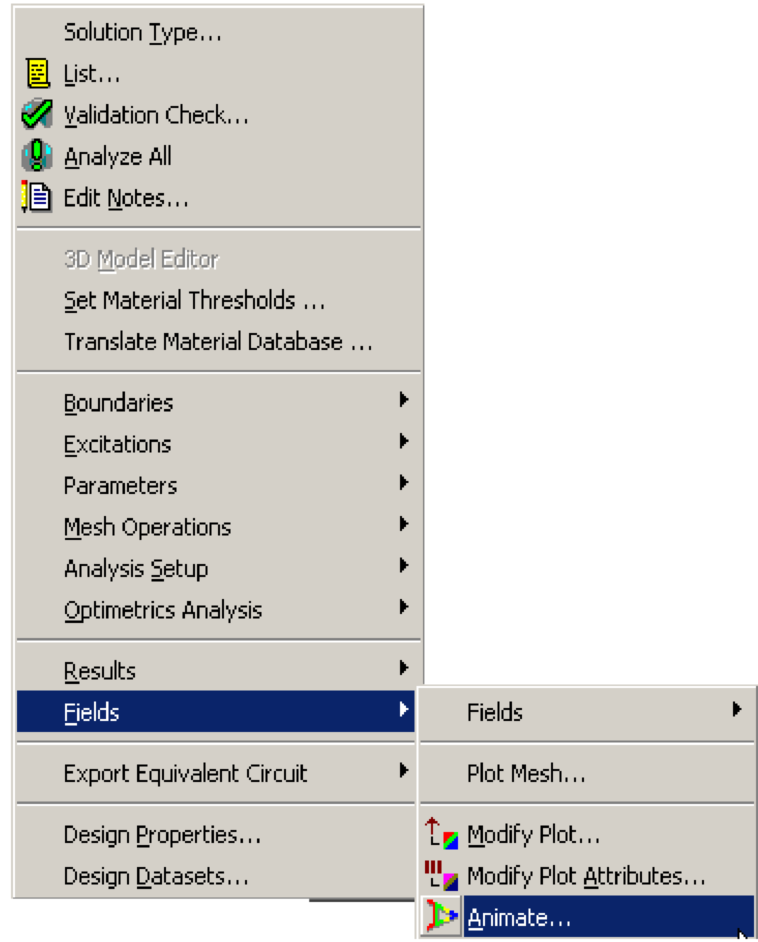

XY Plot 3
4_Partial_Motor_TR
ANSOFT
Curve Info
InputCurrent(Phase_A)
Setup1 : Transient
Thet_deg='0'
InputCurrent(Phase_B)
Setup1 : Transient
Thet_deg='0'
InputCurrent(Phase_C)
Setup1 : Transient
Thet_deg='0'
250.00
125.00
0.00
-125.00
-250.00
Y1 [A]
0.00
2.00
4.00
6.00
8.00
10.00
Time [ms]

- Multiple windows and multiple plots



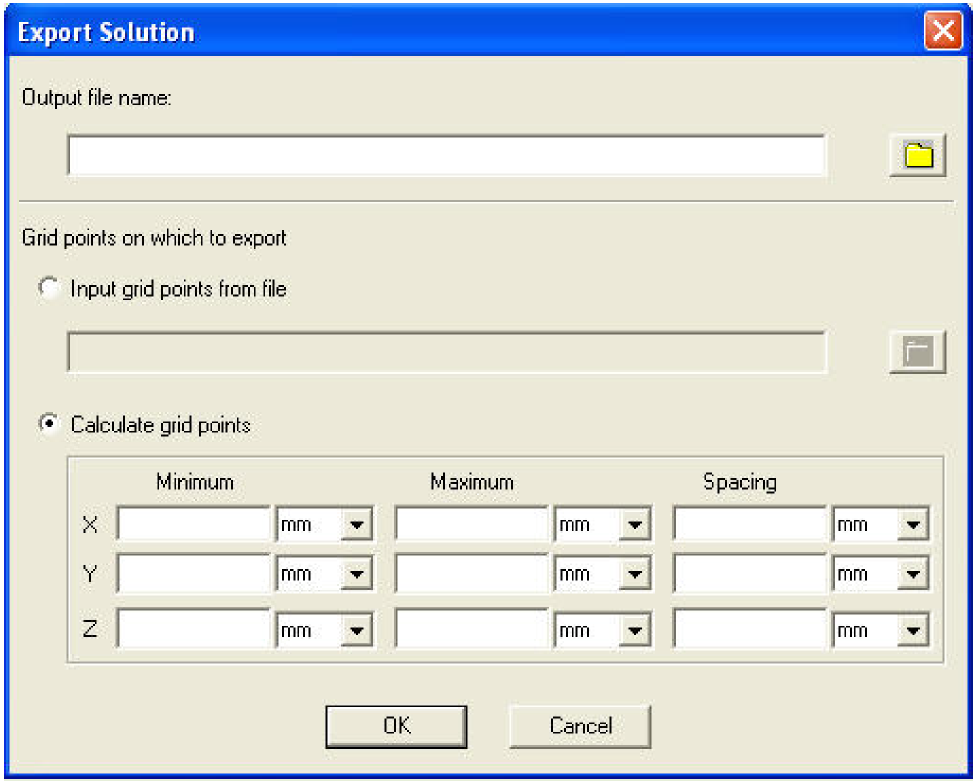
- Animation plot
  - Animate across different surfaces or with different phase angles (eddy solver)
  - Export to .gif or .avi format





- Fields Calculator
  - To bring up the Fields Calculator tool
    1. Select the menu item ***Maxwell 3D > Fields >Calculator***
  - Typical quantities to analyze:
    1. Flux through a surface
    2. Current Flow through a surface
    3. Tangential Component of E-field along a line
    4. Average Magnitude of B-field in a core
    5. Total Energy in an object

- ## Fields Calculator – Export Command
  - Exports the field quantity in the top register to a file, mapping it to a grid of points. Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available:
    1. Grid points from file: Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points.
    2. Calculate grid points: Maps the field quantity to a three-dimensional Cartesian grid. You specify the dimensions and spacing of the grid in the x, y, and z directions.



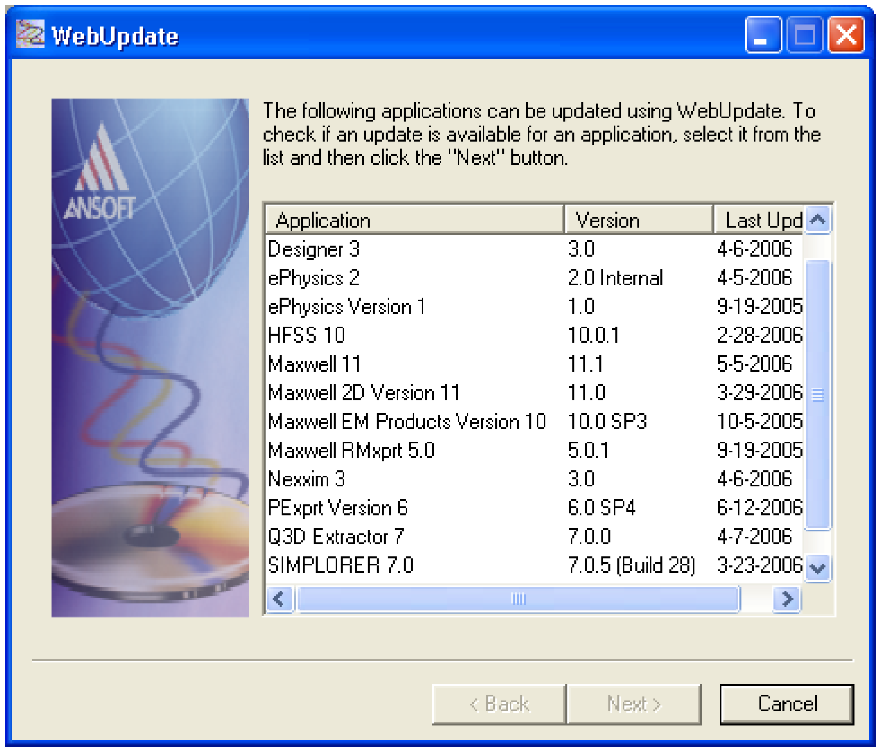
- Export to Grid
  - Vector data <Ex,Ey,Ez>
  - Min: [0 0 0]
  - Max: [2 2 2]
  - Spacing: [1 1 1]
  - Space delimited ASCII file saved in project subdirectory

```
Vector data "<Ex,Ey,Ez>"
Grid Output Min: [0 0 0] Max: [2 2 2] Grid Size: [1 1 1
0 0 0   -71.7231 -8.07776 128.093
0 0 1   -71.3982 -1.40917 102.578
0 0 2   -65.76 -0.0539669 77.9481
0 1 0   -259.719 27.5038 117.572
0 1 1   -248.088 16.9825 93.4889
0 1 2   -236.457 6.46131 69.4059
0 2 0   -447.716 159.007 -8.6193
0 2 1   -436.085 -262.567 82.9676
0 2 2   -424.454 -236.811 58.8847
1 0 0   -8.91719 -241.276 120.392
1 0 1   -8.08368 -234.063 94.9798
1 0 2   -7.25016 -226.85 69.5673
1 1 0   -271.099 -160.493 129.203
1 1 1   -235.472 -189.125 109.571
1 1 2   -229.834 -187.77 84.9415
1 2 0   -459.095 -8.55376 2.12527
1 2 1   -447.464 -433.556 94.5987
1 2 2   -435.833 -407.8 70.5158
2 0 0   101.079 -433.897 -18.5698
2 0 1   -327.865 -426.684 95.8133
2 0 2   -290.824 -419.471 70.4008
2 1 0   -72.2234 -422.674 -9.77604
2 1 1   -495.898 -415.461 103.026
2 1 2   -458.857 -408.248 77.6138
2 2 0   -470.474 -176.115 12.8698
2 2 1   -613.582 -347.994 83.2228
2 2 2   -590.326 -339.279 63.86
```

- ## Getting Help
  - If you have any questions while you are using Maxwell you can find answers in several ways:
    - **ANSYS Maxwell Online Help** provides assistance while you are working.
      - To get help about a specific, active dialog box, click the **Help** button in the dialog box or press the **F1** key.
      - Select the menu item ***Help > Contents*** to access the online help system.
      - **Tooltips** are available to provide information about tools on the toolbars or dialog boxes. When you hold the pointer over a tool for a brief time, a tooltip appears to display the name of the tool.
      - As you move the pointer over a tool or click a menu item, the **Status Bar** at the bottom of the ANSYS Maxwell window provides a brief description of the function of the tool or menu item.
      - The ANSYS Maxwell Getting Started guide provides detailed information about using Maxwell to create and solve 3D EM projects.
      - PDF version of help manual at: ***../Maxwell/Maxwell15/help/maxwell_onlinehelp.pdf*** for printing.
    - **ANSYS Technical Support**
      - To contact ANSYS technical support staff in your geographical area, please log on to the ANSYS customer portal: https://www1.ansys.com/customer/default.asp

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

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



- Application Support for North America
  - The names and numbers in this list may change without notice
    - Technical Support:
      - 9-4 EST:
        - **Pittsburgh, PA**
        - (412) 261-3200 x199

        - **Burlington, MA**
        - (781) 229-8900 x199

      - 9-4 PST:
        - **San Jose, CA**
        - (408) 261-9095 199

        - **Portland, OR**
        - (503) 906-7946 x199

        - **Irvine, CA**
        - (714) 417-9311 x199

Solution Types & Solvers

- This chapter describes the basic generalities of the six different solution types in Maxwell3D. With each solution type is a specific solver - each has its own setup and solution method, yet they can often be integrated to produce powerful solutions. We will discuss the fundamentals of the finite element solution, boundary conditions, excitations and solution setup in this chapter. Then the technical substance for each solver is explained in the next sections.

- There are six different solvers, as mentioned before. They are:
  - Magnetostatic
  - Eddy Current
  - Transient Magnetic
  - Electrostatic
  - DC Conduction
  - Transient Electric

- The last Electrostatic and DC Conduction solvers are joined together because they solve in distinct solution spaces (the DC Conduction in the conductors, and the Electrostatic in the insulators), yet the solution from the DC Conduction solver can be used as a source in the Electrostatic solver (this will be discussed in detail in the Electrostatic and DC Conduction sections).

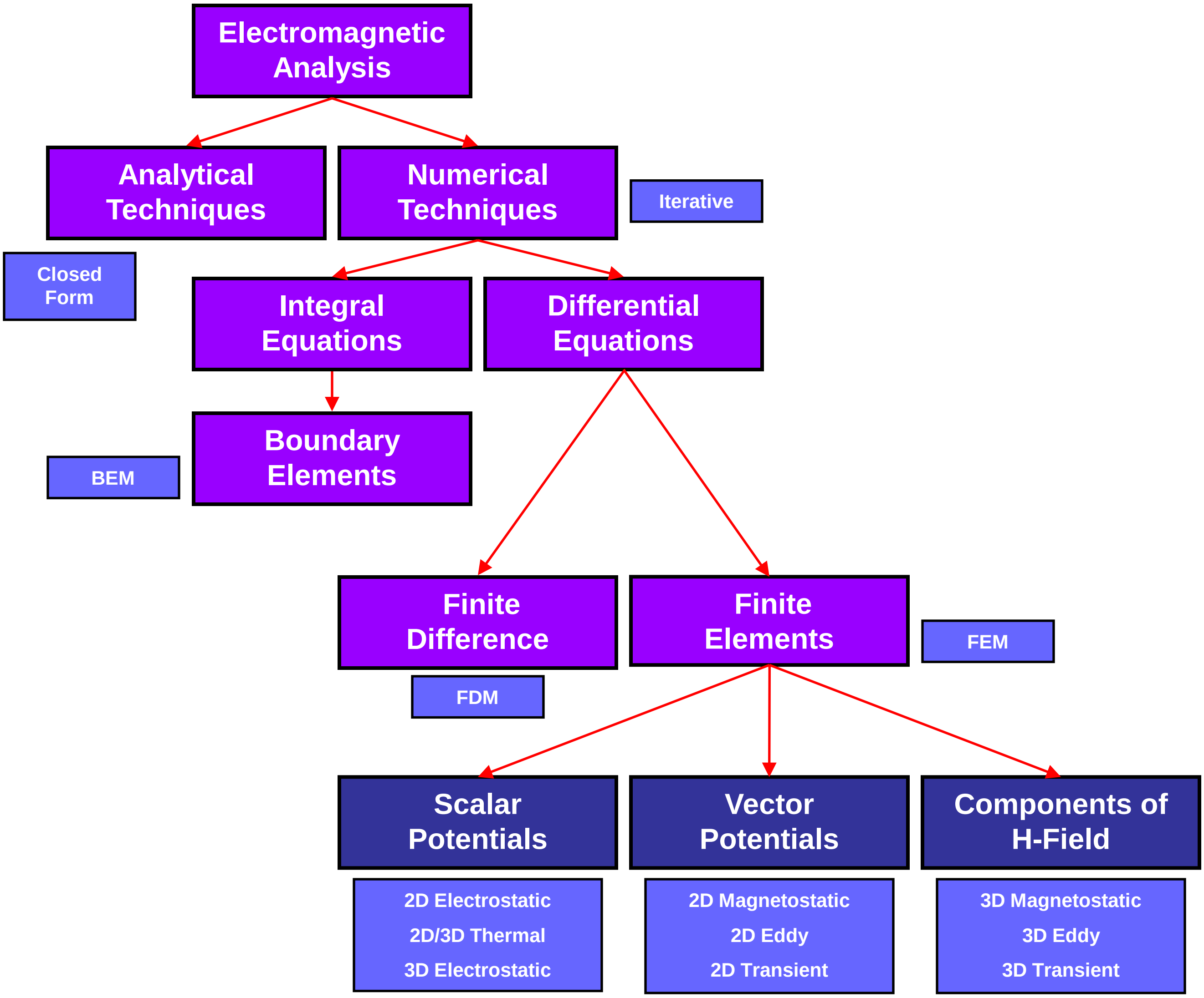
- The Magnetostatic, Eddy Current, Electrostatic, and DC Conduction solvers (every solver except for the Transient solvers) perform an adaptive mesh solution. This means that the finite element mesh is intelligently improved from one adaptive pass to the next. The mesh will be discussed in the Mesh Operations section – including methods to gain benefits of adaptive meshing in a Transient simulation.

- These solvers are designed to solve Maxwell's equations within the scope of the specific solution type.

General Finite Element Method Information

- The Finite Element Method is one of many accepted methods of numerically solving complicated fields where analytically solutions are not sufficient.

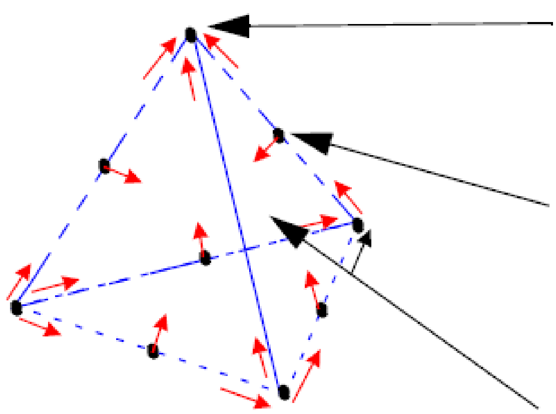
- Below is a chart displaying different methods of electromagnetic analysis – each method with its own strengths and weaknesses. Finite elements have proven to be very robust for general electromagnetic analysis.



General Finite Element Method Information (Continued)

- Finite element refers to the method from which the solution is numerically obtained from an arbitrary geometry. The arbitrary, complicated geometry is broken down into simple pieces called finite elements.

- In Maxwell3D, the fundamental unit of the finite element is a tetrahedron (four-sided pyramid). These elements are constructed together in what is known as the finite element mesh.



The Components of a Field that are tangential to the edges of an element are explicitly stored at the vertices.

The Components of a field that is tangential to the face of an element and normal to an edge is explicitly stored at the midpoint of selected edges.

The Values of a vector field at an interior point is interpolated from the nodal values.

- Equilateral tetrahedra work best for the 2nd order quadratic interpolation that is used between nodes. It is good to keep in mind that a relatively uniform, equilateral mesh is often desired - this will be discussed in more detail in the Mesh section.

- The desired field in each element is approximated with a 2nd order quadratic polynomial (basis function):
  - $H_x(x,y,z) = a_0 + a_1x + a_2y + a_3z + a_4xy + a_5yz + a_6xz + a_7x^2 + a_8y^2 + a_9z^2$

- In order to obtain the basis functions, field quantities are calculated for 10 points in a 3D simulation (nodal values at vertices and on the edges).

- All other quantities are determined from the field solution in part or in all of the solution space.

General Finite Element Method Information (Continued)

- Once the tetrahedra are defined, the finite elements are placed in a large, sparse matrix equation.

$$[S][H]=[J]$$

- This can now be solved using standard matrix solution techniques such as:
  - Sparse Gaussian Elimination (direct solver)
  - Incomplete Choleski Conjugate Gradient Method (ICCG iterative solver)
- It should be noted that the direct solver is the default solver, and is generally the best method. The ICCG solver is included for special cases and for reference.

Error Evaluation

- For each solver, there is some fundamental defining equation that provides an error evaluation for the solved fields. In the case of the magnetostatic simulation, this defining equation is the no-monopoles equation, which says:

$$\nabla \cdot \vec{B} = 0$$

- When the field solution is returned to this equation, we get an error term:

$$\nabla \cdot \vec{B}_{solution} = err$$

- The energy produced by these error terms (these errors act in a sense like sources) is computed in the entire solution volume. This is then compared with the total energy calculated to produce the percent error energy number.

$$percent\,error\,energy = \frac{error\,energy}{total\,energy} \times 100\%$$

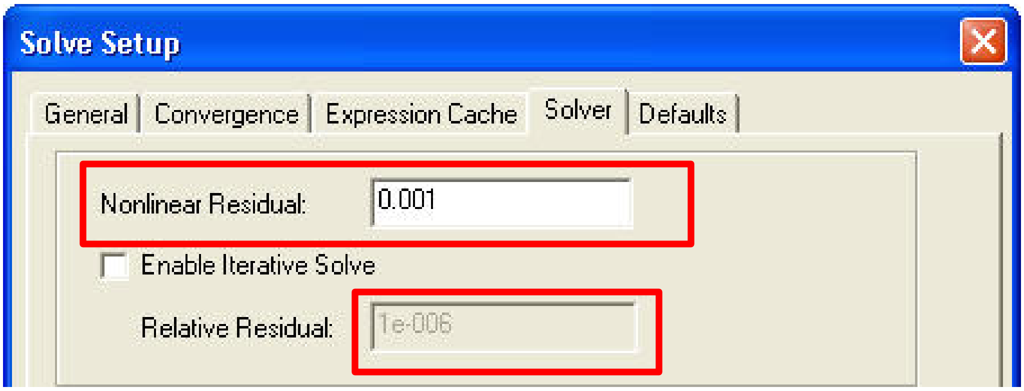
- This number is returned for each adaptive pass along with the total energy, and these two numbers provide some measure of the convergence of the solution.

- Remember that this is a global measure of the convergence – local errors can exceed this percentage.

Solver Residuals

- The solver residuals specify how close a solution must come before moving on to the next iteration.

- There are two types of residuals:
  - Nonlinear – used only for problems with nonlinear BH terms.
  - Linear – used only with the ICCG iterative matrix solver.



- The nonlinear residual is set to 0.001 by default for the Magnetostatic simulator and to 0.005 by default for the Transient simulator.
- The linear residual is set to 1e-5 by default for all solution types.

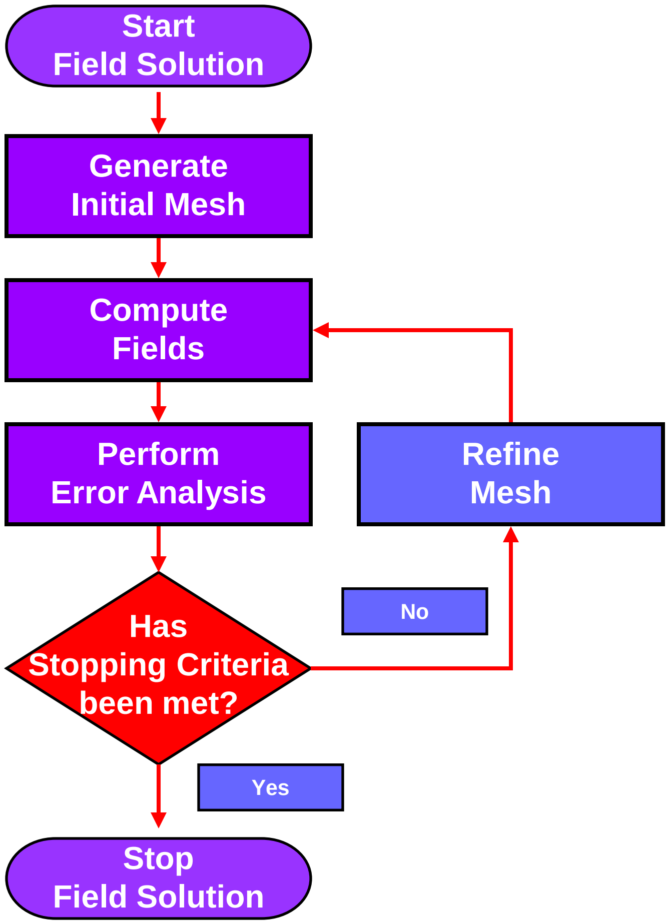
- Decreasing either residual will increase the simulation time, but it will improve the accuracy of the solution to some degree.

- **It is very important to decrease the nonlinear residual if you perform simulations with saturating materials. This will affect your solution.**

Adaptive Refinement Process

- In the Magnetostatic, Eddy Current, Electrostatic, and DC Conduction solvers (every solver except for the Transient solver), the finite element mesh is adaptively redefined to reduce the energy error.
- The solution continues until one of two stopping criteria is met:
  1. The specified number of passes are completed

     OR
  2. Percent error energy AND delta energy are less than specified



Boundary Conditions

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

Why they are Important

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

Common Boundary Conditions

- There are three types of boundary conditions. The first two are largely the user's responsibility to define them or ensure that they are defined correctly. The material boundary conditions are transparent to the user. The following are examples of each type of boundary condition from various solvers.
  - **Excitations**
    1. Current
    2. Voltage
    3. Charge
  - **Surface Approximations**
    1. Symmetry Planes
    2. Master/Slave Boundaries
    3. Insulating Boundary
    4. Default outer boundary
  - **Material Properties**
    1. Boundary between two materials

How the Background Affects a Structure

- The background is the volume that surrounds the geometric model and fills any space that is not occupied by an object. Any object surface that touches the background is automatically defined to be a flux tangential boundary. This forces the flux to stay within the solution region.

- If it is necessary, you can change a surface that is exposed to the background to have properties that are different from the default:
  - In a magnetostatic simulation, to model a field imposed on the region, you can assign tangential H fields to the sides of the region. This imposes a directional field that simulates the response of the structure to an external H field.
  - In an eddy current simulation, to model a surface to allow waves to radiate infinitely far into space, redefine the surface to be radiation boundary.
  - To reduce the model size by splitting the structure in half (or into a periodic unit), use a symmetry boundary (or master/slave boundaries).

- The background can affect how you define the region surrounding your structure. Since it is often desired not to strictly enforce a tangential flux boundary close to the structure (the field is not necessarily tangent near to the structure), the region will have to be defined large enough that fields have largely diminished at the boundary of the region. By making the region large enough, the solution is not perturbed by this finite approximation. This is very important for structures that have large stray fields. However, if most of the fields are guided so that the outside of the region is appropriately shielded, then the region can be smaller to more tightly fit the structure.

Region Object

- Every solver (except the DC conduction solver) needs to have a large region containing all objects. This is important so that everything is defined within one solution domain (the DC conduction solver can have multiple solution domains).
- There is a method of automatically creating a region – select ***Draw > Region*** or select the  icon and input some padding data.
- Keep in mind the above information about the background (it is the volume outside of the region) when defining the padding data of the region.

Boundary Condition Precedence

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

Boundary Conditions in different solvers

- There are different boundary conditions available in each solver, as each solver has a slightly different setup, process, and purpose. The boundary conditions can be assigned in the listed solvers.

- **Zero Tangential H Field** (Magnetostatic , Eddy Current, Transient)
- **Tangential H Field** (Magnetostatic , Eddy Current,)
- **Insulating** (Magnetostatic , Eddy Current, , Transient, DC Conduction)
- **Symmetry** (Magnetostatic , Eddy Current, , Transient, Electrostatic)
- **Master** (Magnetostatic , Eddy Current, , Transient, Electrostatic)
- **Slave** (Magnetostatic , Eddy Current, , Transient, Electrostatic)
- **Radiation** (Eddy Current)
- **Impedance** (Eddy Current)

- The following sections will describe the boundary conditions available in each solver and what they are used for in each case.

Excitations

- Excitations are a unique type of boundary condition that define the sources of the magnetic and/or electric fields. Excitations are general sources – like from Maxwell's equations (current for magnetic fields and charge or voltage for electric fields). Source definitions are directly related to the magnitude and behavior of the field solution.

- There must be at least one source in the simulation (the total energy must be non-zero) for the solver to run. Along with sources found in the excitations for each solution type, boundaries that impose a field or objects that are permanently magnetized act as sources for the field solution.

- For current excitations in the magnetic field solvers it is necessary to solve a separate conduction solution before the field solution starts – this is automatically integrated into the solution process.

- There are two types of excitation assignments – internal excitations and external excitations. Internal excitations can be applied on 2D sheets (usually the cross-section of a conductor) or the surface of objects (in the case of an electrostatic voltage assignment) and are within the solution region. External excitations are often assigned to the face of objects on the edge of the solution region – these usually require multiple excitations to define input and output or difference quantities.

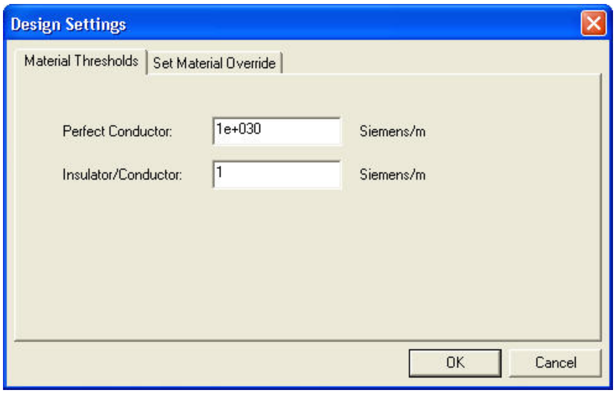
- When external sources are used and exactly two excitations are defined, one excitation will be defined pointing into the conductor and the other will be pointing out of the conductor. The excitation that is pointing into the conductor will be referenced in the matrix parameter assignment and in the solutions.

- When more than two external sources are assigned to one conduction path, an inductance calculation cannot be performed for that conductor path.

- The excitations available are described in the following sections for each individual solution type.

Material Thresholds

- There are two Material Thresholds that can be adjusted for special purposes.
- To adjust these thresholds select ***Maxwell 3D > Design Settings***



- Every magnetic solver has some sort of conduction solver built in (the conduction process is different depending on the type of source used and the solver). This part of the solver determines the current distribution in the conducting solids concurrently with the magnetic field solution (as they both effect each other when eddy currents are taken into consideration). The setting that controls whether an object is considered in the conduction solve or not is an object's conductivity in relation to the **Insulator/Conductor material threshold**. If the object is more conductive than this threshold, the object will be considered as a conductor (able to carry current). If the object is less conductive than this threshold the object will be considered as an insulator (not able to carry current). Different solvers treat conductors differently (i.e. conductors can carry eddy currents in the eddy current and transient solvers), so this threshold can be important in some circumstances. However, this setting is usually set to a value that is applicable for most simulations (the default value is 1 Siemen/meter). This insulator/conductor threshold is also applicable to electric solutions, as objects above this threshold are considered to be perfect conductors (electrostatically) and current carrying (DC conduction).

- The **Perfect Conductor material threshold** is a similar conduction setting. Every solver treats this setting differently (not available in the electrostatic solver). Generally, the solution is not performed in objects that have a conductivity above this threshold, but ideal surface currents are assumed.

- Defaults can also be entered for these numbers by going to ***Tools > Options > Maxwell 3D Options…*** and looking in the General Options tab.

General Solution Setup Procedure

- As discussed before, here is a general solution setup procedure for users new to electromagnetic field simulation:

  1. Based on your application, choose the type of electromagnetic analysis to be performed.
  2. Draw the geometry of the model using the drawing space provided by the 3D Modeler menu and Draw menu commands available through the Maxwell desktop interface.
  3. Assign the material properties to all solid objects in the model, and define new material properties if materials in the default library do not provide the needed material.

     Note: Always make sure the material properties assigned to an object correspond to the real properties of the materials in the electromagnetic device that is being simulated. Material properties supplied in the default library are generic properties and may not always be substituted for actual properties.

  1. Specify the field sources (excitations) and boundary conditions for your unique solution.
  2. Define additional global parameters that you want to calculate (such as force, torque, inductance/capacitance, etc.).
  3. Define mesh operations for special applications (such as seeding in areas/objects of interest).
  4. Specify solution options.
  5. Start the solution process.
  6. When the solution becomes available, perform post processing, such as plotting field quantities and calculating expressions.

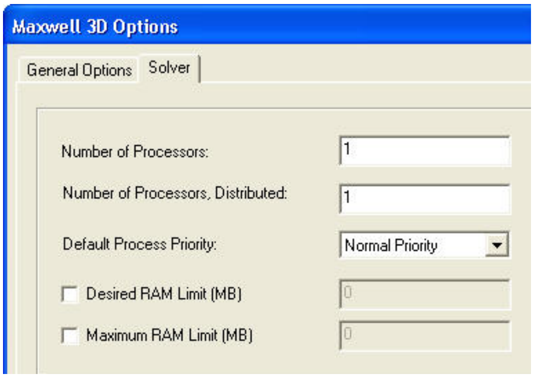
- This procedure works equally well for all solution types. The technical details relevant to each solution type will be discussed in the following sections.

Analyze

- There are many different methods to start an analysis – depending on what all you wish to analyze.
- To analyze a single analysis setup, right-click on the setup name and choose **Analyze**.
- To analyze a single optimetrics setup, right-click on the setup name and choose **Analyze**.
- To analyze all analysis setups, right-click on **Analysis** and choose **Analyze All**.
- To analyze all optimetrics setups, right-click on **Optimetrics** and choose **Analyze > All**.
  - To analyze all parametric setups, right click on **Optimetrics** and choose **Analyze > All Parametric**.
  - The same procedure will work for Optimization analyses, Sensitivity analyses, and Statistical analyses.
- To analyze all setups within a design, there are three options:
  - Right-click on the design name and choose **Analyze All**.
  - Click on the icon from the toolbar.
  - Select ***Maxwell 3D > Analyze All***.
- This procedure will analyze all analysis setups and then all optimetrics setups within the active design.
- To analyze all setups within a project, there are two options:
  - Right-click on the project name and choose **Analyze All**.
  - Select ***Project > Analyze All***.
- This procedure will analyze all analysis setups and optimetrics setups within each design of the active project.

Improved Analysis Configuration

- *Tools > Options > Maxwell 3D Options* and select the **Solver** tab
  - Configure **Number of Processors** (Requires a multi-processor license)
  - Configurable **Default Processor Priority** (Normal Priority is the default)
  - Soft and Hard RAM limits can be controlled here.

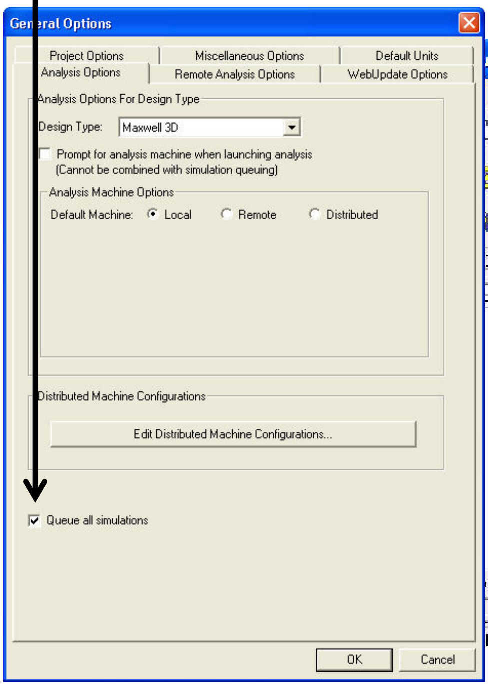


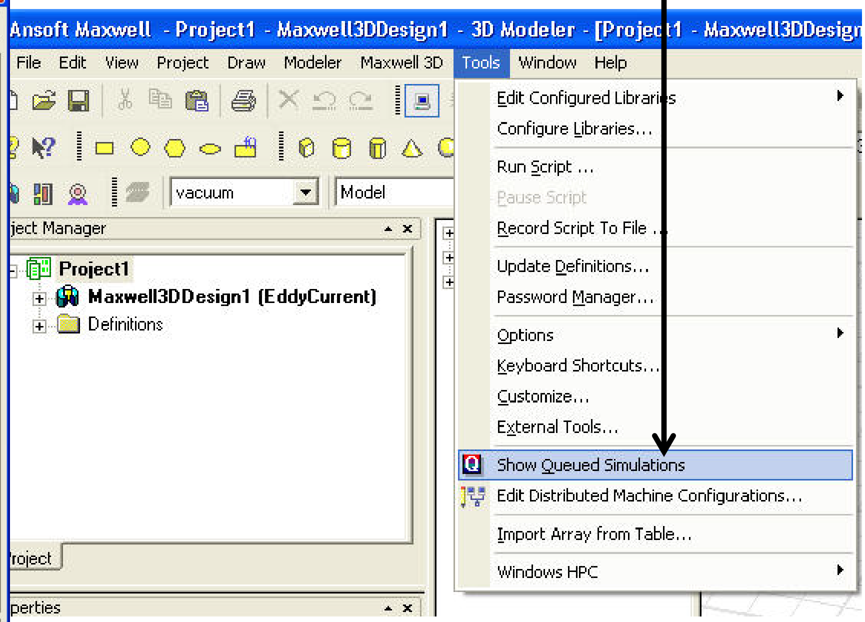
Remote Analysis

- Heterogeneous platform support

Queuing

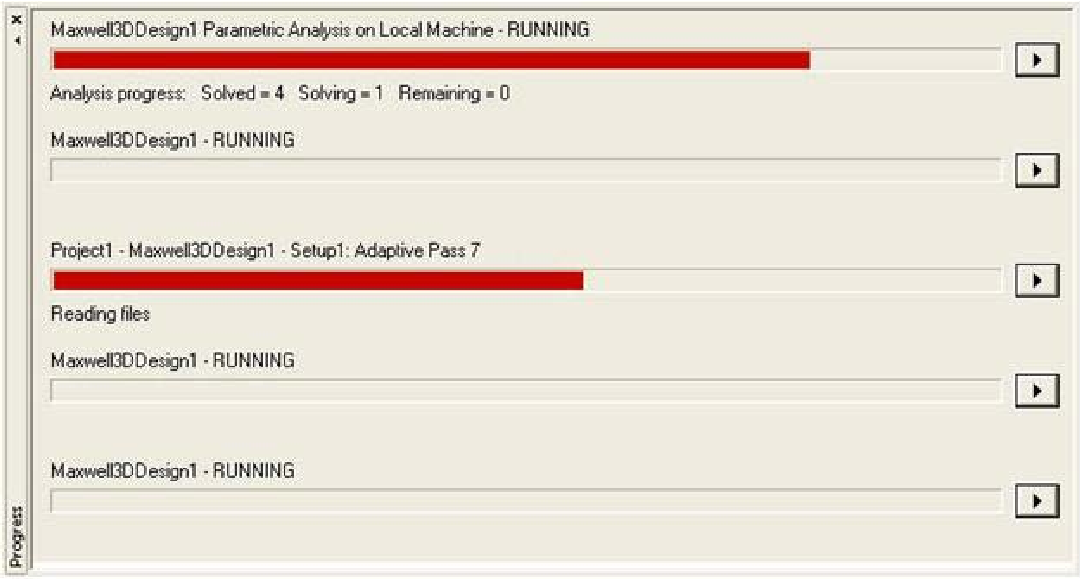
- Queue Projects, Designs, Parametric Sweeps, Frequency Sweeps
- To enable this feature, go to **Tools > Options > General Options > Analysis Options**
- To view queued projects, go to **Tools > Show Queued Simulations**





Distributed Analysis

- 10X Analysis Speed-Up per distributed solve license – it can run as many projects as you have processors.
  - An automated client-server implementation
    - little/no overhead
    - setup easily via remote analysis capability
  - For Optimetrics instances
    - Set the Distributed Machine configuration
    - Set the Analysis to Distributed Mode using the toolbar



Magnetostatic Analysis

- Magnetostatic Analysis is performed by choosing the Magnetostatic solution type.

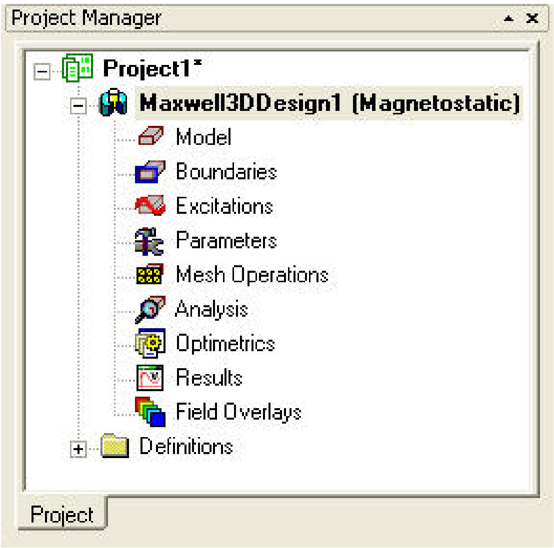
- Applications that use Magnetostatic Analysis can be solenoids, inductors, motors, actuators, permanent magnets, stray field calculations and many others.

Overview

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

Setup

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

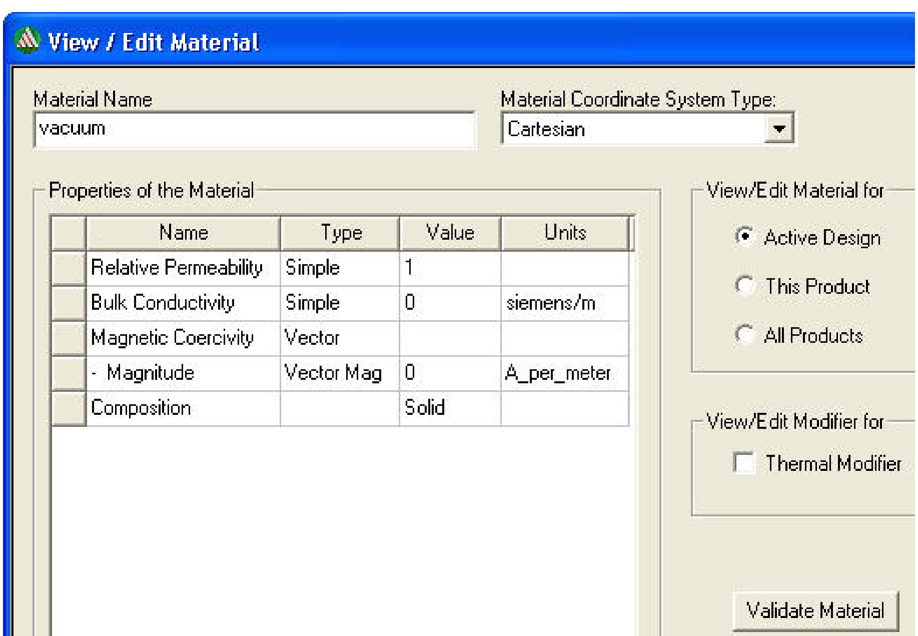


1. The **Model** definition refers to the geometry and material definition.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in a magnetostatic simulation.
3. **Parameters** are values that the solver will automatically calculate after finding the field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

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

Magnetostatic Material Definition

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



Magnetostatic Boundary Conditions

- **Default** – The default boundary conditions for the Magnetostatic solver are:
  - **Natural** boundaries on the interface between objects.
    - This means that the H Field is continuous across the boundary.
  - **Neumann** boundaries on the outer boundaries.
    - This means that the H Field is tangential to the boundary and flux cannot cross it.

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

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

Magnetostatic Boundary Conditions (Continued)

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

- **Symmetry** – There are two magnetic symmetries – odd symmetry (flux tangential) and even symmetry (flux normal). Odd symmetry defines H to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Even symmetry defines H to be normal to the boundary (this is equivalent to the Zero Tangential H Field boundary condition). Remember that geometric symmetry may not mean magnetic symmetry in all cases. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Other considerations for a Symmetry boundary condition:
  - A plane of symmetry must be exposed to the background.
  - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
  - A plane of symmetry must be defined on a planar surface.
  - Only three orthogonal symmetry planes can be defined in a problem

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

Magnetostatic Excitations

- Typical sources for magnetostatic field problems include voltage, current, and current density. When applying the sources for the magnetic field problems, the applied current distribution must be divergence free in the entire space of the solution as it is physical for quasi-stationary conduction current density distributions. Thus, the conduction path(s) for the applied current distributions must be closed when totally contained within the solution space for the problem, or must begin and end at the boundaries.

- Permanent Magnets and externally applied magnetic fields can also act as sources for a magnetostatic analysis, however, those are defined elsewhere.

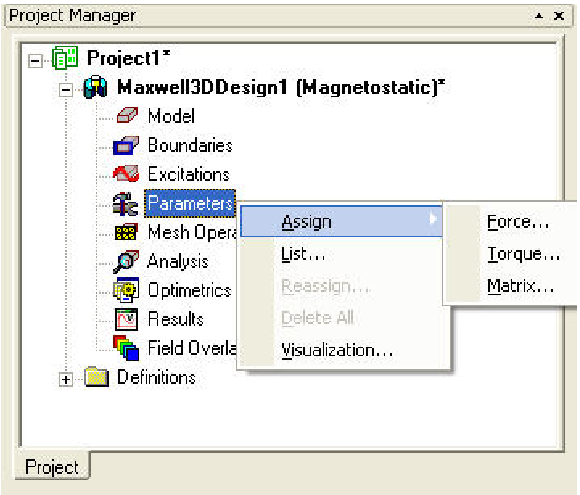
- **Voltage Excitations -** These are used in conjunction with the material conductivity to define the current through a solid conductor. Either multiple Voltage excitations can be used to define a voltage difference across two faces of a conductor (creating a current) or a Voltage Drop can be defined on a 2D sheet object to signify the voltage drop around a conductive loop.

- **Current Excitations** - This excitation can be assigned on any conductor to define the total current (amp-turns) through the cross-section of a loop or to define the current into and out of the opposing, external faces of a conducting object. This is a very general purpose excitation that comes in two flavors – Solid or Stranded. More information about Stranded Magnetostatic Current excitations (along with an example and explanation) can be found in the Magnetostatic Switched Reluctance Motor example.

- **Current Density** - These excitations are used to define a known current density throughout an object and must be used with a Current Density Terminal. The Current Density is defined on the 3D object, and the Terminal is defined on either an internal cross-sectional sheet, or on all external cross-sectional faces.

Parameters

- There are three parameters that can be automatically calculated in a Magnetostatic simulation – **Force**, **Torque**, and Inductance **Matrix**.



- All three quantities are computed directly from the magnetic field solution.
- Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).
- Inductance Matrix has many post-processing options available – this is discussed in detail in the Magnetostatic Inductance Matrix example.
- The results of any parameters can be found by selecting ***Maxwell 3D > Results > Solution Data…*** or by clicking on the icon.
- Further results can be obtained manually through the field calculator.

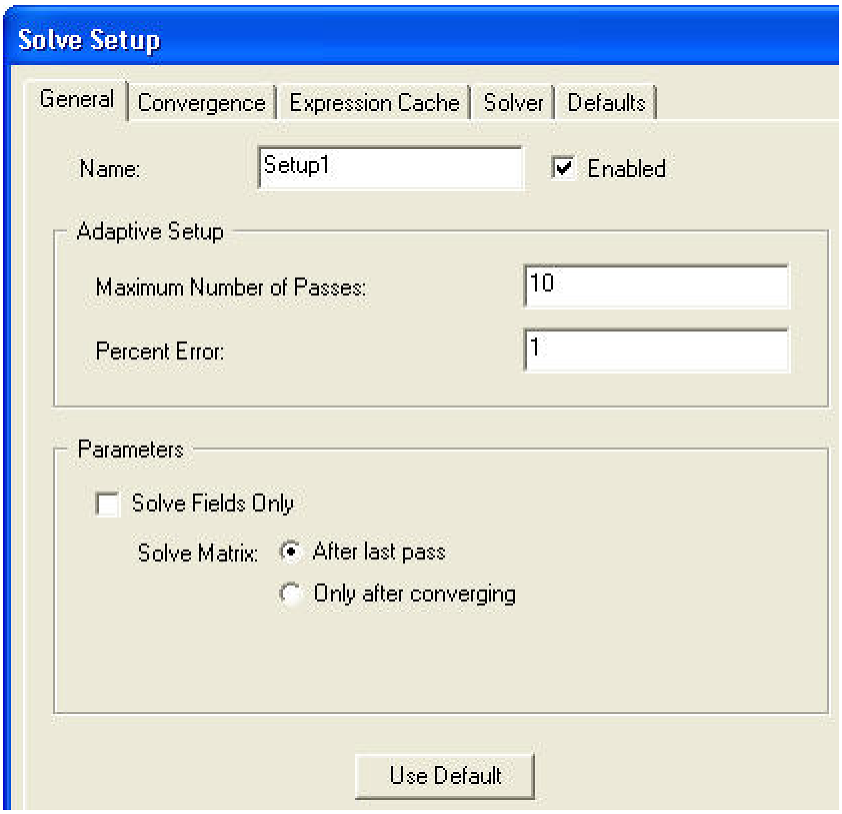
Mesh Operations

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

Solution Setup

- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.

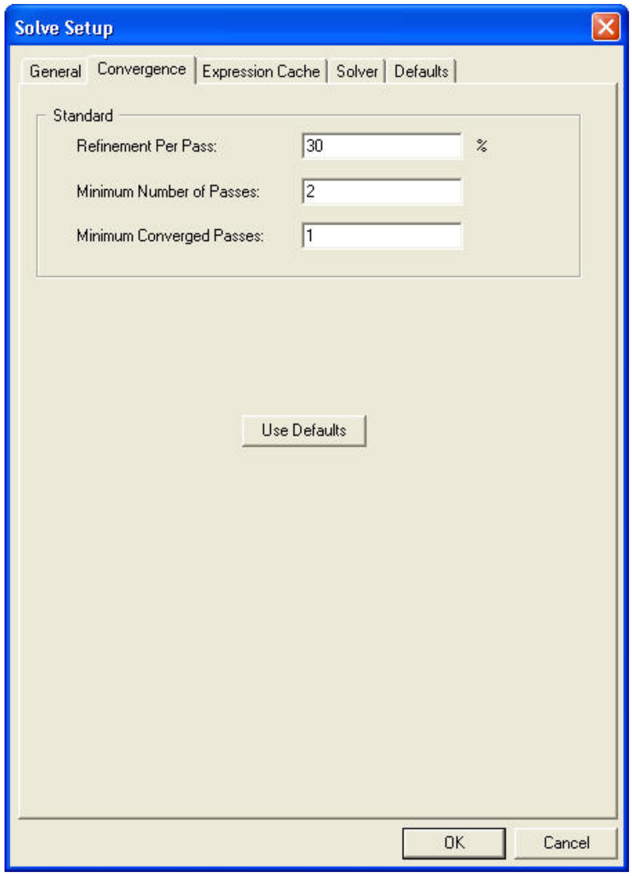
- The following window appears with the General Setup parameters.



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

Solution Setup (Continued)

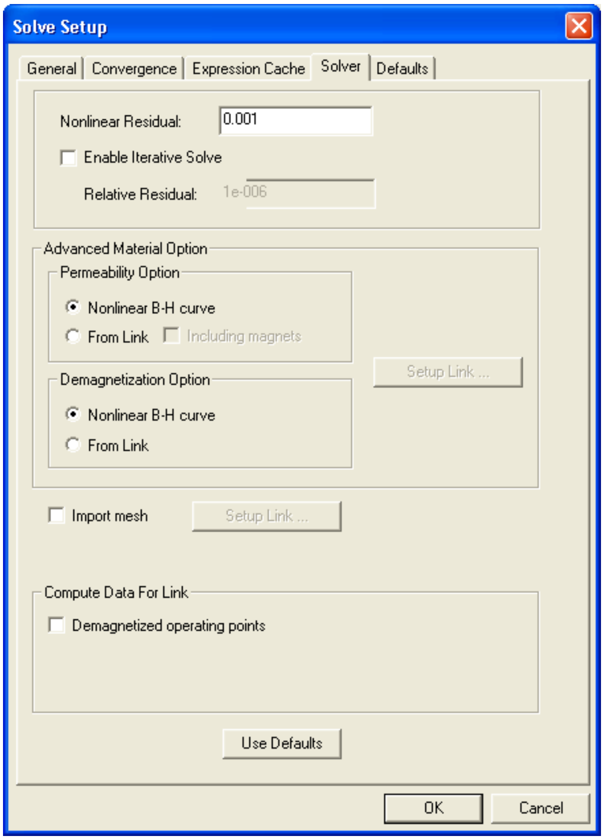
- The second tab of the Solution Setup contains information about Convergence.



- **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
- **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
- **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).

Solution Setup (Continued)

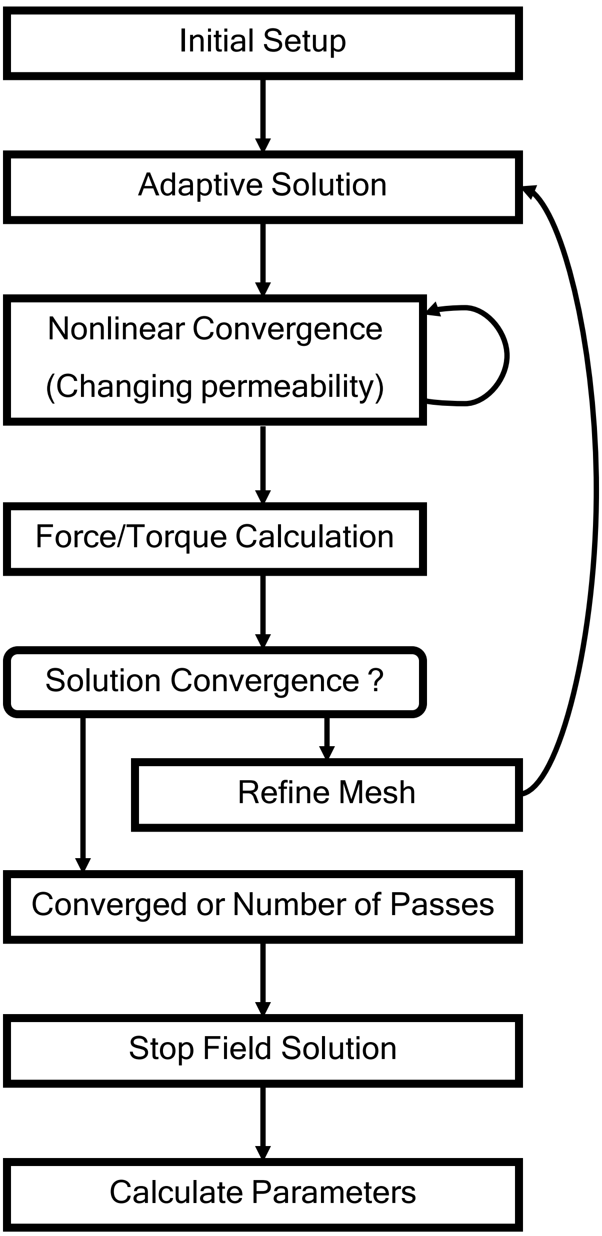
- The third tab of the Solution Setup contains information about the Solver.



- **Nonlinear Residual** defines how precisely the nonlinear solution must define the B-H nonlinear operating points (the default value is 0.001).
- **Enable Iterative Solve** to enable **ICCG** solvers (Direct is the default).
- **Permeability Option** allows the nonlinear B-H operating points either to be calculated by the solver from the **Nonlinear B-H curve** or to use frozen permeabilities **From Link** – the linked solution must have the exact same geometry as the current simulation (Nonlinear B-H curve is the default).
- **Demagnetization Option** allows the permanent demagnetization to be determined from the **Nonlinear B-H curve** or to use demagnetized values **From Link** - where the linked solution must the option **Compute Data for Link - Demagnetized operating points** checked and must have the exact same geometry (Nonlinear B-H curve is the default).
- **Import Mesh** allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **From Link** or **Import Mesh**.

Magnetostatic Solution Process

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



Eddy Current Analysis

- Eddy Current Analysis is performed by choosing the Eddy Current solution type.

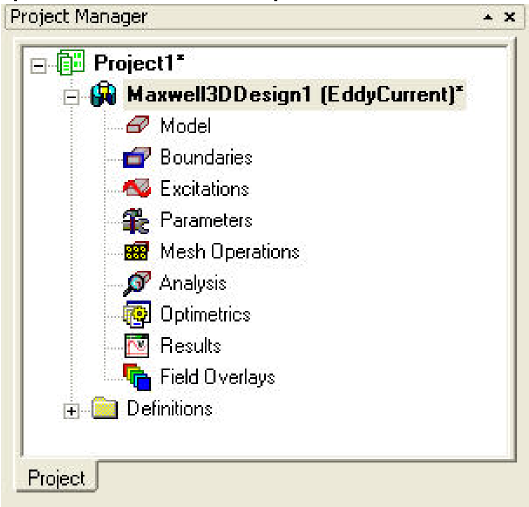
- Applications that use Eddy Current Analysis can be solenoids, inductors, motors, stray field calculations and many others.

Overview

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

Setup

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



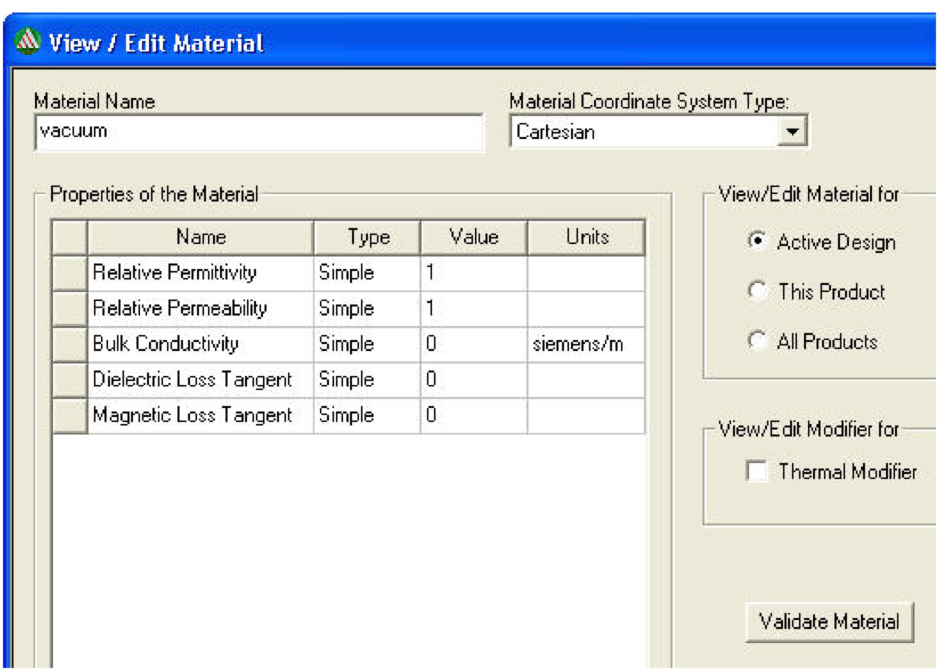
1. The **Model** definition refers to the geometry and material definition.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in a eddy current simulation.
3. **Parameters** are values that the solver will automatically calculate after finding the field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

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

- However, in some cases it is acceptable to work out of the defined order. This is particularly true when defining results, field overlays, or calculated quantities. It is important to think of results when defining the problem setup so that the desired quantities may be obtained in a sufficient manner. Notice, however, that the field calculator is not available until a solution setup is defined.

Eddy Current Material Definition

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



Eddy Current Boundary Conditions

- **Default** – The default boundary conditions for the Eddy Current solver are:
  - **Natural** boundaries on the interface between objects.
    - This means that the H Field is continuous across the boundary.
  - **Neumann** boundaries on the outer boundaries.
    - This means that the H Field is tangential to the boundary and flux cannot cross it.

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

- **Tangential H Field** – This boundary (similar to the Magnetostatic version) is used primarily to model an applied uniform, external field. This would model outer boundaries of the Region that have a defined tangential magnetic field. This boundary should always be placed far from the structure so that the simulation is not over-defined. Faces must be planar and must be defined one at a time, due to the U-V field definition on each face.
  - To apply a uniform field along any orthogonal axis of a bounding box, first, apply a Zero Tangential H Field on the top and bottom faces (with respect to the direction of the axis) of the box. Then apply a Tangential H Field to each side face – define each U vector to be parallel to the selected axis (the V vector does not matter because no field will exist in that direction), and assign the real and imaginary values of U with constant values. The field will be in the direction of the selected axis (perpendicular to the top and bottom faces), and will have a constant value away from the defined objects in the simulation.
  - Note that it is very easy to create impossible field assignments with these boundary conditions. If your simulation does not converge when using these boundaries, try the boundary conditions without any objects included to see if the boundaries are assigned correctly (the simulation will have difficulties converging if boundaries are incorrect).

Eddy Current Boundary Conditions (Continued)

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

- **Symmetry** – There are two magnetic symmetries – odd symmetry (flux tangential) and even symmetry (flux normal). Odd symmetry defines H to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Even symmetry defines H to be normal to the boundary (this is equivalent to the Zero Tangential H Field boundary condition). Remember that geometric symmetry may not mean magnetic symmetry in all cases. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Other considerations for a Symmetry boundary condition are the same as for a Magnetostatic symmetry boundary.

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

- **Radiation** – This boundary condition is specific to the Eddy Current solver. See the discussion of the Radiation boundary in the Radiation Boundary example.

- **Impedance** – This boundary can model induced currents within excluded objects without explicitly solving within the objects. This can decrease simulation time because the difficult to mesh areas near the surface of objects can be ignored and approximated with this boundary. By excluding the object (accomplished by deselecting Solve Inside in the object properties), there will be no solution inside the object. This approximation is good for good conductors (where the skin depth is less than half the width of the excluded conductor).

Eddy Current Excitations

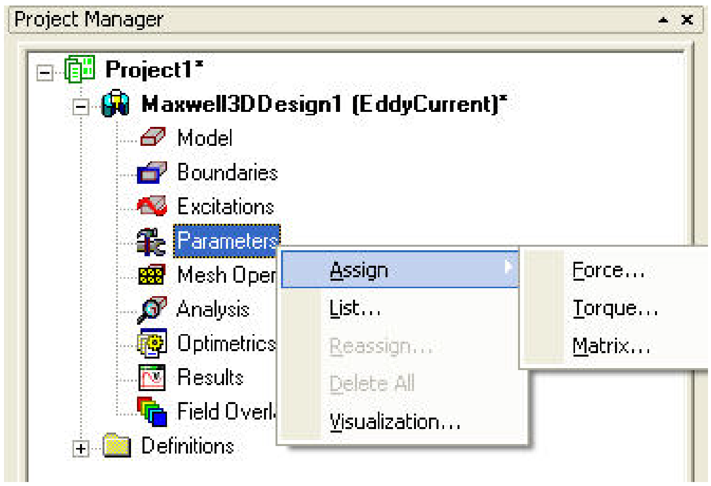
- Typical sources for eddy current problems include current and current density. In applying the sources for the magnetic field problems, keep in mind that the applied current distribution must be divergence free in the entire space of the solution as it is physical for (quasi) stationary conduction current density distributions. Thus, the conduction paths(s) for the applied current distributions must be closed when totally contained within the solution space for the problem or must begin and end at the boundaries. The total current applied to conductors that touch the boundaries doesn't require the existence of terminals at the ends where the current is applied, the respective planar surfaces of the conductors in the plane of the region (background) can be used to apply the excitations.

- **Current Excitations** - This excitation can be assigned on any conductor to define the total peak current (amp-turns) through the cross-section of a loop or to define the current into and out of the opposing, external faces of a conducting object. This is a very general purpose excitation that comes in two flavors – Solid or Stranded. More information about Stranded Current excitations (along with a Magnetostatic example and explanation) can be found in the Magnetostatic Switched Reluctance Motor example. The only difference between the use of stranded and solid conductors between the magnetostatic solver and the eddy current solver, is that in the eddy current solver eddy effects are not considered in stranded conductors, but not in solid conductors.

- **Current Density** - These excitations are used to define a known current density throughout an object and must be used with a Current Density Terminal. The peak Current Density (magnitude and phase) is defined on the 3D object, and the Terminal is defined on either an internal cross-sectional sheet, or on all external cross-sectional faces.

Parameters

- There are three parameters that can be automatically calculated in an Eddy Current simulation – **Force**, **Torque**, and Inductance **Matrix**.



- All three quantities are computed directly from the magnetic field solution.

- Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).

- The results of any parameters can be found by selecting ***Maxwell 3D > Results > Solution Data…*** or by clicking on the icon.

- Further results can be obtained manually through the field calculator.

Mesh Operations

- Mesh operations are described in detail in the Mesh operations section.

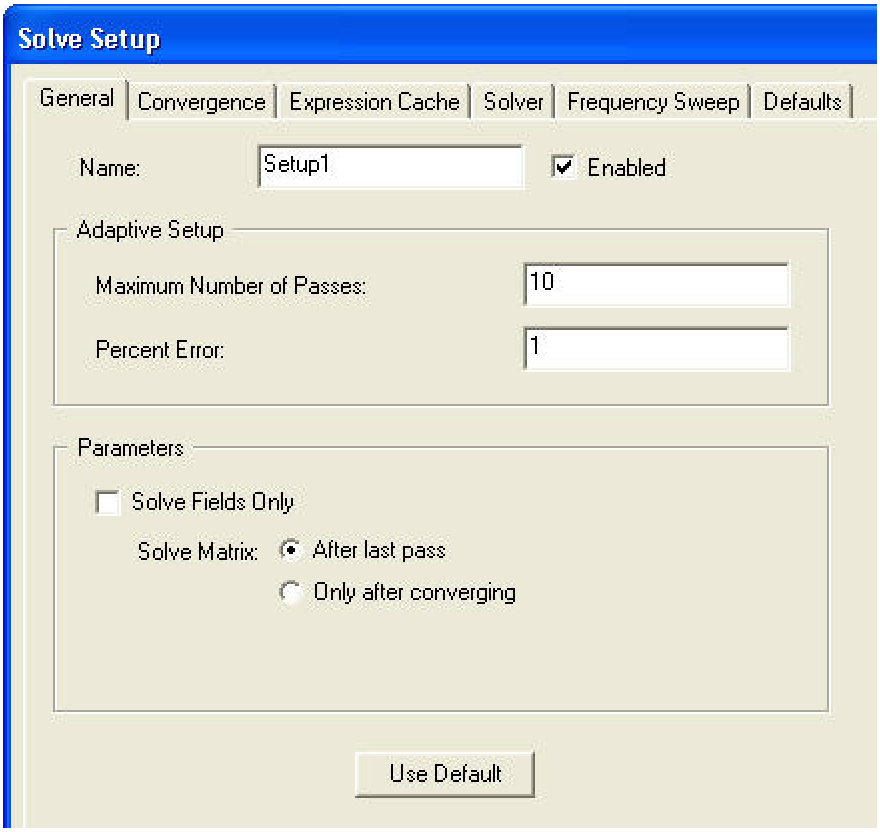
- Remember that the Eddy Current solver has an adaptive mesh solution, so excessive mesh operations are not always required. It can often be worse to over-define the mesh than to not define mesh operations at all (it will take longer to solve, and it will be more difficult to adapt correctly).

- However, it is very important to mesh properly to account for currents with small skin depths – this is important on solid conducting objects.

Solution Setup

- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.

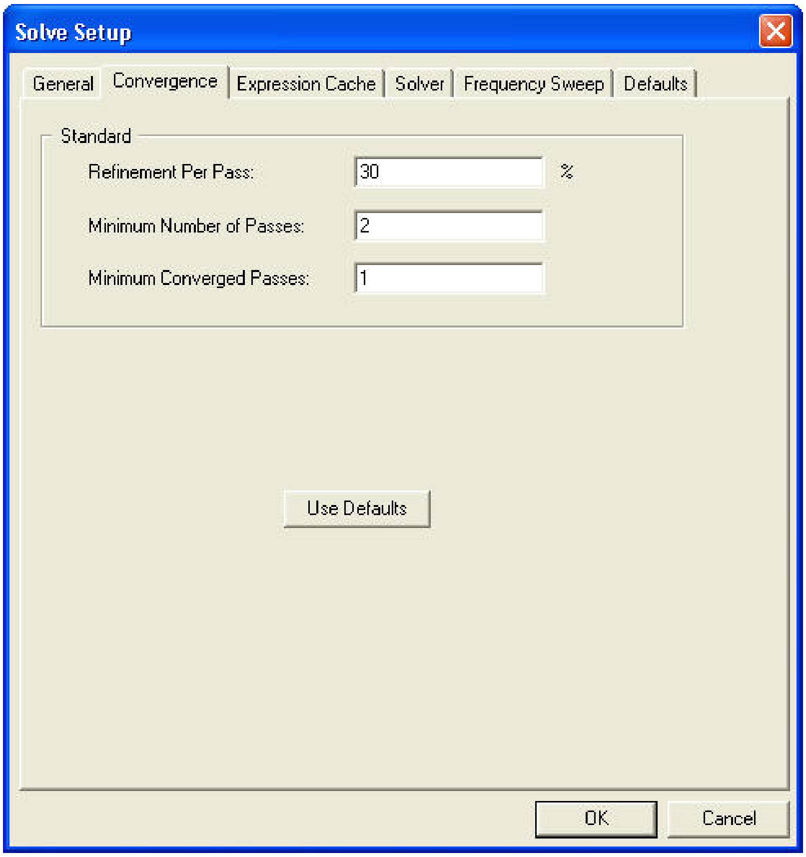
- The following window appears with the General Setup parameters.



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

Solution Setup (Continued)

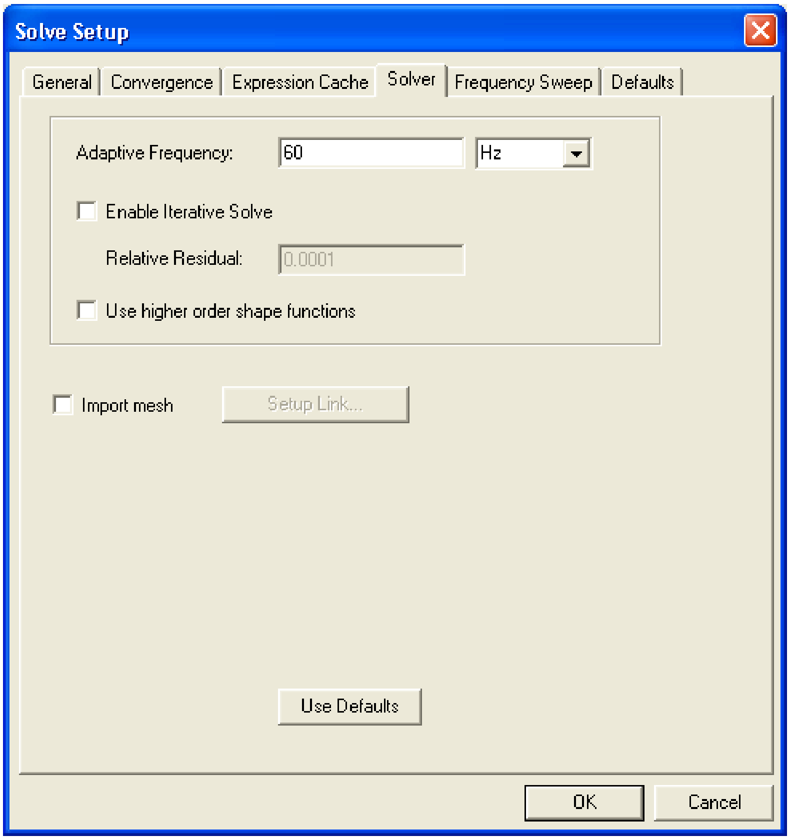
- The second tab of the Solution Setup contains information about Convergence.



- **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
- **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
- **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).

Solution Setup (Continued)

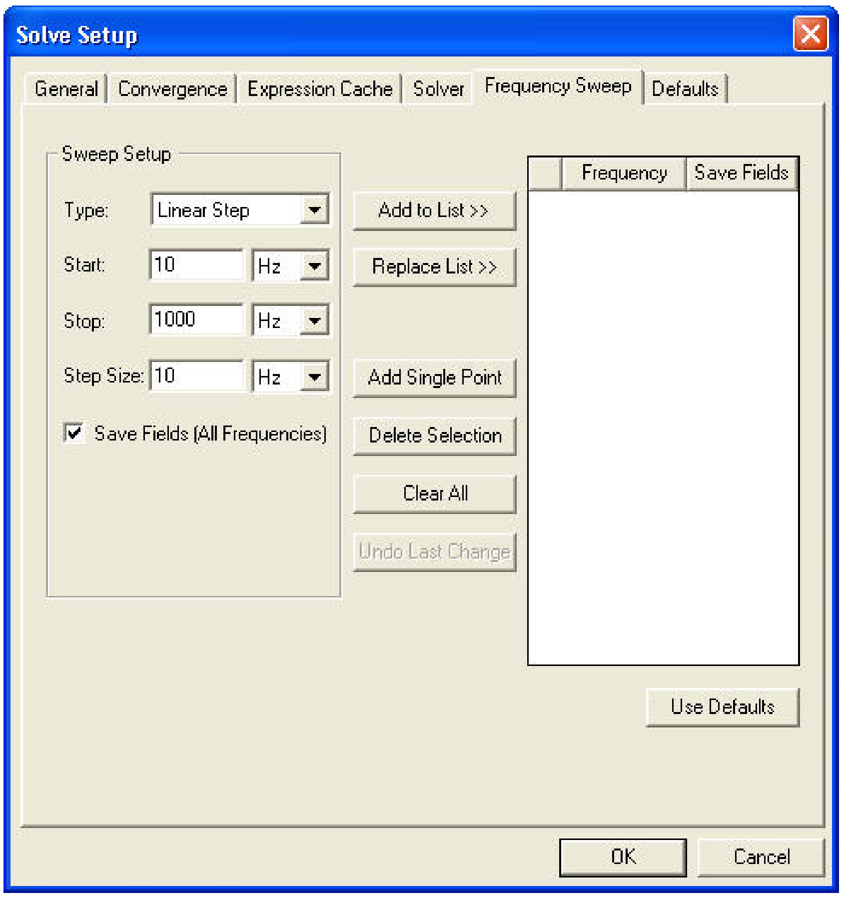
- The third tab of the Solution Setup contains information about the Solver.



- **Adaptive Frequency** defines the frequency at which the mesh is constructed and adapted, and at which solution is obtained (the default value is 60 Hz).
- **Enable Iterative Solve** to enable **ICCG** solvers (Direct is the default).
- **Use higher order shape functions** to enable the higher order option gains better accuracy for eddy current regions.
- **Import Mesh** allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **Import Mesh**.

Solution Setup (Continued)

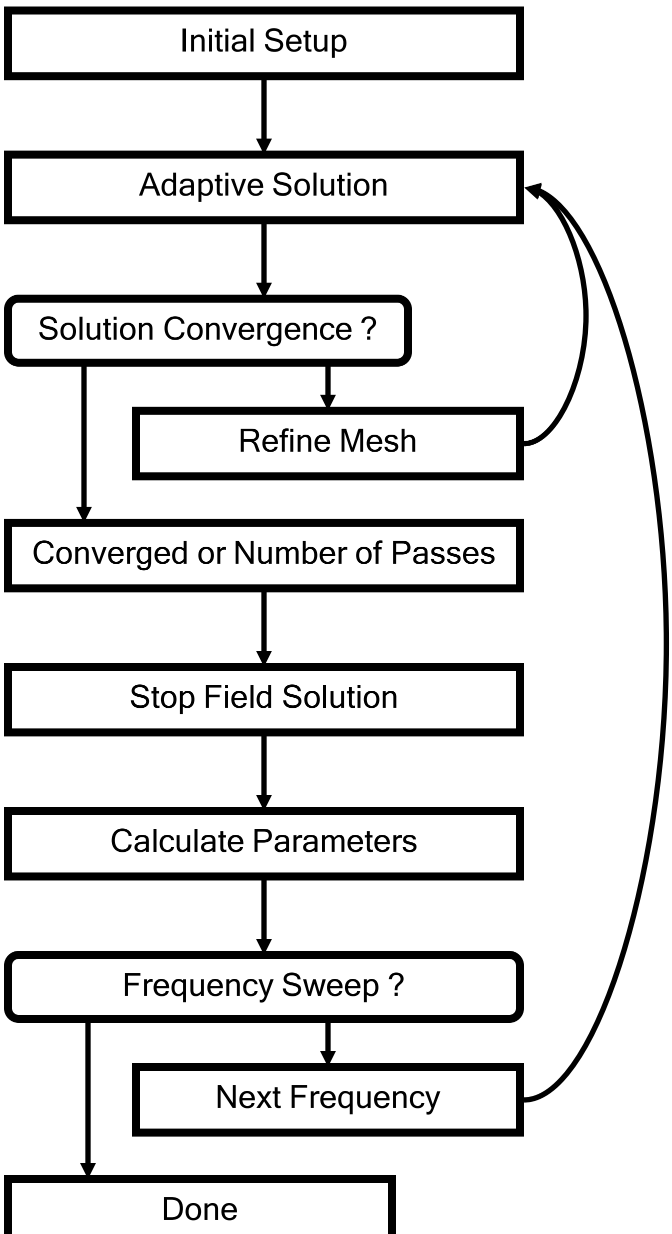
- The fourth tab of the Solution Setup contains information about an optional Frequency Sweep.



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

Eddy Current Solution Process

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



Transient Magnetic Analysis

- Transient Magnetic Analysis is performed by choosing the Transient magnetic solution type.

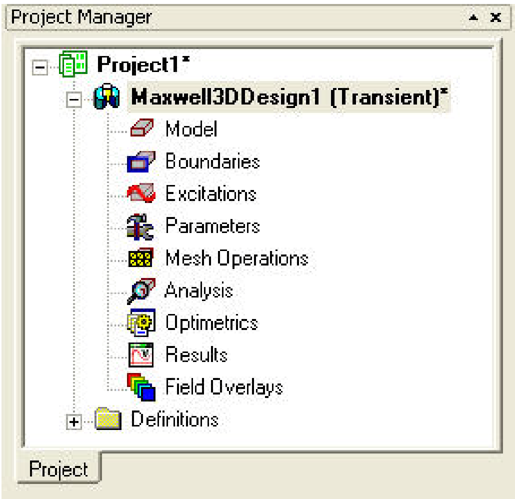
- Applications that use Transient Magnetic Analysis can be solenoids, inductors, motors, actuators, permanent magnets and many others.

Overview

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

Setup

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

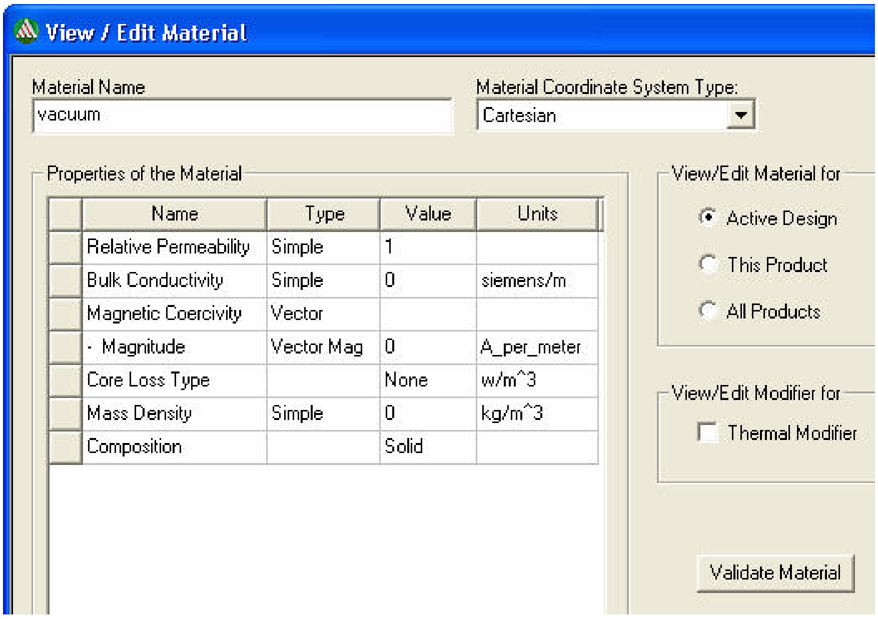


1. The **Model** definition refers to the geometry and material, as well as motion definitions.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in a transient simulation.
3. **Parameters** are values that the solver will automatically calculate after finding the field solution.
4. **Mesh Operations** are discussed in a separate section.
5. **Analysis** defines the solution setup.
6. **Optimetrics** defines any automatic variational analyses.
7. **Results** and **Field overlays** are discussed in a separate section.

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

Transient Material Definition

- In a Transient simulation, the following parameters may be defined for a material:
  - **Relative Permeability** (can be Anisotropic and/or Nonlinear, or Simple)
    - Relative Permeability plays a large role in determining the magnetic field solution.
  - **Bulk Conductivity** (can be Anisotropic or Simple)
    - Bulk Conductivity is used both in determining the current distribution in current carrying conductors and in finding the eddy currents in solid conductors, which affect the magnetic solution.
  - **Magnetic Coercivity** (defined as a vector magnitude and direction)
    - Magnetic Coercivity is used to define the permanent magnetization of magnetic materials. When a non-zero magnitude is entered, the directional entries are visible. The direction (like all directional material properties) are determined by the coordinate system type and the object orientation.
  - **Core Loss Type** (can be Electrical Steel, Power Ferrite, or None)
  - **Mass Density**
  - **Composition** (can be Solid or Lamination)
    - Setting Composition to Lamination creates an anisotropic magnetization effect. This is discussed in the Magnetostatic Anisotropic Material example.



Transient Boundary Conditions

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

Transient Excitations

- Typical sources for transient field problems include coil terminals of type voltage, current or external circuit. When applying the sources for the magnetic field problems, the applied current distribution must be divergence free in the entire space of the solution as it is physical for quasi-stationary conduction current density distributions. Thus, the conduction path(s) for the applied current distributions must be closed when totally contained within the solution space for the problem, or must begin and end at the boundaries.

- Permanent Magnets can also act as sources for a transient analysis, however, those are defined elsewhere.

- **Coil Terminals –** Coil terminals are defined to designate the cross sectional faces of the 3D conductors. These can either be located internally to a closed loop, or on the external faces of a conduction path. The coil terminals are grouped into a Winding that controls the current in one or multiple conduction paths. The only things that are defined by the Coil Terminals are as follows:
  - Number of conductors (solid windings require 1 conductor)
  - Direction of the current
  - A cross section of the conductor

- **Windings** – Windings control the current flowing through the conductors and are therefore crucial to the magnetic solution. There are three different types of winding:
  - **Current** – defines a specified functional current through the conducting paths.
  - **Voltage** – defines a specified functional voltage across the coil terminals (and a series resistance and inductance).
  - **External** – defines that an external circuit will control the current and voltage associated with the conducting path.

- Coil terminals must be added to a Winding to complete the excitation setup.

- Coil terminals will automatically report flux linkage and induced voltage vs. time in the 2D reporter.

Winding Setup

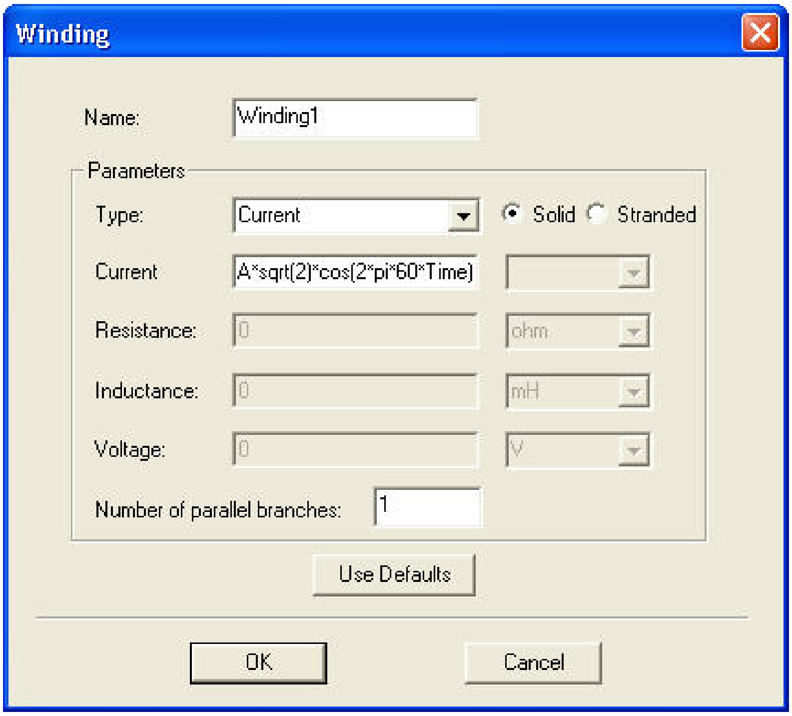
- Assign a coil terminal to a winding by either:
  - Right-clicking on the winding and choosing **Add Terminals…**

    or
  - Right-clicking on the terminal and choosing **Add to Winding…**

- Group coil terminals by adding them to the same winding.

- Grouped conductors are considered in series – the defined current goes through each conductor, voltage is defined across all conductors plus additional resistance and/or inductance – total winding inductance is treated as the sum of each coil's inductance.

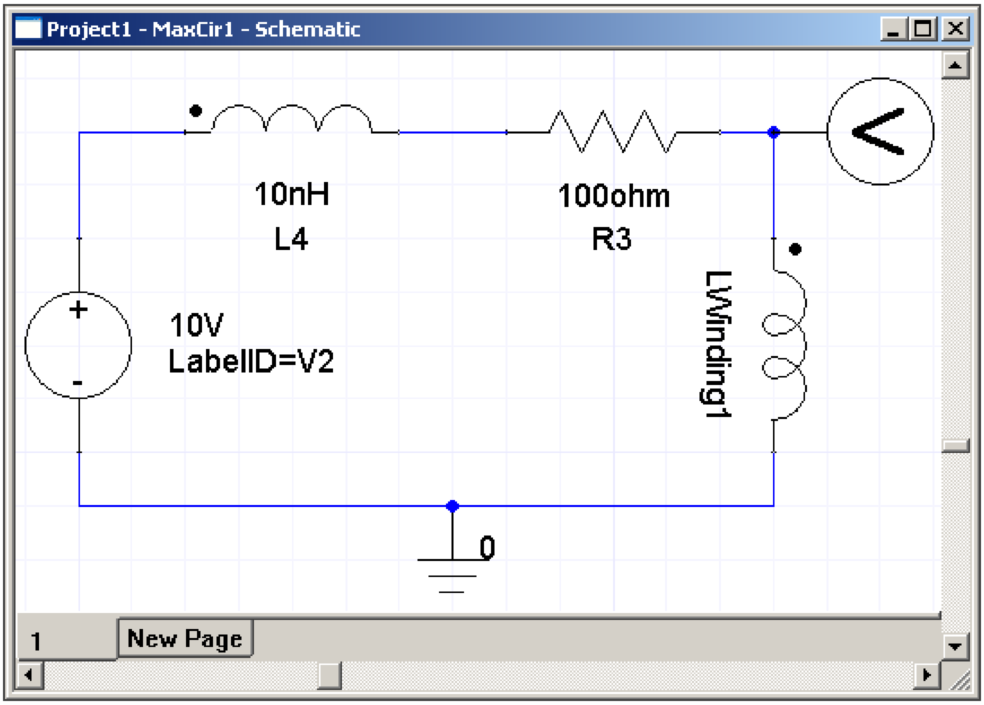
- In the Winding setup:
  - Define the Winding to be the desired Type and Solid vs. Stranded.
  - Then, define the necessary parameters – current or voltage can be a function of time in their respective source types. Resistance and inductance are available for voltage sources (to determine the current), and are considered in series with winding.



External Circuits

- Step1: Open Maxwell Circuit Editor
  (*Windows Start Menu > Programs > Ansoft > Maxwell 15 > Maxwell Circuit Editor*)

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

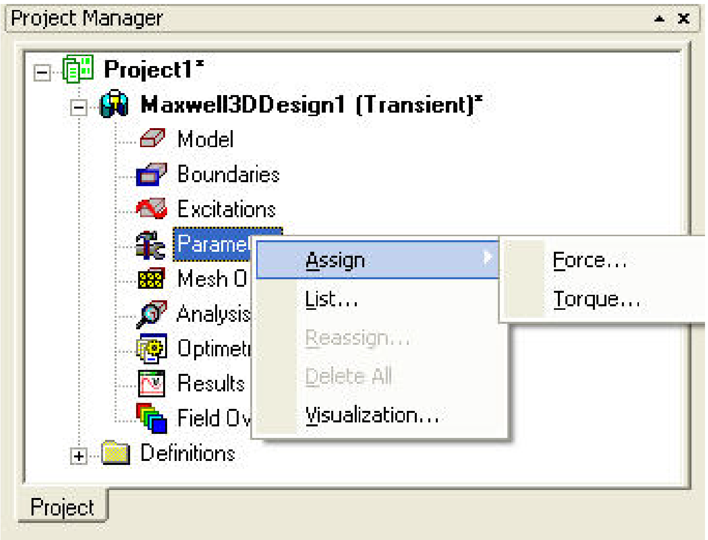


- Step 3: Export netlist from the circuit editor (*Maxwell Circuit > Export Netlist*).
  - Save the Circuit Editor project so that you can edit your circuit later.

- Step 4: Import the netlist into Maxwell (*Maxwell 3D > Excitations > External Circuit > Edit External Circuit…*)
  - Choose **Import Circuit…** to import the netlist.
  - You must re-import the netlist every time that you make a change to the circuit, otherwise the change will not take effect.

Parameters

- There are two parameters that can be automatically calculated in a Transient simulation – **Force** and **Torque**.



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

Mesh Operations

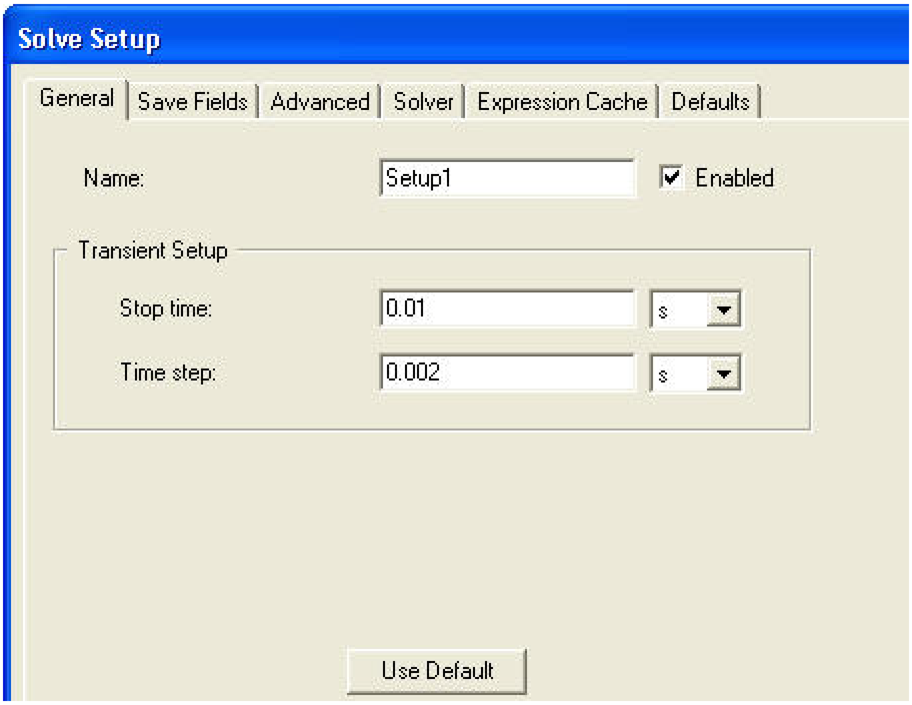
- Mesh operations are described in detail in the Mesh operations section.

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

Solution Setup

- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.

- The following window appears with the General Setup parameters.



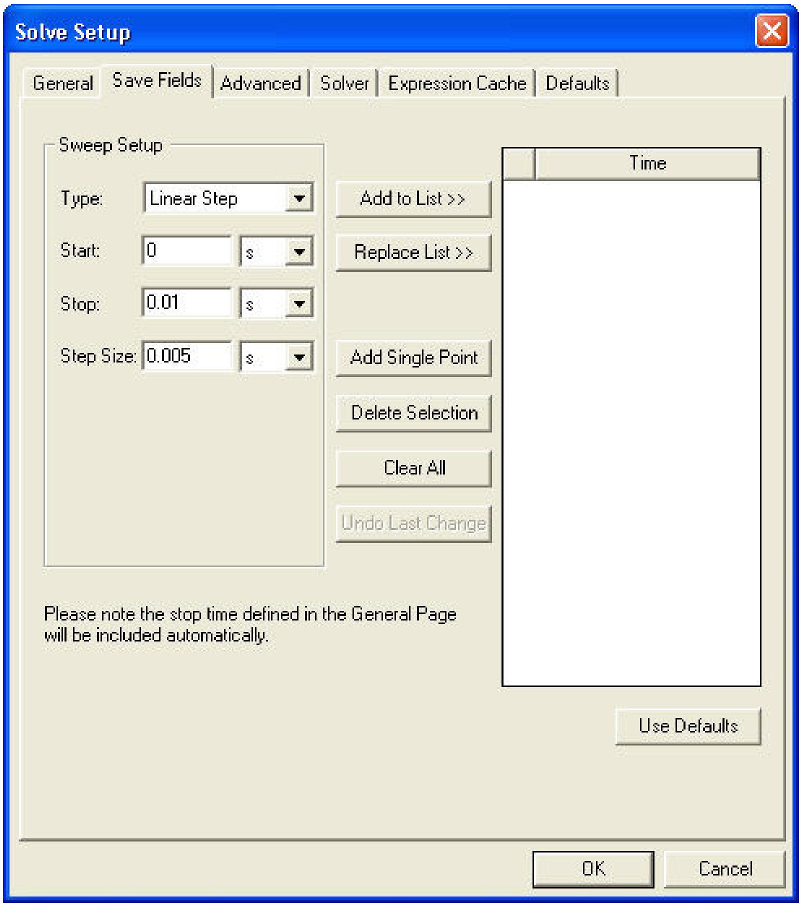
- You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
- **Stop time** indicates the transient simulation time at which the simulation will stop.
- **Time step** indicates the discrete lengths of time used in the transient simulation.

- Notice that the general information is related to the transient nature of this simulation – there is no information about convergence, because there is no adaptive solution to converge with.

- Choose the time steps appropriately for the physical time-constants of the simulation (about 20 timesteps per cycle).

Solution Setup (Continued)

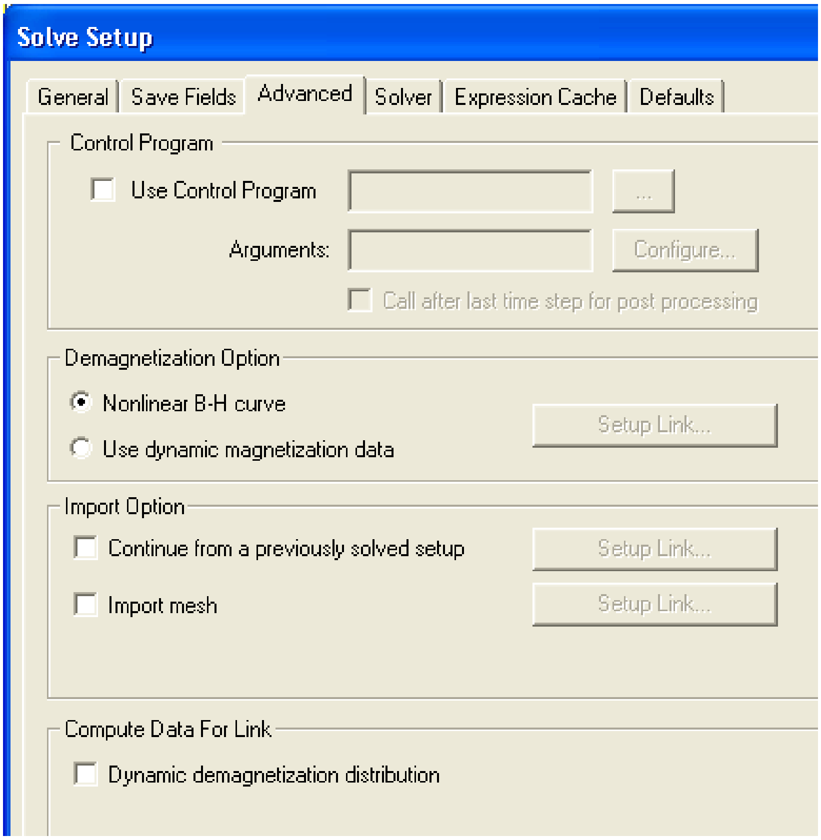
- The second tab of the Solution Setup contains information about Saving Fields.



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

Solution Setup (Continued)

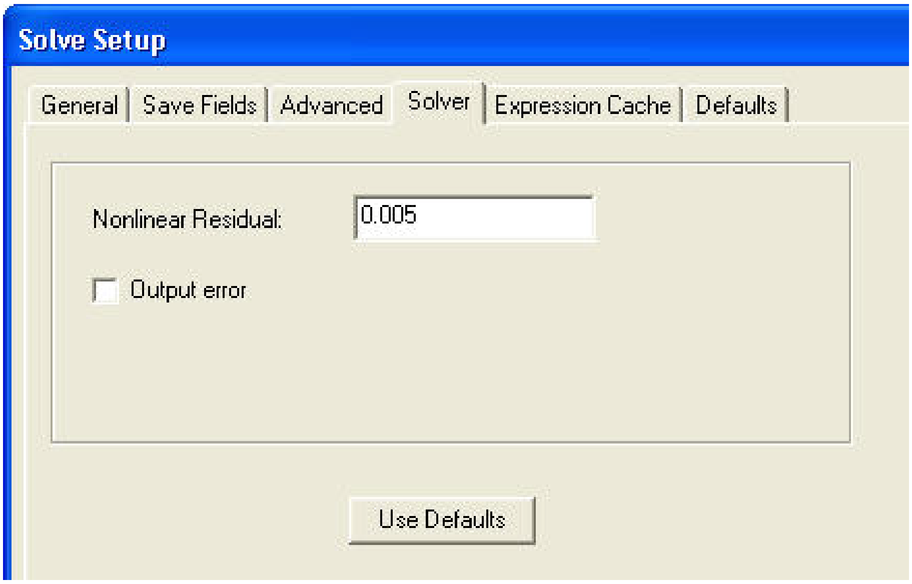
- The third tab of the Solution Setup contains Advanced options.



- **Control Programs** are used to dynamically adjust parameters and control the simulation – information can be found in the Maxwell Help.
- **Demagnetization Option** allows the permanent demagnetization to be determined from the **Nonlinear B-H curve** or to use demagnetized values **From Link** - where the linked solution must the option **Compute Data for Link – Dynamic demagnetization distribution** checked and must have the exact same geometry (Nonlinear B-H curve is the default).
- **Continue from a previously solved setup** enables to continue the solution from a linked Maxwell design setup
- **Import Mesh** allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **From Link** or **Import Mesh**.

Solution Setup (Continued)

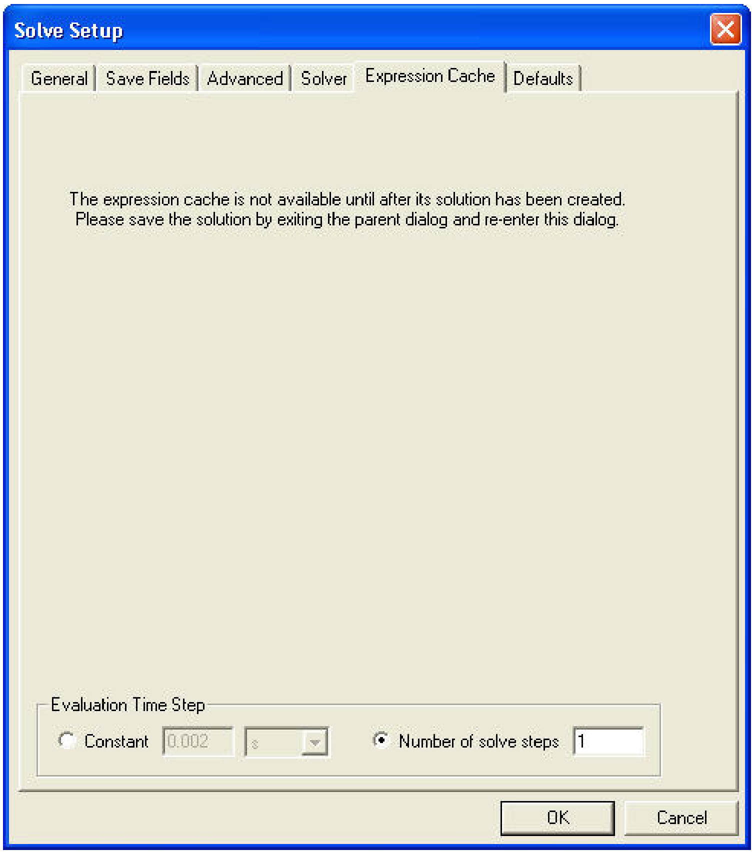
- The fourth tab of the Solution Setup contains information about the Solver.



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

Solution Setup (Continued)

- The fifth tab of the Solution Setup contains Expression Cache options.



- The bottom two tabs are used to evaluate defined Output Variables. Output variables can be added from the menu item ***Maxwell 3D > Results > Output Variables***
- There are two options for defining the time steps at which these output variables are calculated.
  - The first option is to evaluate the output variables at evenly spaced increments of time by using a constant time step.
  - The second option is to evaluate the output variables after every specified number of steps. If we specify say Number of solve setps as **N,** the output variable will then be computed at 0s, and then at step N, step 2*N and so on.

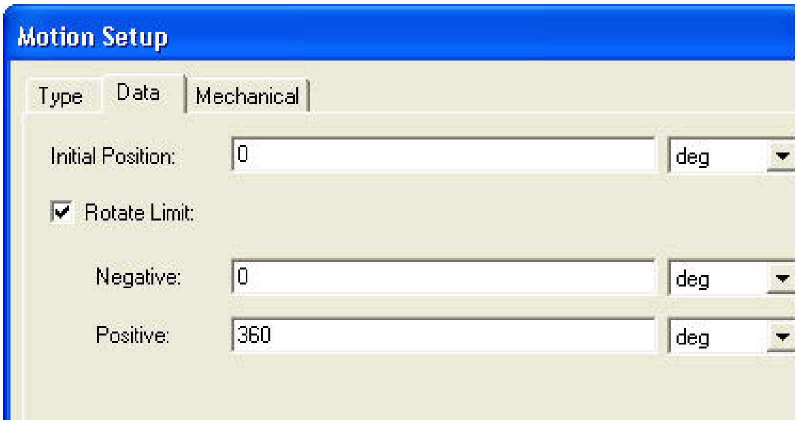
Motion Setup

- Choose **Assign Band** to specify the band object (***Maxwell 3D > Model > Motion Setup > Assign Band…***).

- The Motion Setup window appears when the band is assigned.

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

- Then define the **Initial Position** and translational or rotational limits.



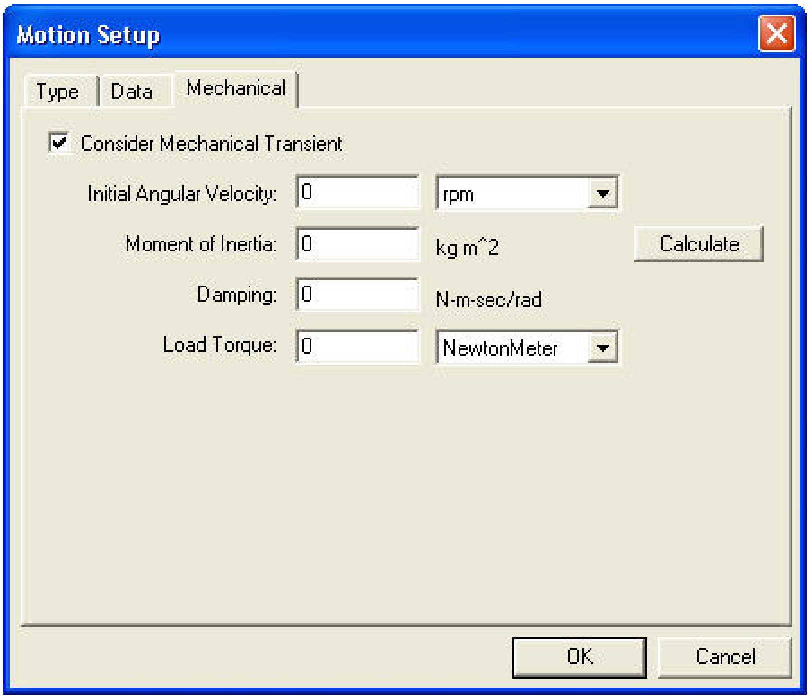
Motion Setup (Continued)

- In the Mechanical tab of the motion setup, there are two fundamental options:
  - Velocity Definition

    or
  - Consider Mechanical Transient

- Velocity Definition is useful when the velocity is constant or follows a known trajectory that can be expressed as a velocity as a function of time. Use this option by de-selecting Consider Mechanical Transient.

- Consider Mechanical Transient is useful for situations when the velocity varies dynamically or is unknown. Requires the following inputs:
  - **Initial Velocity** or **Initial Angular Velocity**
  - **Moment of Inertia**
  - **Damping**
  - **Load Torque**



Motion Setup - Suggestions

- Read the Maxwell Help note called *Meshing Aspects for 3D Transient Applications With Motion*

- When performing translational or non-cylindrical rotational motion, always include a vacuum container object within the band that contains all the moving parts. This container object should be spaced slightly away from all the objects that it contains and should provide clearance when the objects and container move within the band. This container object is necessary for both meshing purposes with multiple objects and to produce a better force calculation surface. A vacuum container object is often useful in cylindrical motion too, but not necessary due to meshing considerations. The objects within the vacuum container are assumed to move as one rigid body (all moving objects are assumed to move as one rigid body by definition of the transient motion solution).

- When conceptualizing translational or non-cylindrical rotational motion, remember that the moving parts within the vacuum container move as one solid body and the remeshing occurs between the surface of the band object and the moving objects (generally a container object).

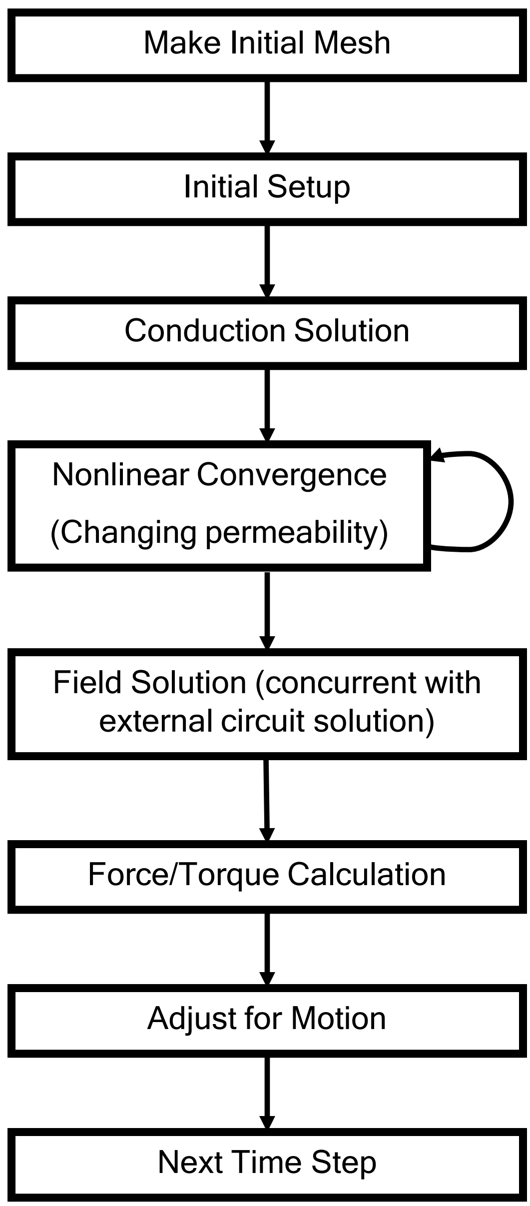
- When conceptualizing cylindrical rotational motion, remember that the band object and all moving parts rotate without remeshing between.

- The conceptual difference between the two is that the band is more solid in the cylindrical case (in that it does not change position with respect to the moving parts), and the band is more fluid in the translational and non-cylindrical case (in that the moving parts change position with respect to the band).

- In the translational and non-cylindrical cases, the mesh the is created within the band is created so that the edge length is not larger than the average edge length for the elements on the surface of the band and the moving parts. If you wish to better define the mesh within the band, you should apply a length-based mesh operation on the surfaces of the band and moving parts (simply on the surface of the vacuum container if used).

Transient Solution Process

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



Electrostatic Analysis

- Electrostatic Analysis is performed by choosing the Electric solution type and selecting the Electrostatic option.

- Applications that use Electrostatic Analysis can be capacitors, high voltage lines, breakdown voltage calculations and many others.

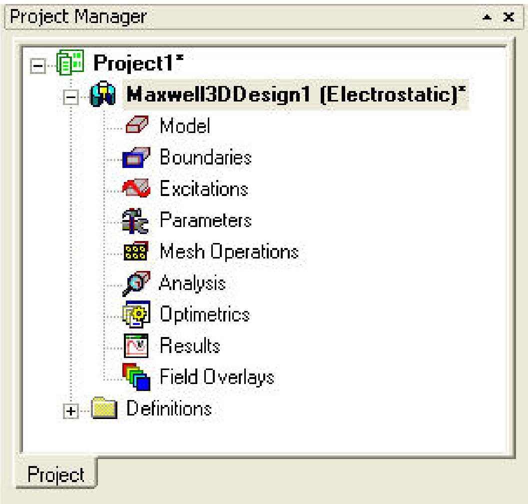
Overview

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

- Can be coupled with a DC conduction simulation, where the electric potential from the DC conduction solution is used as a voltage boundary condition for the electric field solution in the insulators in an electrostatic simulation.

Setup

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

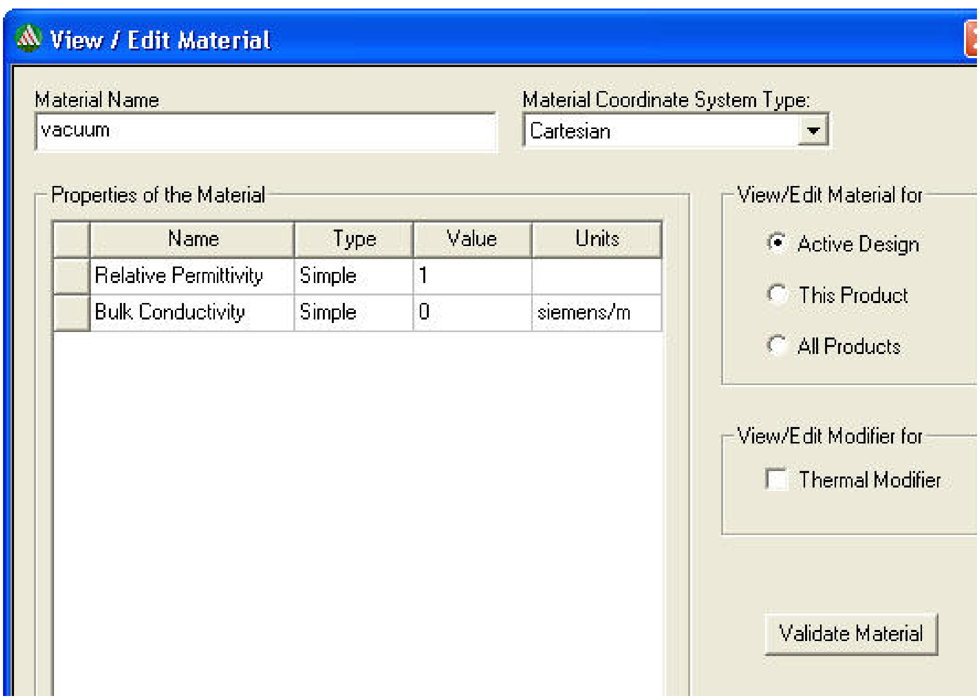


1. The **Model** definition refers to the geometry and material definition.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in an electrostatic simulation.
3. **Parameters** are values that the solver will automatically calculate after finding the field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

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

Electrostatic Material Definition

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



Electrostatic Boundary Conditions

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

- **Symmetry** – There are two Electric symmetries – even symmetry (flux tangential) and odd symmetry (flux normal). Even symmetry defines E to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Odd symmetry defines E to be normal to the boundary. Remember that geometric symmetry may not mean electric symmetry in all cases. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Other considerations for a Symmetry boundary condition:
  - A plane of symmetry must be exposed to the background.
  - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
  - A plane of symmetry must be defined on a planar surface.
  - Only three orthogonal symmetry planes can be defined in a problem

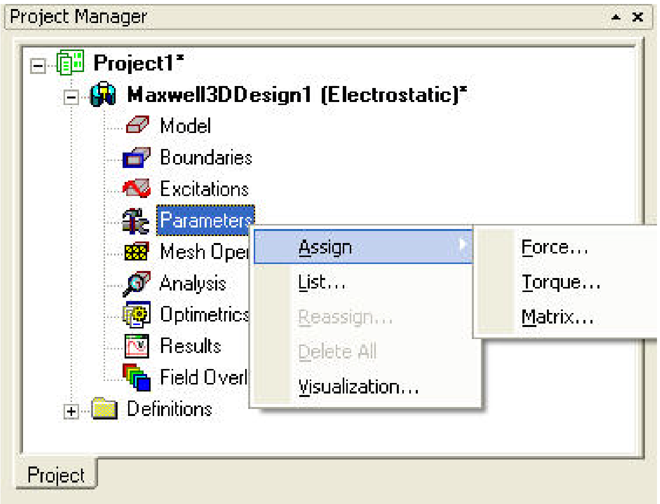
- **Master/Slave** – This boundary condition is also known as a matching boundary condition because it matches the electric field at the slave boundary to the field at the master boundary. The geometry must be identical on each face (the mesh needs to be identical, but the solver takes care of this requirement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

Electrostatic Excitations

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

Parameters

- There are three parameters that can be automatically calculated in an Electrostatic simulation – **Force**, **Torque**, and Capacitance **Matrix**.



- All three quantities are computed directly from the electric field solution.
- Force and torque can be calculated with two different methods - Virtual or Lorentz (Lorentz cannot be used on magnetic materials).
- The results of any parameters can be found by selecting ***Maxwell 3D > Results > Solution Data…*** or by clicking on the  icon.
- Further results can be obtained manually through the field calculator.

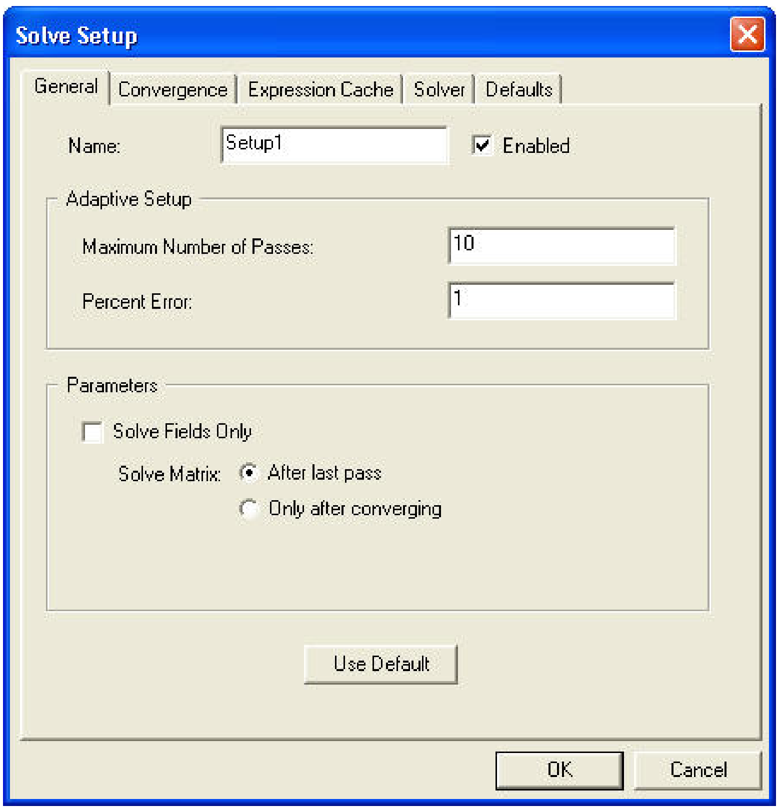
Mesh Operations

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

Solution Setup

- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.

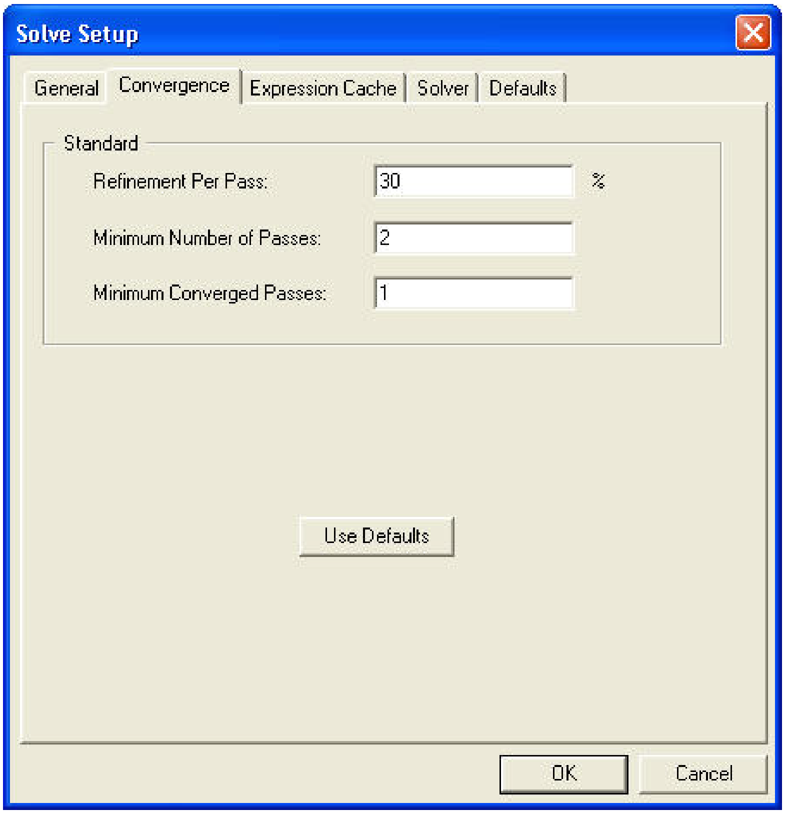
- The following window appears with the General Setup parameters.



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

Solution Setup (Continued)

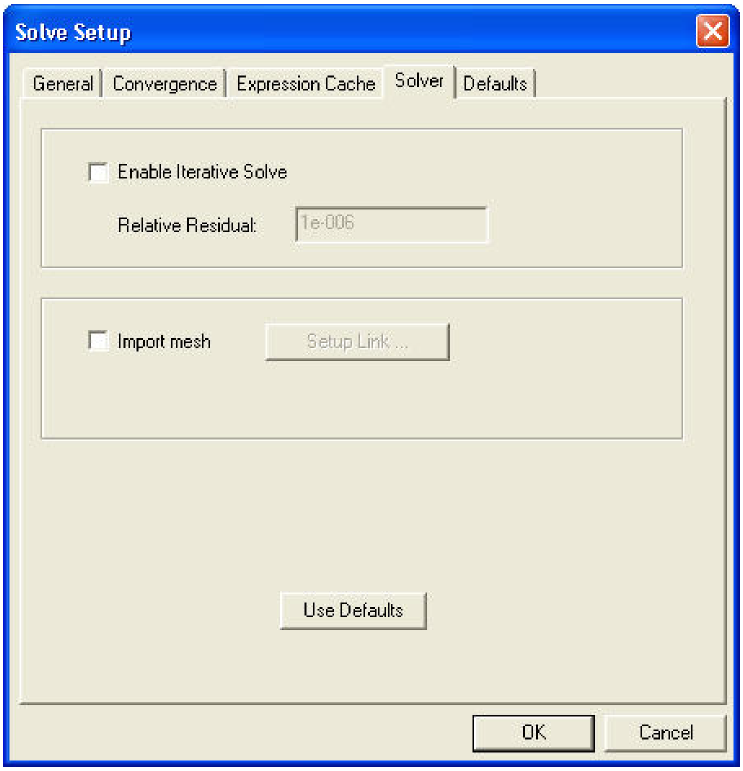
- The second tab of the Solution Setup contains information about Convergence.



- **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
- **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
- **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).

Solution Setup (Continued)

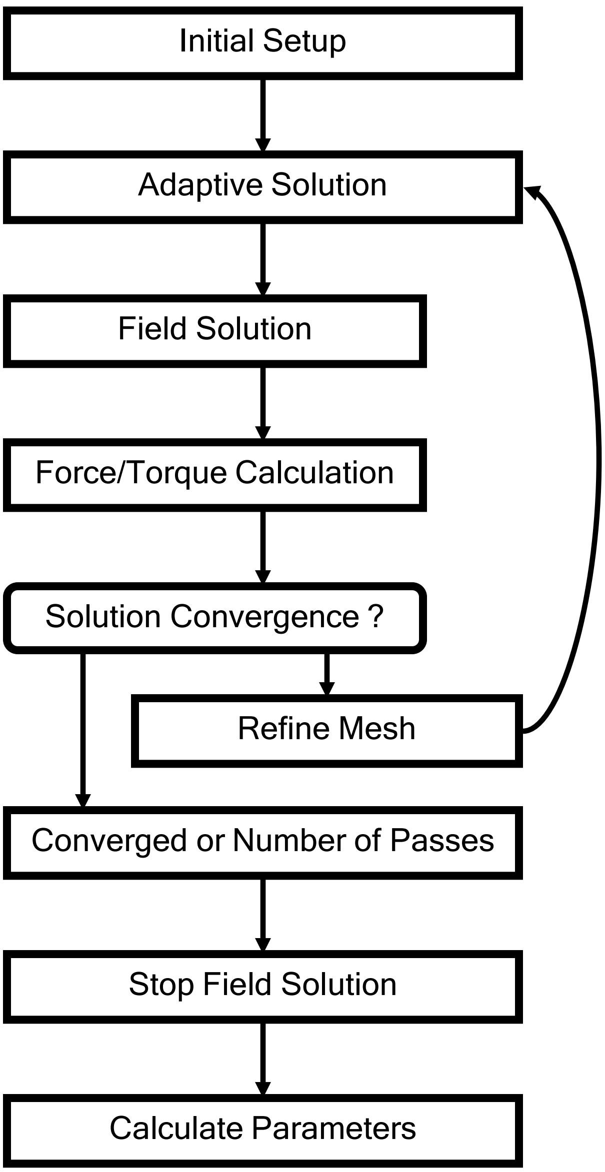
- The fourth tab of the Solution Setup contains information about the Solver.



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

Electrostatic Solution Process

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



DC Conduction Analysis

- DC Conduction Analysis is performed by choosing the Electrostatic solution type and selecting the DC conduction option.

- Applications that use DC Conduction Analysis can be bus bars, power supplies, and many others.

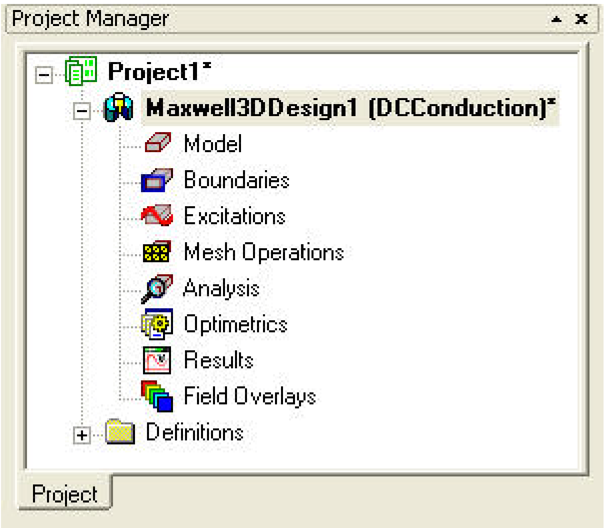
Overview

- The DC Conduction solver computes static (DC) currents in conductors.
- All objects are stationary.
- The source of the static magnetic field can be:
  - Voltages at different ends of solid conductors.
  - Currents applied on surfaces of conductors.
- The quantity solved is the electric scalar potential (ø).
- Current density (J) and Electric Field (E) are automatically calculated from the electric scalar potential (ø).
- Material conductivities can be anisotropic.
- All fields outside of the conductors are not calculated and totally decoupled from the electric field distribution in the conductors – permittivity is irrelevant in this calculation.
- There is non-zero Joule loss (ohmic power loss) in the conductors.

- Can be coupled with an electrostatic simulation, where the electric potentials found in the conductors are used as a voltage boundary condition for the electric field solution in the insulators in an electrostatic simulation.

Setup

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

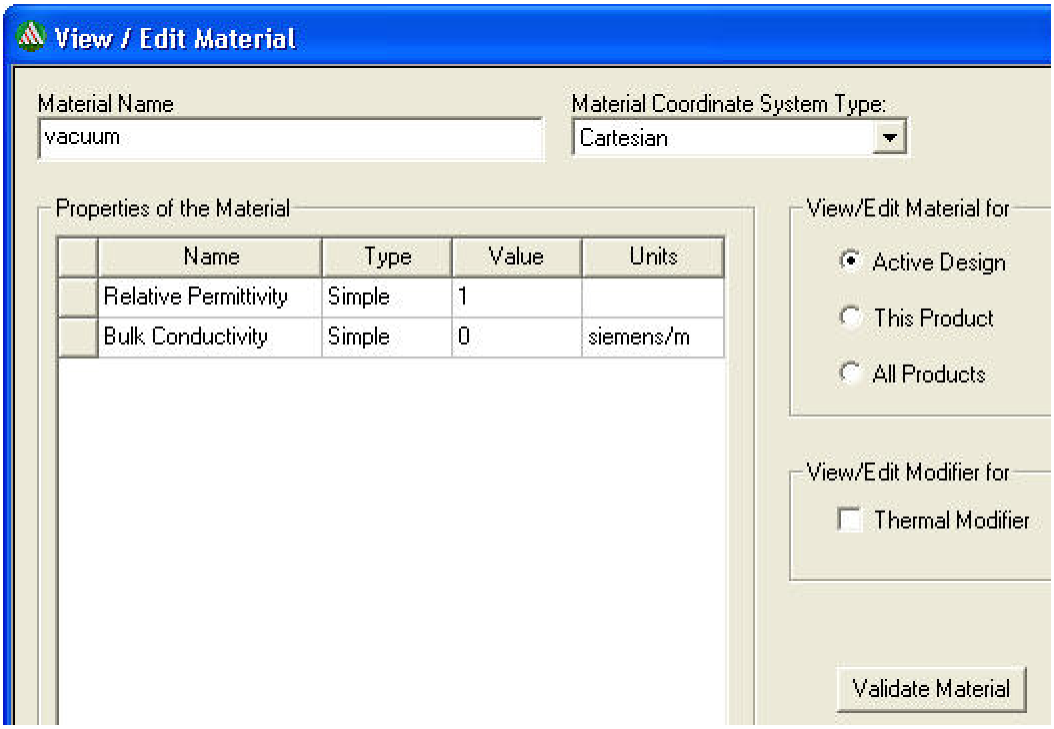


1. The **Model** definition refers to the geometry and material definition.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in a DC Conduction simulation.
3. There are no parameters for a DC conduction simulation.
4. **Mesh Operations** are discussed in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

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

DC Conduction Material Definition

- In a DC Conduction simulation, the following parameters may be defined for a material:
  - **Relative Permittivity** (can be Anisotropic or Simple)
    - Relative permittivity does not affect the DC conduction calculation, but will be important in insulators if this is coupled with an electrostatic simulation.
  - **Bulk Conductivity** (can be Anisotropic or Simple)
    - Bulk Conductivity defines whether an object is a conductor (treated as a perfect conductor in the Electrostatic solver) or an insulator. This separation is determined by the insulator/conductor material threshold setting.



DC Conduction Boundary Conditions

- **Default** – The default boundary conditions for the DC Conduction solver are:
  - **Natural** boundaries on the interface between objects.
    - This means that the normal component of the current density at the boundary is continuous.
  - **Neumann** boundaries on the outer boundaries.
    - This means that the E Field is tangential to the boundary and flux cannot cross it (current cannot leave conductors).

- **Insulating** – This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate distinct conductors (defined on surfaces between the objects).

- **Symmetry** – There are two Electric symmetries – even symmetry (flux tangential) and odd symmetry (flux normal). Even symmetry defines E to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Odd symmetry defines E to be normal to the boundary. Remember that geometric symmetry may not mean electric symmetry in all cases. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Other considerations for a Symmetry boundary condition:
  - A plane of symmetry must be exposed to the background.
  - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
  - A plane of symmetry must be defined on a planar surface.
  - Only three orthogonal symmetry planes can be defined in a problem

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

DC Conduction Excitations

- Typical sources for DC current flow problems are currents applied on surfaces of conductors and voltages (electric potential applied to surfaces of conductors). The direction of the applied current is either “in” or “out”, always normal to the respective surfaces. Multiple conduction paths are allowed. Each conduction path that has a current excitation must also have either a voltage excitation applied or a sink to ensure a unique solution.

- **Voltage Excitations -** These are used in conjunction with the material conductivity to define the current through a solid conductor. Either multiple Voltage excitations can be used to define a voltage difference across two faces of a conductor (creating a current) or a Voltage can be defined along with a current excitation to define a voltage reference for the electric field solution.

- **Current Excitations** - This excitation can be assigned on any conductor to define the total current (amp-turns) through the cross-sectional face of a conductor.

- **Sink** – This excitation is used when only current excitations are defined in a conduction path and there is no voltage excitation. This excitation ensures that the total current flowing through the outside surface of a conduction path is exactly zero.

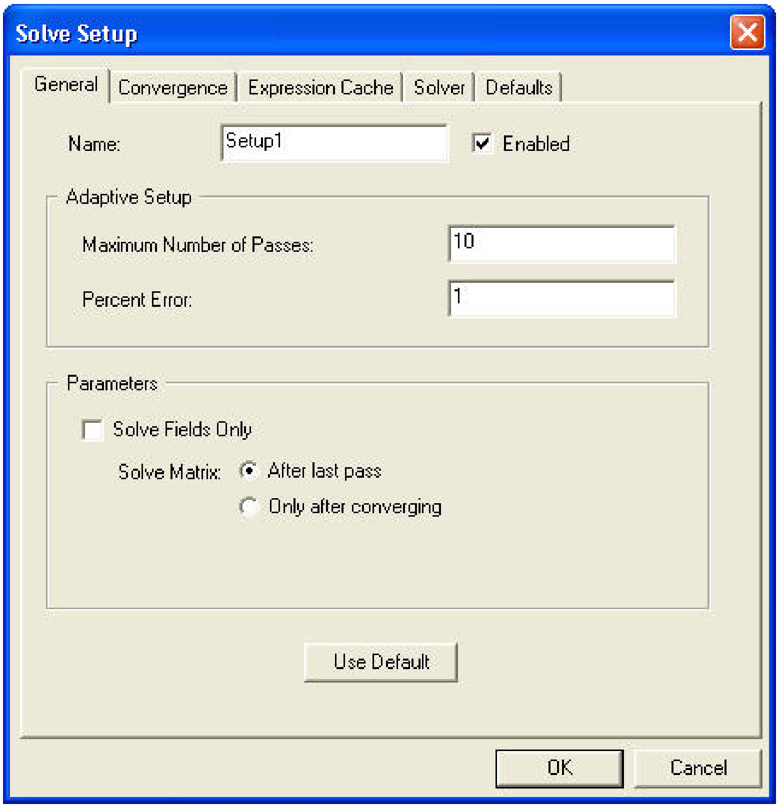
Mesh Operations

- Mesh operations are described in detail in the Mesh operations section.

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

Solution Setup

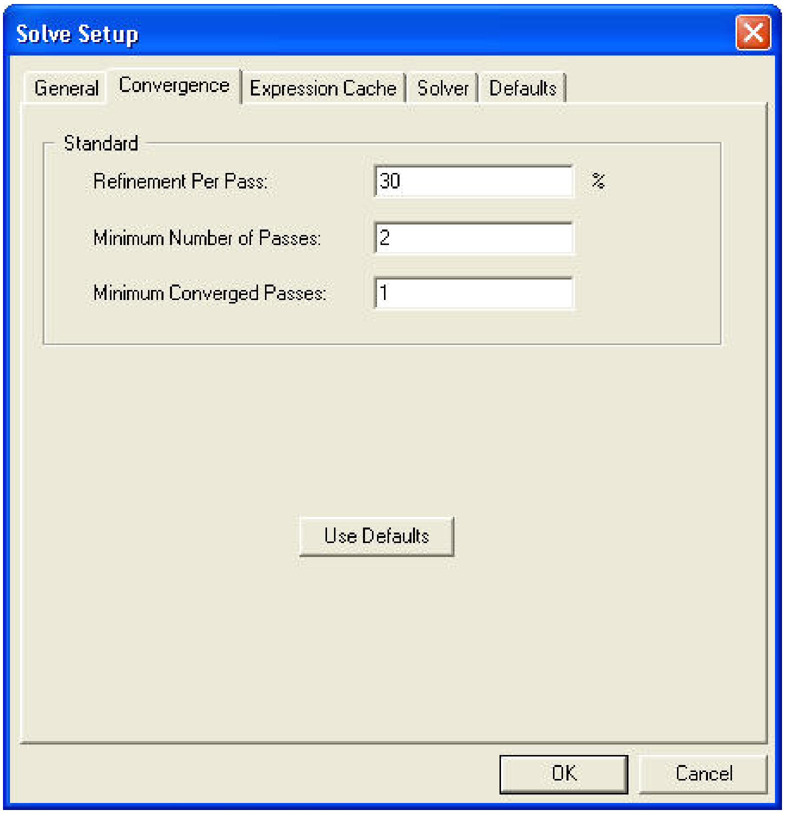
- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.
- The following window appears with the General Setup parameters.



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

Solution Setup (Continued)

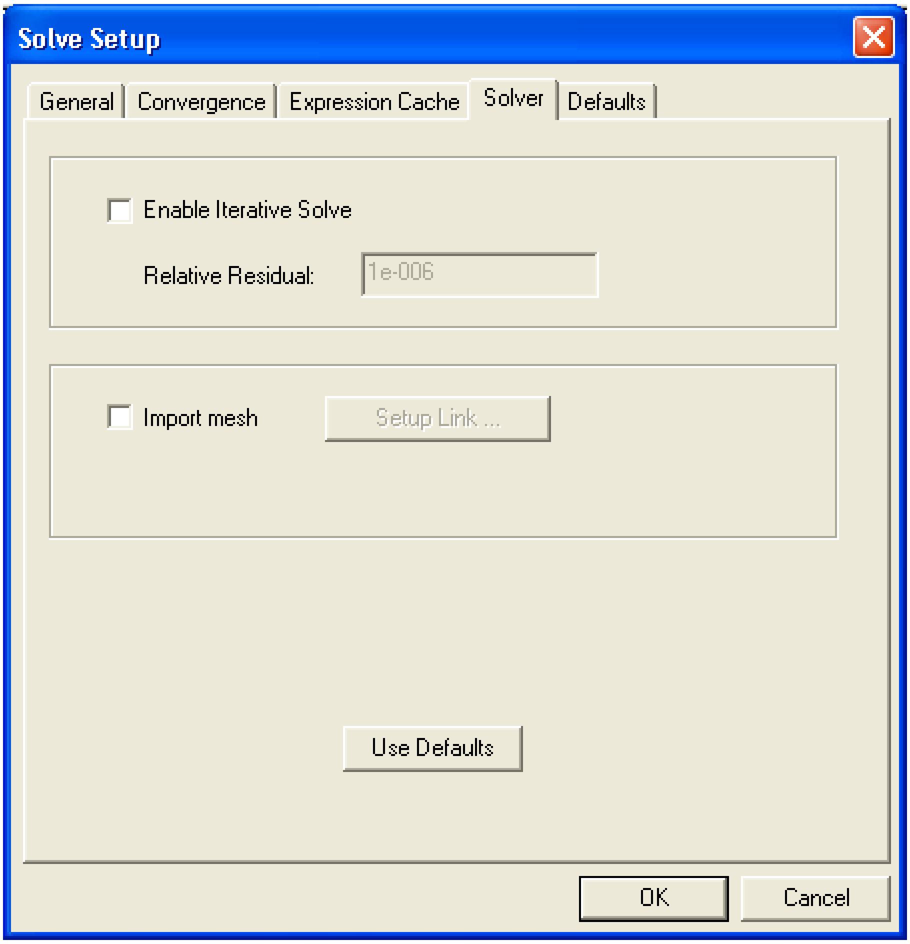
- The second tab of the Solution Setup contains information about Convergence.



- **Refinement Per Pass** defines the number of tetrahedral elements added during mesh refinement as a percentage of the previous pass (30% is the default).
- **Minimum Number of Passes** defines the minimum number of adaptive passes before the solution stops - if there is a conflict, this value is over-ridden by Maximum Number of Passes (the default value is 2).
- **Minimum Converged Passes** defines the minimum number of adaptive passes that have been converged (with respect to the Percent error) before the solution stops (the default value is 1).

Solution Setup (Continued)

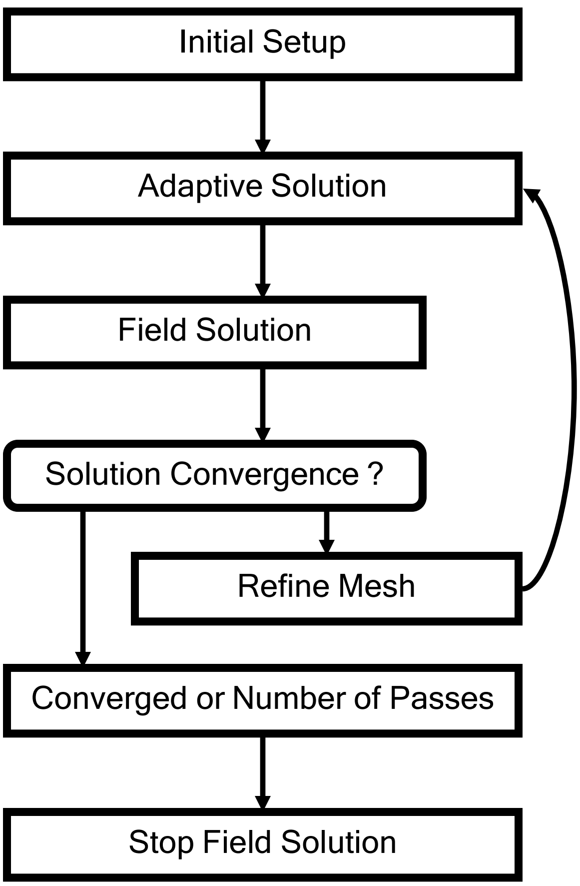
- The third tab of the Solution Setup contains information about the Solver.



- **Enable Iterative Solve** to use **ICCG** solvers (Direct is the default).
- **Import Mesh** allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **Import Mesh**.

DC Conduction Solution Process

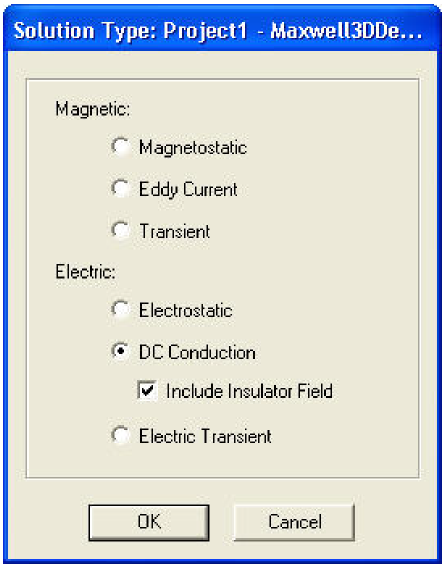
- Unlike pre-processing, the solution process is very automated. Once the problem has been defined properly, Maxwell will take over and step through several stages of the solution process. To start the solution process, right-click on Analysis in the Maxwell Project Tree and select Analyze.

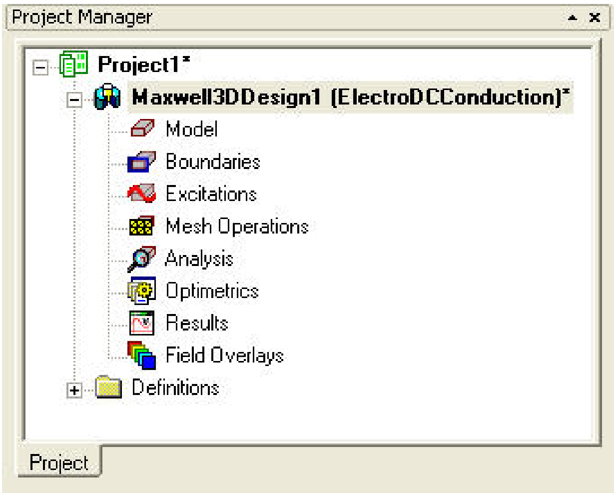


Electrostatic and DC Conduction Combination

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

- To enable combined solution with Electrostatic and DC Conduction, check the box below DC Conduction **Include Insulator Field**





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

Electric Transient Analysis

- Electric Transient Analysis is performed by choosing the Electrostatic solution type and selecting the Electric Transient option.

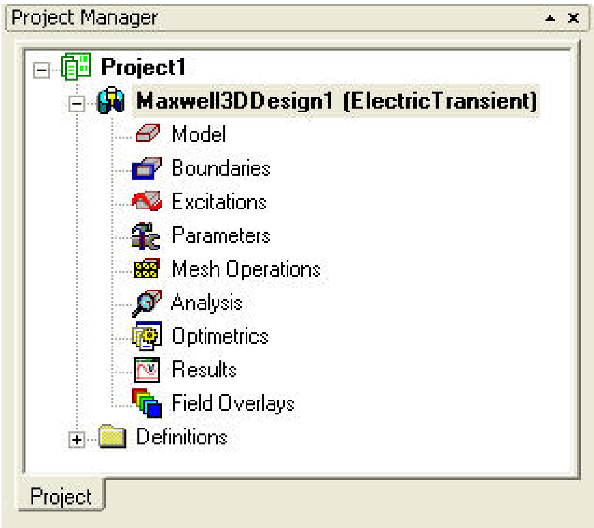
- Applications that use Electric Transient Analysis can be capacitors, power supplies, and many others.

Overview

- The Electric Transient solver computes currents in conductors and time varying electric fields.
- All objects are stationary.
- The source of the time varying electric field can be:
  - Applied potentials.
  - Charge distributions.
  - Voltages at different ends of solid conductors.
  - Currents applied on surfaces of conductors.
- The quantity solved is the electric scalar potential (ø).
- Time varying values of Current density (J), Electric Flux Density (D) and Electric Field (E) are automatically calculated from the electric scalar potential (ø).
- Material permittivities and conductivities can be anisotropic.

Setup

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

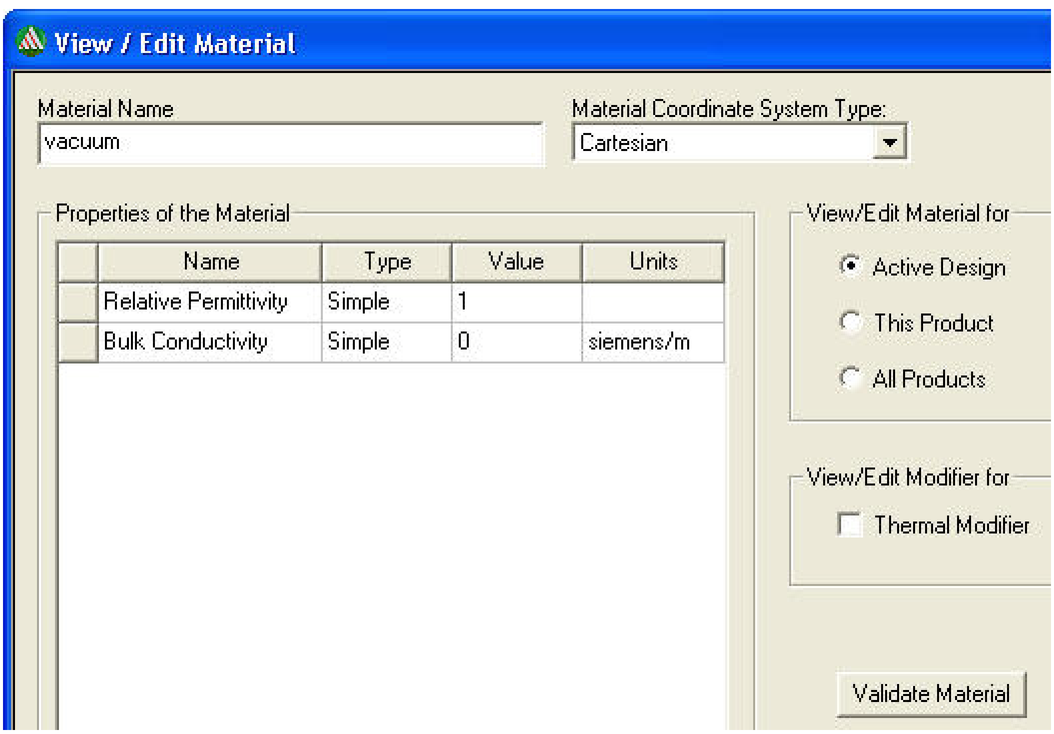


1. The **Model** definition refers to the geometry and material definition.
2. **Boundaries** and **Excitations** refer to the specific boundaries and excitations available in a Electric Transient simulation.
3. **Parameters** are values that the solver will automatically calculate after finding the field solution.
4. **Mesh Operations** are discussed in a separate section.
5. **Analysis** defines the solution setup.
6. **Optimetrics** defines any automatic variational analyses.
7. **Results** and **Field overlays** are discussed in a separate section.

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

Electric Transient Material Definition

- In a Electric Transient simulation, the following parameters may be defined for a material:
  - **Relative Permittivity** (can be Anisotropic or Simple)
    - Relative permittivity determines the electric field solution in the insulators.
  - **Bulk Conductivity** (can be Anisotropic or Simple)
    - Bulk Conductivity defines conductivity of the material.



Electric Transient Boundary Conditions

- **Default** – The default boundary conditions for the Electric Transient solver are:
  - **Natural** boundaries on the interface between objects.
    - This means that the normal component of the current density at the boundary is continuous.
  - **Neumann** boundaries on the outer boundaries.
    - This means that the E Field is tangential to the boundary and flux cannot cross it (current cannot leave conductors).

- **Insulating** – This boundary defines a thin, perfectly insulating sheet between touching conductors. This is particularly useful to separate distinct conductors (defined on surfaces between the objects).

- **Symmetry** – There are two Electric symmetries – even symmetry (flux tangential) and odd symmetry (flux normal). Even symmetry defines E to be tangential to the boundary (this is equivalent to the default boundary condition on the outer boundary). Odd symmetry defines E to be normal to the boundary. Remember that geometric symmetry may not mean electric symmetry in all cases. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time. Other considerations for a Symmetry boundary condition:
  - A plane of symmetry must be exposed to the background.
  - A plane of symmetry must not cut through an object drawn in the 3D Modeler window.
  - A plane of symmetry must be defined on a planar surface.
  - Only three orthogonal symmetry planes can be defined in a problem

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

Electric Transient Excitations

- Typical sources for Electric Transient problems are as given below
- **Voltage Excitations -** These are used in conjunction with the material conductivity to define the current through a solid conductor. Either multiple Voltage excitations can be used to define a voltage difference across two faces of a conductor (creating a current) or a Voltage can be defined along with a current excitation to define a voltage reference for the electric field solution.
- **Current Excitations** - This excitation can be assigned on any conductor to define the total current (amp-turns) through the cross-sectional face of a conductor.
- **Sink** – This excitation is used when only current excitations are defined in a conduction path and there is no voltage excitation. This excitation ensures that the total current flowing through the outside surface of a conduction path is exactly zero.
- **Charge** – The total charge on a surface or object (either a conductor or dielectric).
- **Floating** – used to model conductors at unknown potentials
- **Volume Charge Density** – The charge density in an object.

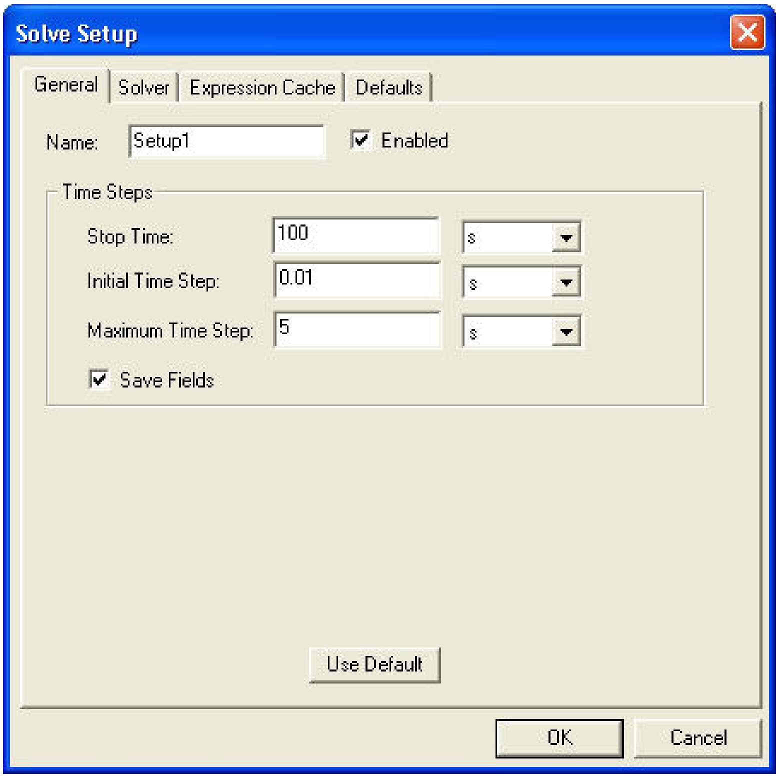
Mesh Operations

- Mesh operations are described in detail in the Mesh operations section.
- Remember that transient solvers do not provide capability of Adaptive meshing. Hence appropriate mesh operations are required before running the solution. Users can also solve the problem using DC Conduction solver to get refined mesh which can be used by electric transient solver.

Solution Setup

- The solution setup defines the parameters used for solving the simulation.
- Add a solution setup by selecting ***Maxwell 3D > Analysis Setup > Add Solution Setup…*** or click on the icon.

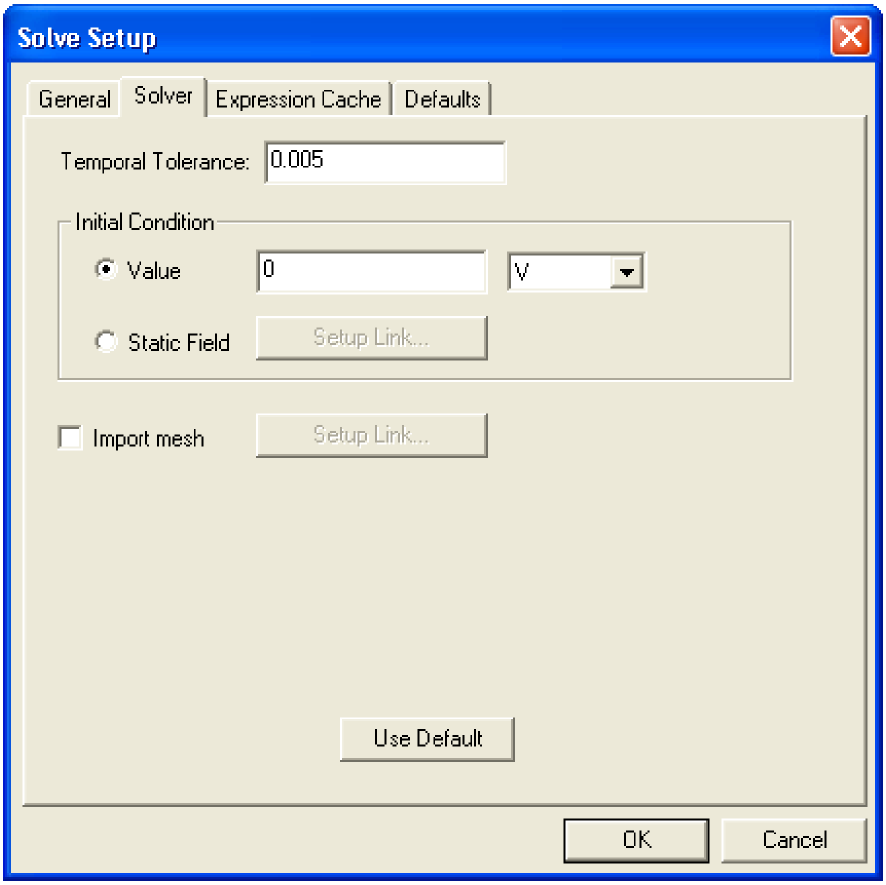
- The following window appears with the General Setup parameters.



- You can **Name** the setup, and you can create multiple setups if you desire (by repeating this procedure).
- **Stop Time** defines the time interval for which the solution is calculated
- **Initial Time Step** defines time step size used by solver at the start of the solution
- **Maximum Time Step** defines maximum size of the time step used by solver
- **Solve Fields** option enables to save fields at each time step enabling to analyze results

Solution Setup (Continued)

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



- **Temporal Tolerance** define tolerance in time sizes
- **Initial Condition** can be a preset **Value** or the solution of a **Static Field. Setup Link** tab can be used to link a static field
- **Import Mesh** allows the initial mesh to be imported from another solution – the linked solution must have the exact same geometry as the current simulation. **Setup Link** must be defined when selecting **Import Mesh**.

Mesh Operations

- This chapter provides details on meshing in the ANSYS Maxwell v15 software. It discusses the default process of creating a mesh, meshing of curvature, user control of meshing, and intermediate methods of manual mesh refinement. The following topics are discussed:
  - Initial Mesh Process
  - Adaptive Mesh Process
  - Mesh Considerations and Impact on Solutions

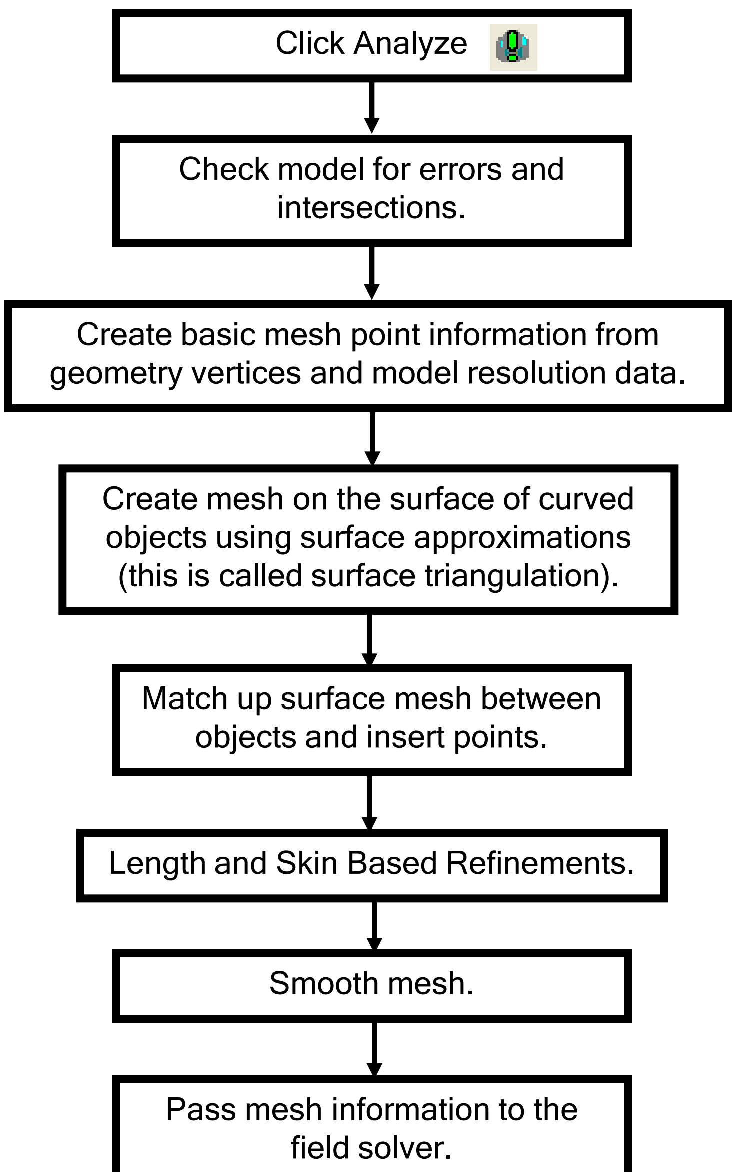
  - Applying Mesh Operations
    - Surface Approximations
      - Curved Geometry Mesh Adaptation
    - Model Resolution
    - Cylindrical Gap Treatment
    - Length Based Mesh Operations
    - Skin Depth Based Mesh Operations

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

  - Mesh failures and suggestions

Initial Mesh Process

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

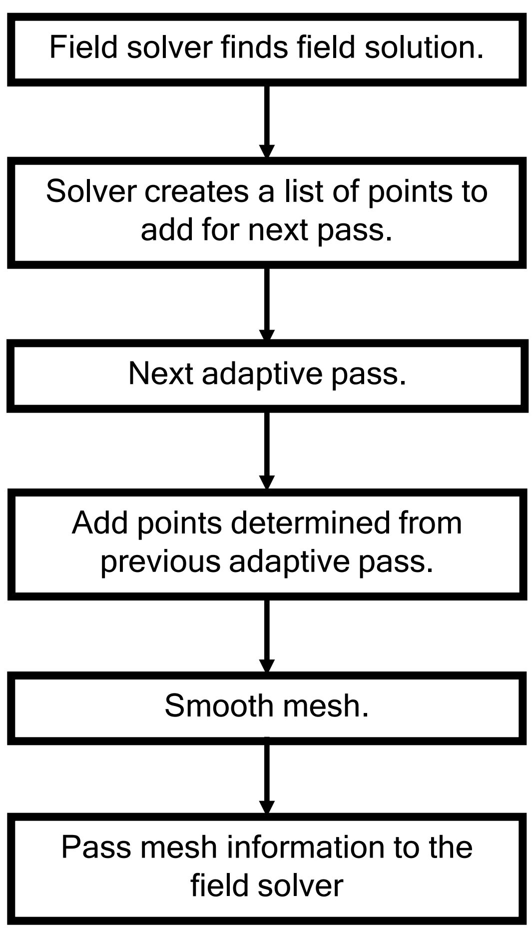


Adaptive Mesh Process

- The adaptive mesh process takes numerous factors into consideration when deciding where to refine the mesh at each pass. There are geometry factors, field solution factors, and everything is based on the percent refinement number defined in the analysis setup dialog.

- If the percent refinement is set to 30%, and there are 1000 elements at pass 1, then 300 points will be added for pass 2. This increase may not be exactly 30% due to smoothing and other factors that will adjust the actual percent refinement.

- The refined mesh points are calculated by the field solver and placed at points where there are strong fields, large errors, large field gradients, or areas that generally have a large impact on the field solution. This list of refined points is then passed on to the meshing procedure which places the points and creates an optimal refined mesh for the next adaptive pass.



Mesh Considerations and Impact on Solutions

- The mesh is important for two separate, yet related reasons – first, the mesh is used to directly determine the numerical field solution, second, the mesh is used to produce all secondary results such as volume integrations or other field calculations.

- There is a trade-off among the size of the mesh, the desired level of accuracy and the amount of available computing resources.

- The accuracy of the solution depends on the size of each of the individual elements (tetrahedra). Generally speaking, solutions based on meshes using thousands of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each element must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

- However, generating a field solution involves inverting a matrix with approximately as many elements as there are tetrahedra nodes. For meshes with a large number of elements, such an inversion requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.

- Generally, uniform mesh elements with equilateral triangular faces are best suited for second order interpolated field solutions. However, these triangles are not easy to produce with complex geometries. The important thing to remember is not to over-define the mesh in any region (this is especially true for non-transient simulations). It is often necessary to define the mesh on specific surfaces or volumes for further calculations, however there are efficient and inefficient methods to achieve good results.

- Inefficient meshes pick up all the details of every curve and joint, even in unimportant areas for the field solution. There are methods that we will discuss that can decrease the mesh in areas of low importance and promote overall convergence of the field solution.

The Initial Mesh and Why Mesh Operations are Often Necessary

- The initial mesh is created only by taking into account the design's geometry. For example, if only one box is included in the simulation, there will be 5 elements generated inside the box in the initial mesh. Only the corners of the geometry are used to create the initial mesh – this is constructed by an initial mesh-maker that only looks at the geometry and knows nothing about the field structure (there is no solution yet, so there can be no knowledge of the field solution).

- It is often the case that the initial mesh is too coarse in the regions of interest to produce an efficient, accurate field solution (this is certainly true in transient simulations). Mesh operations are able to define a manual refinement to the initial mesh that can improve simulation time and provide enhanced solutions in some cases.

- It is sometimes the case that the initial mesh is overly defined in some places, due to joints in the geometry or difficult to mesh areas. Mesh operations can better define the initial mesh in the entire solution region so that the mesh is most efficient and improvements are possible for both simulation time and solution accuracy. This can even allow the simulation of geometries that would be impossible without mesh operations.

- Sometimes the initial mesh has difficulties that can be fixed by assigning mesh operations. This means that problems that are not solvable with the default initial mesh can sometimes be solved with the addition of a few mesh operations (or with mesh considerations in mind during geometry creation).

Applying Mesh Operations

- If you want to refine the mesh on a face or volume you do not necessarily have to generate a solution. Do either of the following after defining mesh operations to apply mesh operations:
  - Select ***Maxwell 3D > Analysis Setup > Apply Mesh Operations*** or right-click on the setup name in the project tree and choose **Apply Mesh Operations**
  - Analyze the design – the mesh operations will take effect when creating the mesh for the current adaptive pass.

- For non-transient simulations, the following behaviors can be expected when applying mesh operations using either of the above methods:
  - If a current mesh has been generated, Maxwell will refine it using the defined mesh operations.
  - If a current mesh has not been generated, Maxwell will apply the mesh operations to the initial mesh.
  - If an initial mesh has not been generated, Maxwell will generate it and apply the mesh operations to the initial mesh.

- For transient simulations, the mesh must be the same from one time step to the next, therefore, adjusting mesh operations of a transient simulation will force the simulation to start from time zero.

- Select ***Maxwell 3D > Analysis Setup > Revert to Initial Mesh*** to clear the mesh information (as well as any solution information) and revert to the initial mesh. This can also be accessed by right-clicking on the setup name and choosing **Revert to Initial Mesh**.

Applying Mesh Operations (Continued)

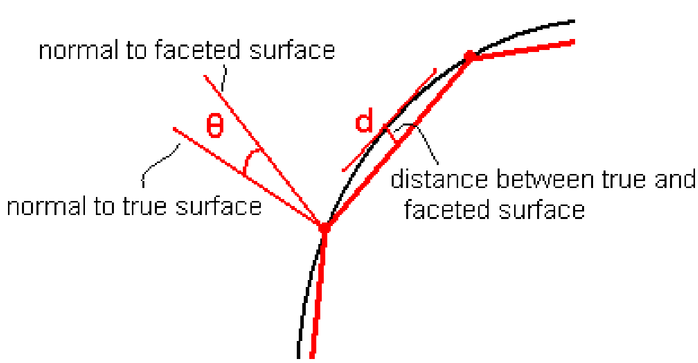
- Note the following:
  - If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.
  - Define a new mesh operation rather than modify an existing mesh operation. Maxwell will not re-apply a modified mesh operation.
  - Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them.
  - Model Resolutions adjust the effective geometry used for the initial mesh (and the mesh maker used in solving the problem), so, applying a model resolution will invalidate any solutions and revert to the initial mesh.
  - Surface approximations only work on the initial mesh. If a surface approximation is applied with an existing current mesh, it will not take effect until the mesh has been cleared.
  - Cylindrical Gap Treatment works only on the band objects which have geometry containing inside it. If there is not geometry found inside the selected object, Cylindrical gap Treatment can not be applied.

- You can look at the mesh by selecting ***Maxwell 3D > Fields > Plot Mesh…*** or by right-clicking on **Field Overlays** in the project tree and choosing **Plot Mesh**. A setup must be created to define the plot. An object or surface must be selected to create the mesh plot. Read the section on Data Reporting to find out more about field plotting on 3D geometries.

- You can view mesh statistics by right-clicking on the setup name and choosing **Mesh Statistics** (except for Transient simulations), or by going to the Solution Data or Analysis Profile and choosing the Mesh Statistics tab. This mesh statistic information represents information about element length (the length of the sides of each element), the tet volume (the volume of the tetrahedral mesh elements), and the total number of elements in each object.

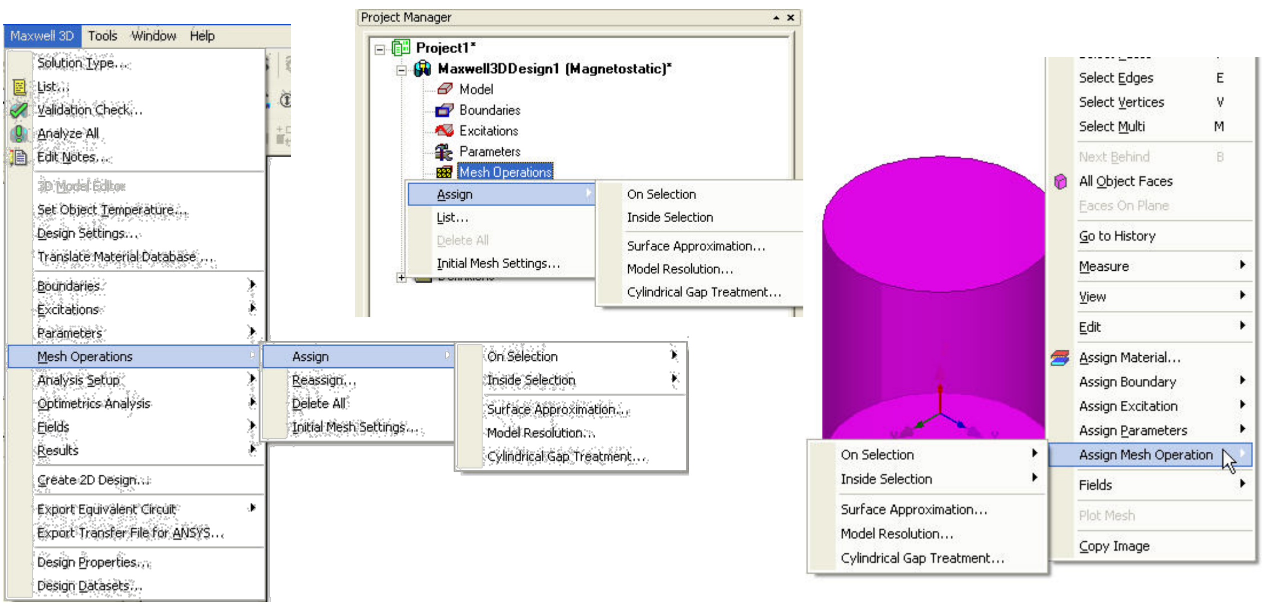
Surface Approximations

- Object surfaces in Maxwell may be planar, cylindrical or conical, toroidal, spherical or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, Maxwell first divides all true surfaces into triangles. These triangulated surfaces are called faceted surfaces because a series of straight line segments represents each curved or planar surface.
- For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.
- The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees (15deg is the default).
- The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner (see the aspect ratio comments after the usage suggestions for a detailed explanation).
- You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click ***Maxwell 3D > Mesh Operations > Assign > Surface Approximation***.)
- The surface approximation settings are applied to the initial mesh.
- Below is a diagram that illustrates the surface deviation, "**d**", and the normal deviation, "*θ*".

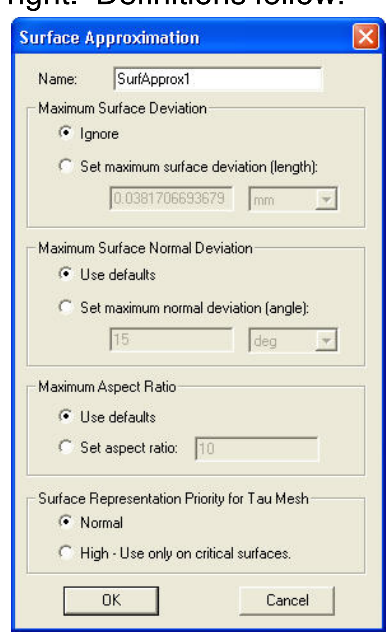


Surface Approximations (Continued)

- Since refining curved surfaces is not always enough to produce an efficient solution, users can control the fidelity to which the initial mesh faceting conforms to geometric curvature by assigning Surface Approximations to appropriate objects and/or object faces
- Mesh Operations can be assigned from the ***Maxwell 3D*** menu, from the Project Tree, or from the geometry interface's context-sensitive menu.



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

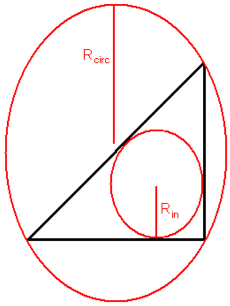


Surface Approximations - Usage Suggestions

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

Surface Approximations – Aspect ratio comments

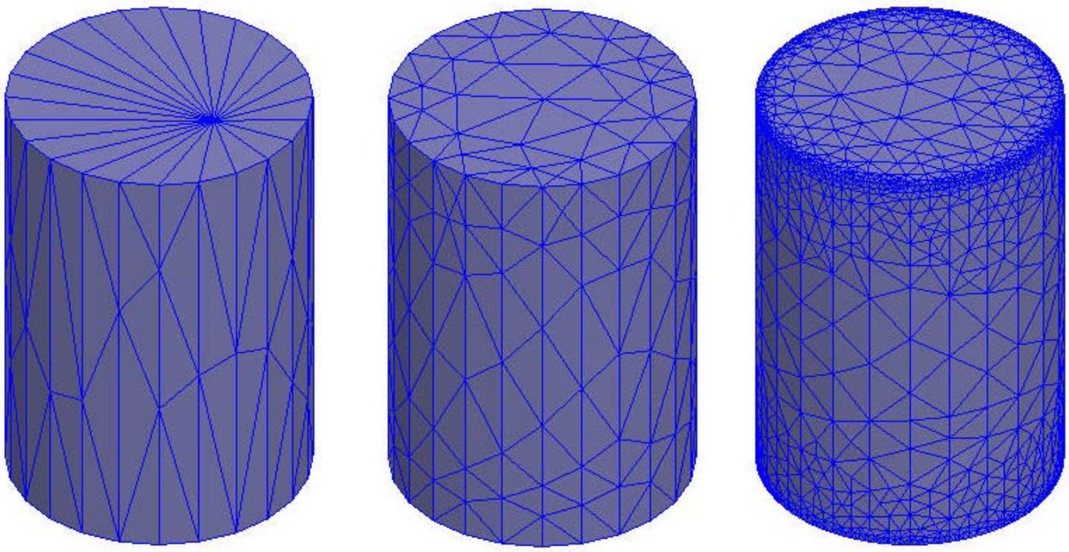
- There is a different definition of aspect ratio on curved and planar surfaces, but in either case, the aspect ratio is defined by the triangles on the selected surfaces.
  - On curved surfaces, the aspect ratio is essentially defined by the height to width ratio of the triangles. The default minimum aspect ratio is around 1.2 and will tend to form nearly isosceles right triangles on the faceted faces. The default aspect ratio on curved surfaces is 10.



  - On planar surfaces, the aspect ratio is equal to the circumscribed radius over twice the inner radius of each triangle – this produces 1 for an equilateral triangle. (See the diagram at left to see what is meant by circumscribed circle and inner circle of a triangle.) The default minimum aspect ratio is around 2. The default aspect ratio on planar surfaces is 200.
- Specifying an aspect ratio of 1 will not produce perfectly equilateral triangles.

Curved Geometry Mesh Adaptation

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



- This example documents a cylindrical, linear magnet oriented in the Z-direction, with a 200% padded region. The pictures correspond to the initial pass, the 10th pass, and the 20th pass, with total mesh sizes of 733, 7074, and 95095 mesh elements for each respective pass.

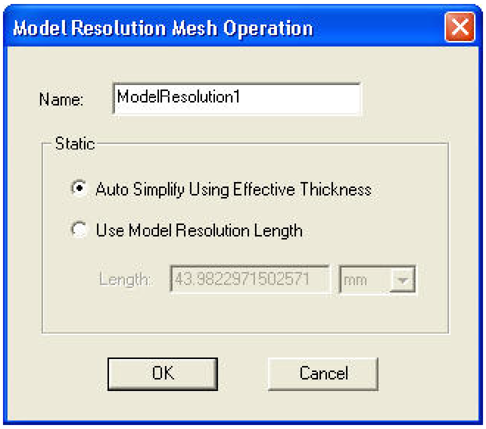
- The increase in mesh size due to the refinement of the cylinder may increase model fidelity, but it may also increase the solution time.

- Note that if you wish to maintain the initial faceting, you can use faceted primitives during geometry creation.

Model Resolution

- In many models (especially imported models), there are small geometric details that are present, yet do not affect the field solution to a large extent.
- Small features in the geometry can lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower.
- For these reasons and more, Model resolution allows the initial mesh to generate without including all the little details that can slow down simulations and do not add to the quality of the solution.
- Model resolution improves model convergence and reduces solve times. If in previous versions the tetrahedron count got too large, the solver could run out of memory. It would be trying to add meshing elements to areas of the model that are not important. Now with model resolution, we are able to crank down the mesh size in order to actually get convergence before memory becomes an issue.
- Model resolution is a length based value that modifies the initial mesh. This meshing operation allows the user to specify a minimum edge length of any tetrahedron used for the mesh. By specifying a minimum edge length for a tetrahedron, the mesher will have to coarsely mesh geometric detail that may not be important for the field solution. This saves the solver a tremendous amount of solution time since the initial mesh is smaller, and the mesher does not have to add mesh elements to areas that are not important for the field solution.

- Model resolution can be assigned from the ***Maxwell*** menu, from the **Project Tree**, or from the geometry interface's context-sensitive menu, and is found immediately below Surface Approximation in the Mesh Operations list.
- The Model Resolution dialog is presented below. The suggested value for the resolution length is determined by the smallest edge length on the selected objects. This may not be the best value to use, but it might be a good place to start or, at least, a good reference.



Model Resolution (Continued)

- Model resolution capabilities are an expanded capability of the ACIS engine and are incorporated into the new fault tolerant meshing features. By default, ACIS uses 1.0e-6 model units as the smallest possibly resolved length. Any two geometry points that are closer than this absolute resolution length (ResAbs) will not be resolved by ACIS (so choose the model units appropriately).

- By default, if there are difficulties creating an initial mesh, a second initial mesh will be attempted with a model resolution automatically applied to all model objects with 100 times the absolute resolution length (ResAbs). If continued difficulties are experienced, a third initial mesh will be attempted with an automatic model resolution of 1000 times ResAbs on all model objects. After the third attempt a mesh failure is reported (you will be able to see three distinct "mesh_init" lines in the analysis profile).

- If the user specifies the model resolution length so that the mesh grossly misinterprets the model or changes the contacts between the objects, Maxwell will detect this and report this as an error (*"MRL too large or object_name lost all surface triangles"*).

Model Resolution – Usage Suggestions

- Apply model resolution to one specific object or a group of objects. Be aware of what the size of important details are when assigning a new model resolution to an object or group of objects. Measure various lengths (***Modeler > Measure***) to survey some nominal lengths for the specific objects. Then, when applying the model resolution, look at the number suggested in the model resolution assignment dialog – this number is the smallest edge found on the selected objects. If this edge is very much smaller than some of the measured and expected lengths, apply a model resolution that is larger than the suggested value, yet significantly smaller than the expected length (for example, if you have an object that is cube-like with 10in sides, yet the smallest length is returned as 1e-6in, you should apply a model resolution of about 0.1in to the object).

- If the mesh fails with a vague error, occasionally, applying a model resolution to a complicated object will assist the initial mesh generation.

- Model resolution is very useful when dealing with imported geometries.

- When dealing with models that have very high aspect ratios due to small geometric detail, use a model resolution of 1/10 to 1/20 of the thinnest conductor to start with. Then adjust the value accordingly.

- It is often good to be conservative with model resolutions as a large model resolution can cause mesh errors.

- Model resolution is often most useful when fillets, chamfers, and other small features are included at joints and corners – these can often be ignored for electromagnetic simulations and are often small enough to use an appropriate model resolution.

- Always create the initial mesh and look at the surface mesh on objects before solving the problem. If too aggressive of a model resolution is used, then important features can be skipped with a model resolution – it is important to know that the mesh is appropriate for the simulation.

Cylindrical Gap Treatment

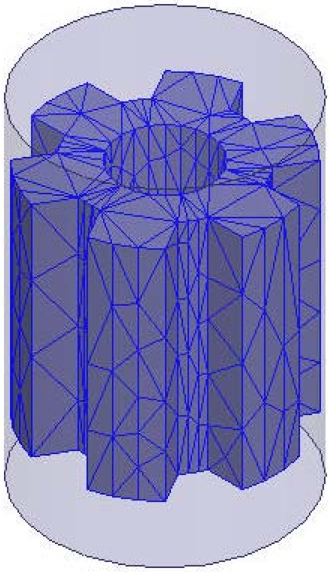
- In The geometries of motors or any geometries involving rotational motion simulation, normally a Band object is created to separate the moving objects from stationary objects.
- In most of the cases, the gap between band object and Moving/Stationary objects is very thin. This thin air gap needs to resolved well in meshing in order to achieve good convergence
- The mesh operations become more important in cases of motion simulation with transient solver which does not provide capability of adaptive mesh refinement
- In such cases, it is always better to use Cylindrical Gap Treatment mesh operation in order to refine the mesh in thin air gap region

- Cylindrical Gap Treatment mesh operation when assigned, will search for the proximity of the faces between the object to which operation is assign and the objects inside it. The refinement of mesh is done based on the closeness of the faces in order to have better mesh in proximity region.
- Note that in order to assign this mesh operation the assigned object should contain some geometry inside it. If there is no geometry inside the object, this mesh operation can not be assigned.

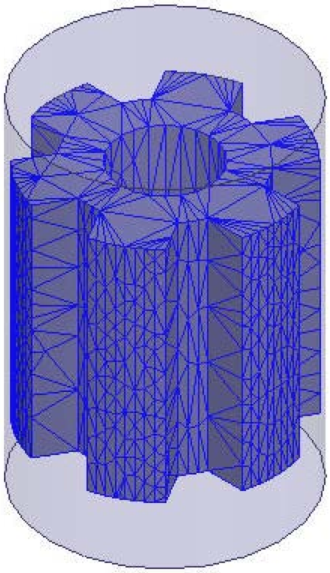
- Cylindrical Gap Treatment mesh operation with transient solver is automatically assigned once the motion setup is done. To set the motion, seletc the menu item ***Maxwell 3D > Model > Motion Setup > Assign Band***. Once the band object is specified with rotational motion, a cylindrical Gap Treatment mesh operation is automatically setup for the band object. Note that the mesh operation is not created for translational motion.
- For steady solvers this mesh operation can be assign through menu item ***Maxwell 3D > Mesh Operations > Assign > Cylindrical Gap Treatment***. Same operation can also be applied by right clicking on **Mesh Operations** tab in Project Manager tree

Cylindrical Gap Treatment (Continued)

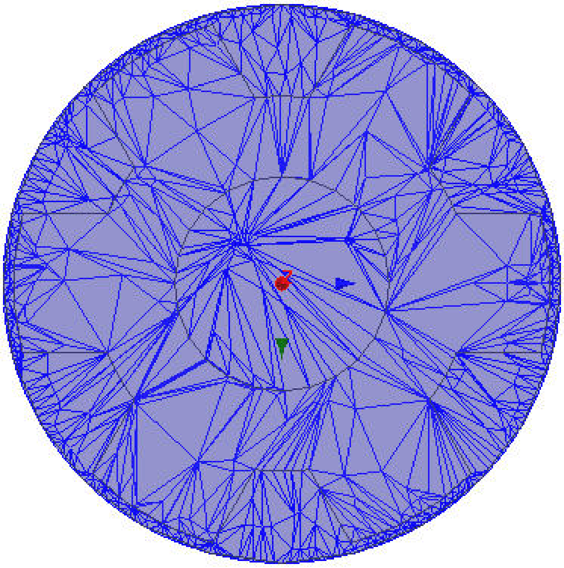
- Below shown is a simple case with Cylindrical Gap Treatment mesh operation specified on the object. The gap between inner object and the band object is very small and resolved using Cylindrical gap treatment



Initial Mesh



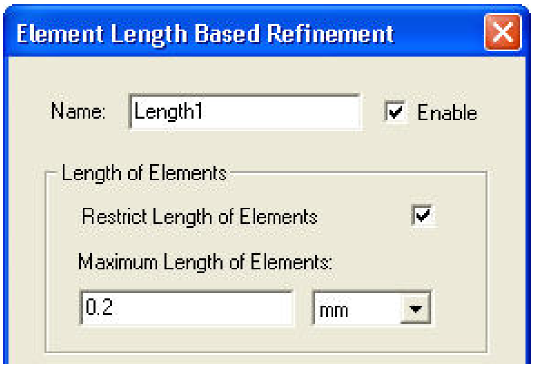
Refined Mesh with Cylindrical Gap Treatment



Mesh section seen from top

Length Based Mesh Operations

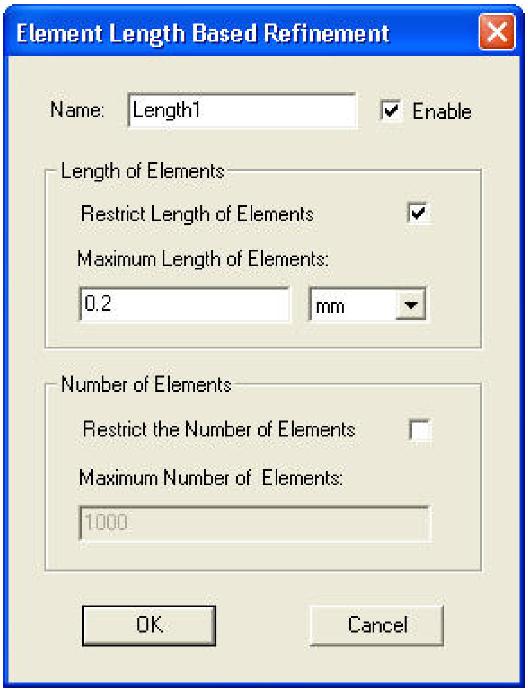
- Length based mesh operations are perhaps the most basic conceptual mesh refinement option. The assignment of a length based mesh operation limits the edge length of all tetrahedral elements inside an object or on an object's surface.
- There are two types of length based mesh operations:
  - On Selection
  - Inside Selection
- The Length-based On-selection refinement will limit the edge length of all triangles formed on the surface of a selected object (or group of objects), or any selected faces.
- The Length-based Inside-selection refinement will limit the edge length of all tetrahedral elements inside the selected object (or group of objects).
- The assignment of these two types are similar – simply choose either:
  Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based … or
  Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based …
- The same dialog appears for both types of assignment - the only difference is whether it affects all edges within an object or all edges on a face (or surface).



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

Length Based Mesh Operations (Continued)

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



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

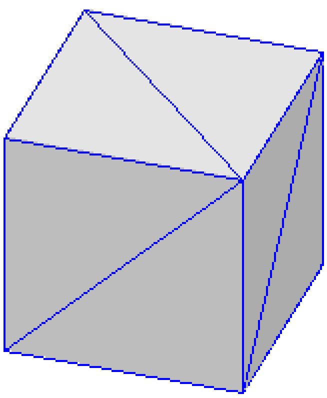
- For example: if the restriction to the length of elements requires 3000 additional elements, yet the number of elements is restricted to 1000, then only 1000 elements will be added, and the elements will not meet the Length of Elements requirement (they will be larger than specified).

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

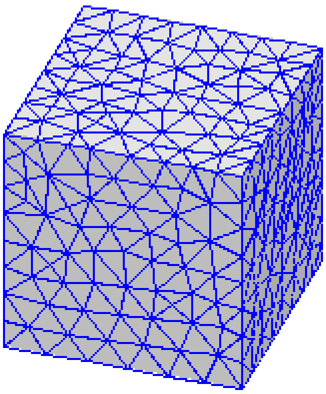
- This second method of adding elements is good and bad, because it allows control over the number of elements that are inserted into the defined objects for the particular mesh operation, however, it does not necessarily need to meet any length requirements (so the elements could still be too large).

Length Based Mesh Operations - Example

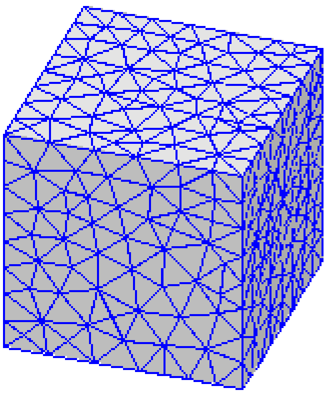
- This example uses a cube inside a Region with 100% padding. An initial mesh was created for four different cases of applying length-based mesh operations.
- First, a default initial mesh was created without defining any mesh operations.
- Second, an initial mesh was created by defining a length-based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



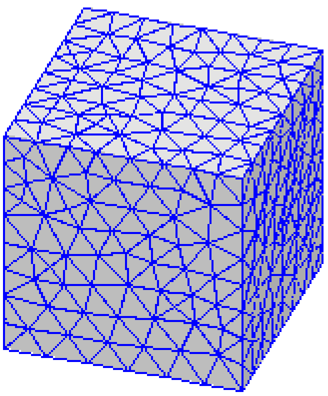
Default Initial mesh
42 elements



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



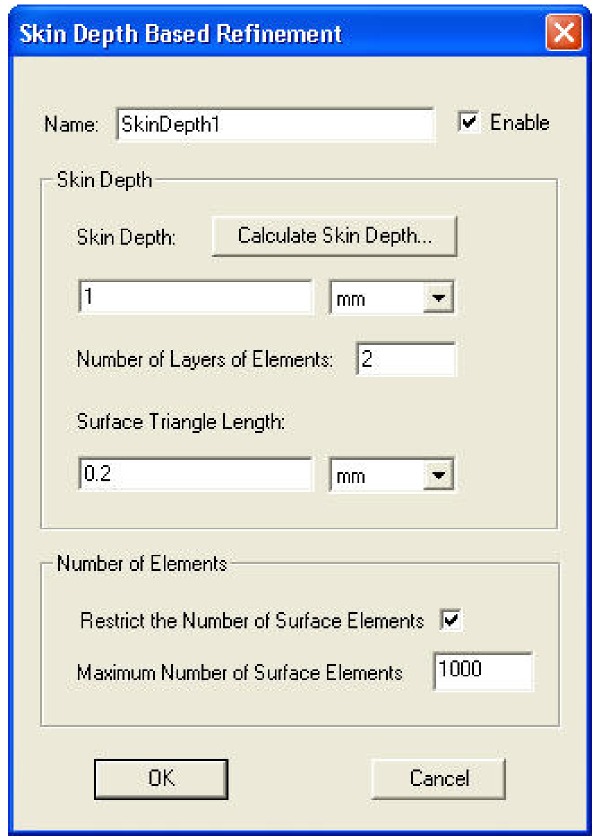
Initial mesh with Length-Based
Inside-Selection set to 20%
6698 elements



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

Skin Depth Based Mesh Operations

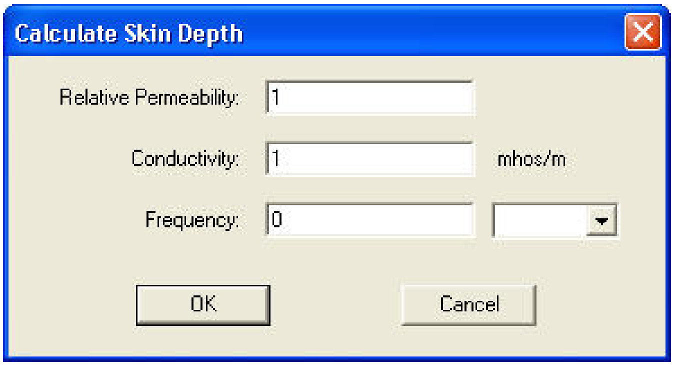
- Skin depth based mesh operations are similar to On-selection length-based mesh operations. These operations are defined by both the triangles on the surface and a seeding inside the selected faces.



- This refinement method creates layers of mesh within the selected surfaces of objects – this is useful for modeling induced currents near the surface of a conducting object.
- **Skin Depth** defines the classical skin depth value (discussed on the next page).
- **Number of Layers** defines the number of layers of points that are placed at distances according to the skin depth value.
- The **Surface Triangle Length** is equivalent to the surface length restriction for length-based refinement.
- The **Number of Elements** is equivalent to the surface element restriction for length-based refinement. The total number of elements added may exceed this number by a factor of the Number of Layers.

Skin Depth Based Mesh Operations (Continued)

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



- The skin depth of an object is calculated as:

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

- Currents are concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.

- The skin depth based mesh operation is most important in eddy current and transient simulations where eddy currents and proximity effects are important to the solution of the simulation.

Mesh Reduction Techniques

- Because the mesh size affects the simulation time, it is often desired to decrease the total number of elements in a simulation to also decrease total simulation time. This can be done either at the expense of accuracy or not depending on the efficiency of the mesh construction. There are several ways to reduce the mesh size – here are a few examples.

1. Mitigate large aspect ratios with intelligent geometry construction.
   - If you have a thin conductor that is far from other objects, this conductor should not be drawn with curved surfaces. Draw a square or regular polygon and sweep the square along the path of the conductor. This will limit the number of triangles necessary in a large aspect ratio object and will decrease the total number of mesh elements in the simulation.

2. Add a vacuum container object (dummy object) around all objects in simulation.
   - This can be a good method to limit the aspect ratios of triangles in the region. The region needs to be defined a certain distance away from the components (due to fringing fields) – so this vacuum container can assist the mesh construction and reduce aspect ratios in the region near to the structure.

3. Use a model resolution when it will reduce the size of the mesh but not affect the solution of the simulation.
   - Look at the Model Resolution usage suggestions for more information.

4. Use faceted objects instead of curved surfaces when the surface is not in an area of high fields.
   - Faceted objects are easier to mesh and do not add more surface elements in the same manner as curved objects. This can reduce the mesh significantly in many cases. Curved surfaces may provide a more true representation of objects, especially important in areas with large fields. However, using faceted surfaces even with large fields may be appropriate and may reduce the mesh in many cases.

5. Add small mesh operations on conductors and inside magnetic materials.
   - These will intelligently assist the initial mesh formation, so that less adaptive passes are necessary.

Dummy Objects

- Placing an object of the same material inside another object does not affect the field solution for the field solver. However, this additional object defines another set of surfaces on which the initial mesh will form. This object can also have additional mesh operations defined on it. For these reasons and more, adding additional objects may assist the mesh formation process, and, moreover, increase the amount of control with which you can define the initial mesh. These objects are called dummy or virtual objects.

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

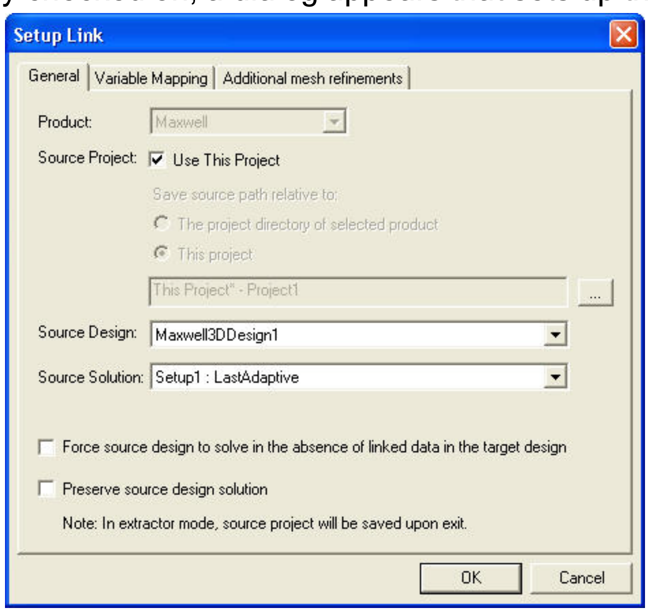
- These dummy object examples serve different purposes – the first three examples in the above list are intended to improve the general convergence of the solution, while the last three examples are intended to improve the post-processing results due to the local mesh in the vicinity of the measurement.

- When placing dummy objects, use simple objects (such as rectangles, or regular polyhedrons) when possible. The size of the dummy objects should be comparable to the objects around it. The placement is sometimes best to be placed slightly away from model objects (such as a container object for a rotational machine – which should have its radius in the middle of the gap, or equally spaced in the gap with the band object) where the placement creates an extra layer of mesh elements that allow for better calculation of changing fields. Other times the placement is best when the dummy object touches the model objects (such as in the gap of a magnetic core) where the dummy object helps to both reduce the aspect ratio in the surroundings and to provide an object to apply mesh operations if desired.

Linking Mesh - Transient

- Every solution type has a mesh linking capability. It is sometimes necessary to have a simulation linked (when using demagnetization options, or otherwise). However, it can be beneficial for mesh purposes to link a mesh from another simulation. This is nowhere as powerful as in a transient simulation.
- Because a transient simulation does not adaptively refine its mesh, the mesh that it uses is constrained the initial mesh. Mesh operations can assist in manually refining the mesh, but this can be insufficient sometimes. What is provided with mesh linking, is the power of an adaptively refined mesh used in a transient simulation.

- A transient simulation can be linked to any other simulation with the exact *__same geometry__*. This means that a magnetostatic or eddy current simulation can be adaptively solved to any accuracy, then the design can be copied and converted to a transient simulation that uses the same adaptively refined mesh.
- In the transient simulation Analysis Setup, simply check **Import Mesh**. When this box is initially checked on, a dialog appears that sets up the mesh link.



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

Linking Mesh – Non-Transient

- There are several important reasons why mesh linking capabilities are important in non-transient simulations. One example is if DC power flow calculations are performed, both an electrostatic and a magnetostatic solution are required, and the mesh must be identical to perform the necessary calculations (see the Field Calculator topic for an example). Another example is if the exact same mesh is required in a parametric sweep (especially important with distributed analysis).

- Note that if mesh operations are included in a linking simulation, the model resolution and surface approximations are ignored – however, the length-based and skin-depth based resolutions are performed. If identical meshes are desired, delete mesh operations from the linking simulation.

- You cannot link two analysis setups within the same design.

- Any simulation can be linked to any other simulation with the exact *same geometry* (regardless of solution type).

- In the Analysis Setup, simply check **Import Mesh**. When this box is initially checked on, a dialog appears that sets up the mesh link. In the dialog, either select **This Project** or browse to the desired project file. Then choose the design name within the selected project and the desired setup name within the design. Adjust any parameters if necessary and then select **OK**. The mesh in the linking simulation will now be identical to the mesh in the selected project/design/setup.

- To guarantee identical meshes in a parametric sweep (with no geometry variations) do the following:
  1. Create a nominal design and solve it.
  2. Copy the design and delete mesh operations.
  3. Set up the mesh link in the copied design.
  4. Create a parametric sweep in the linking design.
- This will guarantee that every row of the parametric sweep (assuming no geometry variations) will use an identical mesh.

- To re-initialize the linked data, select ***Maxwell 3D > Analysis Setup > Clear Linked Data*** or right-click on **Analysis** and choose **Clear Linked Data**.

Mesh Failures and Suggestions

- There are several different points at which the mesh can fail even if the design passes all model verifications (no ACIS errors, no non-manifold objects, no partial intersections).

- If an ACIS error or other geometry error occurs, please see the section on geometry import/healing.

- Information about a mesh failure will appear in ***Modeler > Model Analysis > Show Analysis Dialog > Last Simulation Mesh…***.

- Generally, the mesh will fail at the point where it tries to match the surface mesh between objects. The error for this failure will state *"Volume Meshing Failed – Stitching flag <#> failure mode <#>"*. This is often the case if surfaces are not perfectly coincident – if there is a small gap between two objects, the mesh will have difficulties filling the gap yet retaining a reasonable aspect ratio and form.

- Another semi-frequent mesh error is *"Incompatible Faceting"*. This error is almost exclusively related to coincident true surfaces. There is some information on coincident true surfaces in the geometry import/healing section.

- You can determine which part of the model is causing problems for the mesh by making an object non-model (double-click on the object to bring up the Properties dialog – then deselect the **Model** check-box). If it meshes, then that object is the problem.

- Can you move the faces of the object or move the object so that it is not coincident (or is fully coincident)? Try to change the geometry a little bit without changing the solution.

Mesh Failures and Suggestions (Continued)

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

Follow up on what the dialog says

Consider eliminating tiny details through healing

Consider aligning faces through healing

Determine which object or part of object is the trouble maker

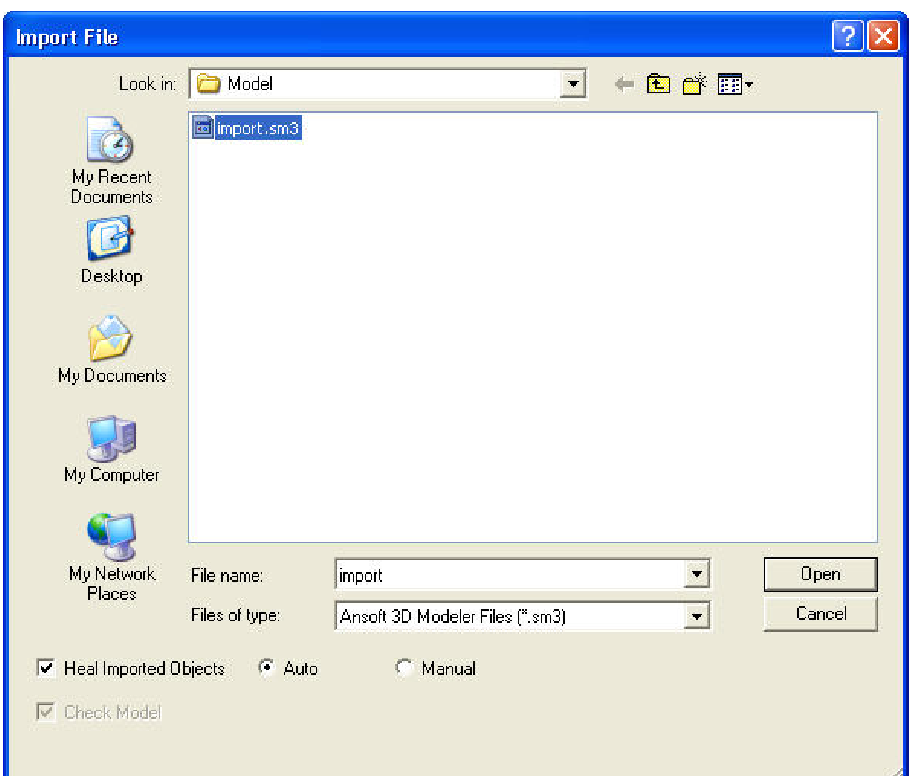
Vary Model Resolution and Surface Deviation

Try miscellaneous modifications

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

Importing a geometry into the 3D modeler and Healing of geometries

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



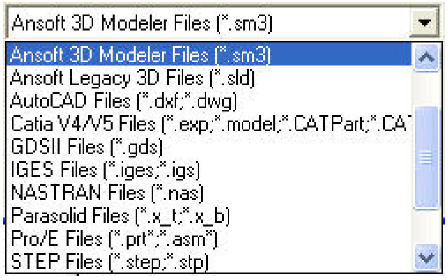
ANSYS Maxwell Design Environment

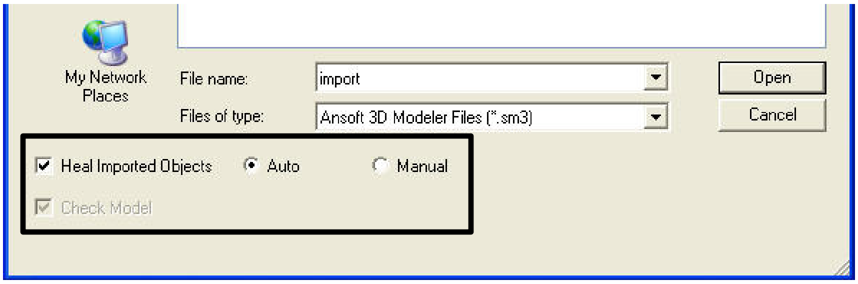
- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - Edit
    - Duplicate: **Around Axis**
  - 3D Solid Modeling
    - **Import**
    - **Purge History**
    - Model Analysis: **Analyze Objects**
      Heal
      Align Faces
    - Surface Operations: **Move Faces**
    - Boolean Operations: **Unite**
      Subtract
      Intersect
      Separate Bodies
  - Mesh Operations
    - **Surface Approximation**
    - **Model Resolution**
    - **Cylindrical Gap Treatment**
  - Analysis
    - **Validation Check**

Importing

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

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



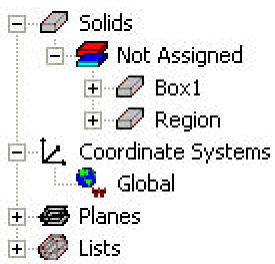


   - On the right is the first available option for healing an imported model. This will be discussed in detail later.

1. Select the desired model and then **Open**.
   1. The model should appear in the window with any defined healing applied to it.

Characteristics of Imported Objects

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



Suggestions for Exporting Geometries from Drawing Tools

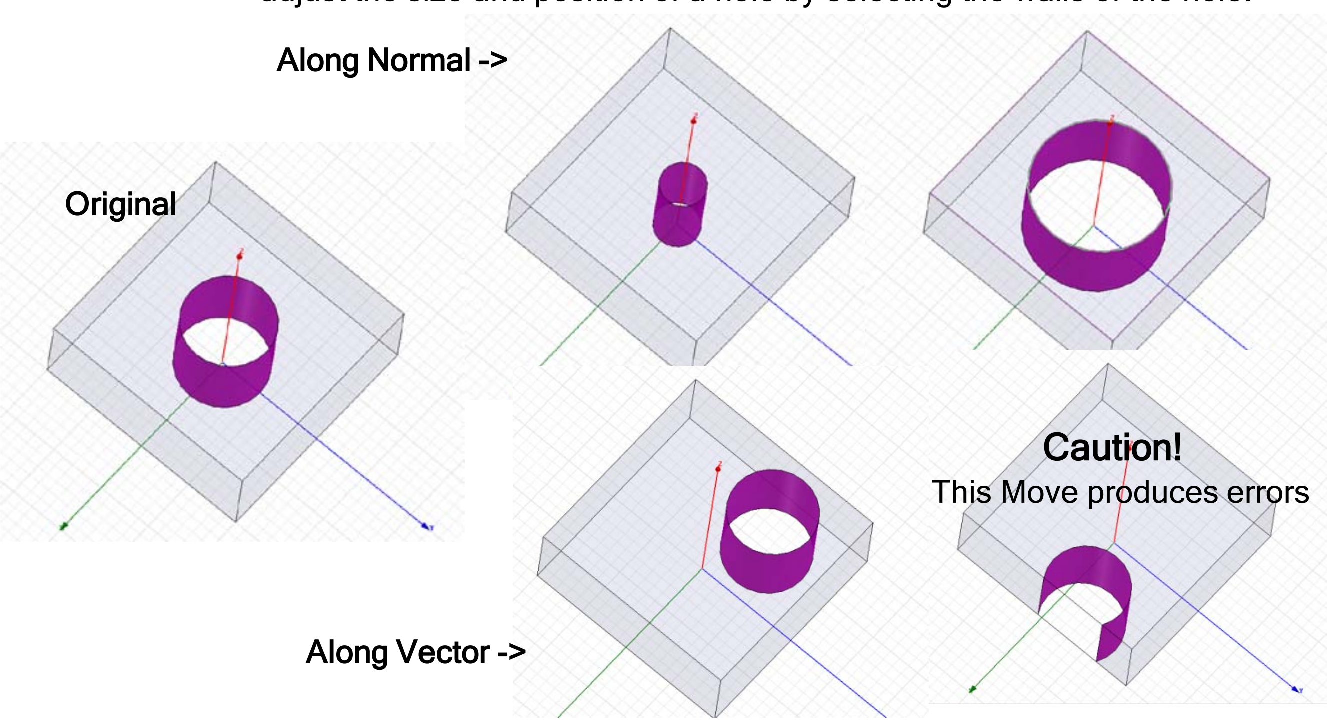
- De-feature the model to get rid of details that are not important for EM simulations (e.g. small rounds, chamfers, fillets, and mechanical connectors). Small features will result in a large and inefficient mesh.
- Use caution when exporting parts with touching (coincident) parts. Overlapping objects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

Adjusting Imported Geometries

- Several methods can be used to adjust an imported geometry – two methods are described below. Creativity can expand the following basic strategies to adjust most imported geometries and even allow parametric manipulation of these static imports.

- The first method is to use a combination of the geometry primitives (cylinders, cubes, etc.) and Boolean expressions to add, subtract, and manipulate the geometries and therefore allow access to many dimensional variables.

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



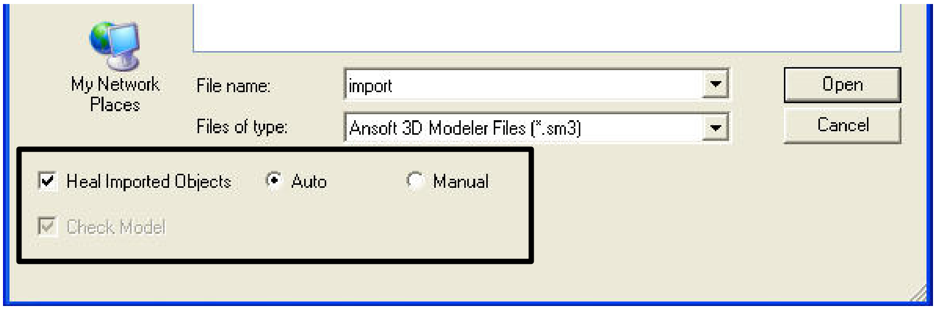
Common Model Errors

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

Imported Geometry Errors

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

Healing during Geometry Import



- In the Import File dialog there is a check box "**Heal Imported Objects**"
- This will heal small errors that occur when converting the imported file from its original format to the format of the 3D Modeler (ACIS SAT v21.0).
- There are two modes for healing the imported object – "auto" and "manual".
  - Auto healing will try to address ACIS errors and non-manifold errors, the first two classes of potential errors listed earlier.
  - Manual healing adds small-feature removal to the auto-healing. You can remove small features at this stage if you wish. However, the usual approach is to apply auto-healing at this stage and leave small-feature removal until later.

- After you import a part, you should perform a **Validation Check** as described in the next section. This allows you to target problems before the model is altered or becomes too complex.

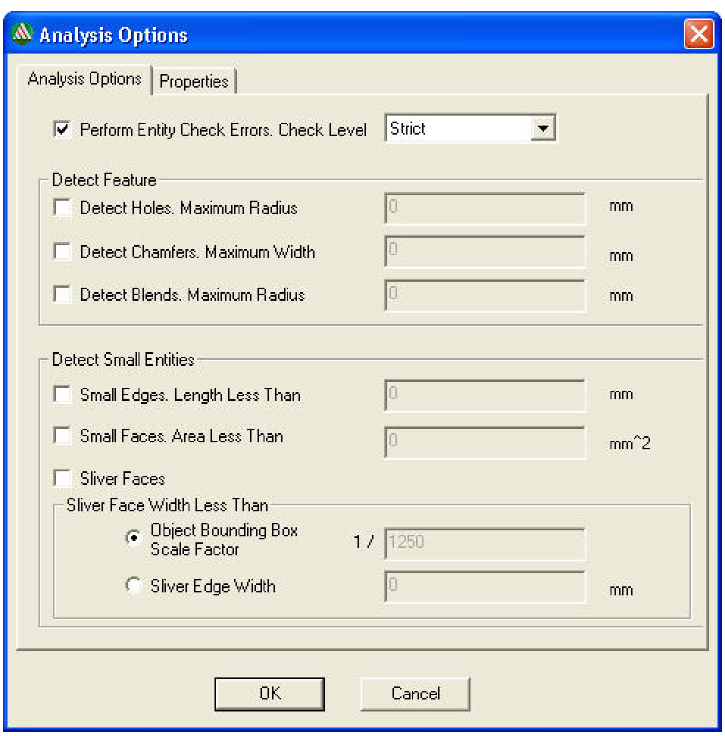
Healing after Geometry Import

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

Fixing api_check_entity() errors

- The objects that fail api_check_entity() should be analyzed via the Analyze Objects menu item.

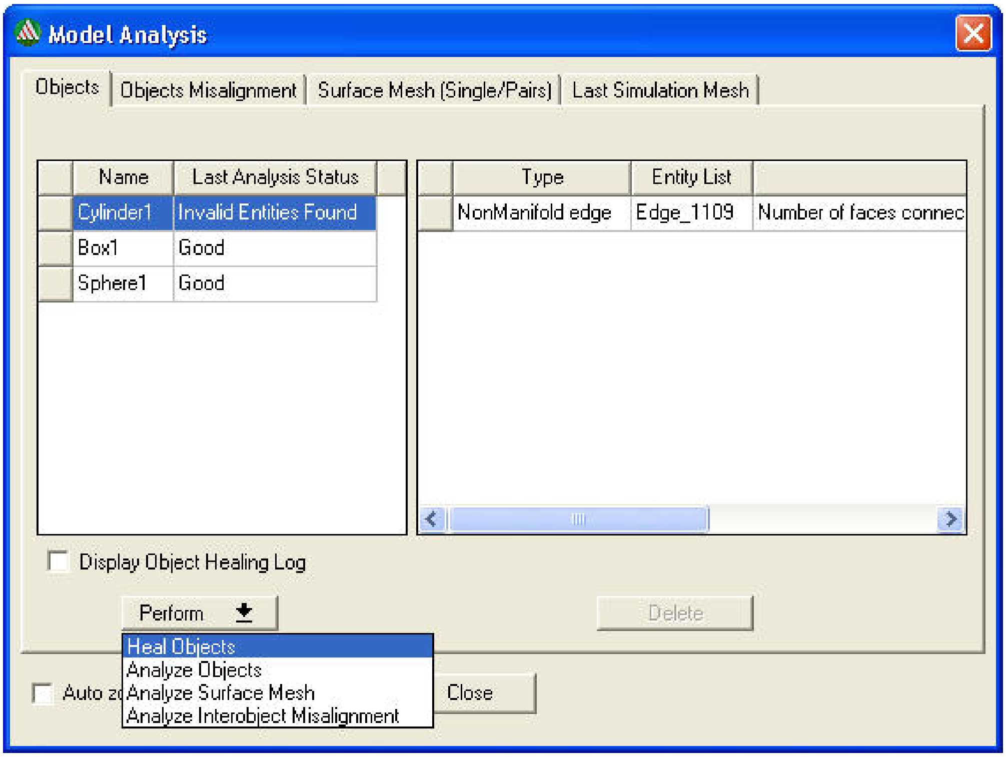
1. Select the objects that have ACIS errors.
2. Select ***Modeler > Model Analysis > Analyze Objects***.
3. The **Analysis Options** dialog box appears with options for small feature detection.



4. Select **OK** to proceed.

Fixing api_check_entity() errors (Continued)

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



- This dialog produces a list of problems affecting faces, edges, and vertices.
- There is also the option to auto zoom to regions where problems exist.

6. Select the objects marked with “Invalid Entities Found” and click ***Perform > Heal Objects***. The **Healing Options** dialog box appears. Adjust parameters as necessary. Click **OK**. The **Model Analysis** dialog box reappears.
7. In most cases, the objects are healed, and the errors are fixed.
8. If errors persist, select the edges and faces still containing errors and click **Delete**. This replaces each selected face/edge object by a tolerant edge/vertex, respectively. In some cases, the replacement of face/edge by edge/vertex fails.

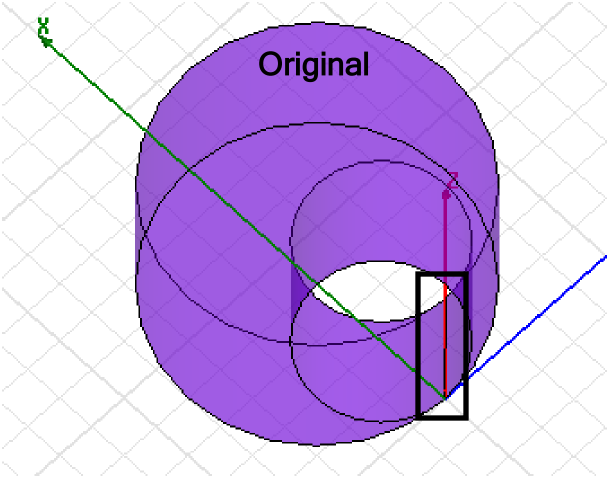
- Objects in the Model Analysis can have the following statuses: Good, Null Body, Analysis not performed, Invalid entities found, Small-entity errors.
- Invalid-entity errors are ACIS and non-manifold errors.
- Invalid-entity errors must be fixed before a mesh can be generated.

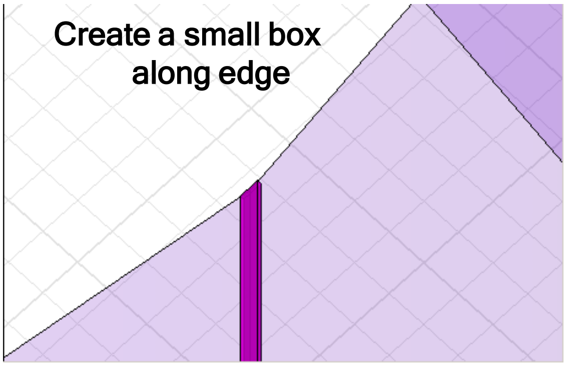
Fixing non-manifold errors

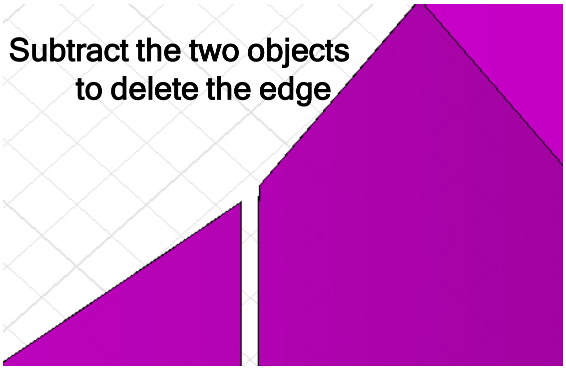
- The main strategy for fixing non-manifold errors is to slightly adjust the geometry to either add thickness to the object or delete the non-manifold edge.

- Find the non-manifold face/edge/vertex either visually or with the Analyze Objects menu item.
- Create a small box to contain the non-manifold edge.
- Either unite or subtract the non-manifold object with the small box.
- A unite operation will add thickness to the object at the required place.
- A subtract operation will delete the non-manifold edge (and a small portion of the model).

- This example shows a non-manifold edge along the z-axis





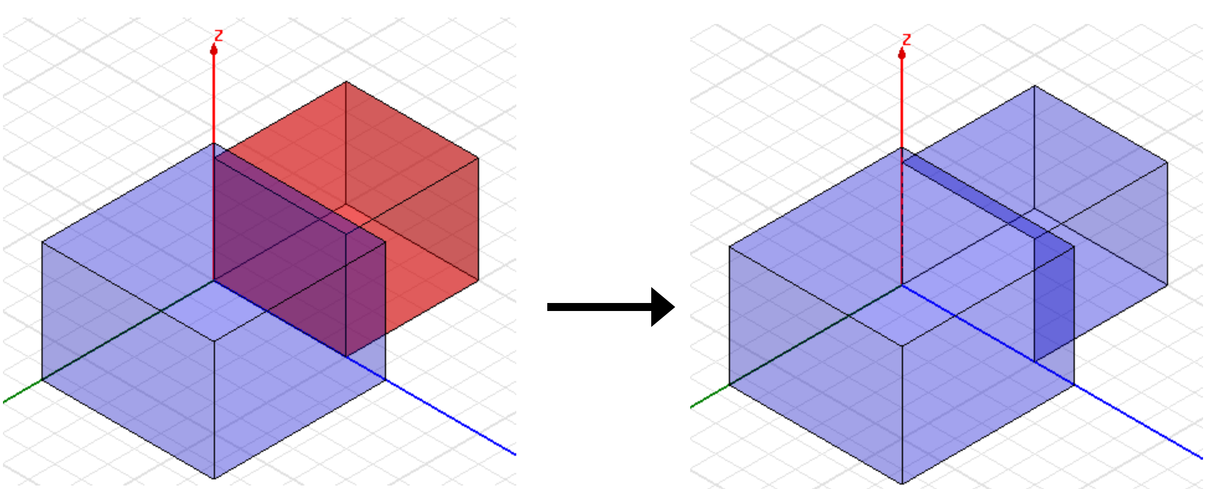


- Use engineering judgment to decide whether this edge is in an area of high fields, and is therefore important. The importance of this area can decide the size of the box and whether to add or subtract the box.

Fixing Object Pair intersection

- This type of error is easily seen in a Validation Check, where the intersecting objects are designated with an error notification in the message window.
- There are several reasons why objects could be intersecting when they are imported. If care is taken with drawing and exporting the model from the desired CAD tool (i.e. no interference fit and no overlap in the model), then the problem is most likely in the translation.
- You can identify which parts of the objects are intersecting by performing an **Intersection** of the two objects (in the case where one object is supposed to be entirely inside the other you should perform a **Subtraction** to see what part of the inner object is not within the outer object).
- There are several ways to fix this issue. They are as follows:

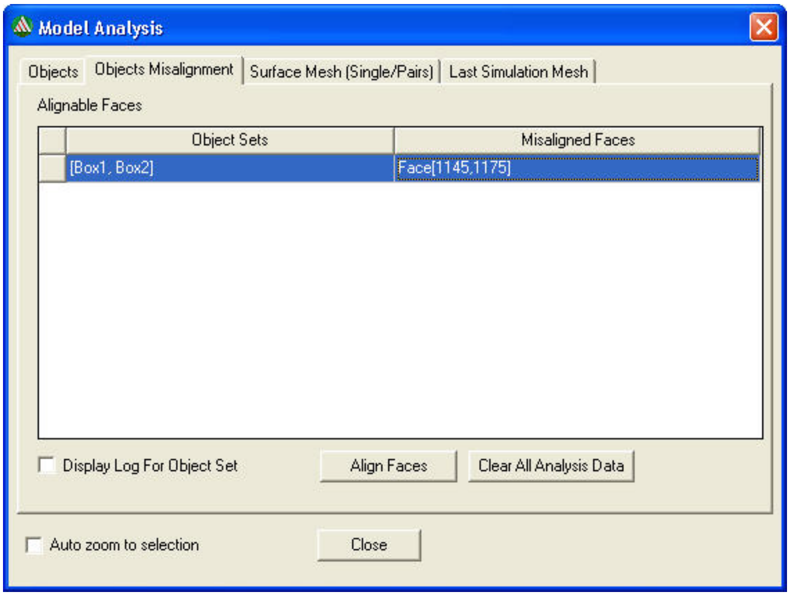
- The easiest circumstance for intersecting objects is if the objects are of the same material with no need for any distinction between the objects.
- To fix intersecting objects of the same material:
  1. Select the intersecting objects.
  2. Select ***Modeler > Boolean > Unite***.
- This will group all the selected objects and there will be no intersection.

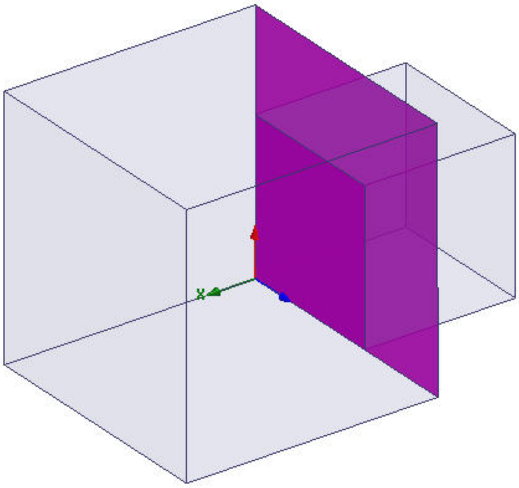


- In circumstances requiring separate objects the following techniques apply.

Fixing Object Pair intersection (Continued)

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



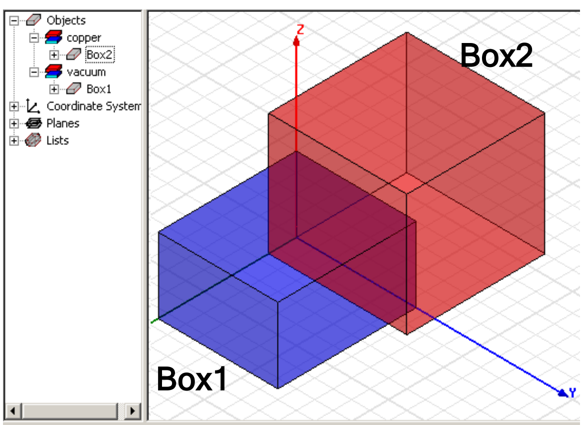


  1. Select **Align Faces** to make the faces coincident.
  2. Proceed with any other intersecting, mis-aligned faces.
  3. Select **Close**.
- This set of operations should fix the intersecting errors.
- Run a **Validation Check** to determine that the aligned objects do not intersect.

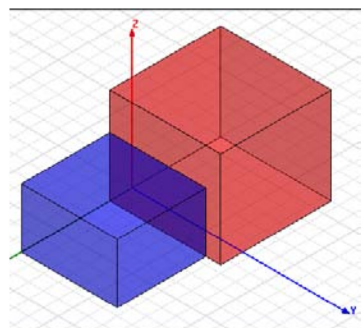
Fixing Object Pair intersection (Continued)

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

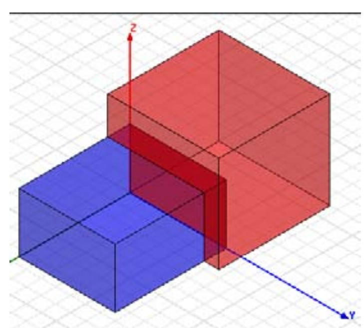
(a)



(b)



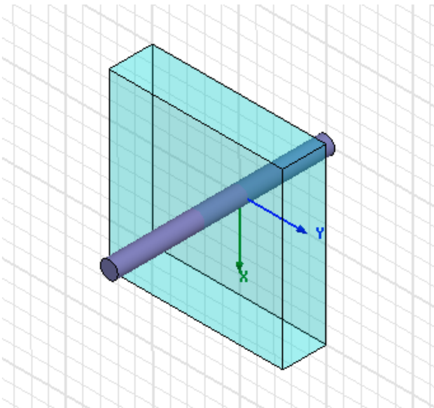
(c)

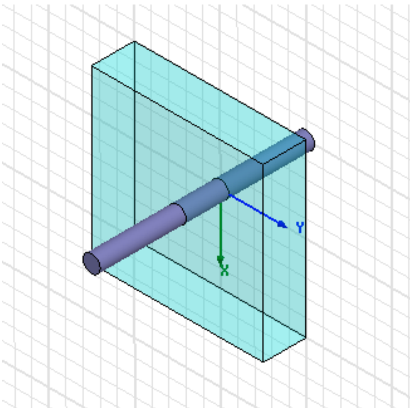


- **Note:** Either Select **Clone tool objects before subtracting** while subtracting or Duplicate the object (Select the tool object, then ***Edit > Duplicate > Around Axis*** with an Angle of 0 and a Total number of 2, or use **Ctrl+C**, **Ctrl+V** if you are in the original object's coordinate system)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

Fixing Object Pair intersection (Continued)

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





Meshing difficulties

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

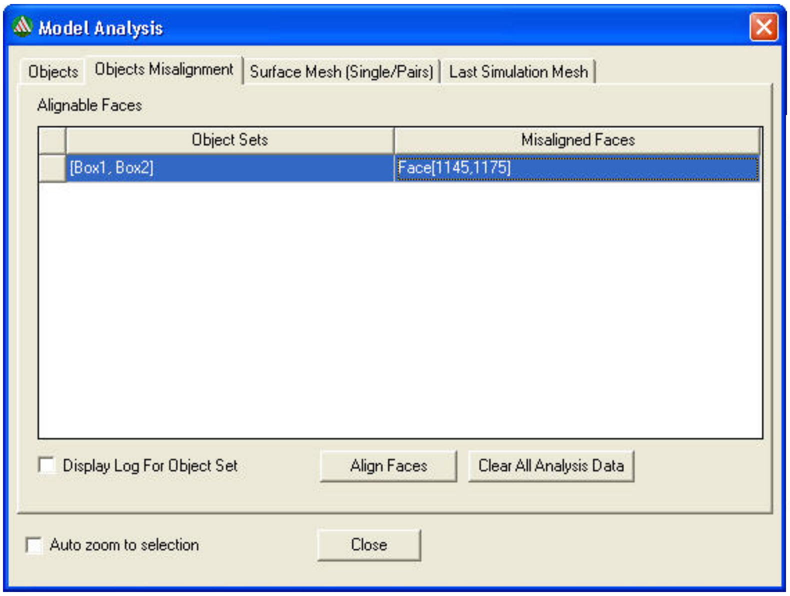
Small features

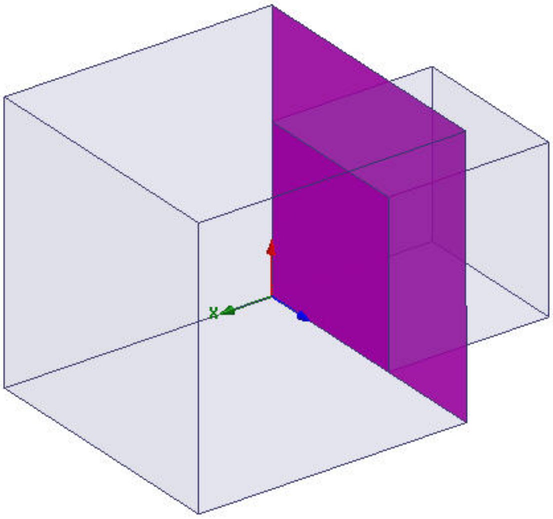
- Small features in the geometry can lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower.
- Small features may even cause the mesh generation to fail.
- By small, we mean details on an object that are thousands of times smaller than the main features of the object, and that, in most cases, are unintended consequences of the drawing history in another CAD tool.
- Therefore, it is advantageous to remove small features.
- This step is entirely optional, and although it could have been accomplished when an object is imported (by selecting the **Manual** radio button instead of **Auto**), we present this information here because the previous steps were necessary to invoke mesh generation, while this one is optional.

- To start the small-feature analysis:
  1. Select the objects (their histories must be purged) and invoke Object Analysis through ***Modeler > Model Analysis > Analyze Objects***.
  2. In the **Model Analysis Window**, select **Perform > Analyze Objects**.
  3. The software will report the smallest edge length and the smallest face area, and enable you to set thresholds for the detection of short edges, small faces, and sliver faces.
  4. Click **OK**, and the analysis is performed.
- This will report small features in the analyzed objects that can be Deleted automatically or repaired with Boolean operations in the Modeler.

Mis-aligned entities

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





Coincident True Surfaces

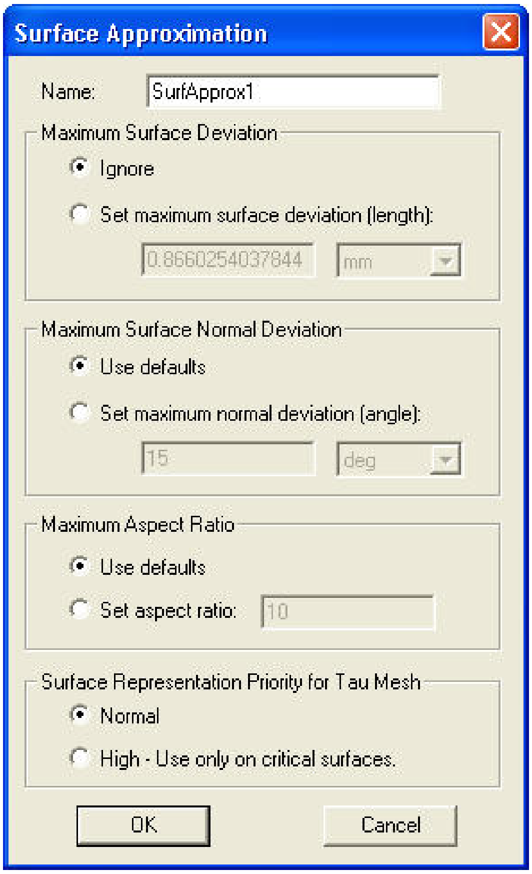
- True Surfaces are curved surfaces in the Modeler (e.g. spheres have a true surface, while boxes do not).
- Due to the finite nature of the mesh, these true surfaces are approximated with facets and tetrahedra (as explained in the section on Mesh Operations).
- When two true surfaces are coincident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

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

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

Coincident True Surfaces (Continued)

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



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

Mesh Operations

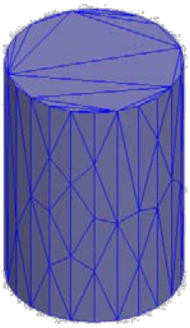
- This chapter provides details on meshing in the ANSYS Maxwell 3D software. The following topics are discussed:
  - Curved Geometry Mesh Adaptation
  - Faceting Default Settings
  - User-Defining Surface Approximations
  - Application Recommendations
  - Model Resolution
  - Applying Mesh Operations

Examples

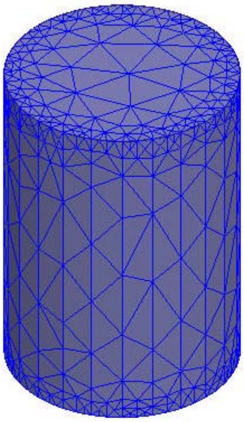
- The following examples are provided to demonstrate the topics discussed in the chapter:
  - Skin effect and adjacent effect for two bars (Eddy Current)
  - LR Matrix of a bar of PCB (Eddy Current)
  - 3 Phase Linear PM Motor (Transient)
  - 1.5kw 4p24s IPM Synchronous Motor (Transient)

Curve Mesh Adaptation in Maxwell v15

- Maxwell v15 meshes handles curved surfaces differently than prior versions of Maxwell. Proper understanding of the differences in curved mesh handling and the advantages of new capabilities in the Maxwell v15 solver are essential to obtaining accurate results.
  - The new graphical drawing interface offers the use of true-curved drawing, by removing the option to assign a facet count to the construction of primitives such as circles, cylinders, spheres, and ellipses.
  - Faceted primitives are still suggested as polyhedrons and polyhedral solids for few cases. For examples, in translational motion analysis with Transient solver, band have to be made as a faceted solid for better remeshing after object motion
  - Initial meshing is constrained by faceting decisions made by the first pass of the meshing algorithms as shown below.

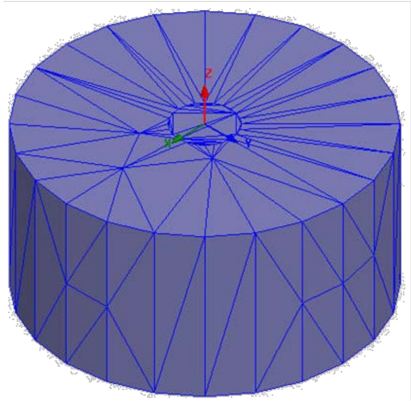


  - For Maxwell v15 in order to provide more robust meshing with respect to more complex geometries, the initial faceting selections made for curved objects are respected throughout the adaptation process, so that the adapted mesh is a subdivided variation of the *same meshed volume* as the initial mesh. An adapted mesh from Maxwell v15 is shown below.



Faceter Default Settings

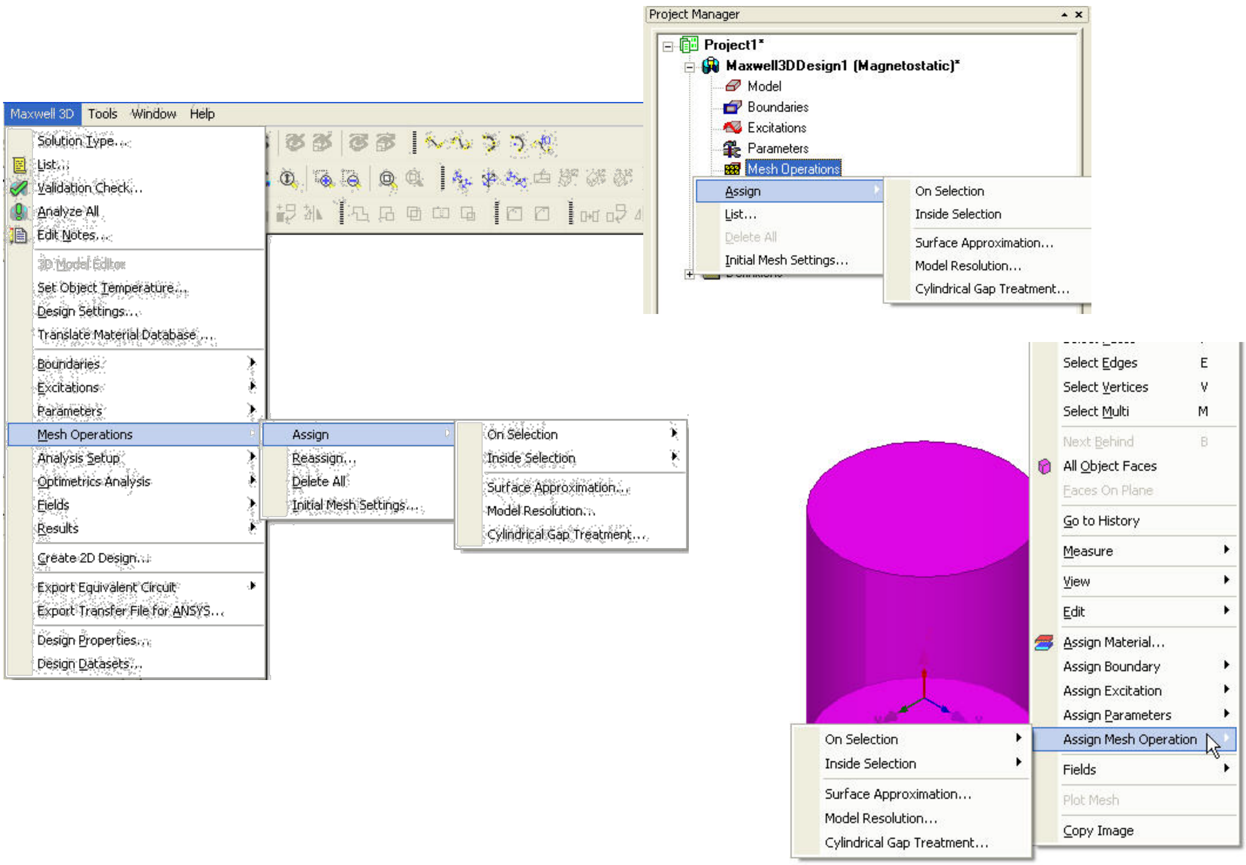
- In order to resolve the true surfaces better, the initial faceting default setting is to constrain the mesh surface normals to fall within ***15 degrees*** of the true-curved surface normals.
    - This means that a cylindrical surface would be ‘faceted’ into 24 segments about its circumference, as illustrated below.
    - The normal for each flat segment cannot be off by more than 15 degrees from the normal for the curved true surface which that segment’s face is approximating.



- The default is sufficient for many applications in which the curved geometry is not itself highly critical to the simulation result, yet where more faceting would result in an unnecessary large initial mesh. Special cases might need added meshing operations to achieve better results.

Mesh Operations

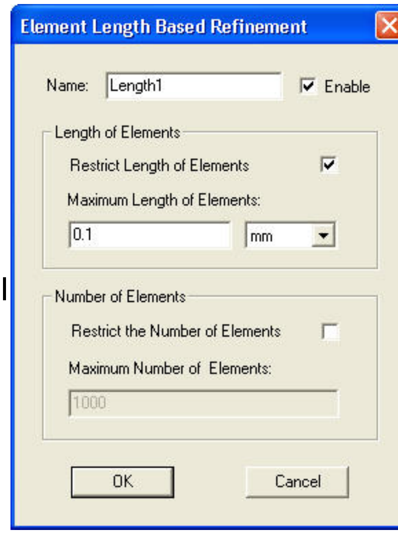
- Mesh Operations can be assigned from the ***Maxwell 3D*** menu, from the **Design Tree**, or from the geometry interface's context-sensitive menu.



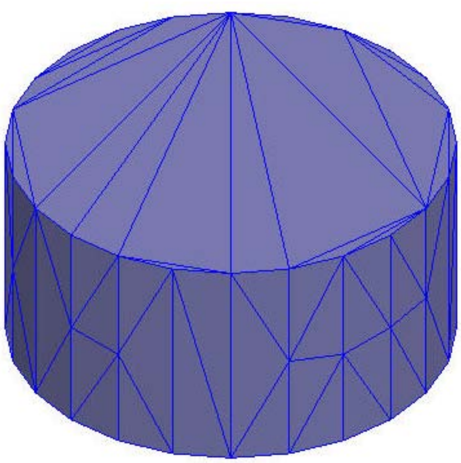
- Maxwell v15 can offer user 6 types of Mesh Operations:
  - On Selection/ Length Based;
  - On Selection / Skin Depth Based
  - Inside Selection / Length Based
  - Surface Approximation
  - Model Resolution
  - Cylindrical Gap Treatment

Mesh Operations Examples

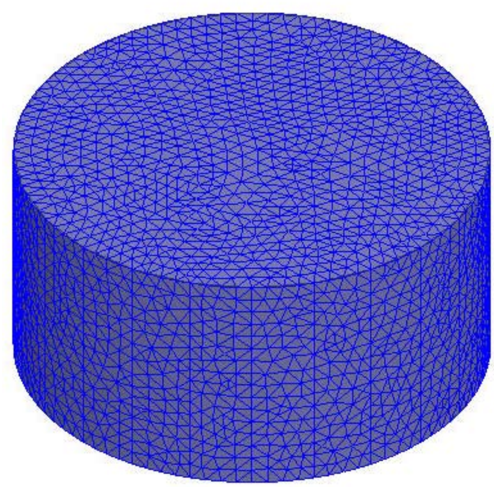
- **On Selection / Length Based**
  After user selected one or some Objects, this mesh operation can be setup as a maximum length of the edges of the mesh for it.



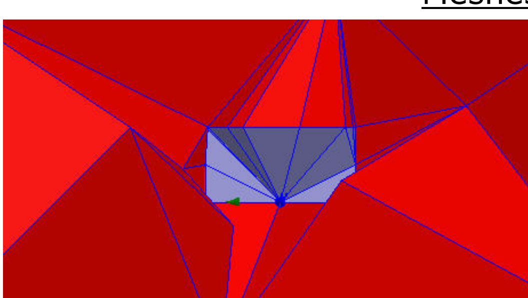
- The Cylinder with initial mesh below was created by Draw cylinder with R = 1 mm, H = 1 mm, are the meshes on surface and the center cross- section. The right fig. of them are the results after using the mesh option **On Selection / Length Based = 0.1 mm**. User can see the surface of it was meshed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



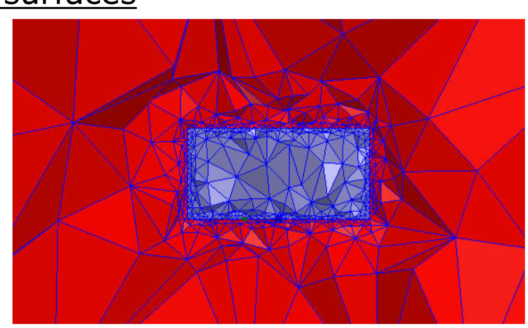
Without Mesh Operation



Meshes on the surfaces



Without Mesh Operation

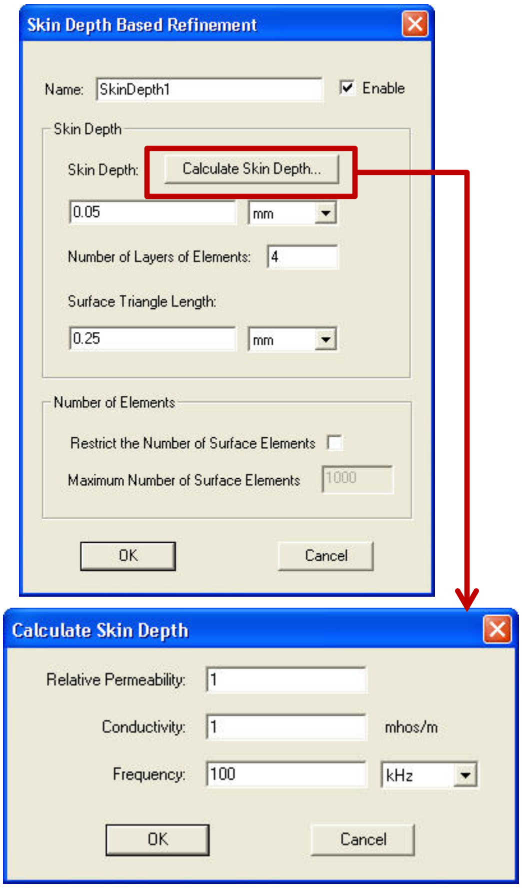


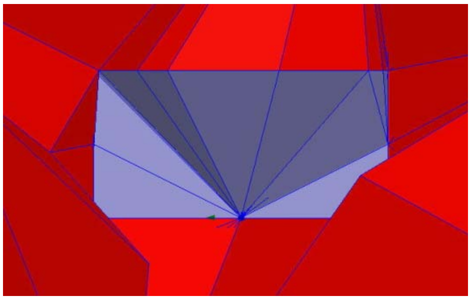
After Mesh Operation

Meshes on the center cross-section of them

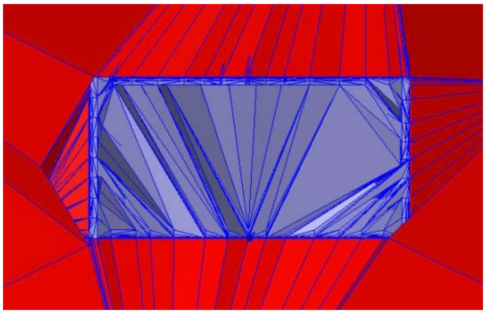
Mesh Operations Examples

- **On Selection / Skin Depth Based**
- The mesh will be created with the skin depth effect in the object.
- The Cylinder with initial mesh left below was created by Draw regular Polyhedron with R = 1 mm, H = 1 mm,
  The right one was the results with the setting in **On Selection / Skin Depth Based:**
  Skin Depth = 0.25mm;
  Number of Layer of Element = 4;
  Surface Triangle Length = 0.4mm.
  The inner mesh will be 4 layers where the skin depth effects will be considered.
  if user select **Calculate Skin Depth**, the skin depth will be automatically calculated due to the frequency, permeability and conductivity set by user.



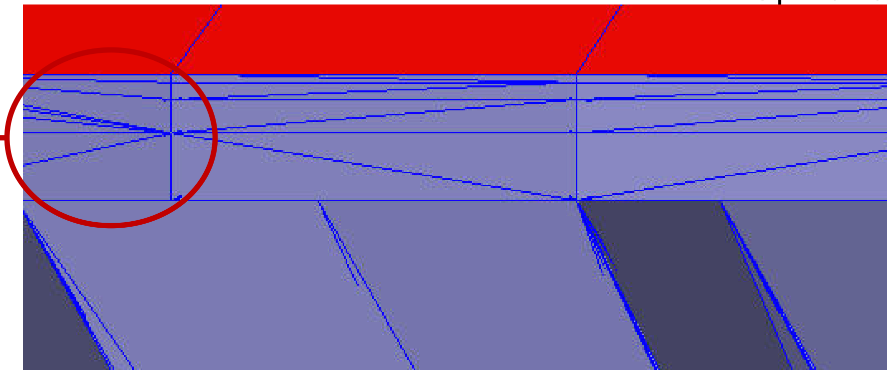


Without Mesh Operation



With Mesh Operation

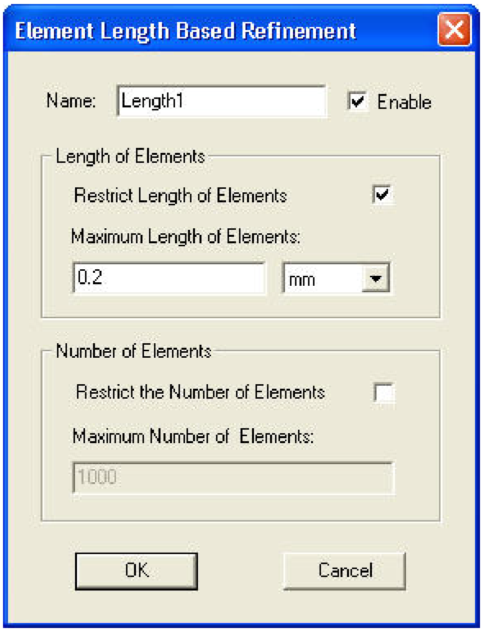
Four Layers of Skin Depth Mesh



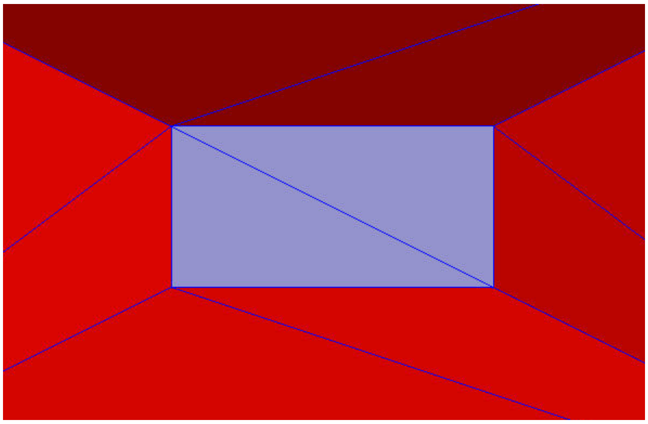
Cross-section of the mesh

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

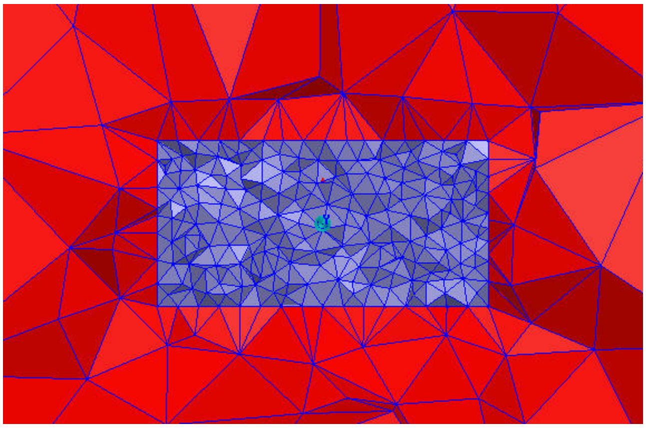
- Below is the 2x2x1 mm box with initial mesh. The right below is result with the command of **Inside Selection / Length Based = 0.25mm**



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



Without Mesh Operation



With Mesh Operation

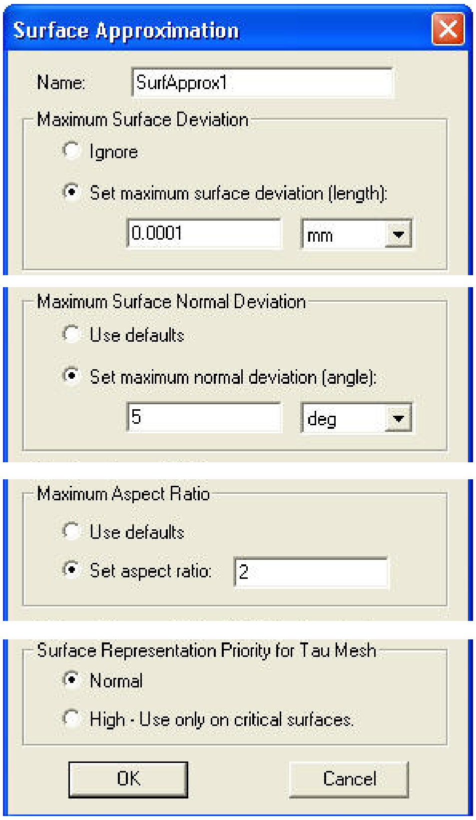
Cross-section of the mesh

Mesh Operations Examples

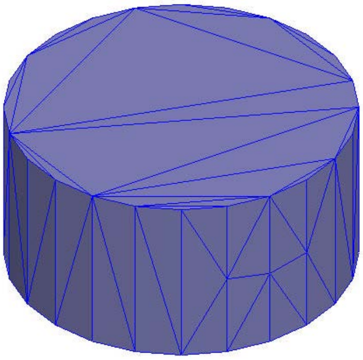
- **Surface Approximation**

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

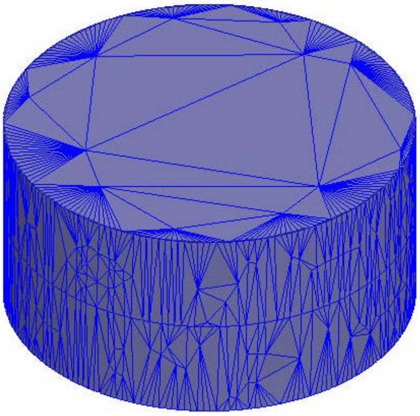
  - *Surface Deviation* is the maximum spacing, in drawing units, that the tetrahedral surfaces may be from the true-curved geometry's surface.
  - *Normal Deviation* is the maximum angular difference, in degrees, that a tetrahedral face's normal can have from the surface normal for the true geometry which it is meant to represent.
  - *Aspect Ratio* refers to the maximum allowed aspect ratio of all faces of all tetrahedral of the selected object or face. This setting influences mesh quality rather than actual meshed volume or surface locations.
  - *Surface Representation Priority* for Tau Mesh
    In most cases, meshing is done by Tau Mesh.
    You can set the surface representation as normal or high



- The Cylinder with initial mesh on the right is created by Draw cylinder with **R = 1 mm, H = 1 mm**



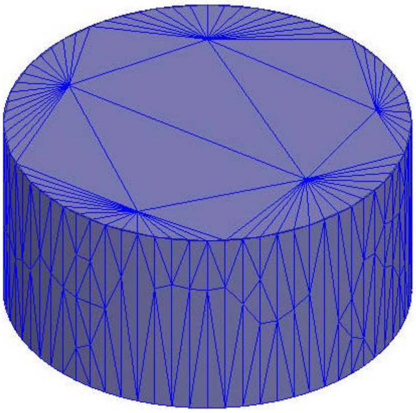
- The right is the result with **Maximum Surface Deviation** = **0.000125mm.**



Maximum Surface Deviation

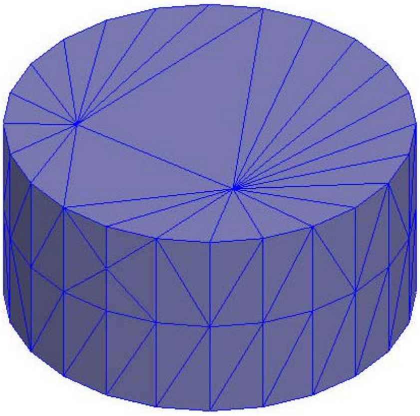
Mesh Operations Examples

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



Maximum Surface Normal Deviation

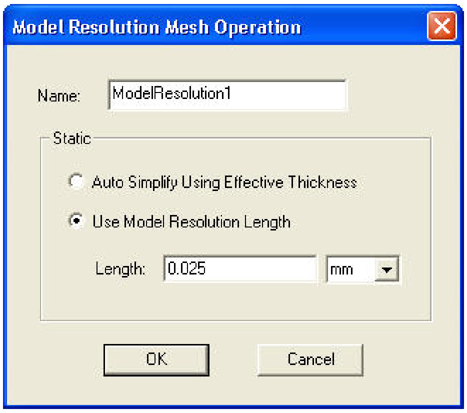
The last in the right is the surface result with **Maximum Aspect Ratio = 2.**



Maximum Aspect Ratio

Model Resolution

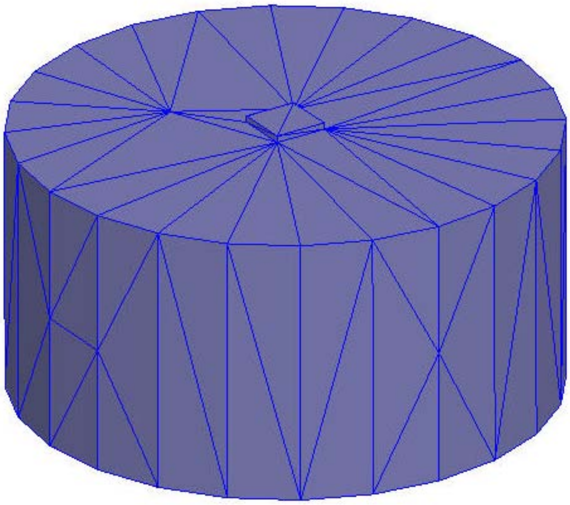
- Model resolution is a length based value that modifies the initial mesh. This meshing operation allows the user to specify a minimum edge length of any tetrahedron used for the mesh. By specifying a minimum edge length for a tetrahedron, the mesher will have to coarsely mesh geometric detail that may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



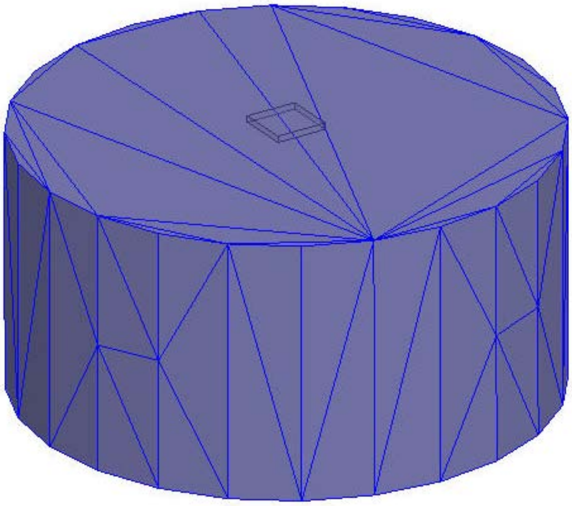
- Simple example:

There is a small stub (0.2x0.2x0.025mm) on a cylinder (R=1mm, H=1mm). Below is the initial mesh without Model Resolution.

Then with **Model Resolution = 0.025 mm**, the mesh of it will be as below right, the stub will be neglected.



Without Mesh Operation



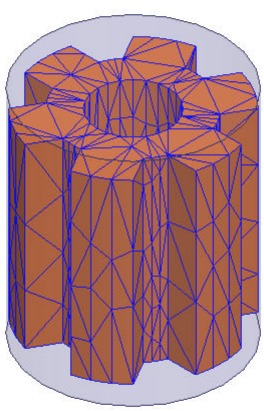
With Mesh Resolution

Mesh on the surface

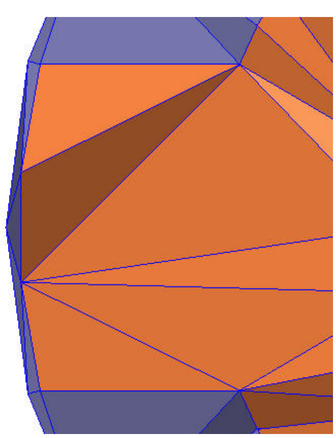
Cylindrical Gap Treatment

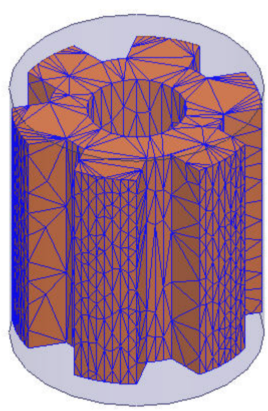
- Cylindrical Gap Treatment mesh operation is a proximity based mesh refinement. The refinement of mesh is done on the applied objects based on the closeness of the geometry lying inside it.
- These mesh operation can be used effectively in motor geometry specially when the gap between Band object and Stator/Rotor is very small. In such cases meshing may fail or generate poor mesh in gap region. With this mesh operation those problems can be avoided.
- For Transient Solver involving rotational motion, this mesh operation is automatically created once the rotational motion is defined
- The mesh operation can be accessed through menu item similar to other mesh operations.
- In order to specify this mesh operation on any object, the selected object must contain some geometry (Other Objects) inside it. Otherwise the mesh operation can not be applied as proximity is not detected.

Below shown is a simple core surrounded by a cylinder. The gap between the core and cylinder is around 1mm. A cylindrical Gap Treatment mesh Operation is applied on the cylinder. The refinement can be seen on the faces of core which are close to cylinder

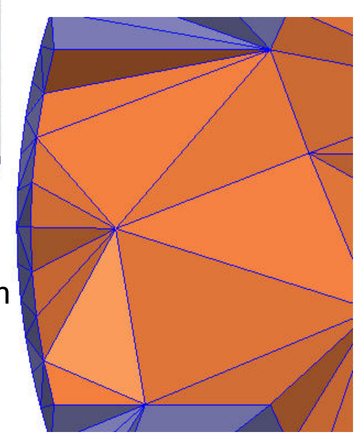


Without
Mesh Operation





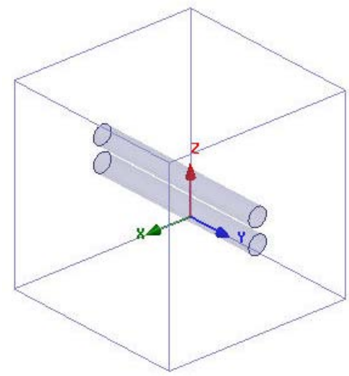
With
Mesh Operation



Mesh Operation Examples

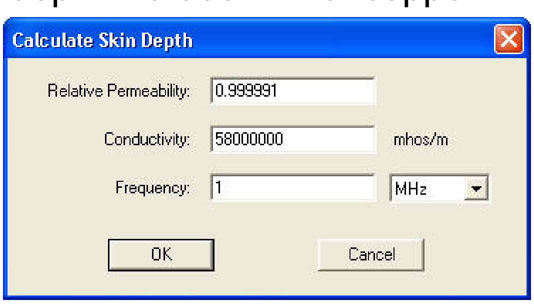
EX1. Skin effect and adjacent effect for two bars (Eddy Current)

- **Model:**
  Two parallel copper bars (named coil1 & coil2) are created by Draw Cylinder with R = 0.15 mm, H = 3 mm,. The distance between their axes is 0.4mm (Gap is 0.1mm)
  The air box (3x3x3mm) Is created for analysis region

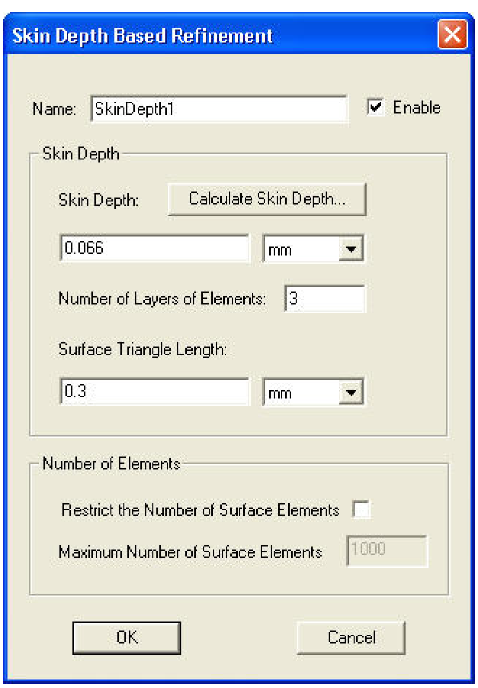


- **Excitations**
  For each end terminal of bars is set as 1A current in same Y direction (Design named “same”)
  or Y direction for bar1, -Y direction for bar2 (Design named “opposite”)
- **Mesh Operation**
  Select cylindrical surfaces of bars except their end terminals as in right,

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



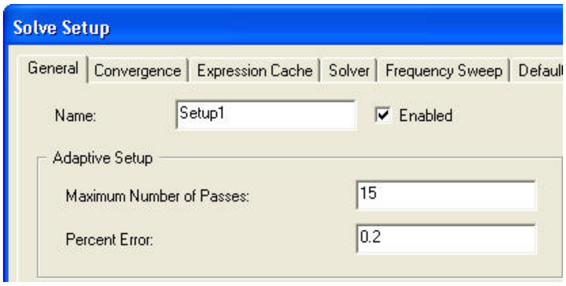
Then set **Skin Depth Based Refinement** as below:
Skin Depth = 0.066mm;
Number of Layer of Element = 3;
Surface Triangle Length = 0.3mm.
and without Number Restriction

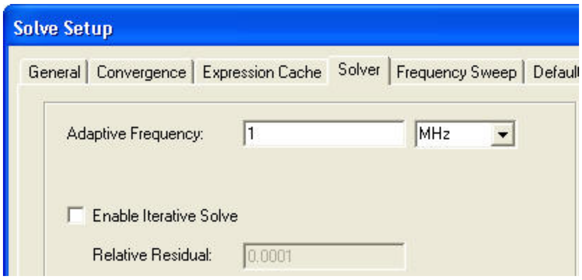


EX1. Skin effect and adjacent effect for two bars (Con't)

- Solve Setup

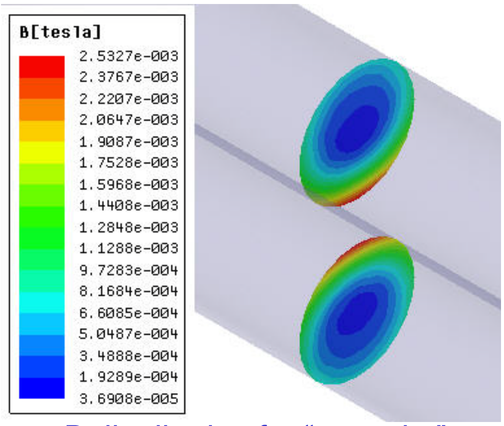
  Solve Setup is set as below:



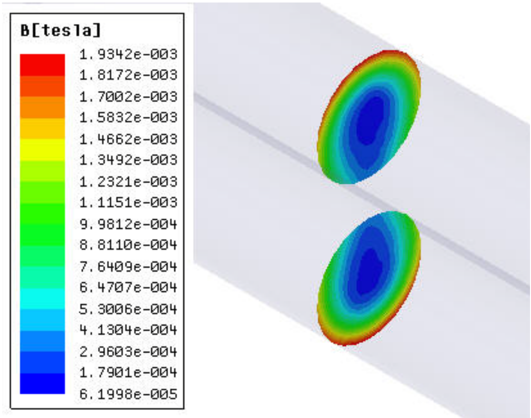


- Results

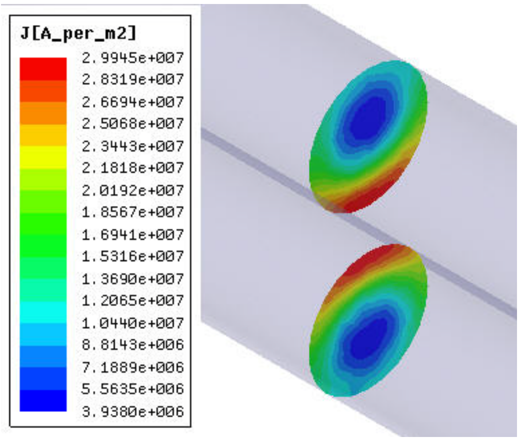
  The B & J distribution on the XZ cross-section of two bars will be as below:



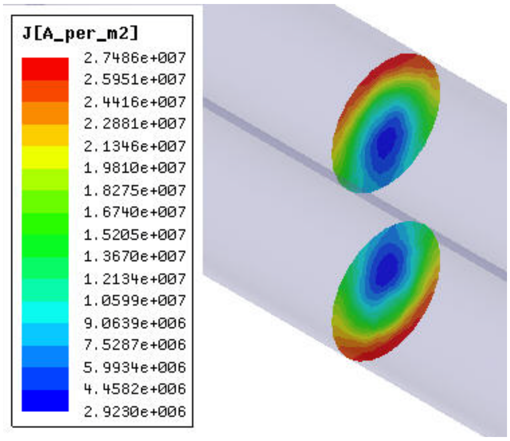
B distribution for "opposite"



B distribution for "same"



J distribution for "opposite"
With mesh plot



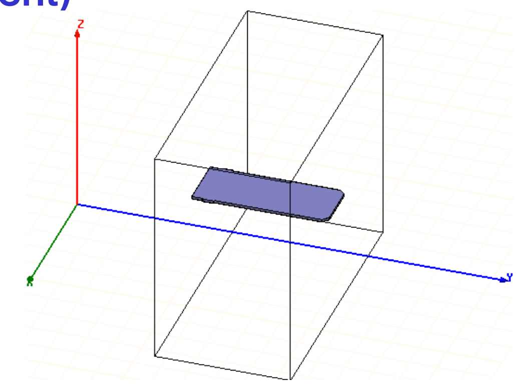
J distribution for "same"
With mesh plot

EX2. LR Matrix for print bar (Eddy Current)

- Model:

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

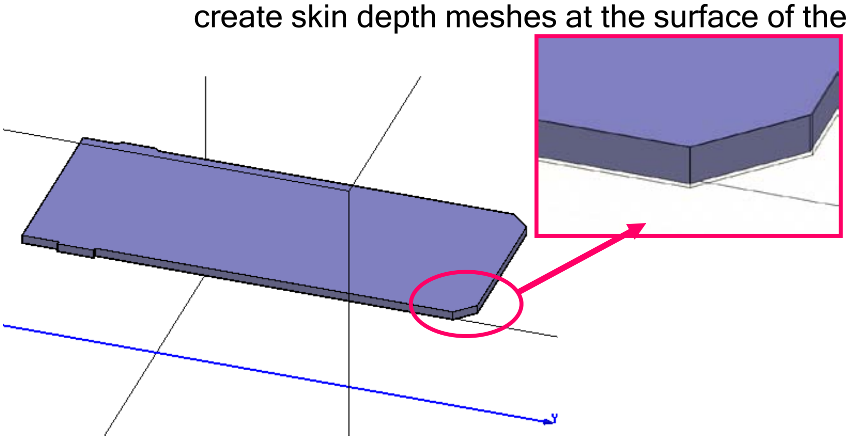
  An air box (50x24.5x40 mm) Is created as the analysis region

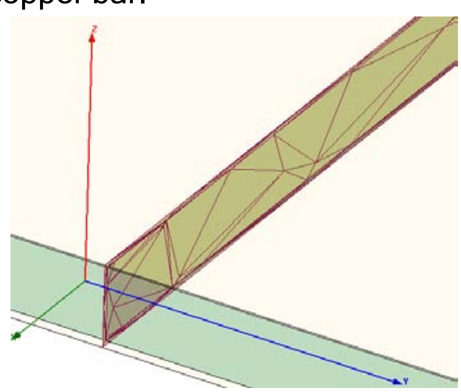


- Mesh Operation

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

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





Mesh in cross-section of bar

- Excitations

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

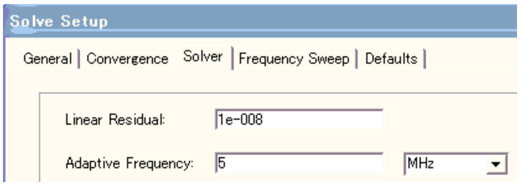
- Matrix setup

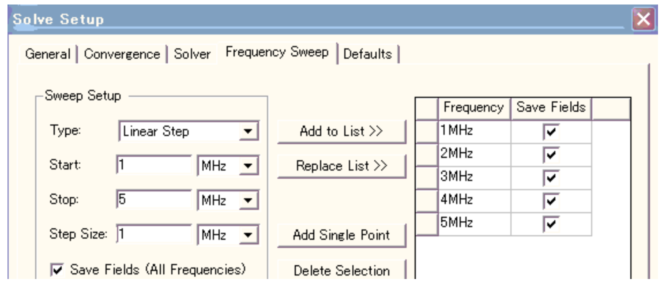
  Select the copper bar and do **/Assign Parameters/Matrix** with Current1 as **Include**

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

- Solve Setup

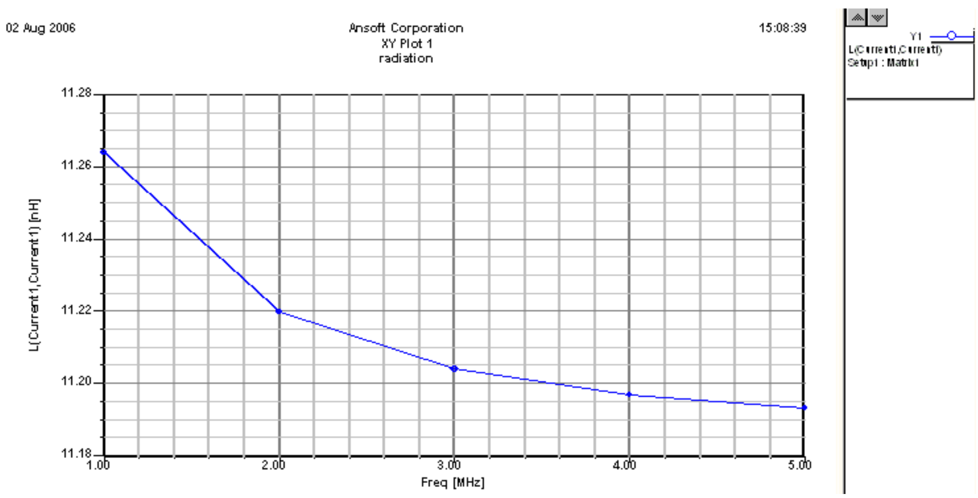
  Solve Setup is set as below: (Set Default In General & Convergence)



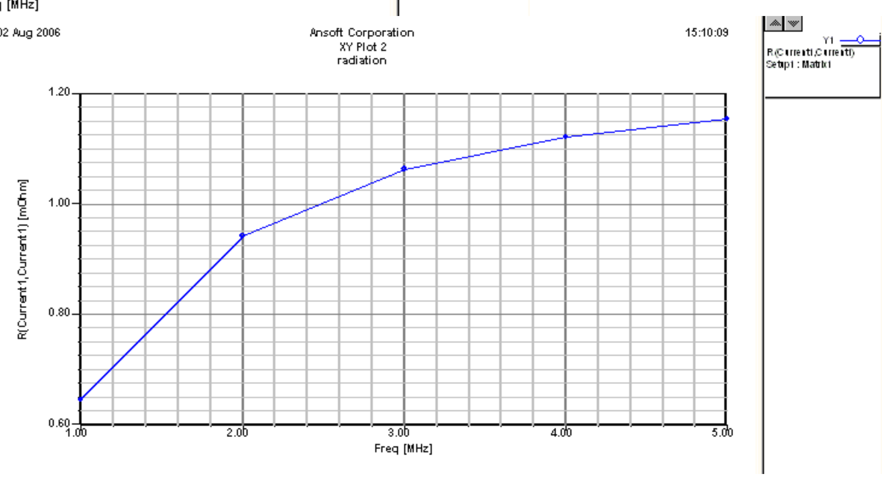


- Results

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



L vs. Frequency

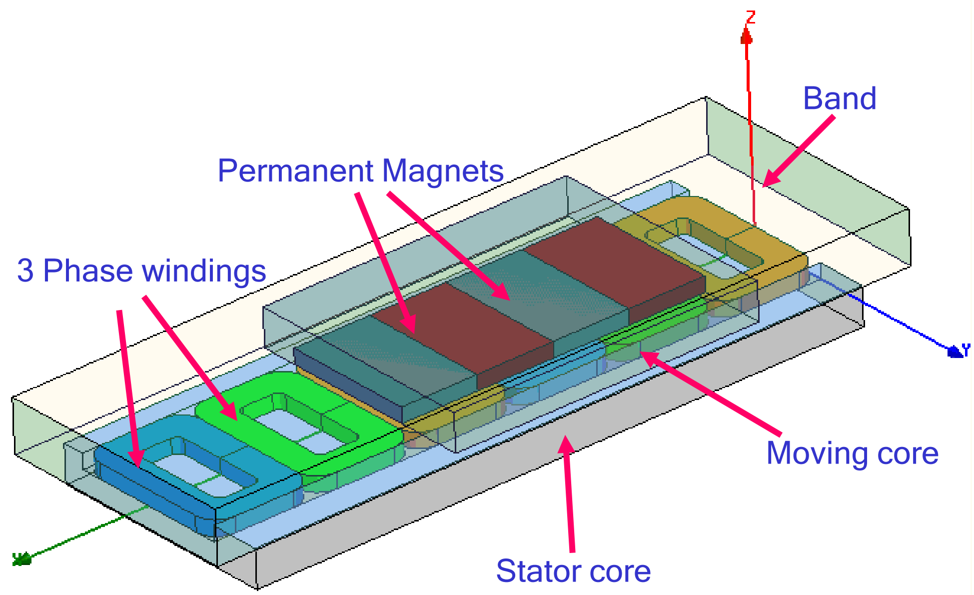


R vs. Frequency

EX3. 3ph Linear PM Motor (Transient)

- Model:

  The basic parts of it are as below:



- Mesh Operation

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

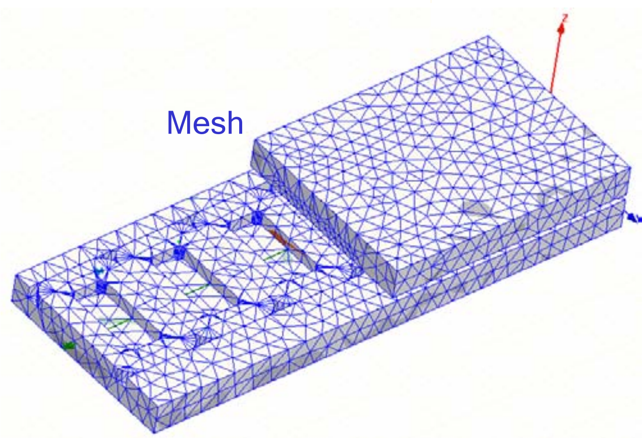
  windings: 4 mm;

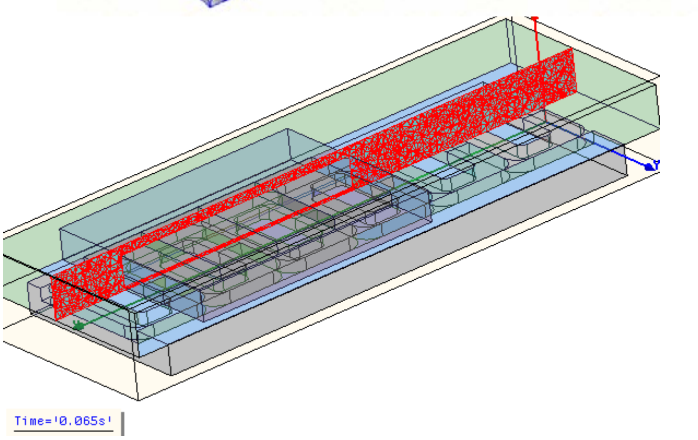
  magnets: 2 mm;

  cores : 4 mm:

  Band : 3 mm

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





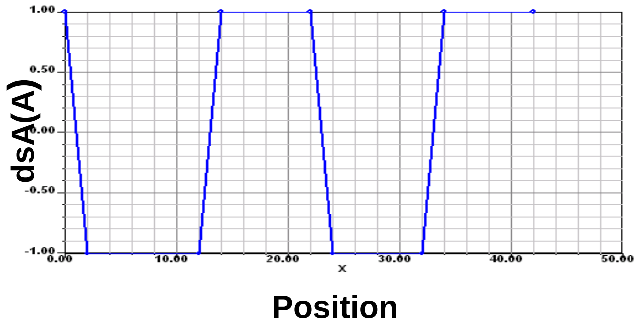
Band mesh in XZ cut-plane at some time step

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

- Exciation

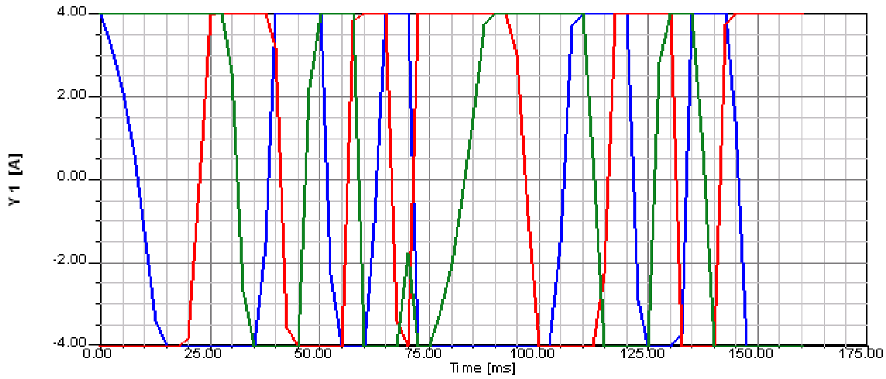
  The current exciation is set as a time function:

  **U Phase: if((Speed>0&&Position<=41.5), 4*pwl_periodic(dsA,Position),
  -4*pwl_periodic(dsA,Position))**

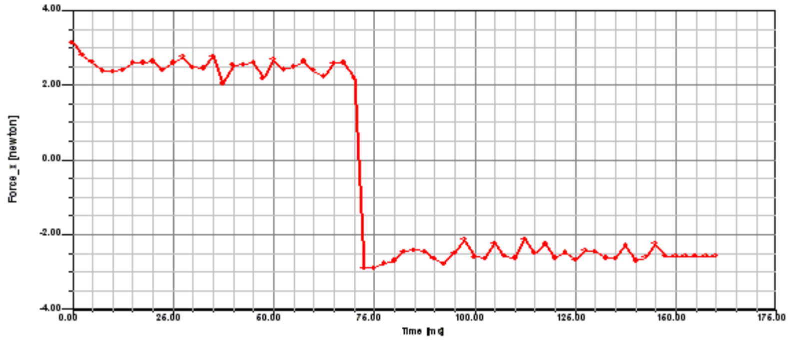


.

- Results



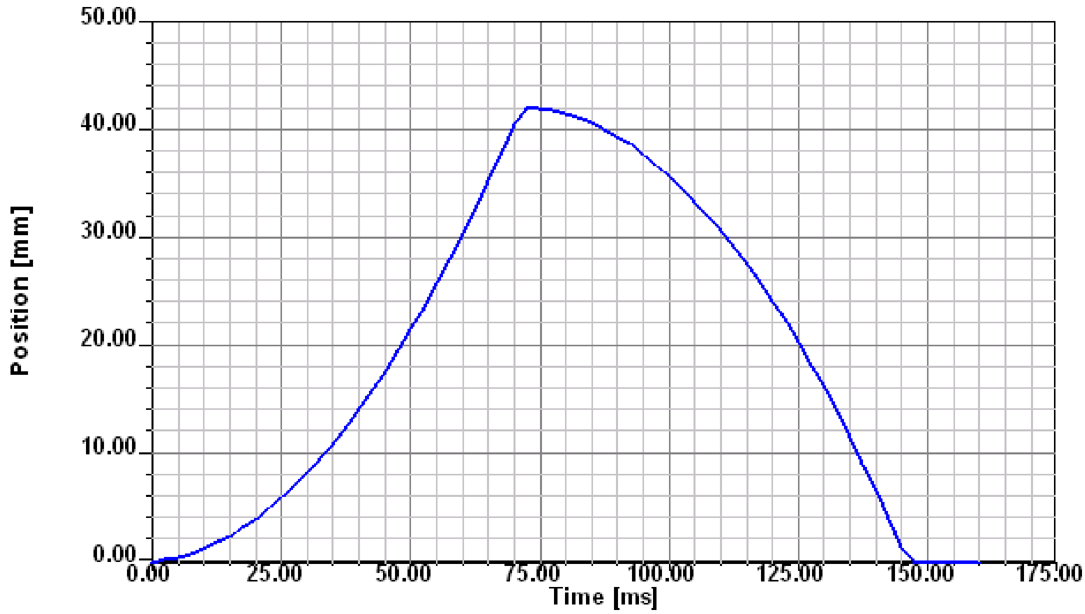
Transient current of 3 phase winding



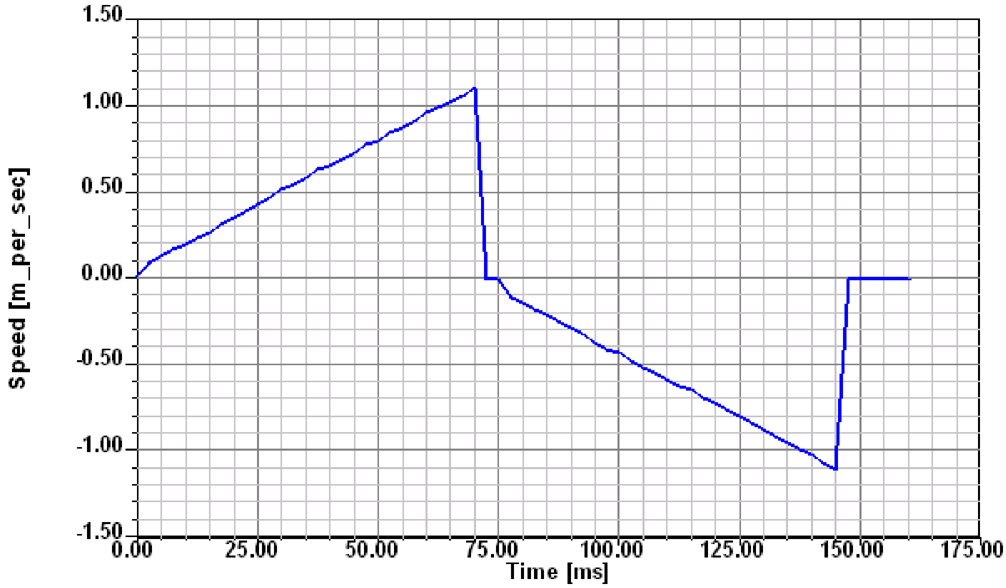
Force in X direction (|fx|>2N)

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

- Results



Position vs. Time

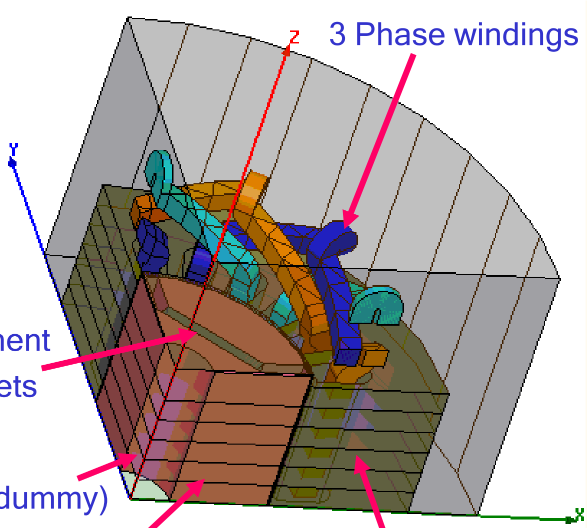


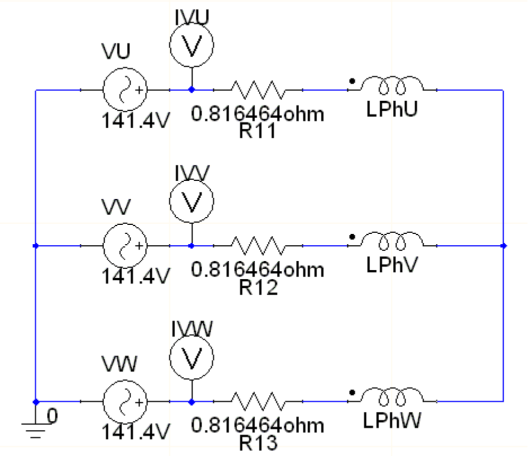
Speed vs. Time

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

- Model:

  Please see the model as below:





Circuit model of 3 Phase Y connection

- Mesh Operation

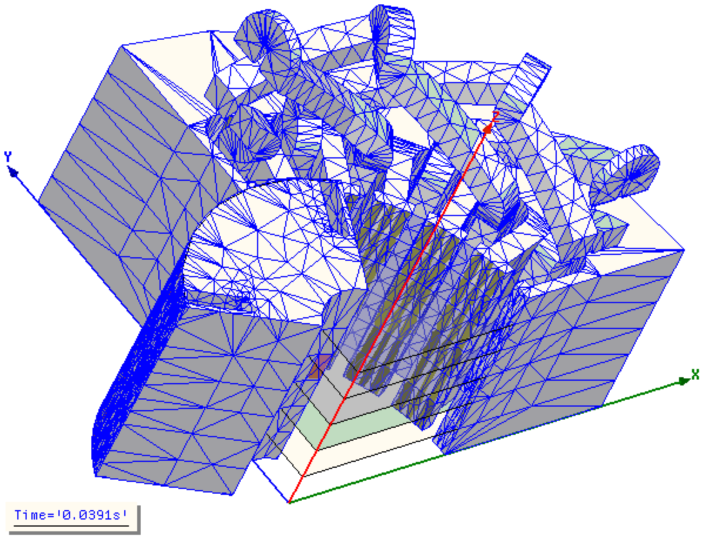
  For making the mesh as 6 levels in motor, 5 dummy circular sheets are created. Using **Inside Selection / Length Based** to set the length of the maximum of element edge for windings: 5 mm;

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

  Set maximum surface deviation (length): 2.5 mm;

  Set maximum surface deviation (angle): 3 deg

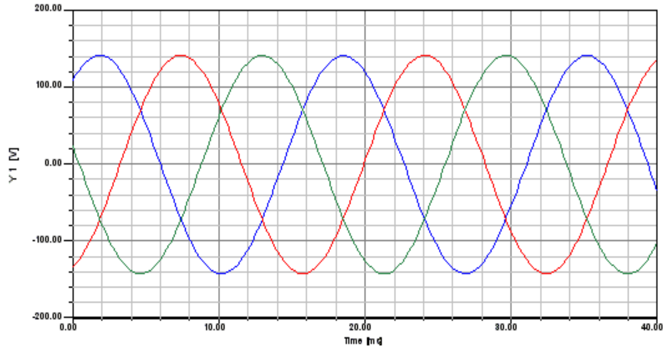
  The mesh of the main parts will be shown in the right.



Mesh

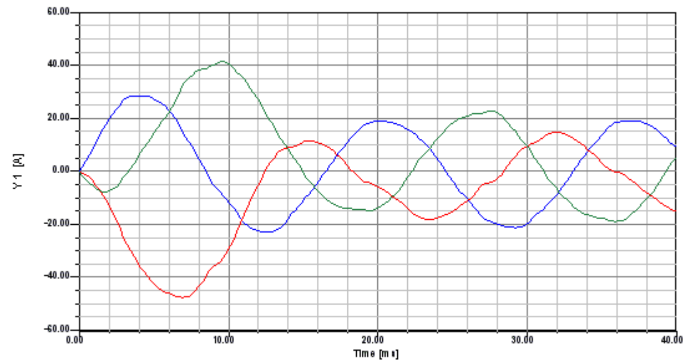
EX4. 1.5KW 4p24s IPM Synchronous Motor (Transient) (con't)

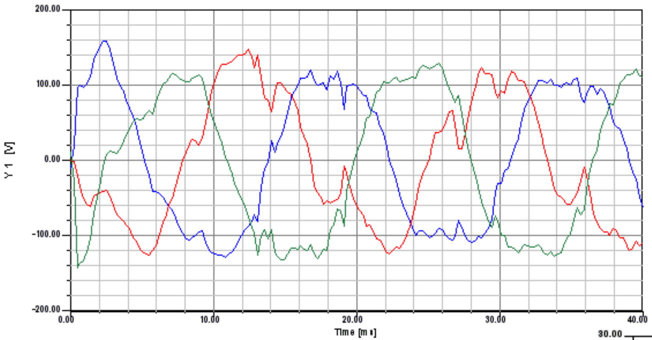
- Results at Speed = 1800 rpm



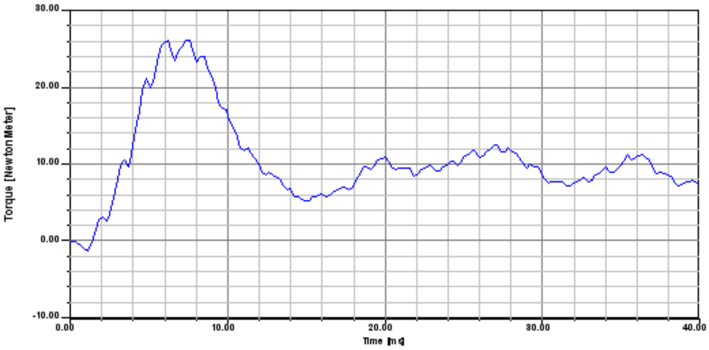
Input node voltages

Current of 3 phase





Induced Voltages



Output Torque

Maxwell Adaptive Meshing

Maxwell and the Finite Element Method

- Maxwell uses the Finite Element Method (FEM) to solve Maxwell's equations. The primary advantage of the FEM for solving partial differential equations lies in the ability of the basic building blocks used to discretize the model to conform to arbitrary geometry.

- The arbitrary shape of the basic building block (tetrahedron) also allows Maxwell to generate a coarse mesh where fewer cells are needed to yield an accurate solution, while creating a finely discretized mesh where the field is rapidly varying or higher accuracy is needed to obtain an accurate global solution.

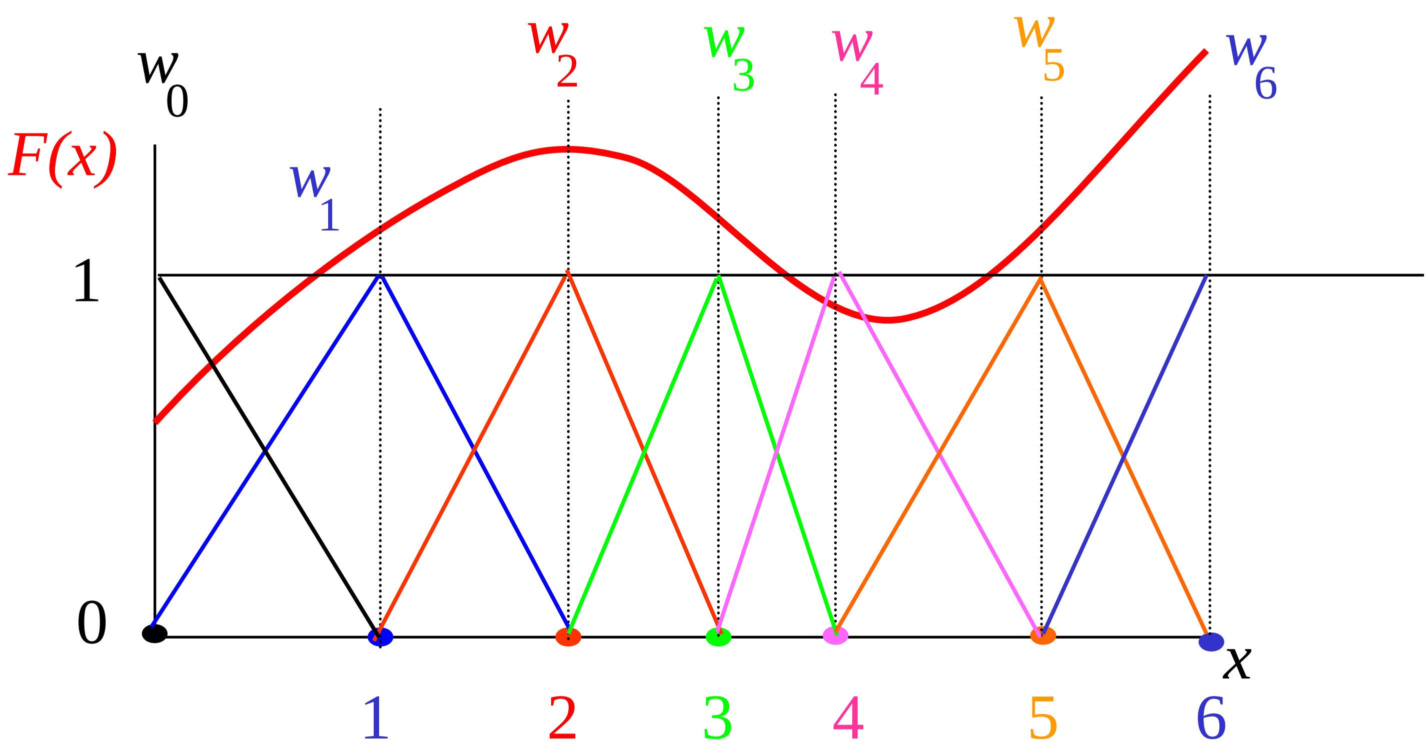
- The FEM has been a standard for solving electromagnetic problems since the 1970s. Maxwell appeared on the market in 1984 pioneering the use of adaptive meshing in a commercial product.

- The FEM has been a standard for solving problems in structure mechanics since the mid 1950's (imagine defining the 3D structure of a bridge with punch cards)!

- First see the one-dimensional example…..

The Finite Element Method in 1-D

- The finite element method (FEM) can be used to approximate the unknown curve *F(x)*

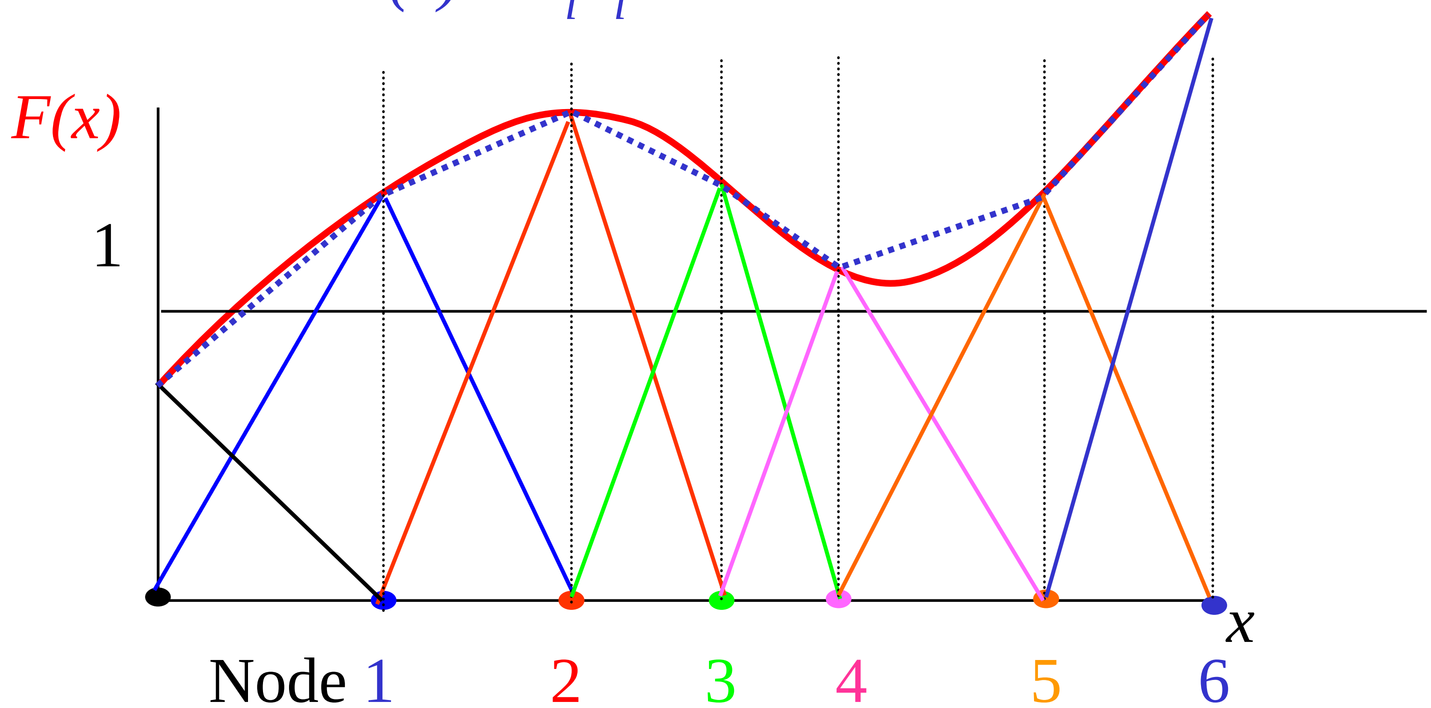


- The model is "discretized into 6 individual cells. $w_0 - w_6$ are the piecewise linear "basis functions" from which the approximate solution will be built.

The Finite Element Method in 1-D

- In the FEM, the unknown function is expressed as a weighted sum of the piecewise continuous basis functions.

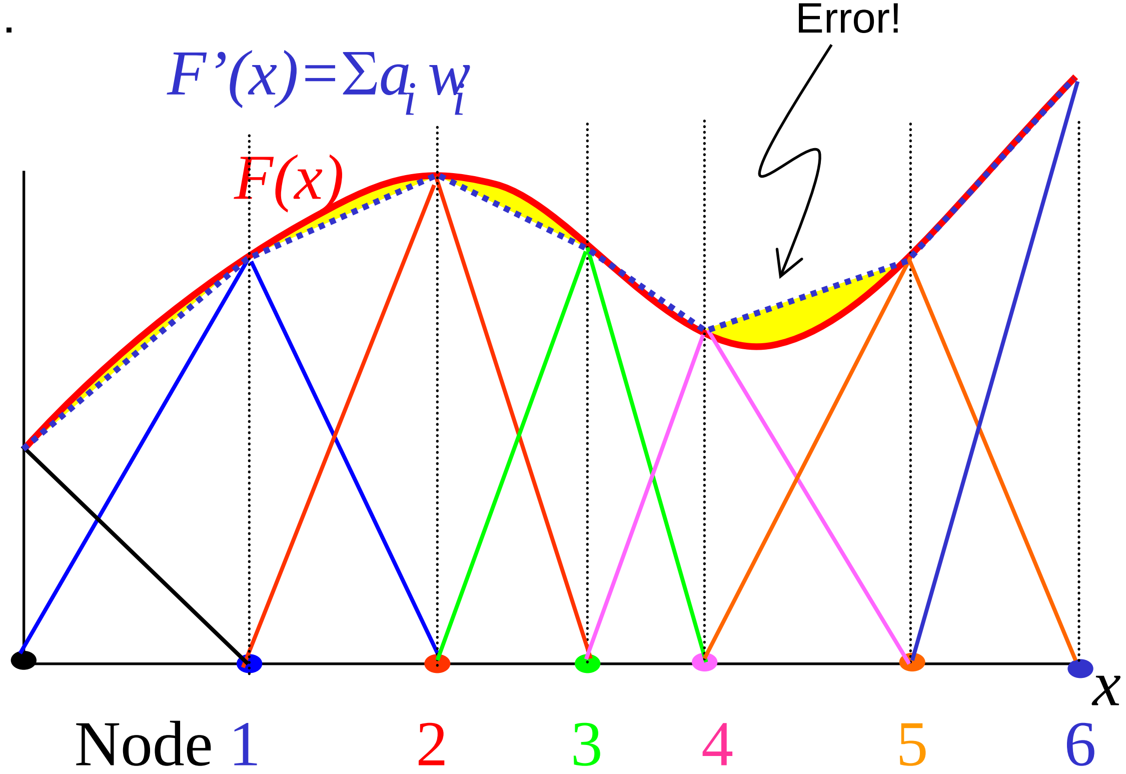
$$F'(x)=\Sigma a_i w_i$$



- The approximate solution resulting from the FEM calculation is $F'(x)$ shown as a dotted line.

The Finite Element Method in 1-D

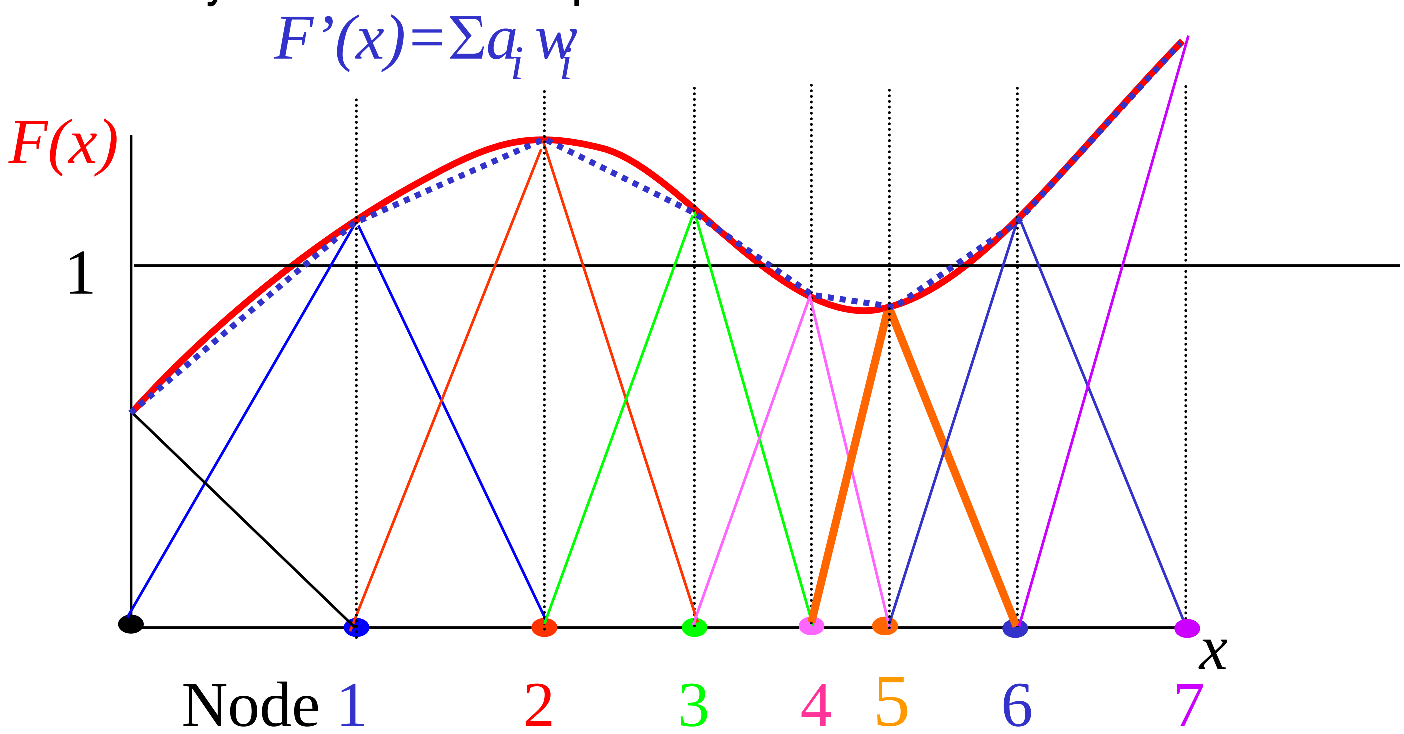
- A key feature of the FEM as it is implemented in Maxwell is the ability to locally determine the error. Recall that *F(x)* is not known, but the ERROR can be determined[1].



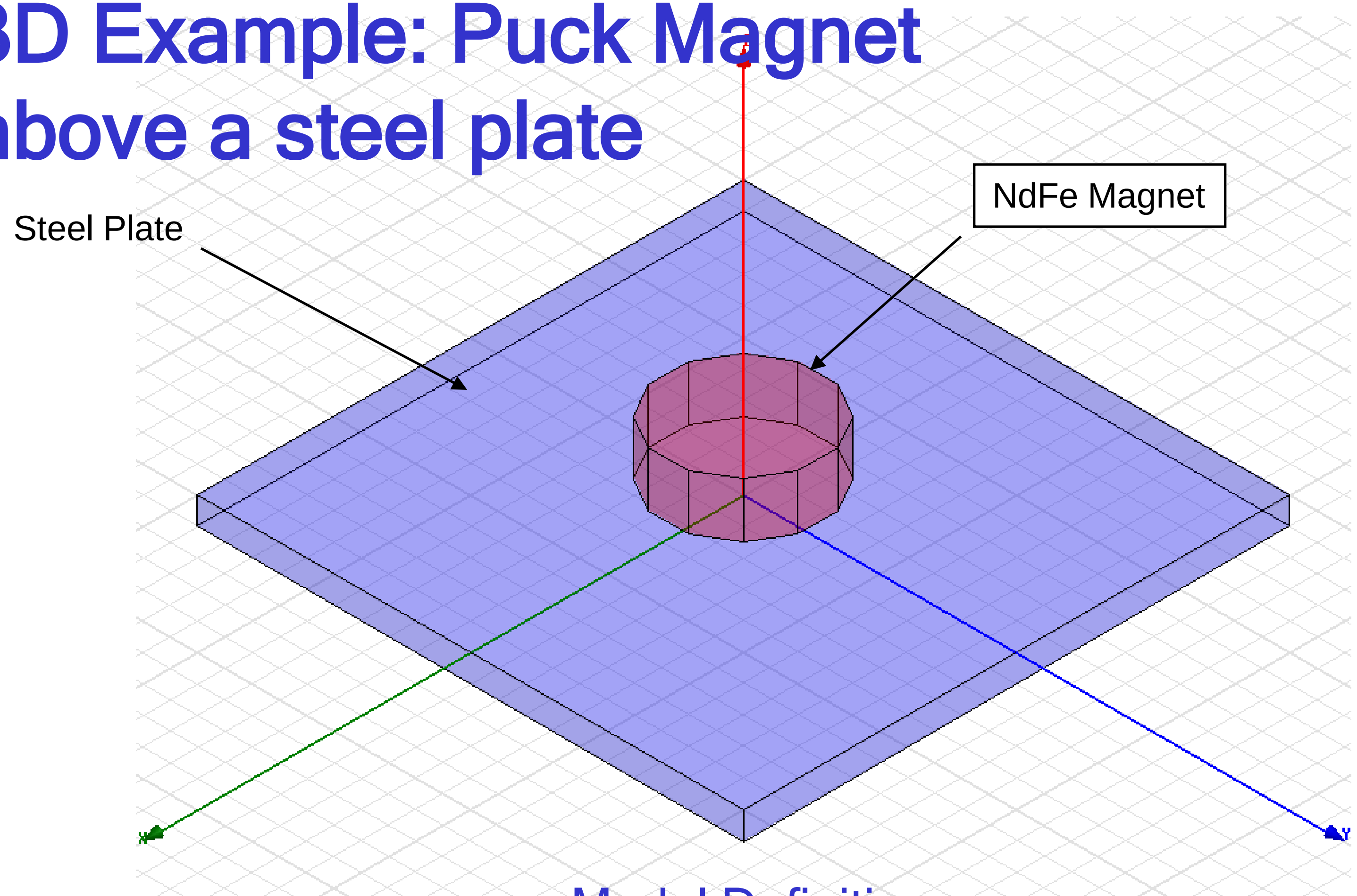
[1] Cendes, Z.J., and D.N. Shenton. Adaptive mesh refinement in the finite element. computation of magnetic fields; IEEE Transactions on magnetics (T-MAG), Pages 1811-1816 , Volume 21, Issue 5, Sept 1985

The Finite Element Method in 1-D

- The mesh density is increased where the error is largest. Hence, the final (and computationally most expensive) solution will have a mesh that yields the greatest accuracy with the fewest possible mesh cells.



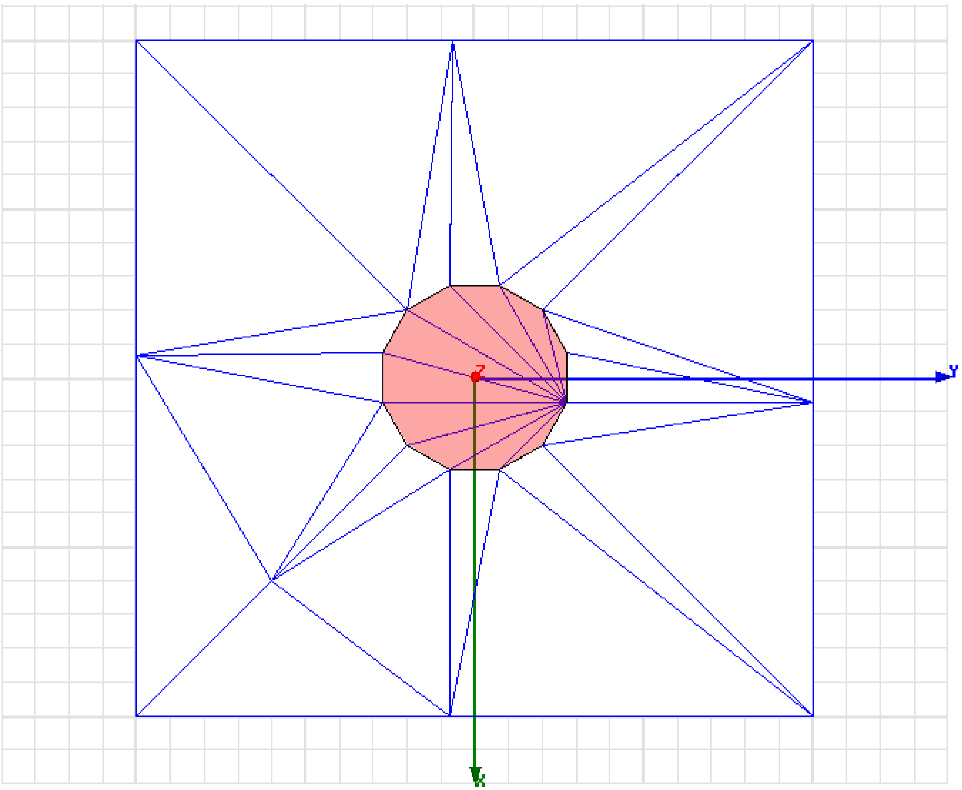
3D Example: Puck Magnet above a steel plate



Model Definition

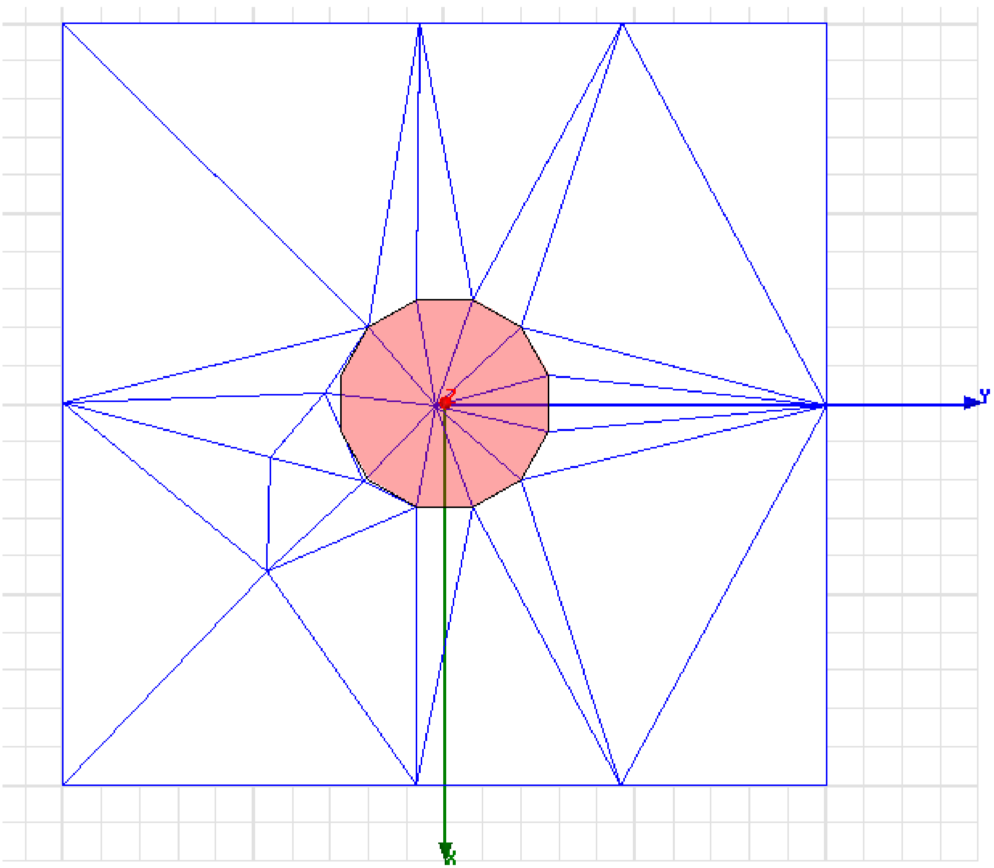
Adaptive Mesh Refinement

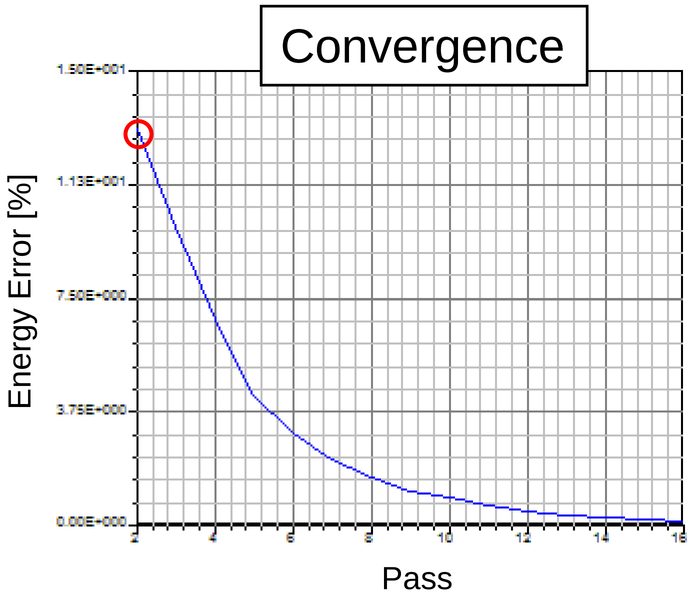
- View of **initial mesh**
  - The projection of the 3-dimensional mesh onto the plate surface creates a triangular mesh. The initial mesh is as coarse as possible while still representing all geometric features.



Adaptive Mesh Refinement

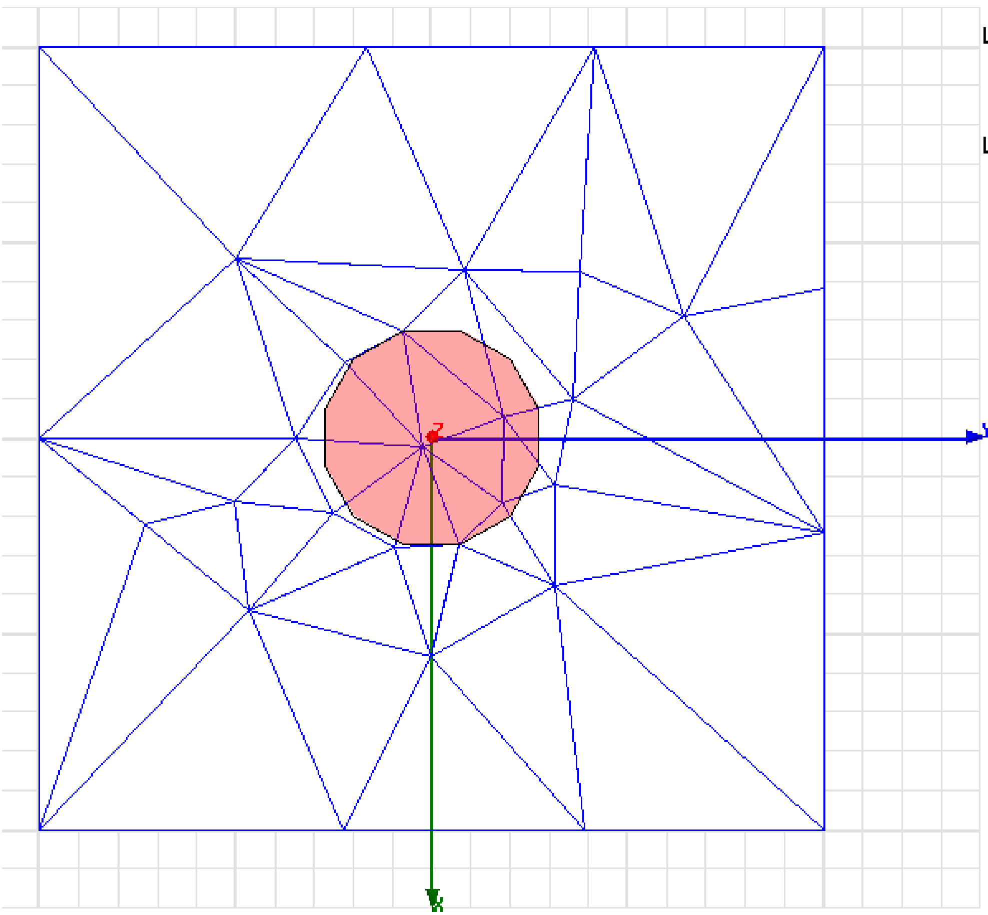
- Mesh after 2 passes

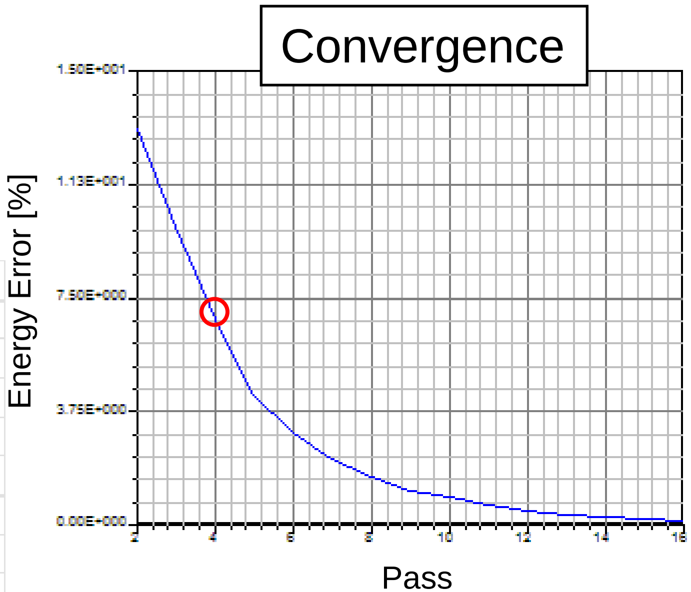




Adaptive Mesh Refinement

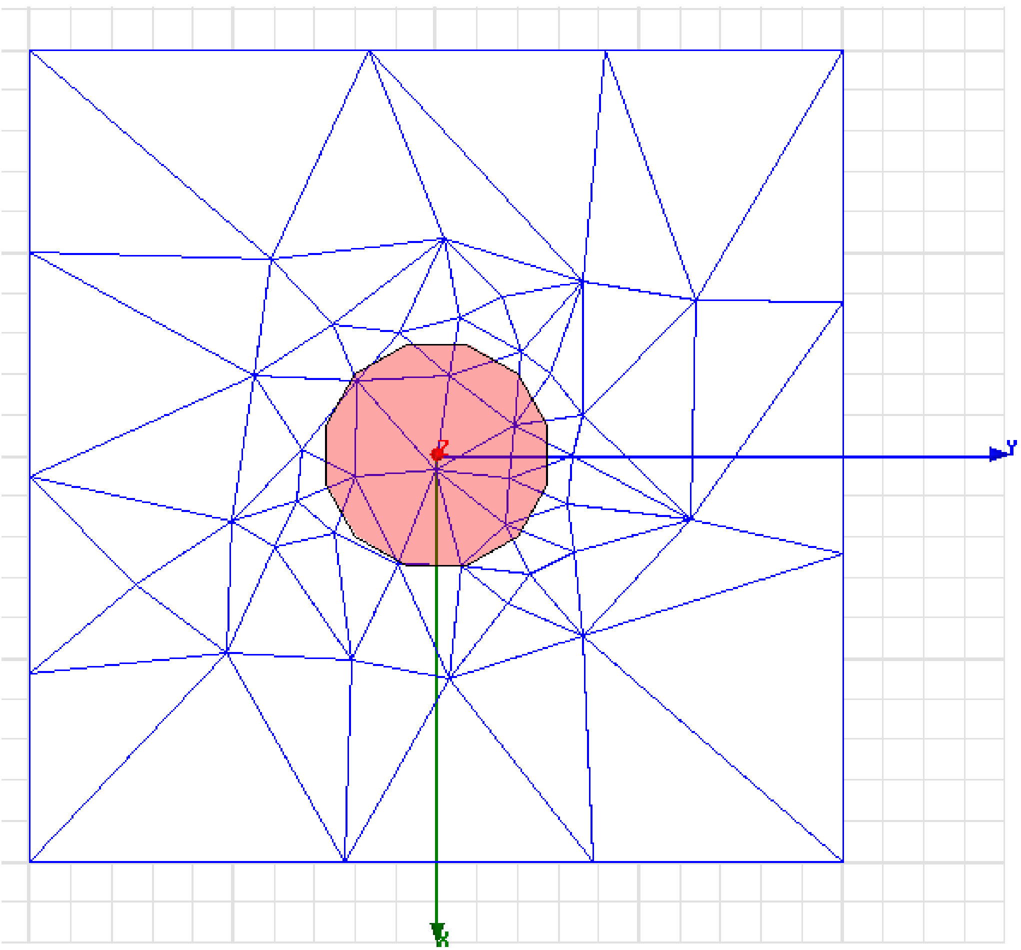
- Mesh after 4 passes

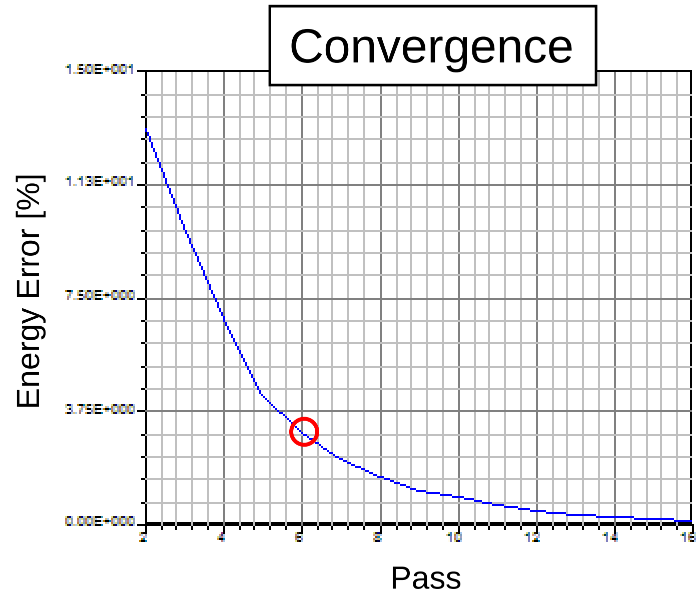




Adaptive Mesh Refinement

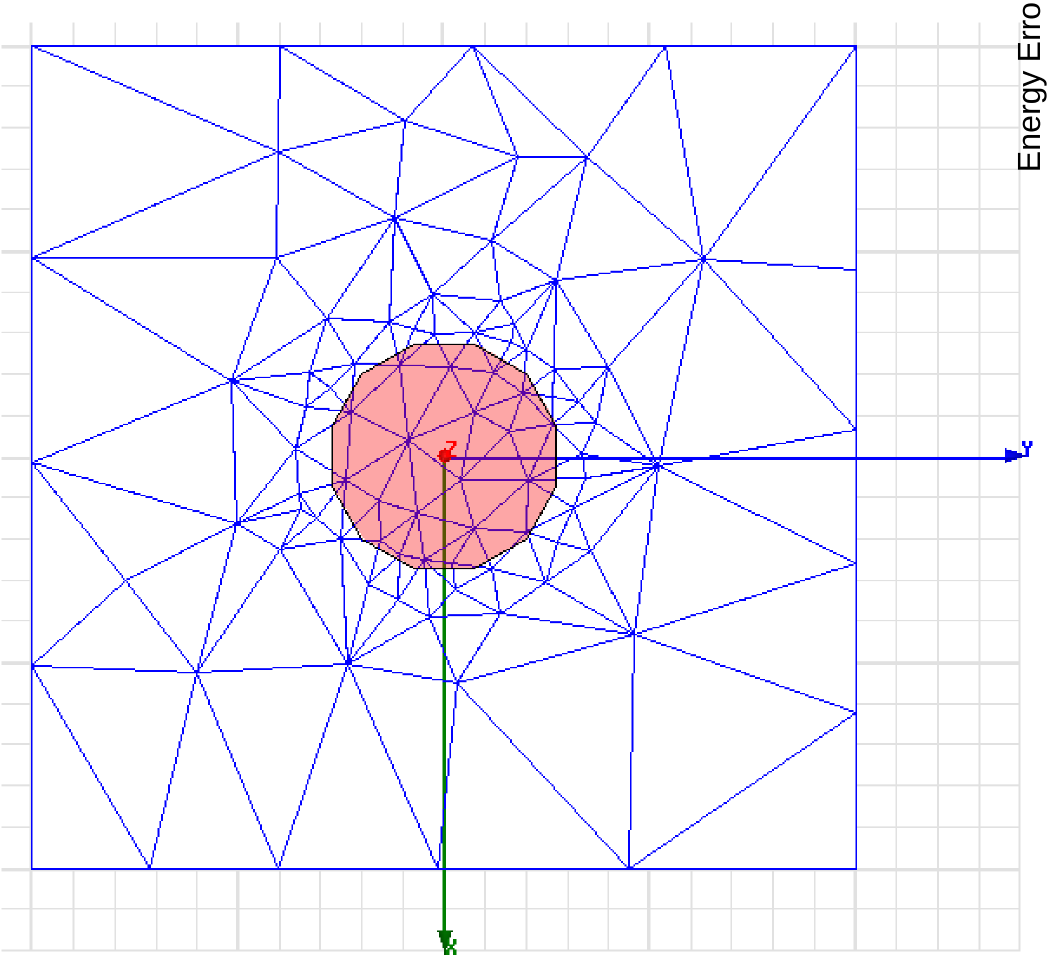
- Mesh after 6 passes

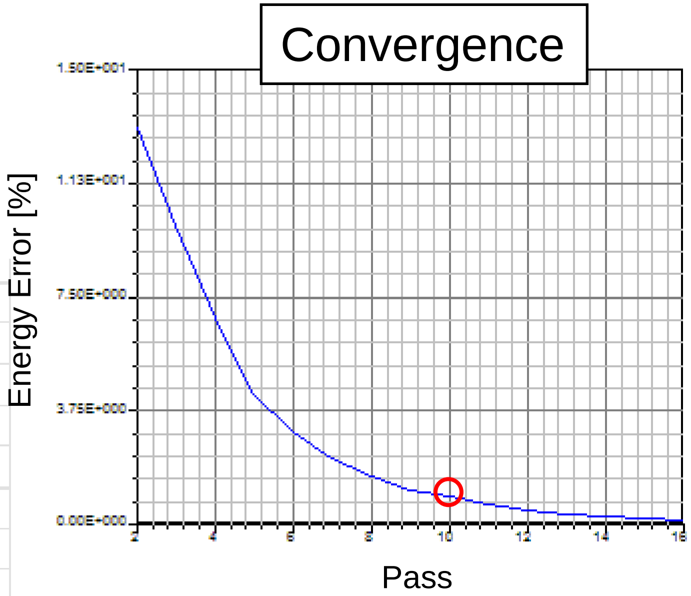




Adaptive Mesh Refinement

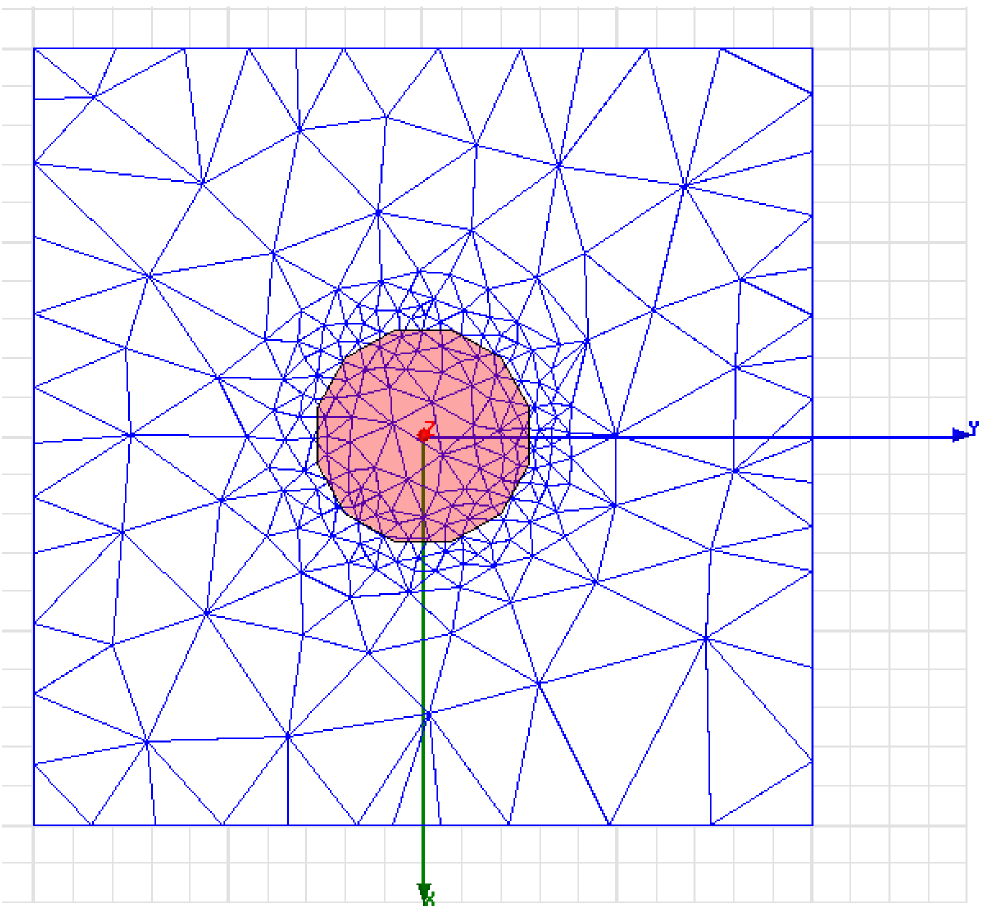
- Mesh after 10 passes

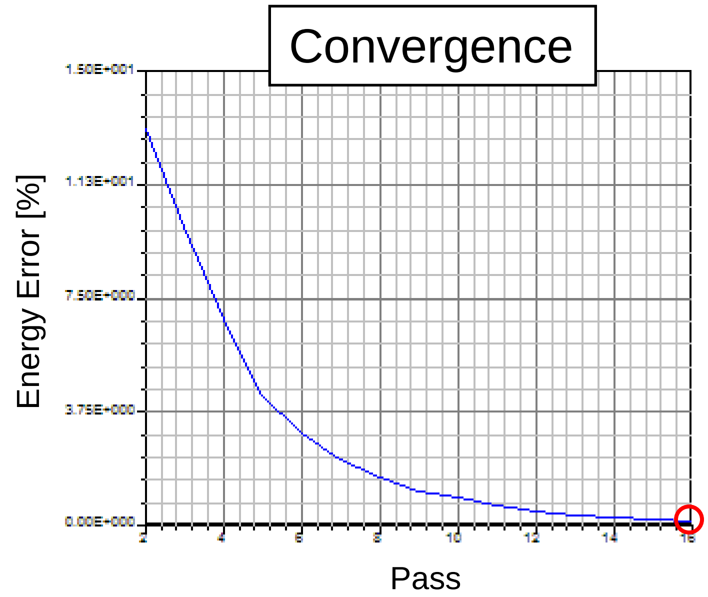




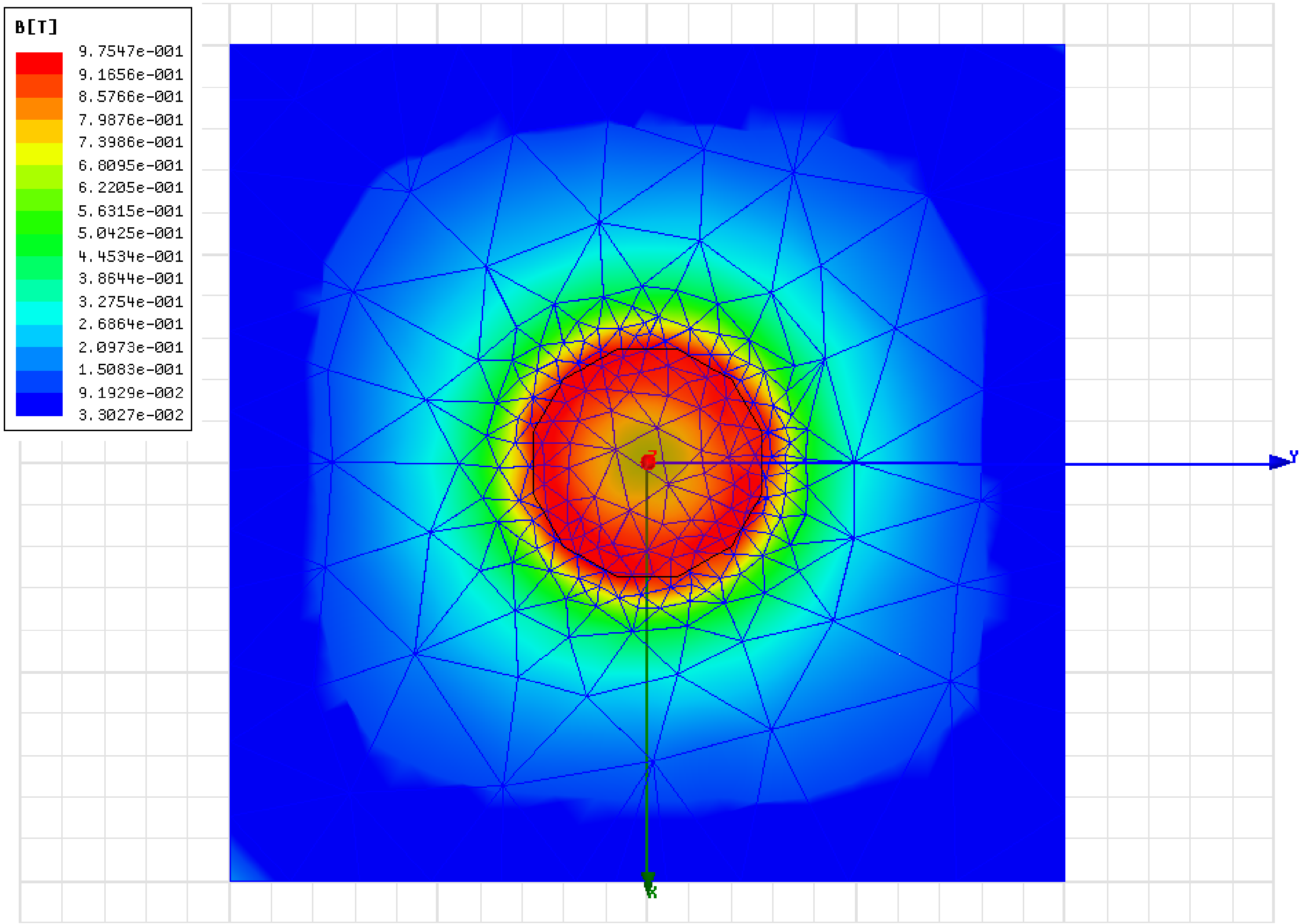
Adaptive Mesh Refinement

- Mesh after 16 passes.
  - Desired Energy Error of 0.2% is reached



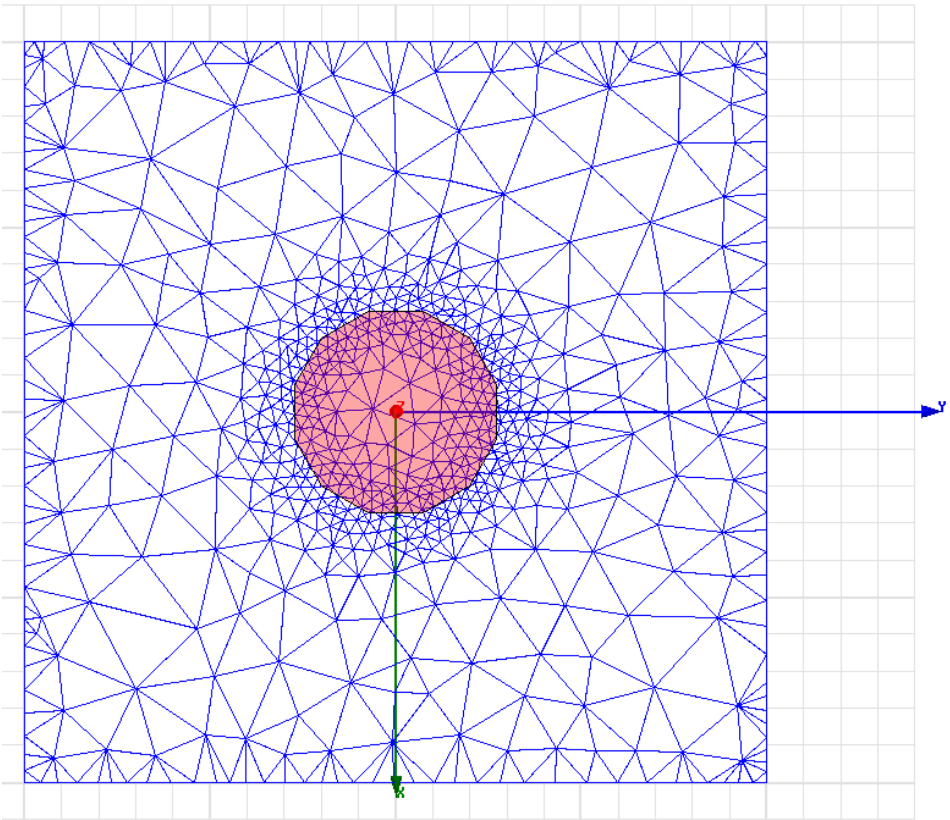


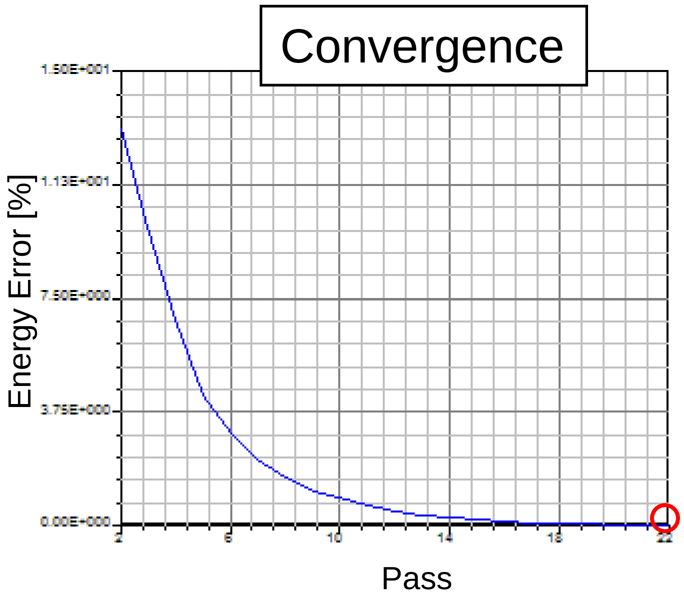
Plot of |B| on surface of the Plate (DC after 16 passes)



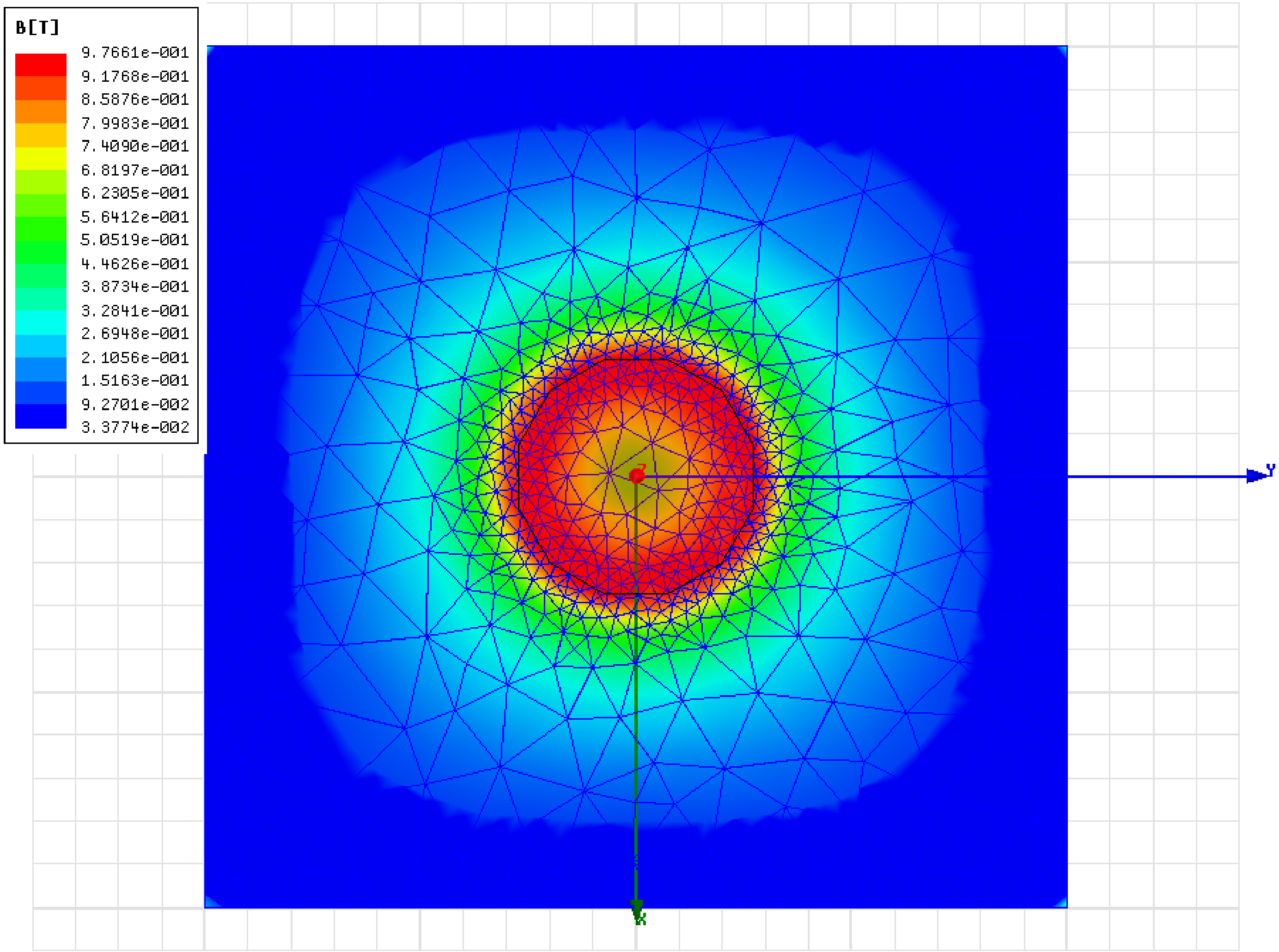
Adaptive Mesh Refinement

- Energy Error goal reduced further.
- Mesh after 22 passes. Desired **Energy Error of 0.03%** is reached

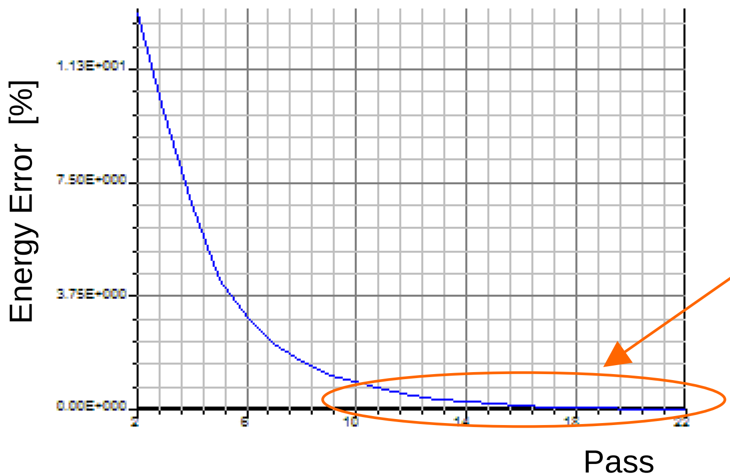


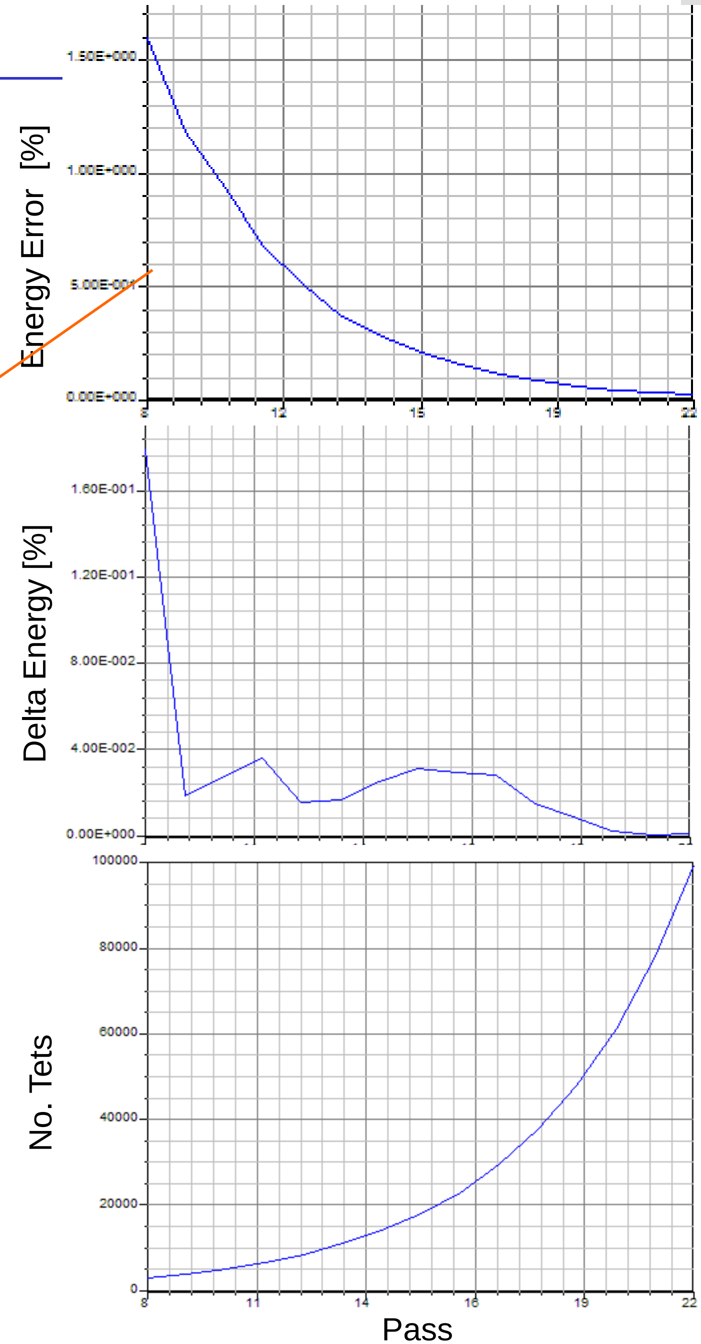


Plot of |B| on surface of the Plate (DC after 22 passes)



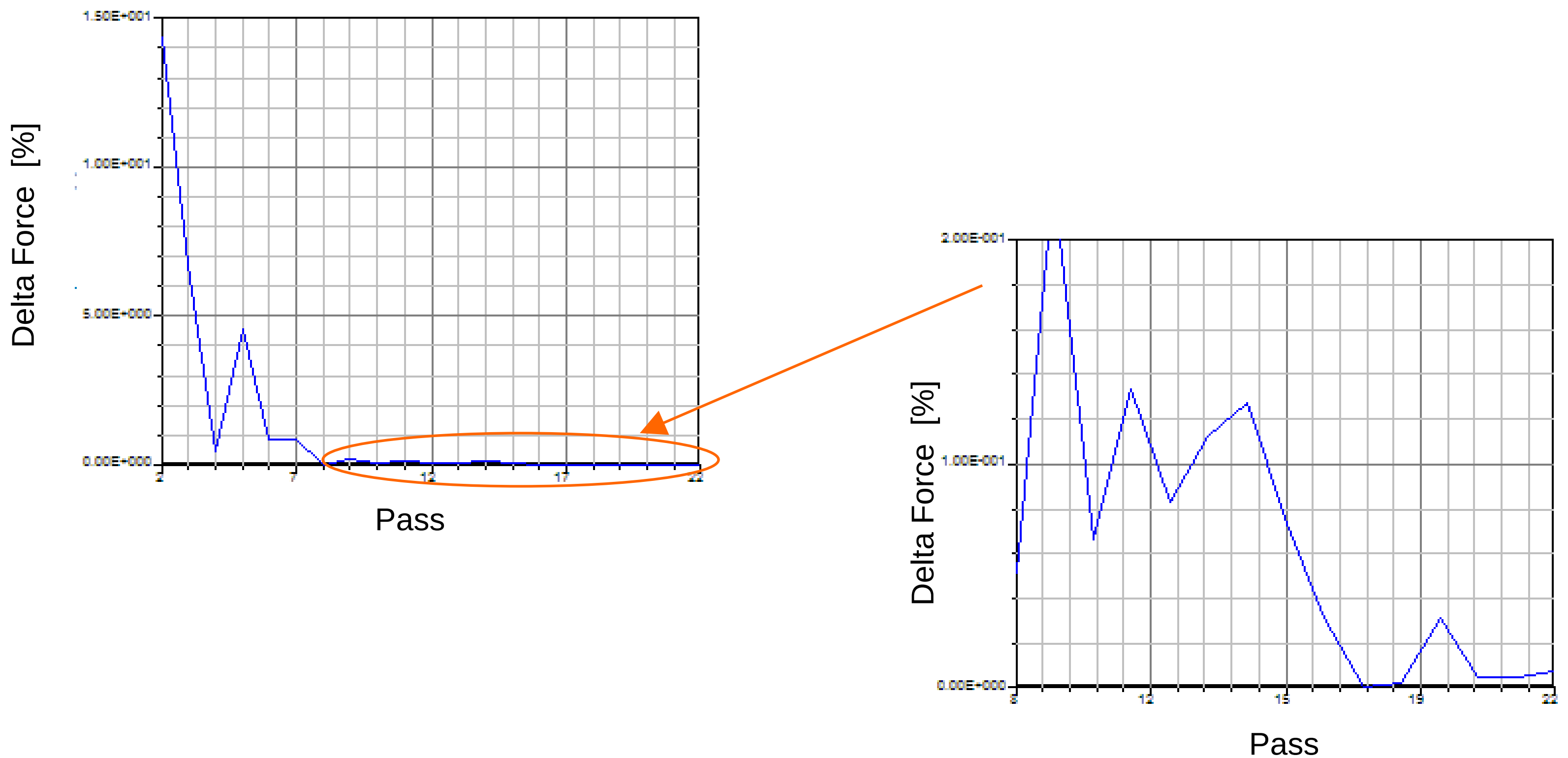
Convergence





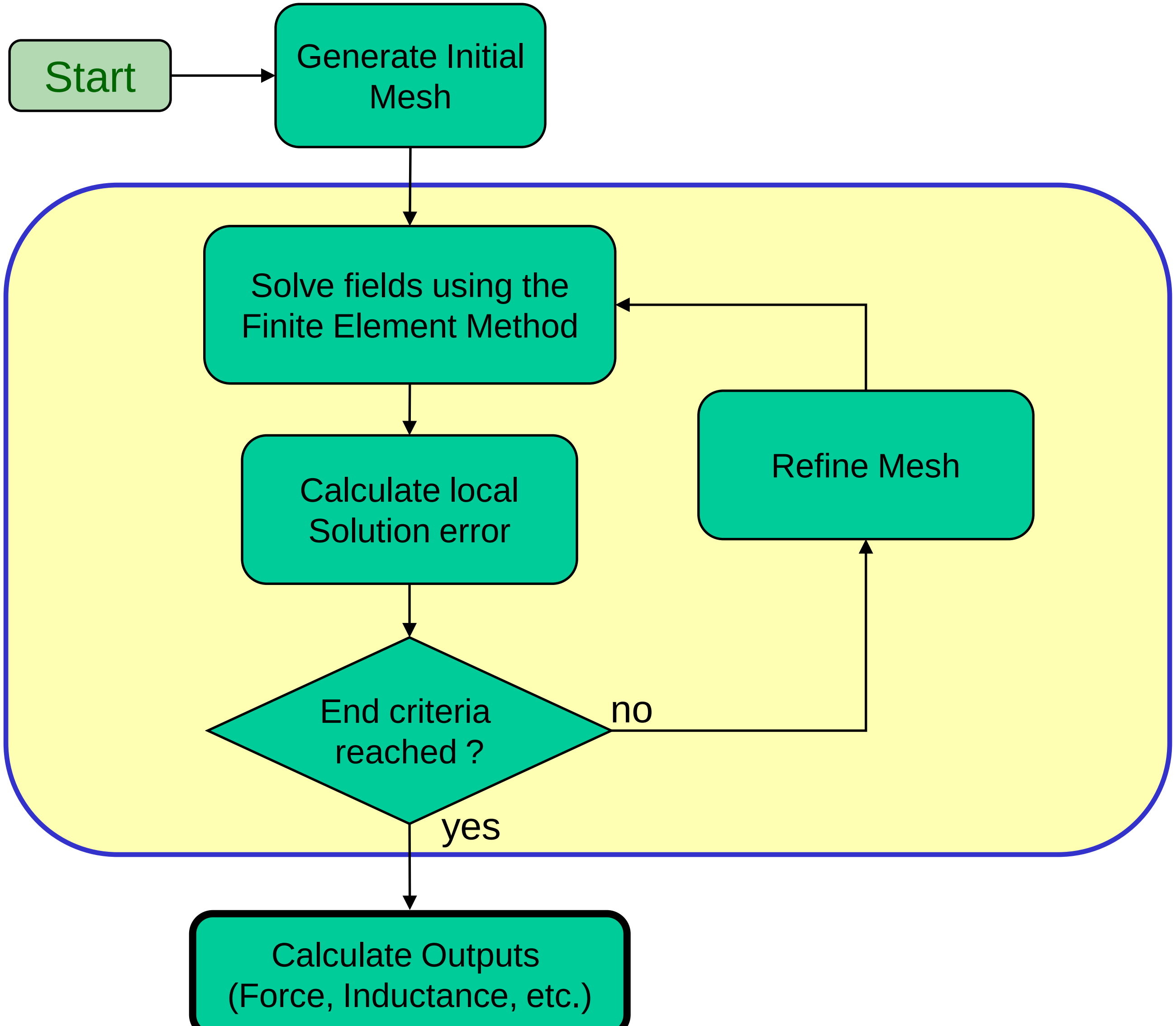
- Even in the region of the convergence curve above pass number 16, a gradient in the energy error from pass to pass can be observed.
- Continued iteration results in higher accuracy but at a cost of a large number of tetrahedrons.
- Clearly a tradeoff must be found

Convergence definition through use of additional variables



- % Change in the Force on the magnet can be used to determine a more effective stopping criteria since this value can be tied directly to the acceptable numerical tolerance on a physical variable

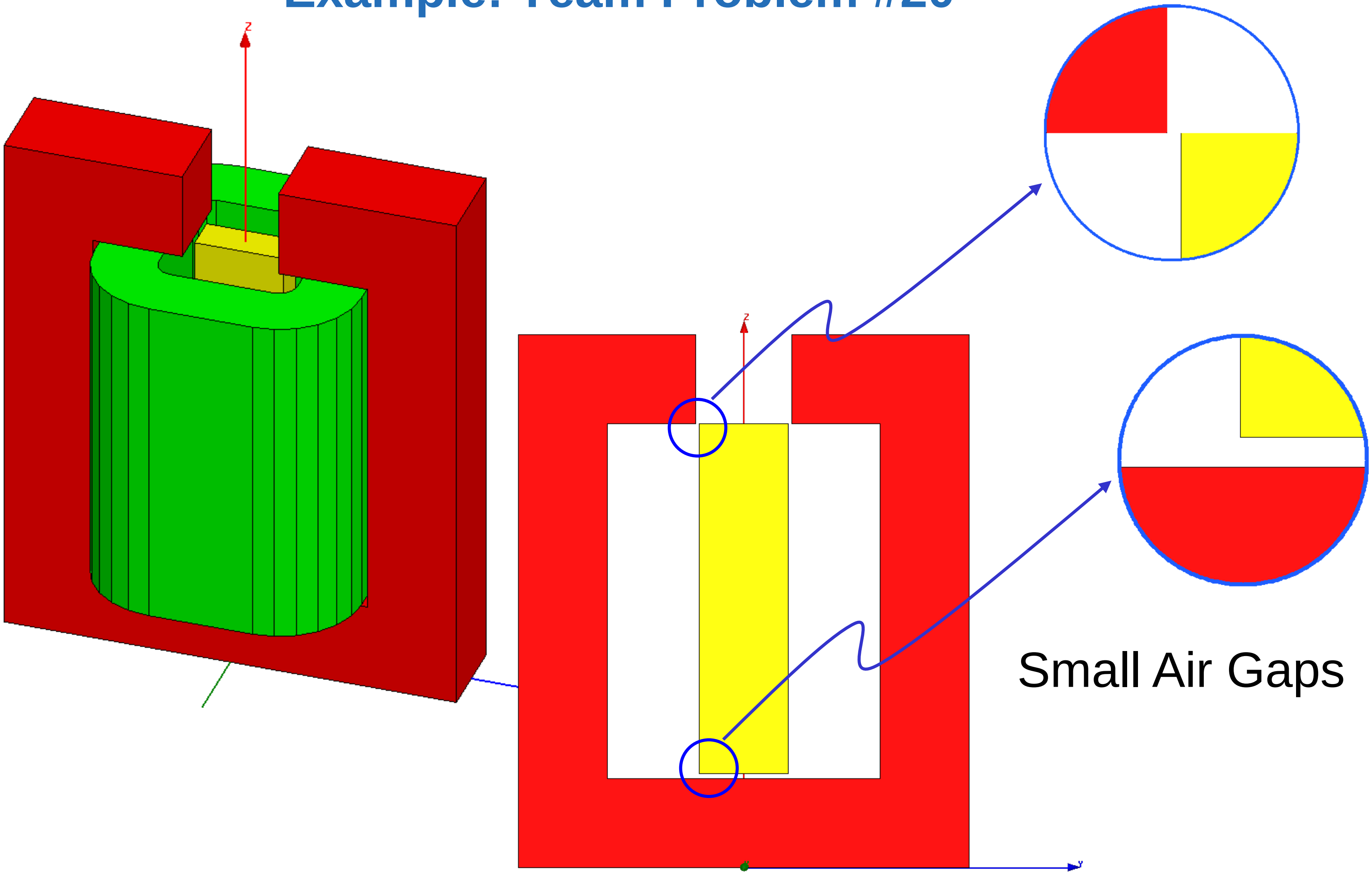
The “Solve” Procedure in Maxwell



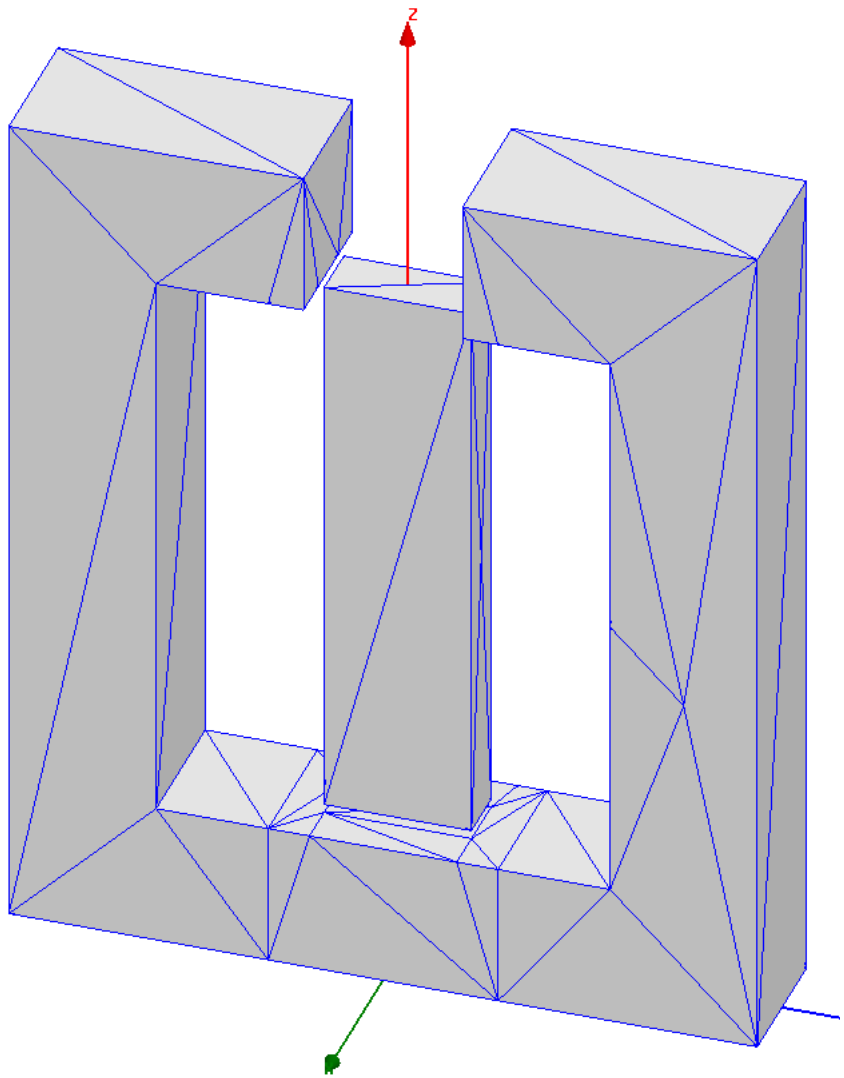
Summery

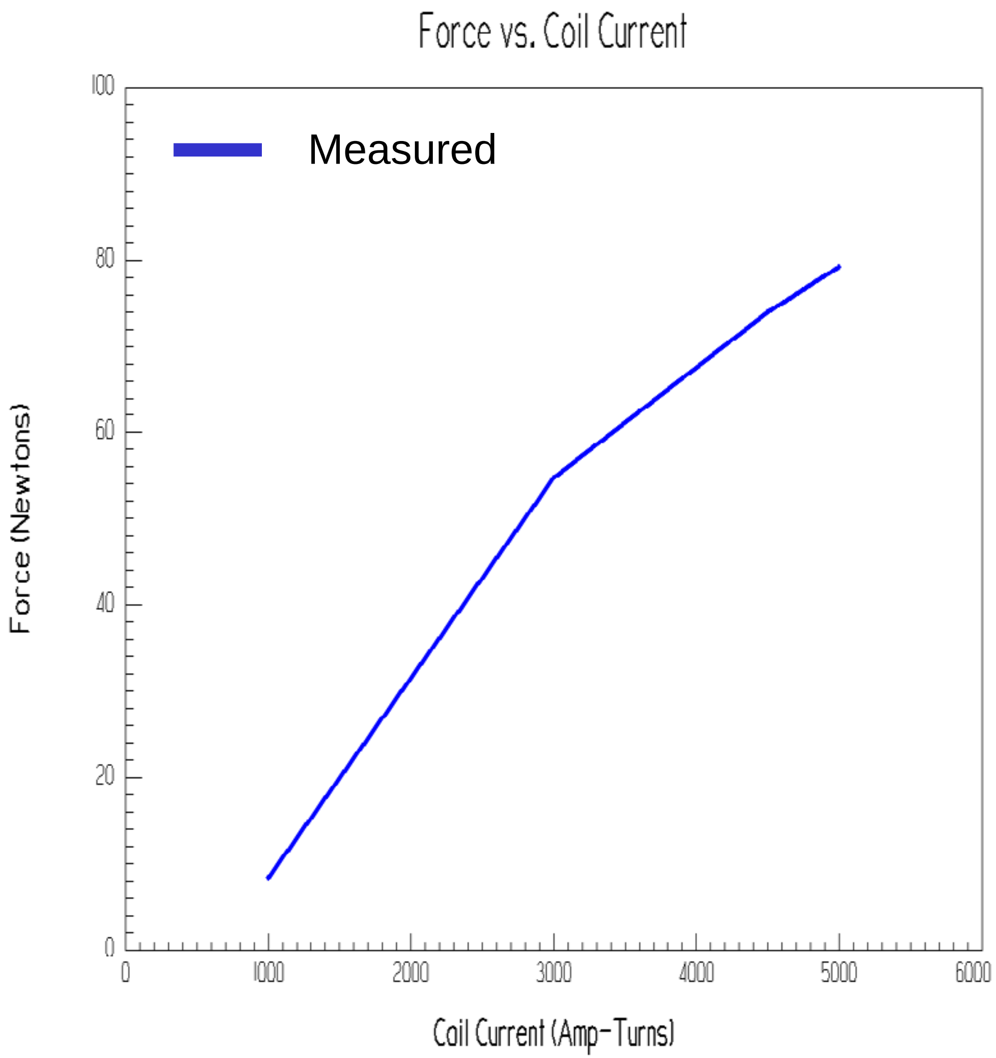
- Most of the execution time in a field solver is spent on the “final” solution. This is the solution that uses the largest mesh (i.e. the most finely discretized model) and hence yields the most accurate results.
- Autoadaptive mesh refinement generates an efficient mesh without requiring the user to be an “expert mesher”.
- User defined convergence criteria enable quantification of the trade-off between accuracy and calculation time.

Example: Team Problem #20

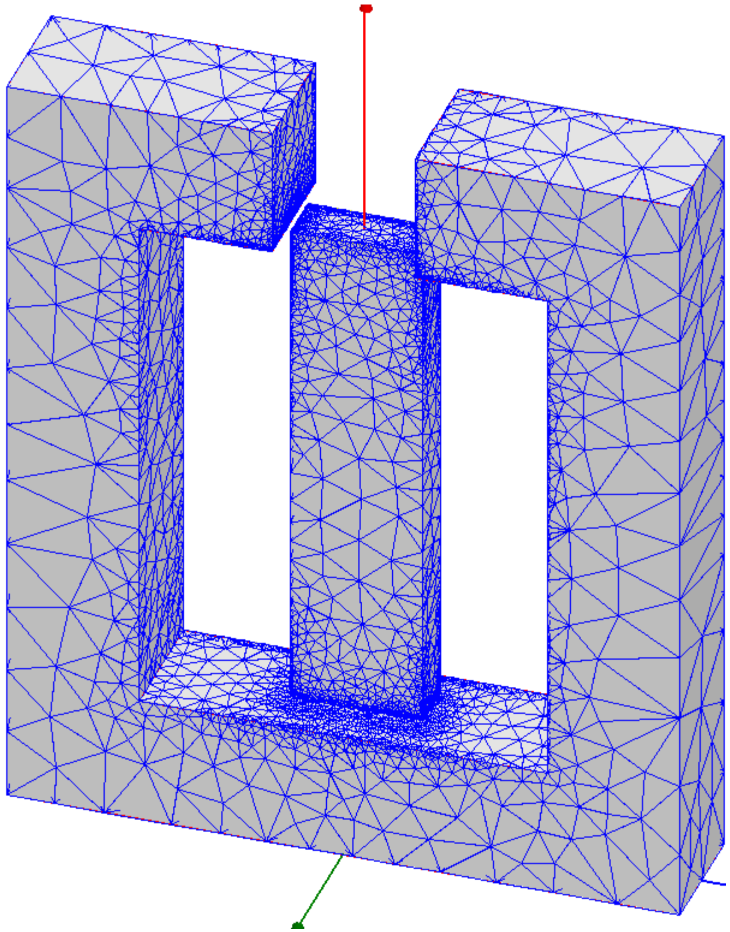


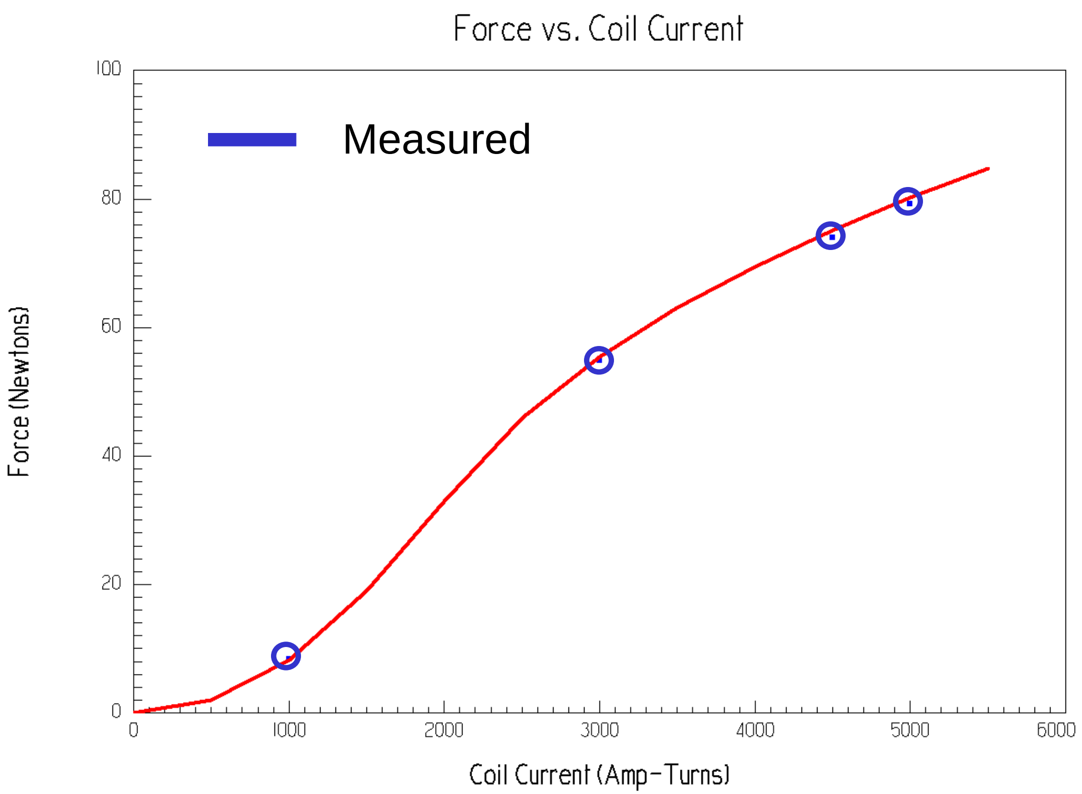
Automatic Adaptive Meshing





Comparison to Measurement





Overview

- ANSYS Maxwell has very powerful and flexible data management and plotting capabilities. Once understood, it will make the whole solution process much easier, and will help craft the entire problem setup.

Topics of Discussion

- 2D Plotting
- 3D Plotting
- Field Plotting

Data Plotting

- Data plotting can take a variety of forms. The most often used format is 2D Cartesian plotting, but we also have the capability to plot in 3D as well. Below is a list of all the quantities that can be plotted on various graphs. For definitions of each of these quantities, see the online help or the respective solution type chapter.
  - **Magnetostatic, Eddy Current, Electric**
    - Force
    - Torque
    - Matrix
  - **Transient**
    - Motion
      - Position
      - Force/Torque
      - Velocity
    - Winding
      - Flux Linkage
      - Induced Voltage
    - Core Loss
    - Power Loss
    - Branch Current (external meter required)
    - Node Voltage (external meter required)
  - **Fields**
    - Mag_H
    - Mag_B
    - Mag_J
    - Energy Density
    - Coenergy Density
    - Apparent Energy Density
    - Ohmic Loss Density
    - Named Expressions
  - **NOTE**: For all data plots of Fields data, a Line must be defined before creating the plot.

Data Plotting (Continued)

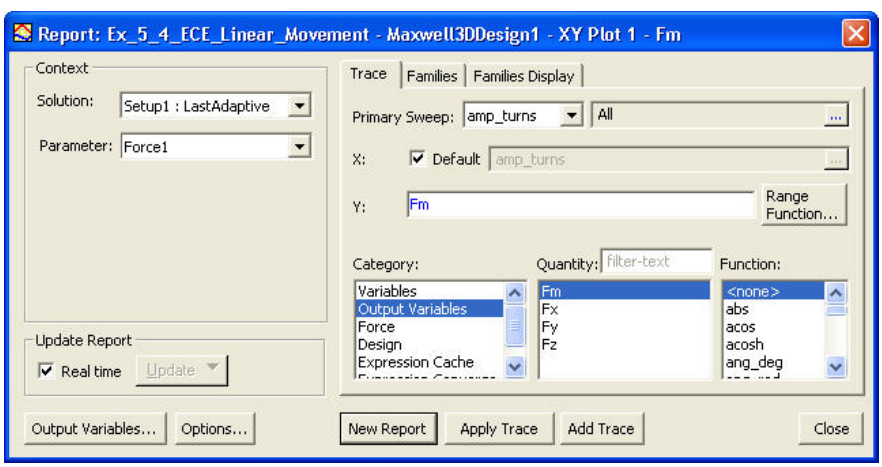
- **Types of Plots:**
  - Rectangular Plot
  - Data Table
  - 3D Rectangular Plot
- **To Create a Plot:**
  1. Select ***Maxwell 3D > Results > Create Magnetostatic Report***
  2. Reports Dialog will be displayed (options on the next page)

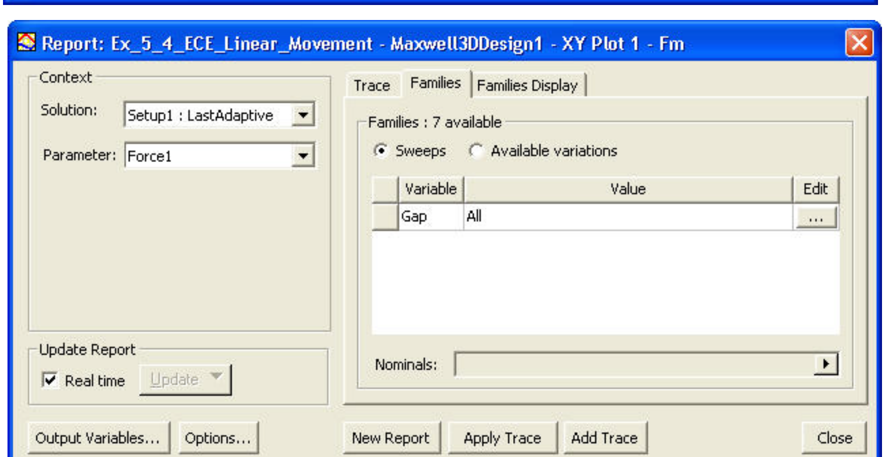
  OR

  1. Select ***Maxwell 3D > Results > Create Fields Report***
  2. Reports Dialog will be displayed (options on the next page)

  OR

  1. Select ***Maxwell 3D > Results > Create Quick Report*** (when available)
  2. Choose Quick Report type and click OK.

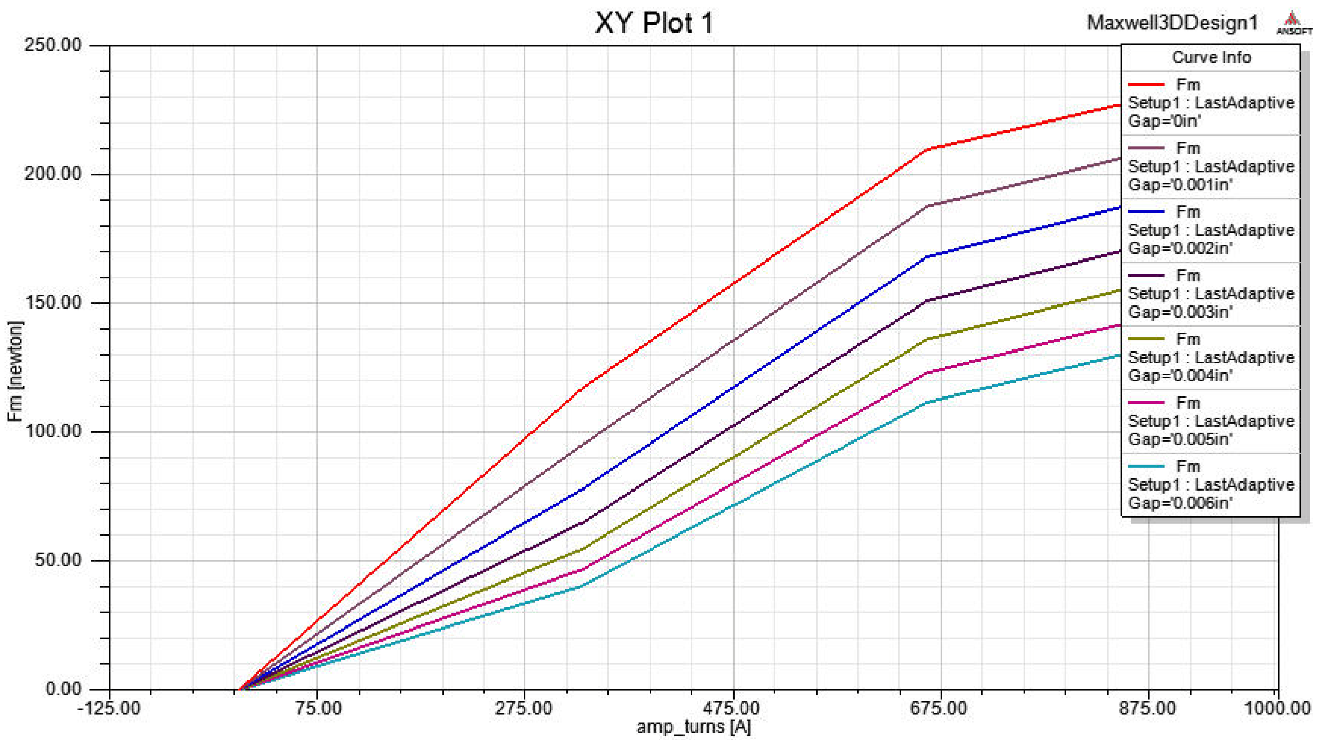




Data Plotting (Continued)

- **Creating a Plot** (continued)
  4. **Context**
     - **Design** – choose from available designs within a project
     - **Solution** – choose from available setups or solution types
     - **Geometry** – if plotting Fields data, select an available Line from the list.
  5. **Trace Tab**
     - **Primary Sweep** – controls the source of the independent variable in the plot.
       - **NOTE:** By default, the Report editor selects **Use Current Design and Project variable values**. This will select the primary sweep of distance for a field data plot, or an undefined variable X for other plot types, and the current simulated values of the project variables
       - To display a plot with multiple traces for different variable values, select the **Families** tab and select the **Sweeps** for variables to be plot with. The Plot will have Primary variable as X axis and multiple traces corresponding to other variables selected from Families tab. You can also select Particular values of Sweep variable by selecting the radio button to **Available variations**
     - **X** – controls any functional operator on the independent variable (this is usually set to **Primary Sweep**).
     - **Y** – select the value to be plotted and any operator.
       - **Category:** Select the category of the variable which needs to be plot
       - **Quantity:** to Select the variable
       - **Function:** To perform arithmetic operations on variable before plot
  6. Select **Add Trace** for as many values as you would like to plot
  7. Select **Done** when finished
- An example of a multi-trace plot of the sweep tab shown on the previous page is shown next

Data Plotting (Continued)



Examples of the Field Calculator in Maxwell3D

- The Field Calculator can be used for a variety of tasks, however its primary use is to extend the post-processing capabilities within Maxwell beyond the calculation / plotting of the main field quantities. The Field Calculator makes it possible to operate with primary vector fields (such as H, B, J, etc) using vector algebra and calculus operations in a way that is both mathematically correct and meaningful from a Maxwell's equations perspective.

- The Field Calculator can also operate with geometry quantities for three basic purposes:
  - plot field quantities (or derived quantities) onto geometric entities;
  - perform integration (line, surface, volume) of quantities over specified geometric entities;
  - export field results in a user specified box or at a user specified set of locations (points).

- Another important feature of the (field) calculator is that it can be fully script driven. All operations that can be performed in the calculator have a corresponding "image" in one or more lines of VBscript code. Scripts are widely used for repetitive post-processing operations, for support purposes and in cases where Optimetrics is used and post-processing scripts provide some quantity required in the optimization / parameterization process.

- This document describes the mechanics of the tools as well as the "softer" side of it as well. So, apart from describing the structure of the interface this document will show examples of how to use the calculator to perform many of the post-processing operations encountered in practical, day to day engineering activity using Maxwell. Examples are grouped according to the type of solution. Keep in mind that most of the examples can be easily transposed into similar operations performed with solutions of different physical nature.

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to interact with the calculator as covered in this topic
  - Analysis
    - Electrostatic
    - DC Conduction
    - Magnetostatic
    - Eddy Current
    - Transient
  - Results
    - Output Variables
    - **Field Calculator**
  - Field Overlays:
    - Named Expressions
    - Animate

Description of the interface

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

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

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

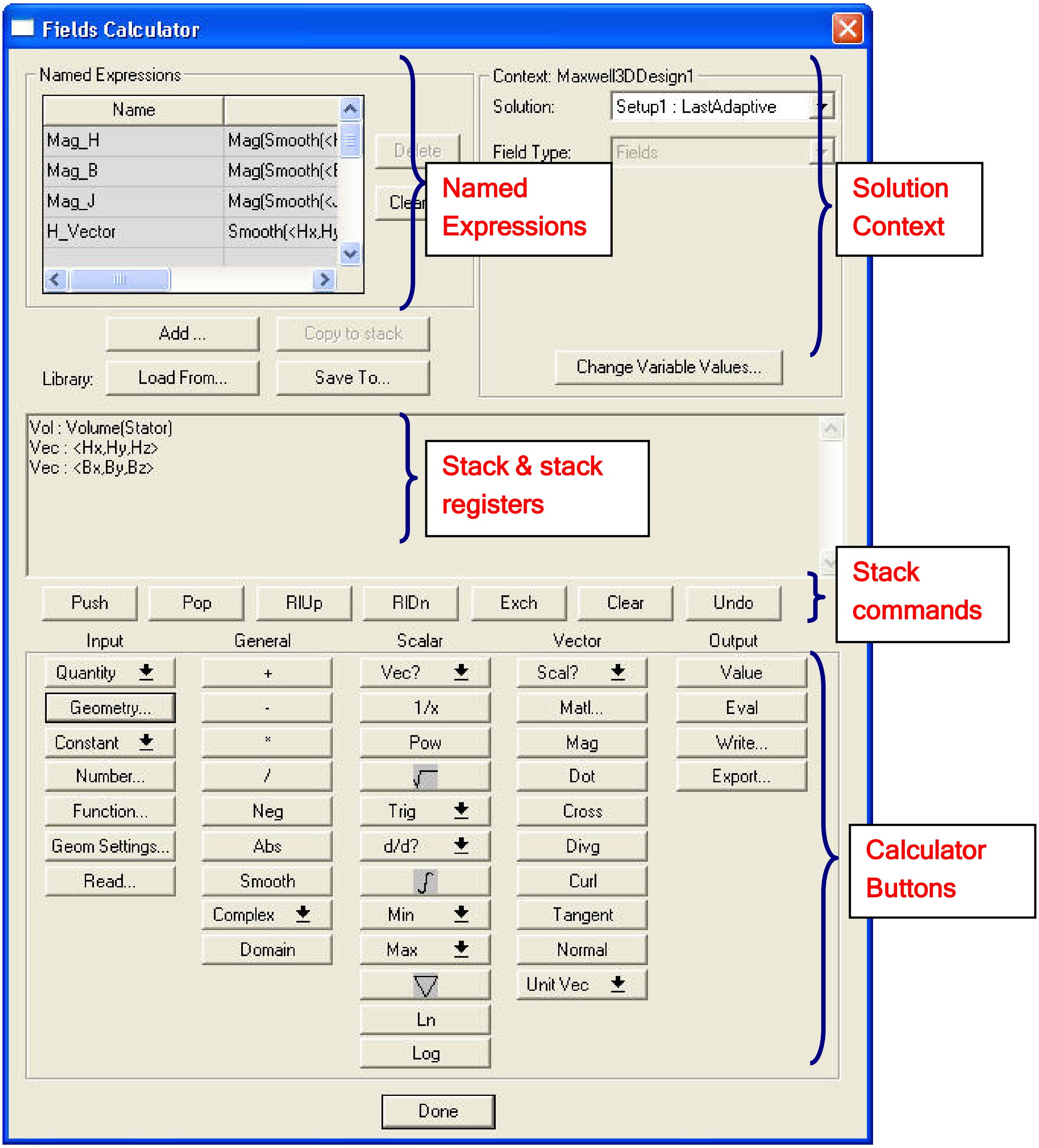


Fig. I1 Field Calculator Interface

- **The calculator buttons are organized in five categories as follows:**

  - **Input** contains calculator buttons that allow the user to enter data in the stack; sub-categories contain solution vector fields (B, H, J, etc.), geometry(point, line, surface, volume, coordinate system), scalar, vector or complex constants (depending on application) or even entire f.e.m. solutions.
  - **General** contains general calculator operations that can be performed with "general" data (scalar, vector or complex), if the operation makes sense; for example if the top two entries on the stack are two vectors, one can perform the addition (+) but not multiplication (*);indeed, with vectors one can perform a dot product or a cross product but not a multiplication as it is possible with scalars.
  - **Scalar** contains operations that can be performed on scalars; example of scalars are scalar constants, scalar fields, mathematical operations performed on vector which result in a scalar, components of vector fields (such as the X component of a vector field), etc.
  - **Vector** contains operations that can be performed on vectors only; example of such operations are cross product (of two vectors), div, curl, etc.
  - **Output** contains operations resulting in plots (2D / 3D), graphs, data export, data evaluation, etc.

- **As a rule, calculator operations are allowed if they make sense from a mathematical point of view. There are situations however where the contents of the top stack registers should be in a certain order for the operation to produce the expected result. The examples that follow will indicate the steps to be followed in order to obtain the desired result in a number of frequently encountered operations. The examples are grouped according to the type of solution (solver) used. They are typical medium/higher level post-processing task that can be encountered in current engineering practice. Throughout this manual it is assumed that the user has the basic skills of using the Field Calculator for basic operations as explained in the on-line technical documentation and/or during Ansoft basic training.**

- **Note: The f.e.m. solution is always performed in the global (fixed) coordinate system. The plots of vector quantities are therefore related to the global coordinate system and will not change if a local coordinate system is defined with a different orientation from the global coordinate system.**

Electrostatic Examples

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

- *Description*: Assume an electrostatic (3D) application with separate metallic objects having applied voltages or floating voltages. The task is to calculate the total electric charge on any of the objects.
- Calculate/plot the charge density distribution on the object; the sequence of calculator operations is described below:

- **Input > Quantity > D** (load D vector into the calculator);
- **Input > Geometry**
  - In **Geometry** window,
    1. Select radio Button to **Surface**
    2. Select the Surface of interest and Press **OK**
- **Vector > Unit Vec > Normal** (creates the normal unit vector corresponding to the surface of interest)
- **Vector > Dot** (creates the dot product between D and the unit normal vector to the surface of interest, equal to the surface charge density)
- **Add** *(input "charge_density" as the name)* -> **OK** (creates a named expression and adds it to the list)
- **Done** (leaves calculator)
- *(select the surface of interest from the model)*
- ***Maxwell 3D > Fields > Fields > Named Expressions*** (a Selecting calculated expression window appears)
- *(select "charge_density" from the list)* -> **OK** (A Create Field Plot window appears)
- **Done**

- Calculate the total electric charge on the surface of an object
- **Input > Quantity > D** (load D vector into the calculator);
- **Input > Geometry > Surface…** *(select the surface of interest)* **-> OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

- **Example ES2: Calculate the Maxwell stress distribution on the surface of an object**
- *Description:* Assume an electrostatic application (for ex. a parallel plate capacitor structure). The surface of interest and adjacent region should have a fine finite element mesh since the Maxwell stress method for calculation the force is quite sensitive to mesh.
- The Maxwell electric stress vector has the following expression for objects without electrostrictive effects:

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

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

- **Input > Quantity > D**
- **Input > Geometry > Surface** *(select the surface of interest)* **> OK**
- **Vector > Unit Vec > Normal** (creates the normal unit vector corresponding to the surface of interest)
- **Vector > Dot**
- **Input > Quantity > E**
- **General > *** (multiply)
- **Input > Geometry > Surface…** *(select the surface of interest)* **> OK**
- **Vector > Unit Vec > Normal** (creates the normal unit vector corresponding to the surface of interest)
- **Input > Number >Scalar** (0.5) **OK**
- **General > ***
- **Input > Constant > Epsi0**
- **General > ***
- **Input > Quantity > E**
- **Push**
- **Dot**
- **General > ***
- **General > -** (minus)
- Continued on next page.

Example ES2: Continued

- **Add …** *(input "stress" as the name)* > **OK** (creates a named expression and adds it to the list)
- **Done** (leaves calculator)
- *(select the surface of interest from the model)*
- ***Maxwell 3D > Fields > Fields > Named Expressions*** (a Selecting calculated expression window appears)
- *(select "stress" from the list)* > **OK** (A Create Field Plot window appears)
- **Done**

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

- *(select "stress" from the Named Expressions list)*
- **Copy to stack** (inserts the named expression to the stack)
- **Input > Geometry > Surface** *(select surface of interest)* > **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

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

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

where Tn is the local tension force acting along the normal direction to the surface and F is the total force acting on object(s) inside Σ.
- The above results for the electrostatic case hold for magnetostatic applications if the electric field quantities are replaced with corresponding magnetic quantities.

Current flow Examples

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

- *Description:* Assume a given conductor geometry that extends between two terminals with applied DC currents.
- In DC applications (static current flow) one frequent question is related to the calculation of the resistance when one has the field solution to the conduction (current flow) problem. The formula for the analytical calculation of the DC resistance is:

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

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

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

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

- The sequence of calculator commands to compute the power loss P is given below:
- **Input > Quantity > J**
- **Push**
- **Input > Number > Scalar** (1e7) **OK** (conductivity assumed to be 1e7 S/m)
- **General > /** (divide)
- **Vector > Dot**
- **Input > Geometry > Volume** *(select the volume of interest)* **> OK**
- **Scalar >** $\int$
- **Output > Eval**
- The resistance can now be easily calculated from power and the square of the current.
- Continued on next page.

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

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

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

- ## Example CF2: Export the field solution to a uniform grid

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

- Continued on next page.

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

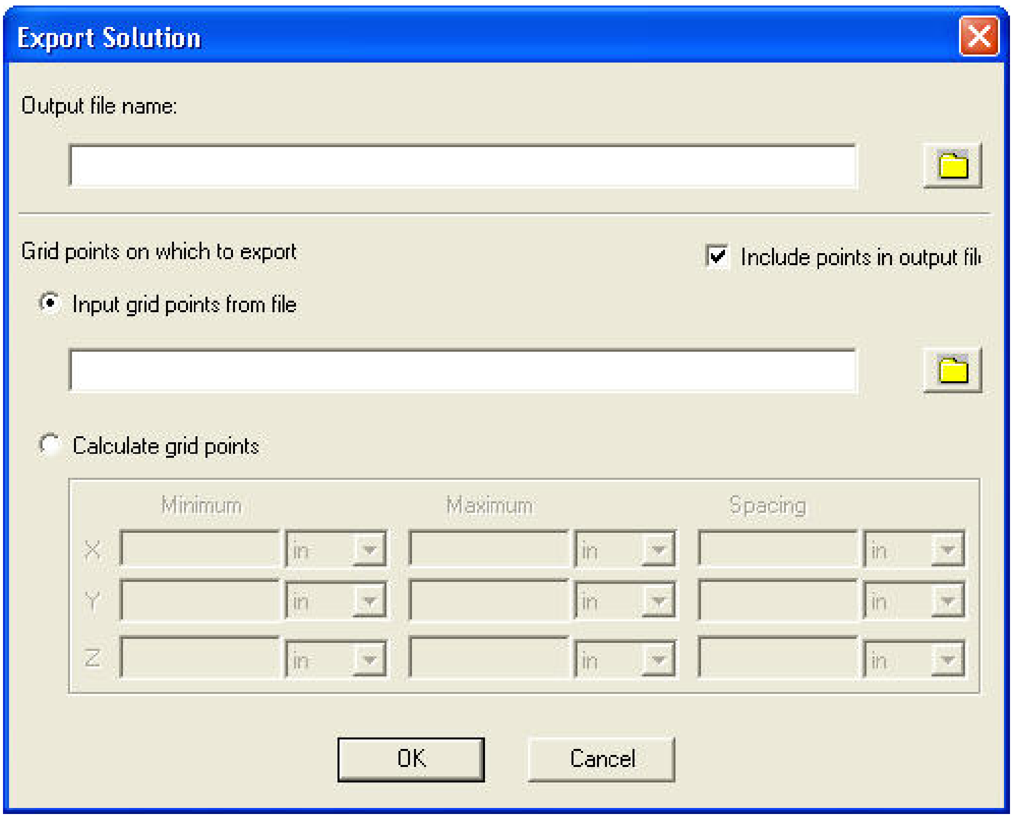


Fig. CF2 Define the size of the export region (box) and spacing within

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

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

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

- **Input > Quantity > J**
- **Input > Geometry > Volume** (choose the volume of the branch of interest) **OK**
- **General > Domain** (this is to limit the subsequent calculations to the branch of interest only)
- **Input > Geometry > Surface > yz** (choose axis plane that cuts perpendicular to the branch) **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

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

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

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

Magnetostatic examples

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

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

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

  where Γ is the closed contour (polyline) and $S\Gamma$ is an open surface bounded by Γ but otherwise of arbitrary shape. $I_{S\Gamma}$ is the total current intercepting the surface $S\Gamma$.
- To calculate the (closed) line integral of H, the sequence of field calculator commands is:

- **Input > Quantity > H**
- **Input > Geometry > Line** *(choose the closed polygonal line around the conductor)* **OK**
- **Vector > Tangent**
- **Scalar >** $\int$
- **Output > Eval**

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

Example MS2: Calculate the magnetic flux through a surface

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

- **Input > Quantity > B**
- **Input > Geometry > Surface** *(specify the integration surface)* **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

- The result is positive or negative depending on the orientation of the B vector with respect to the normal to the surface of integration.
- The above operation corresponds to the following mathematical formula for the magnetic flux:

$$\Phi_S = \int_S \vec{B} \cdot \vec{n}\, dA$$

Example MS3: Calculate components of the Lorentz force

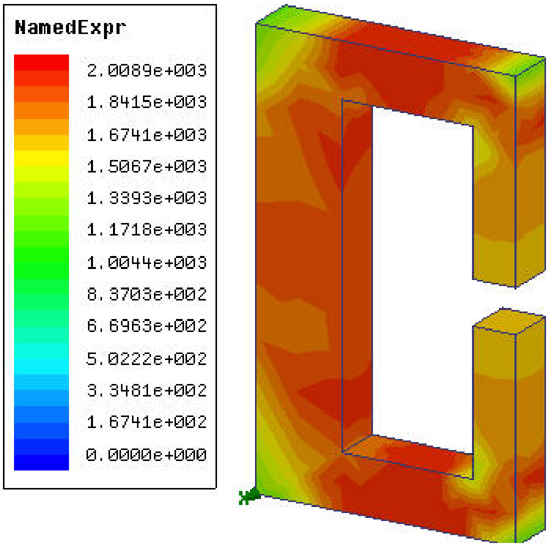
- *Description:* Assume a distribution of magnetic field surrounding conductors with applied DC currents. The calculation of the components of the Lorentz force has the following steps in the field calculator.

- **Input > Quantity > J**
- **Input > Quantity > B**
- **Vector > Cross**
- **Vector > Scal? > ScalarX**
- **Input > Geometry > Volume** *(specify the volume of interest)* **OK**
- **Scalar >** $\int$
- **Output > Eval**

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

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

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



- Input > Quantity > B
- Vector > Scal? > ScalarX
- Input > Quantity > H
- Vector > Scal? > ScalarX
- Input > Constant > Mu0
- General > * (multiply)
- General > / (divide)
- General > Smooth
- Add *(input "rel_perm" as the name)*
- OK (creates a named expression and adds it to the list)
- Done (leaves calculator)

- *(select the surface of interest from the model)*
- ***Maxwell 3D > Fields > Fields > Named Expressions*** (a Selecting calculated expression window appears)
- *(select "rel_perm" from the list)* > OK (a Create Field Plot window appears)
- Done

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

Frequency domain (AC) Examples

Example AC1: Calculate the radiation resistance of a circular loop

- *Description:* Assume a circular loop of radius 0.02 m with an applied current excitation at 1.5 GHz;
- The radiation resistance is given by the following formula:

$$R_r = \frac{P_{av}}{I_{rms}^2} \qquad P_{av} = \frac{1}{2}\oint\!\!\oint_S \mathrm{Re}\left(\vec{E} \times \vec{H}^*\right) dS = \frac{1}{2}\oint\!\!\oint_S \mathrm{Re}\left[\frac{1}{j\omega\varepsilon_0}\left(\nabla \times \vec{H}\right) \times \vec{H}^*\right] dS$$

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

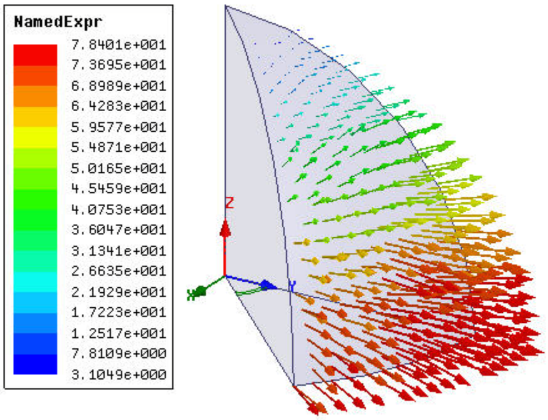
- Assuming that a half symmetry model is used, no ½ is needed in the above formula. The sequence of calculator commands necessary for the calculation of the average power is as follows:

- **Input > Quantity > H**
- **Vector > Curl**
- **Input > Number > Complex (0 , -12) OK**
- **General > ***
- **Input > Quantity > H**
- **General > Complex > Conj**
- **Vector > Cross**
- **General > Complex > Real**
- **Input > Geometry > Surface** *(select the surface of interest)* **> OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

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

Example AC2: Calculate/Plot the Poynting vector

- *Description:* Same as in Example AC1.
- To obtain the Poynting vector the following sequence of calculator commands is necessary:



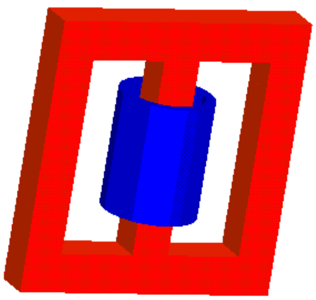
- **Input > Quantity > H**
- **Vector > Curl**
- **Input > Number > Complex (0 , -12) OK**
- **General > ***
- **Input > Quantity > H**
- **General > Complex > Conj**
- **Vector > Cross**

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

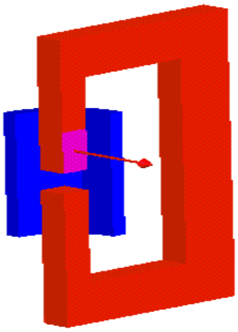
- **General > Complex > Real**
- **Input > Number > Scalar (0.5) OK**
- **General > ***
- **Add** *(input "Poynting" as the name)* > **OK** (creates a named expression and adds it to the list)
- **Done** (leaves calculator)
- *(select the surface of interest from the model)*
- ***Maxwell 3D > Fields > Fields > Named Expressions*** (a Selecting calculated expression window appears)
- *(select "Poynting" from the list)* > **OK** (a Create Field Plot window appears)
- **Done**

Example AC3: Calculate total induced current in a solid

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



a) full model



b) quarter model

Fig. AC3 Geometry of inductor model

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

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

- For the real part of the induced current:

- **Input > Quantity > J**
- **General > Complex > Real**
- **Input > Geometry > Surface** *(select previously defined integration surface)* **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

- For the imaginary part of the induced current:

- **Input > Quantity > J**
- **General > Complex > Imag**
- **Input > Geometry > Surface** *(select previously defined integration surface)* **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

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

- **Input > Quantity > J**
- **Input > Number > Scalar** (45) **OK** (assuming a calculation at 45 degrees phase angle)
- **General > Complex > AtPhase**
- **Input > Geometry > Surface** *(select previously defined integration surface)* **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Output > Eval**

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

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

- For the real part of the voltage:

- **Input > Quantity > J**
- **Vector > Matl > Conductivity > Divide OK**
- **General > Complex > Real**
- **Input > Geometry > Line >** *(select the applicable line)* **OK**
- **Vector > Tangent**
- **Scalar >** $\int$
- **Output > Eval**

- For the imaginary part of the voltage:

- **Input > Quantity > J**
- **Vector > Matl > Conductivity > Divide OK**
- **General > Complex > Imag**
- **Input > Geometry > Line >** *(select the applicable line)* **OK**
- **Vector > Tangent**
- **Scalar >** $\int$
- **Output > Eval**

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

- **Scalar > Trig > Atan2**

- **Example AC5: Calculate the AC resistance of a conductor**
- *Description:* Consider the existence of an AC application containing conductors with significant skin effect. Assume also that the mesh density is appropriate for the task, i.e. mesh has a layered structure with 1-2 layers per skin depth for 3-4 skin depths if the conductor allows it. Here is the sequence to follow in order to calculate the total power dissipated in the conductor of interest.

- **Input > Quantity > OhmicLoss**
- **Input > Geometry > Volume** *(specify volume of interest)* **OK**
- **Scalar >** $\int$
- **Output > Eval**
- *Note:* To obtain the AC resistance the power obtained above must be divided by the squared rms value of the current applied to the conductor. Note that in the Boundary/Source Manager peak values are entered for sources, not rms values.

Time Domain Examples

Example TD1: Plotting Transient Data

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

- **Input > Quantity > J**
- **Input > Geometry > Surface** *(enter the surface of interest)* **OK**
- **Vector > Normal**
- **Scalar >** $\int$
- **Add** *(input "Induced_Current" as the name)* > **OK** (creates a named expression and adds it to the list)
- **Done** (leaves calculator)
- ***Maxwell 3D > Results > Output Variables…*** (an Output Variables Dialogue appears)
- *(select Fields from the Report Type pull down menu)*
- *(select Calculator Expressions as the Category and "Induced_Current" as the Quantity)*
- **Insert Quantity Into Expression**
- *(name your expression by writing "Ind_Cur" in the Name textbox)*
- **Add** (adds an output variable and its expression to the list)
- **Done**

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

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

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

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

- **Input > Quantity > B**
- **Vector > Mag**
- **Input > Geometry > Volume** *(enter volume of interest)* **OK**
- **Scalar > Max > Value**
- **Output > Eval**

- To get the location of the maximum:

- **Input > Quantity > B**
- **Vector > Mag**
- **Input > Geometry > Volume** *(enter volume of interest)* **OK**
- **Scalar > Max > Position**
- **Output > Eval**

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

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

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

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

- Then in the calculator:
- **Input > Quantity > B**
- **Write** *(enter the name of the file)* **OK**
- Exit the calculator and choose a different time step, say t2.

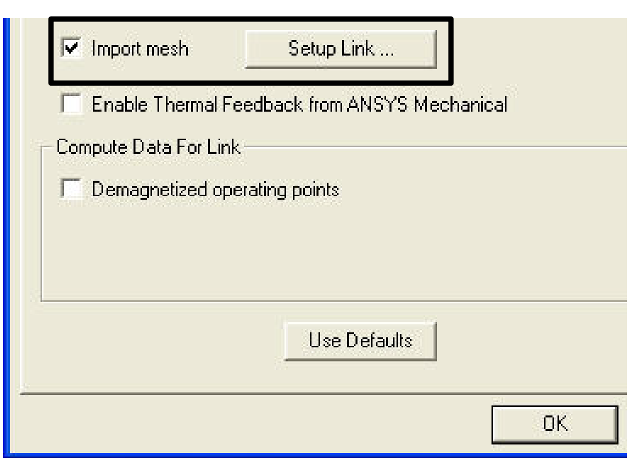
- Then, in the calculator:
- **Input > Quantity > B**
- **Read** *(specify the name of the .reg file to be read in)* **OK**
- **General > + (add)**
- **Vector > Mag**
- **Input > Geometry > Surface >** *(enter surface of interest)* **OK**

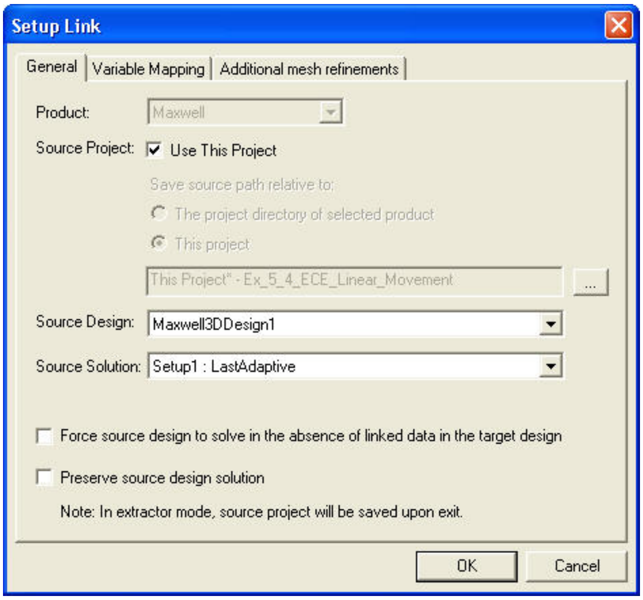
- *Note 1:* For this operation to succeed it is necessary that the respective meshes are identical. This condition is of course satisfied in transient applications without motion since they do not have adaptive meshing. It should be noted that this capability can be used in other solutions sequences –say static- if the meshes in the two models are identical.
- The whole operation is numeric entirely, therefore the nature of the quantities being “combined” is not checked from a physical significance point of view. It is possible to add for example an H vector solution to a B vector solution. This doesn’t have of course any physical significance, so the user is responsible for the physical significance of the operation.
- For the particular case of time domain applications it is possible to study the “displacement” of the (vector) solution from one time step to another, study the spatial orthogonality of two solution, etc. It is a very powerful capability that can be used in many interesting ways.

Example TD3: Continued

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

- As example one can consider the case of a cylindrical conductor above the ground plane with 1 Amp current, the voltage with respect to the ground being 1000 V. As well known, one can solve separately the magnetostatic problem (in which case the voltage is of no consequence, and only magnetic fields are calculated) and the electrostatic problem (in which case only the electric fields are calculated). With Maxwell it is possible to “combine” the two results in the post-processing phase if the assumption that the electric and magnetic fields are totally separated and do not influence each other. One possible reason that such an operation is meaningful from a physical point of view might be the need for an analysis of power flow.
- Assume that a magnetostatic and electrostatic problem are created with identical geometries. Link the mesh from one simulation to the mesh of the other, so that they will be identical – this can be accomplished in the Setup tab of the Analysis Setup properties. Select the Import mesh box, and clicking on the Setup Link … button. Then specify the target Design and Solution in the Setup Link dialog. To assure that the mesh is identical in both the linked solution and the target solution, make sure to set the Maximum Number of Passes to one (1) in the linked solution.





Example TD3: Continued

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

- **Input > Quantity > E**
- **Output > Write…** *(enter the name of the .reg file to contain the solution)* **OK**

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

- **Input > Read** *(specify the name of the file containing electrostatic E field)* **OK**
- **Input > Quantity > H**
- **Vector > Cross**
- **Input > Geometry > Volume > background > OK**
- **General > Domain**
- **Input > Geometry > Surface > yz**
- **Vector > Normal**
- **Scalar > $\int$**
- **Output > Eval**

- A result around 1000 W should be obtained, corresponding to 1000 W of power being transferred along the wire but NOT THROUGH THE WIRE! Indeed the power is transmitted through the air around the wire (the Poynting vector has higher values closer to the wire and decays in a radial direction). The wire here only has the role of GUIDING the power transfer! The wire absorbs from the electromagnetic field only the power corresponding to the conduction losses in the wire.
- When the integration of the Poynting vector was performed above, a domain operation was also performed limiting the result to the background only. This shows clearly that the distribution of the Poynting vector in the background is responsible for the power transfer. Displaying the Poynting vector in different transversal planes to the wire shows also the direction of the power transfer.
- This type of analysis can be very useful in studying the power transfer in complex devices.

- ## Example TD4: Create an animation from saved field solutions

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

- *(select surface of interest)*
- ***Maxwell 3D > Fields > Fields > J > Mag_J***
- **Done**
- ***Maxwell 3D > Fields > Animate …*** (a Setup Animation dialogue appears)
- *(select desired time steps from those available in the list)*
- **Done**

Miscellaneous Examples

Example M1: Calculation of volumes and areas

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

- **Input > Number > Scalar** (1) **OK** (enters the scalar value of 1)
- **Input > Geometry > Volume** *(enter the volume of interest)* **OK**
- **Scalar >** $\int$
- **Output > Eval**

- The result is expressed in m^3.

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

- **Input > Number > Scalar** (1) **OK** (enters the scalar value of 1)
- **Input > Geometry > Surface** *(enter the surface of interest)* **OK**
- **Scalar >** $\int$
- **Output > Eval**

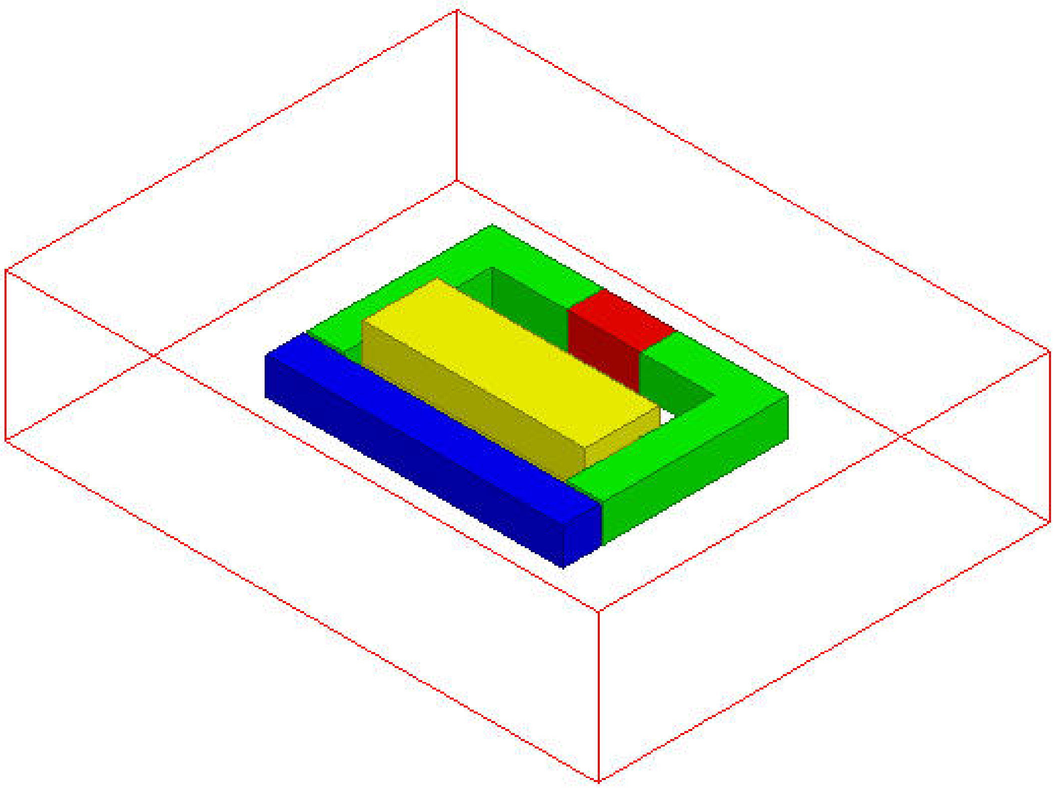
- The result is expressed in m^2.

Chapter 5.0 – Magnetostatic Examples

Example (Magnetostatic) – Magnetic Force

Magnetic Force

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



ANSYS Maxwell Design Environment

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

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

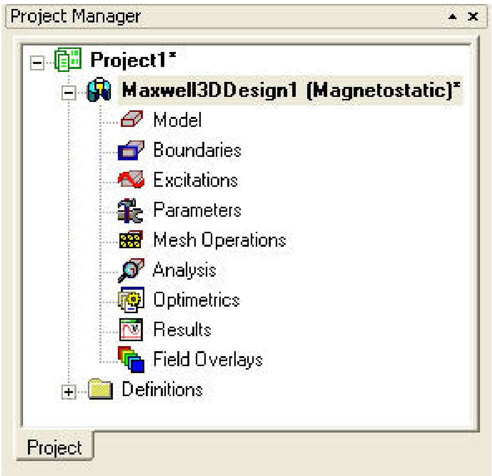
Setting Tool Options

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

Example (Magnetostatic) - Magnetic Force

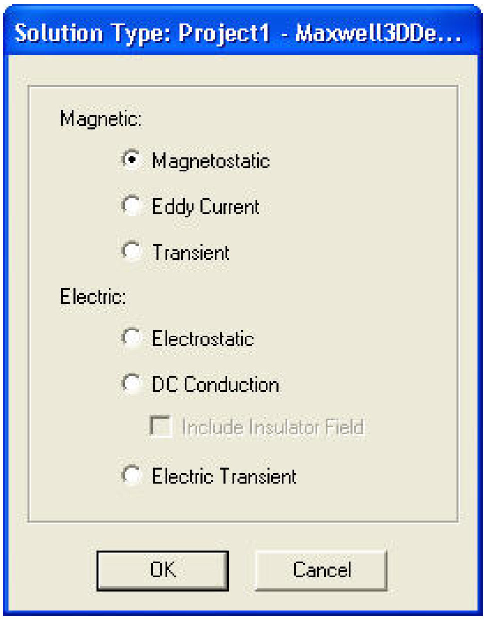
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



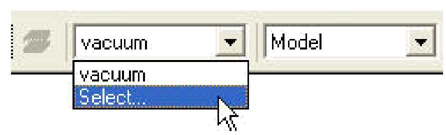
Example (Magnetostatic) - Magnetic Force

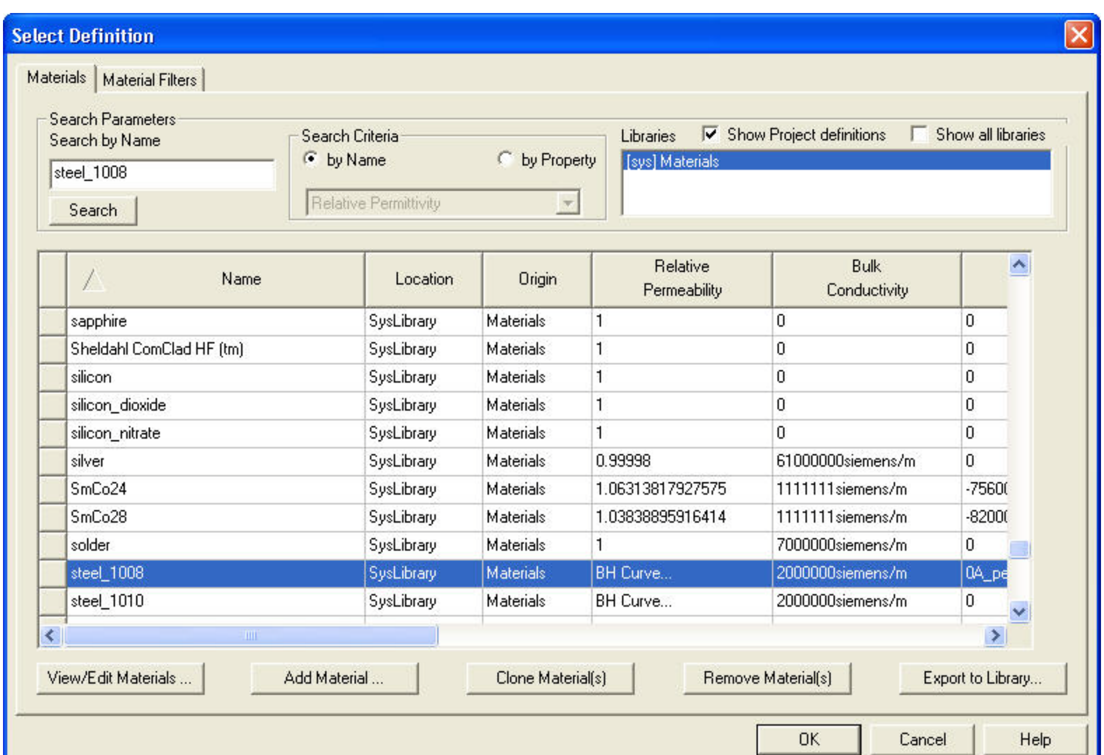
Set Model Units

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

Set Default Material

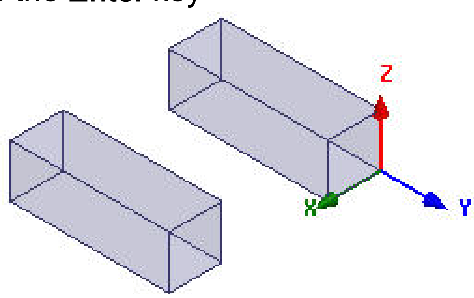
- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **steel_1008** in the **Search by Name** field
    2. Click the **OK** button



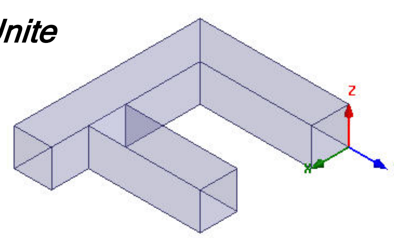


Create Core

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **-5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **10**, dY: **-30**, dZ: **10**, Press the **Enter** key
  - Select the menu item ***View > Fit All > Active View***.
- **Duplicate Box**
  - Select the object **Box1** from the history tree
  - Select the menu item ***Edit > Duplicate Along Line***
    1. Using the coordinate entry fields, enter the first point
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **30**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Total Number: **2**
    4. Click the **OK** button

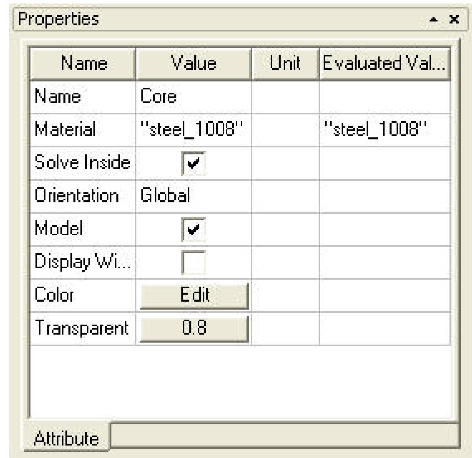


- **Create another box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **-30**, Z: **-5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **50**, dY: **-10**, dZ: **10**, Press the **Enter** key
- **Unite Objects**
  - Select the menu item ***Edit > Select All***
  - Select the menu item, ***Modeler > Boolean > Unite***



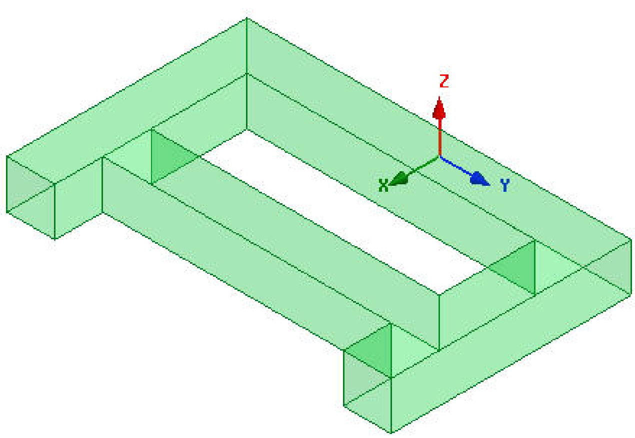
Example (Magnetostatic) - Magnetic Force

- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Core**
    2. Change its color to **Green**



- **Mirror object**
  - Select the Object **Core** from the history tree
  - Select the menu item, ***Edit > Duplicate > Mirror***
    1. Using the coordinate entry fields, enter the first point
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the normal point
       - dX: **0**, dY: **1**, dZ: **0**, Press the **Enter** key
  - Select the menu item ***View > Fit All > Active View.***

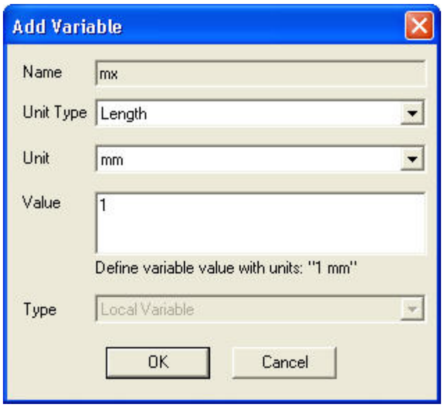
- **Unite Objects**
  - Press **Ctrl** and select the objects **Core** and **Core_1** from the history tree
  - Select the menu item, ***Modeler > Boolean > Unite***

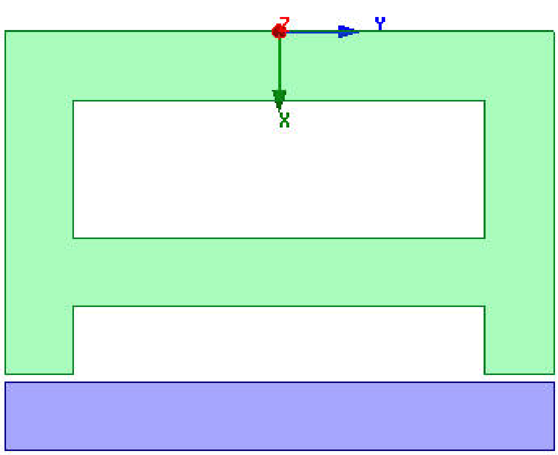


Create Bar

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **51**, Y: **-40**, Z: **-5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **10**, dY: **80**, dZ: **10**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **Bar**
    2. Change its color to **Blue**

- **Parameterize Object**
  - Expand the history tree of the object **Bar**
  - Double click on the command **CreateBox** from the tree
  - For **Position**, type: **50mm+mx, -40, -5**, Click the **Tab** key to accept
  - In Add variable window,
    1. Unit Type: **Length**
    2. Unit: **mm**
    3. Value: **1**
    4. Press **OK**
  - Press **OK** to exit

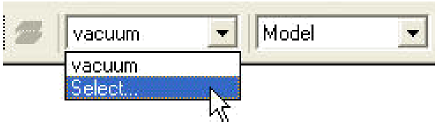




Example (Magnetostatic) - Magnetic Force

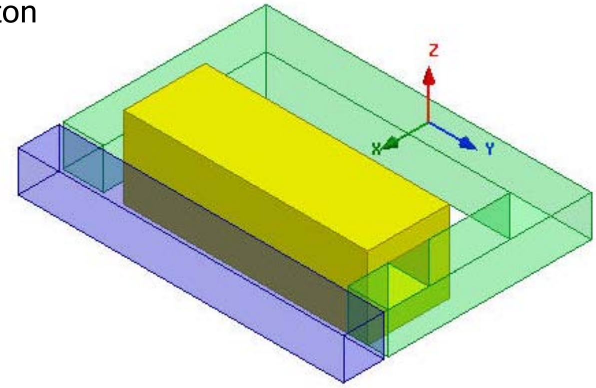
Set Default Material

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



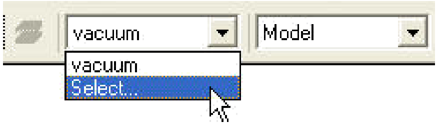
Create Coil

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **45**, Y: **30**, Z: **10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **-20**, dY: **-60**, dZ: **-20**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **Coil**
    2. Change its color to **Yellow**
- **Subtract Core**
  - Press **Ctrl** and select the objects **Core** and **Coil** from the history tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Coil**
    2. Tool Parts: **Core**
    3. Clone tool objects before subtracting**: ☑ Checked**
    4. Click the **OK** button



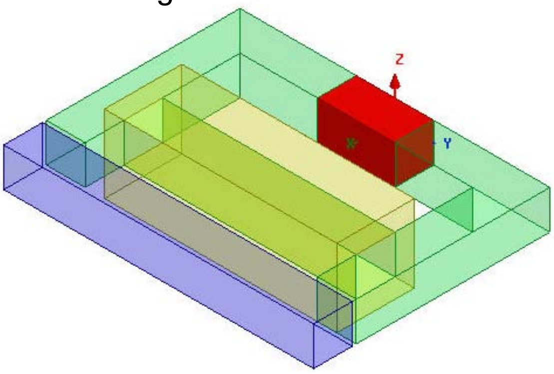
Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **NdFe35** in the **Search by Name** field
    2. Click the **OK** button



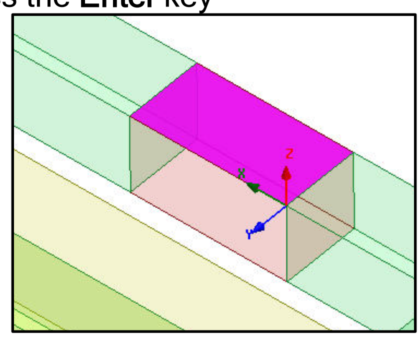
Create Magnet

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **-10**, Z: **-5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **10**, dY: **20**, dZ: **10**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **Magnet**
    2. Change its color to **Red**
- **Subtract Object**
  - Press **Ctrl** and select the objects **Magnet** and **Core** from the history tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window
    1. Blank Parts: **Core**
    2. Tool Parts: **Magnet**
    3. Clone tool objects before subtracting: ☑ **Checked**
    4. Click the **OK** button

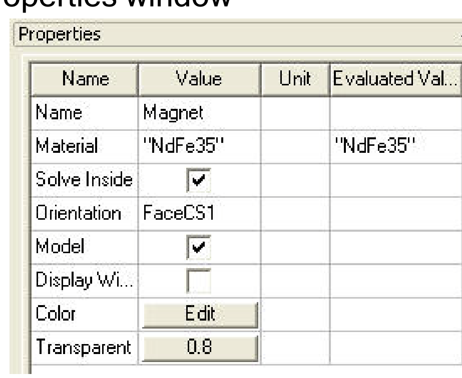


Orient Magnet

- **Note:** By default all objects in Maxwell will have their orientation with respect to global co-ordinate system and all magnetic materials are magnetized in X direction. If actual direction of magnetization is different from Global axis, we need to create Local coordinate system in that direction and orient the magnet with respect to local coordinate system.
- **Create Local coordinate system**
  - Change Selection from Object to faces
    1. Select the menu item ***Edit > Select > Faces*** or
    2. Press shortcut key **“F”** from keyboard
  - Using the mouse, select the top face of the Magnet from graphic window
  - Select the menu item ***Modeler > Coordinate System > Create > Face CS***
    1. Using the coordinate entry fields, enter the origin
       - X: **10**, Y: **10**, Z: **5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the axis:
       - dX: **0**, dY: **-20**, dZ: **0**, Press the **Enter** key



- **Change Orientation of Magnet**
  - Select **Magnet** from the tree and goto Properties window
  - Change Orientation to **FaceCS1**



  - Change Selection to Objects
    1. Select the menu item ***Edit > Select > Objects*** or
    2. Press shortcut key **“O”** from keyboard

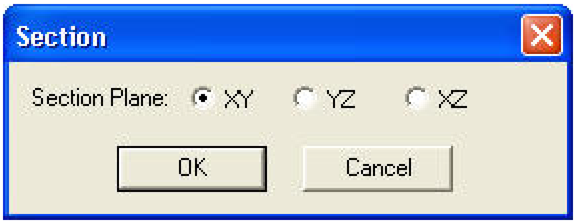
Example (Magnetostatic) - Magnetic Force

Assign Boundaries

- **Assign Insulating boundary to prevent current leakage out of the Coil**
  - Select the object **Coil** from the tree
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Insulating***
  - In Insulating Boundary window,
    1. Name: **Insulating1**
    2. Click the **OK** button

Create Excitations

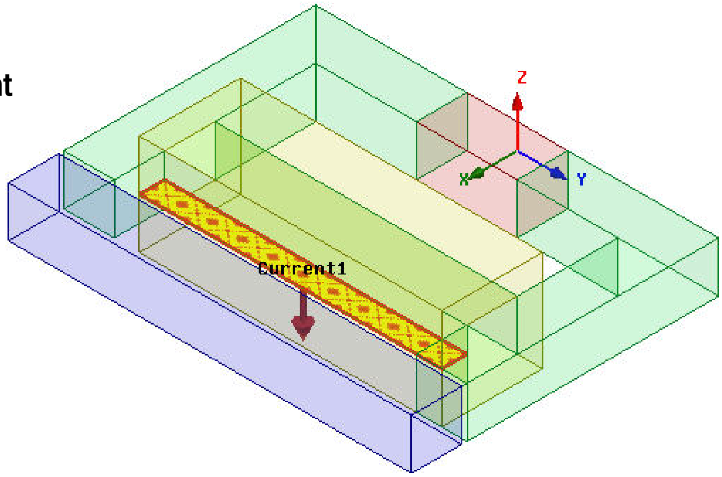
- **Change Work Coordinate System**
  - Goto ***Modeler > Coordinate System > Set Working CS***
  - In Select Coordinate System Window
    1. Select **Global**
    2. Press **Select**
- **Create Section of coil for assigning Current**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **XY**
    2. Click the **OK** button



- **Change Attributes**
  - Select the object **Coil_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Coil_Terminal**
- **Separate Sheets**
  - Select the sheet **Coil_Terminal**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Coil_Terminal_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***

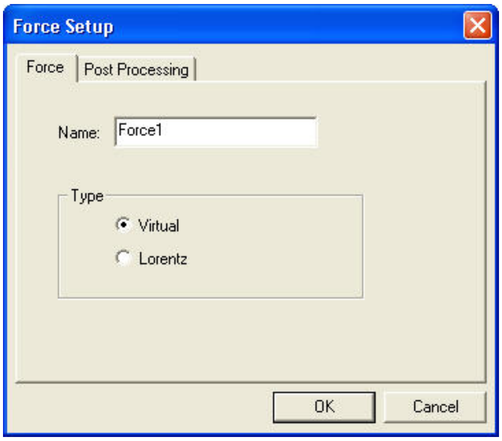
Example (Magnetostatic) - Magnetic Force

- **Assign Excitations**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **c1**
    3. Type: **Stranded**
    4. Current Direction: **negative Z** (Use **Swap Direction** if needed)
    5. Press **OK**
  - In Add variable window,
    1. Unit Type: **Current**
    2. Unit: **A**
    3. Value: **100**
    4. Press **OK**



Create Force Parameter

- **Create Parameter to calculate Force**
  - Select the Object **Bar** from the history tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Force***
  - In Force Setup window,
    1. Name: **Force1**
    2. Type: **Virtual**
    3. Press **OK** button



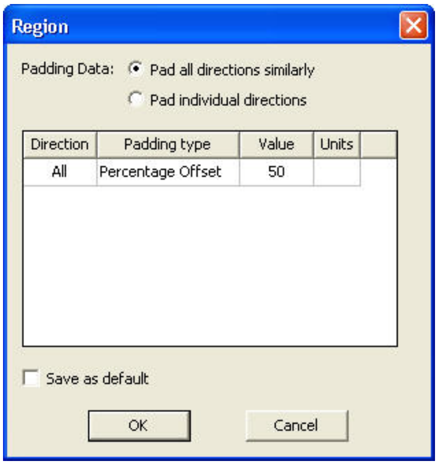
- Note: Virtual forces are useful on any object while Lorentz forces are only applicable on current carrying objects which have permeability = 1.

Set Default Material

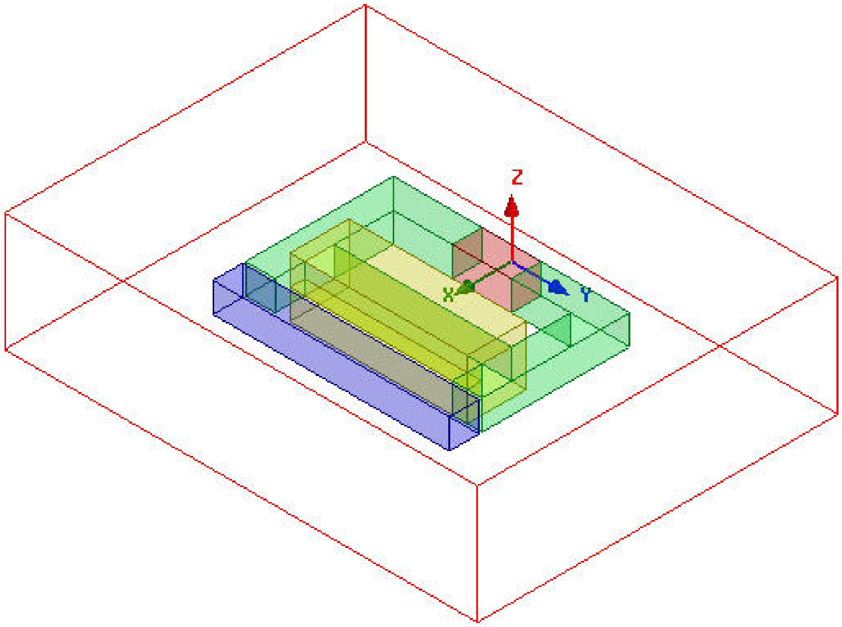
- **To Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **50**
    4. Press **OK**



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



Analysis Setup

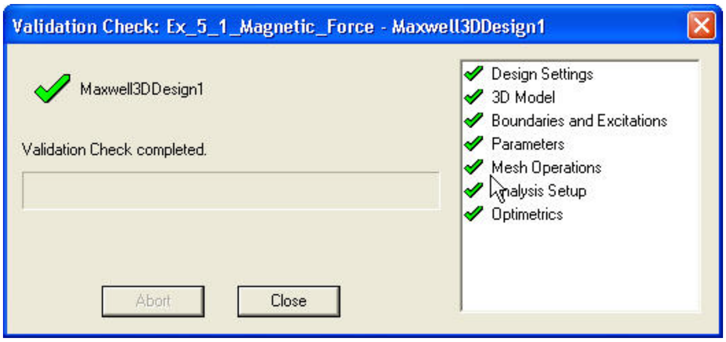
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **Ex_5_1_Magnetic_Force**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



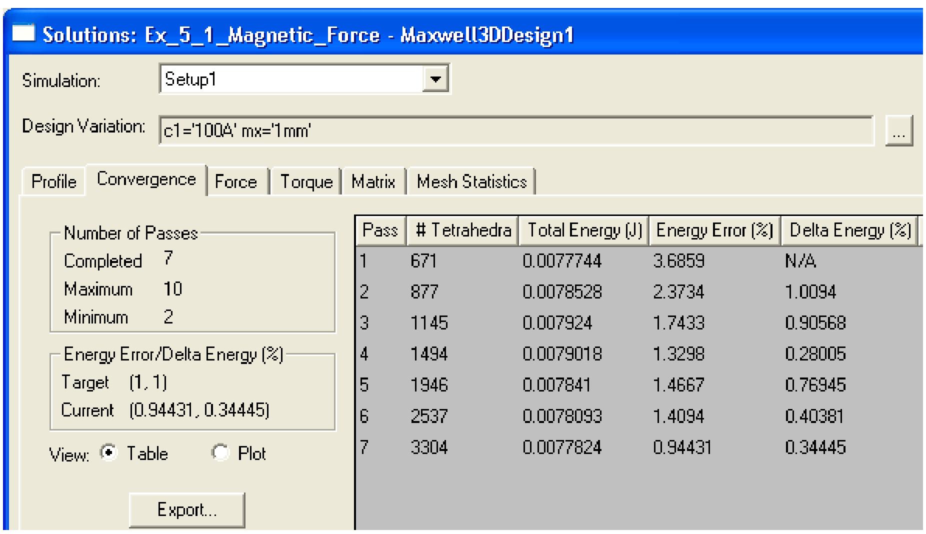
Note: To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

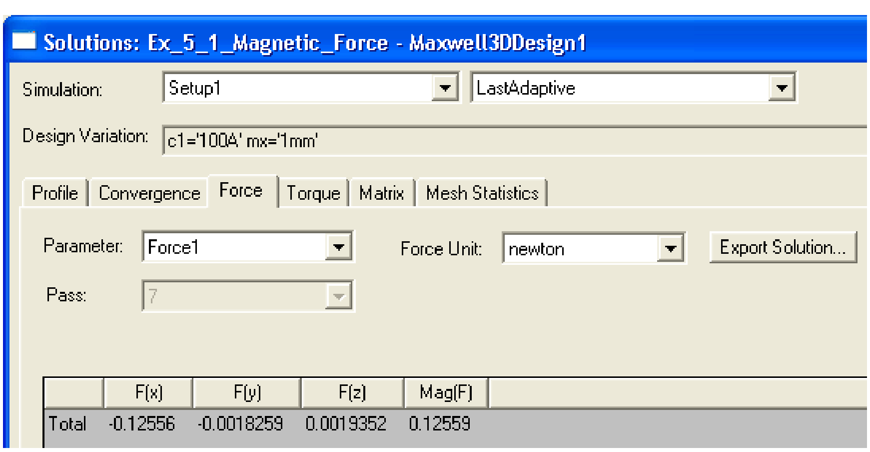
Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.



Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	671	0.0077744	3.6859	N/A
2	877	0.0078528	2.3734	1.0094
3	1145	0.007924	1.7433	0.90568
4	1494	0.0079018	1.3298	0.28005
5	1946	0.007841	1.4667	0.76945
6	2537	0.0078093	1.4094	0.40381
7	3304	0.0077824	0.94431	0.34445

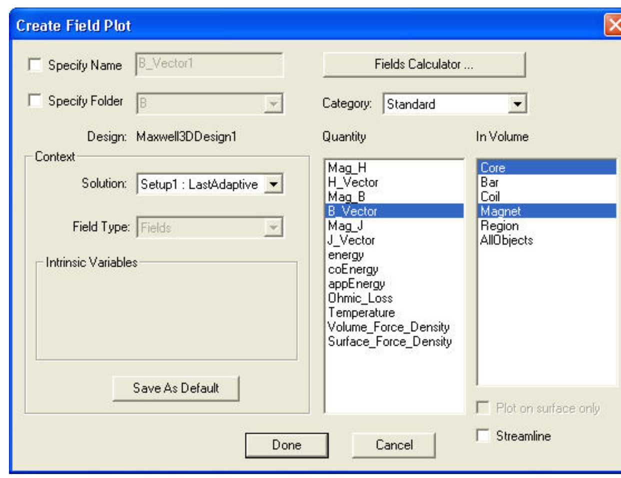
- To view the Force values:
  1. Click the **Force**Tab



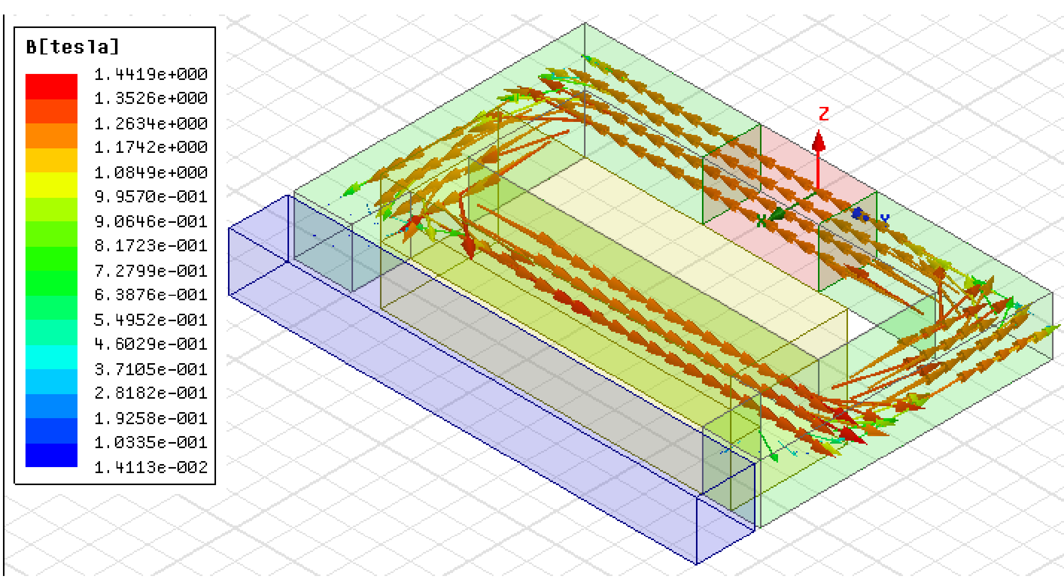
	F(x)	F(y)	F(z)	Mag(F)
Total	-0.12556	-0.0018259	0.0019352	0.12559

Plot Flux Density on Core Cross section

- To ensure that the current direction and magnetization are correct, create a plot the flux density vector on the core cross section.
  - In the history tree, expand Planes and select the geometry on which to create the plot: **Global:XY**
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > B_Vector***
  - In Create Field plot window
    1. Quantity: **B_vector**
    2. In Volume: Press **Ctrl** and select **Core** and **Magnet**
    3. Press **Done**



Note: This field plot is for the nominal design case with: mx = 1mm and c1 = 100A.



Optimetrics Setup – Parametric Sweep

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

Add a Parametric Sweep

- Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Parametric***
- In Setup Sweep Analysis window,
  1. On the **Sweep Definitions** tab**:**
     1. Click the **Add** button
     2. Add/Edit Sweep Dialog
        1. Select Variable: **c1** (pull down menu to to change from “mx” if necessary)
        2. Select **Linear Step**
        3. Start: **0**
        4. Stop: **500**
        5. Step: **100**
        6. Click the **Add** button
        7. Click the **OK** button
  2. Click the **Options** tab**:**
     1. Save Fields And Mesh: ☑ **Checked**
  3. Click the **OK** button

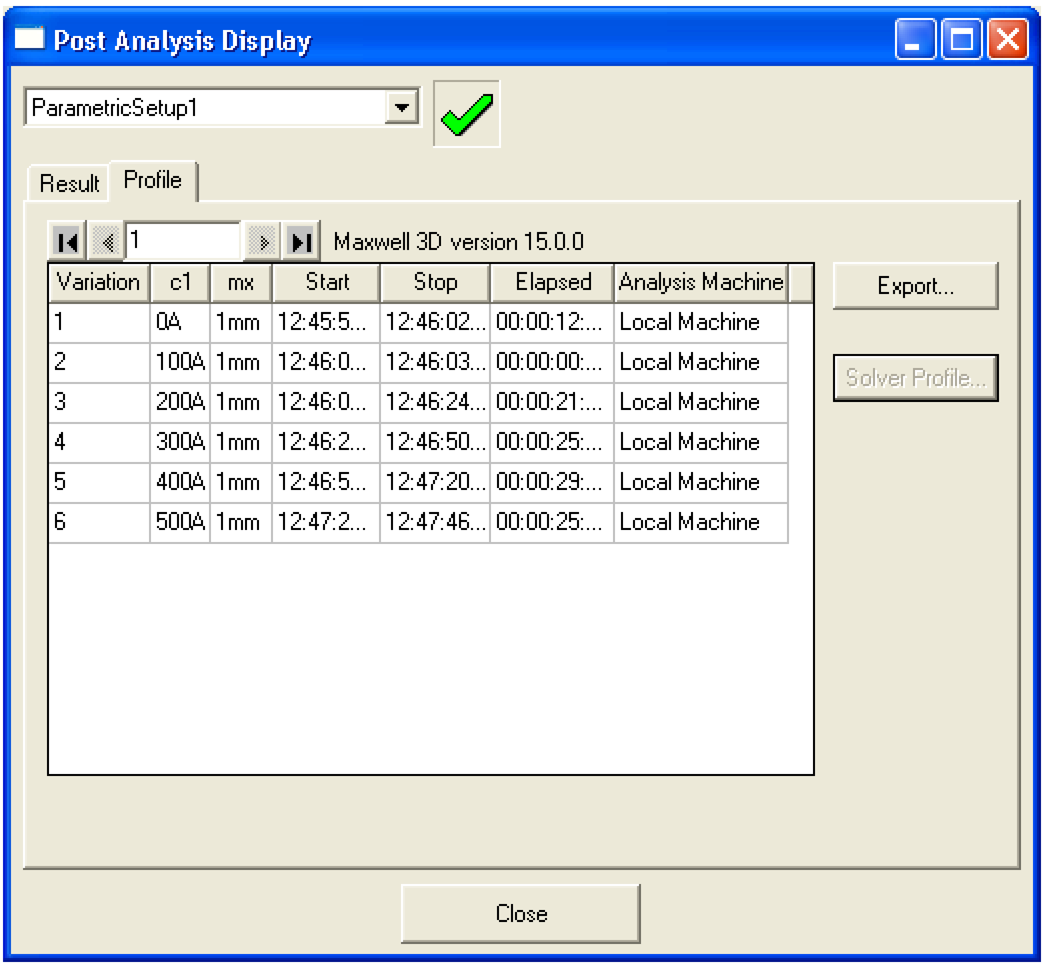
Analyze Parametric Sweep

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

Example (Magnetostatic) - Magnetic Force

Optimetrics Results

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

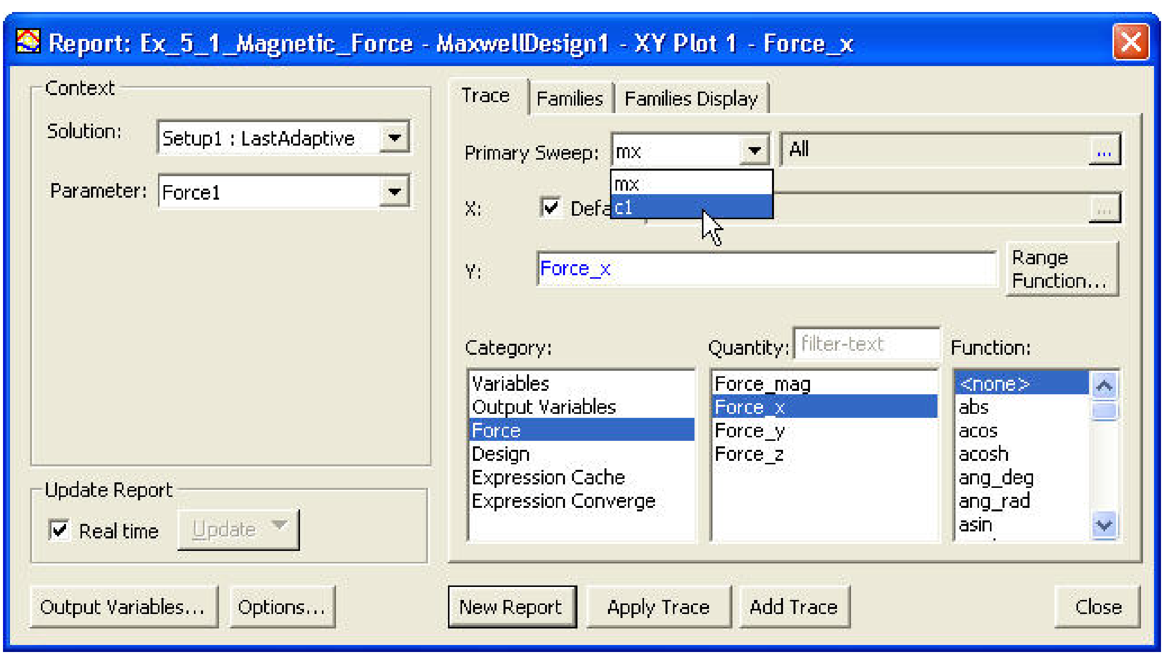


Variation	c1	mx	Start	Stop	Elapsed	Analysis Machine
1	0A	1mm	12:45:5...	12:46:02...	00:00:12:...	Local Machine
2	100A	1mm	12:46:0...	12:46:03...	00:00:00:...	Local Machine
3	200A	1mm	12:46:0...	12:46:24...	00:00:21:...	Local Machine
4	300A	1mm	12:46:2...	12:46:50...	00:00:25:...	Local Machine
5	400A	1mm	12:46:5...	12:47:20...	00:00:29:...	Local Machine
6	500A	1mm	12:47:2...	12:47:46...	00:00:25:...	Local Machine

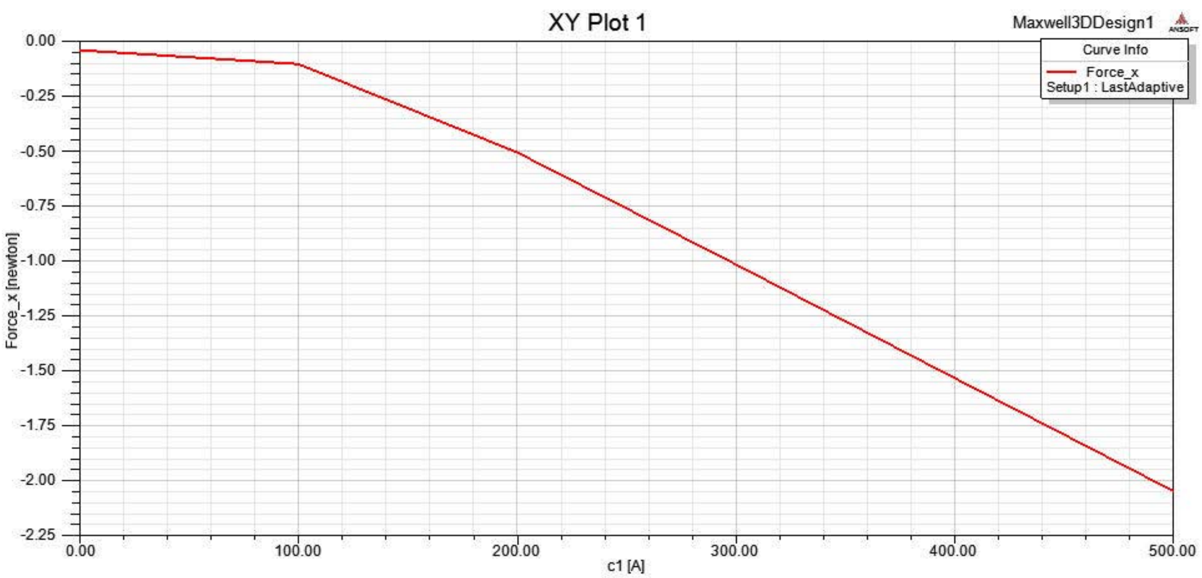
Example (Magnetostatic) - Magnetic Force

Create Plot of Force at each Current

- **To create a report:**
  - Select the menu item *Maxwell 3D> Results > Create Magnetostatic Report > Rectangular Plot*
  - In Reports window
    1. Solution: **Setup1: LastAdaptive**
    2. Parameter: **Force1**
    3. Click the **Trace** tab
        1. Parametric Sweep : **c1**
        2. X Axis: **Default**
        3. Y Axis:
            1. Category: **Force**
            2. Quantity: **Force_x**
            3. Function: **<none>**
    4. Click the **New Report** button

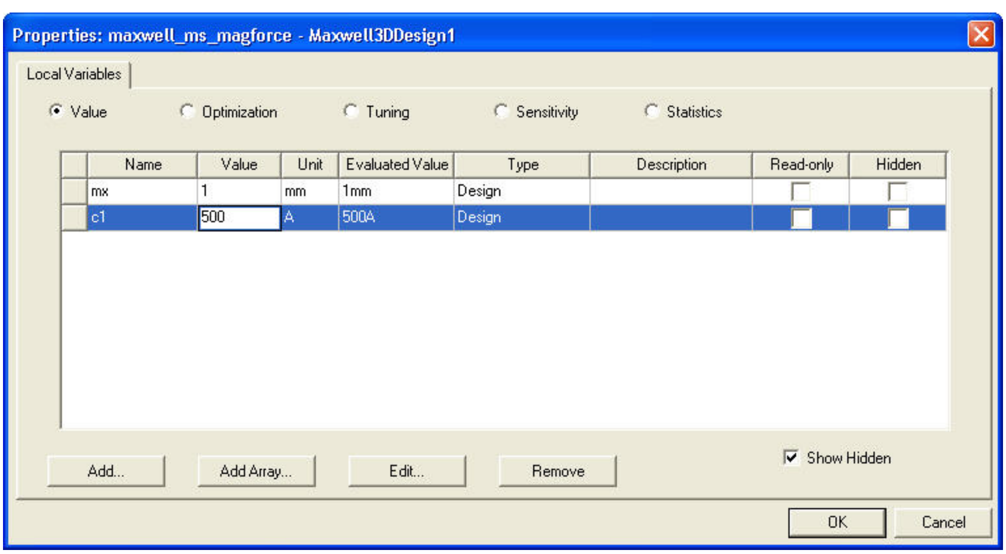


Example (Magnetostatic) – Magnetic Force



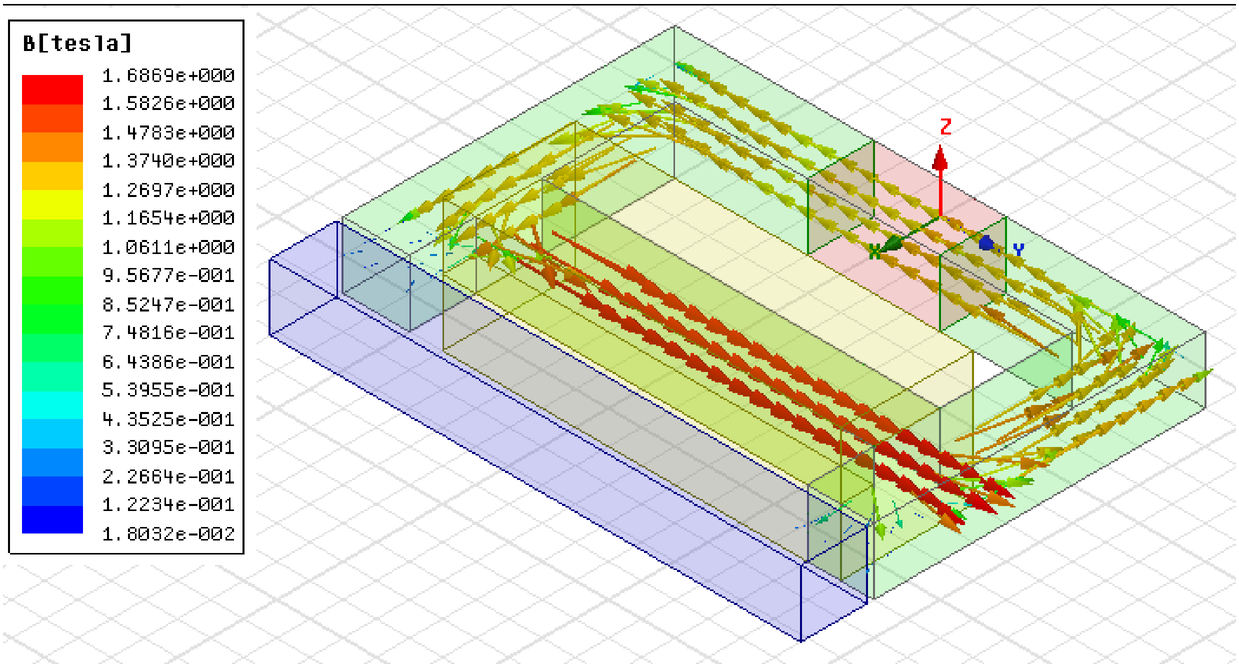
Plot Flux Density on Core Cross section at a different current

- Finally, create a plot of B_vector in the core corresponding to the last row in the parametric table with c1 = 500A.
- Switch to the solution for the last row in the parametric table:
  - Select ***Maxwell 3D > Design Properties > Local Variables***. Change the value for c1 = 500 and click on **OK**.



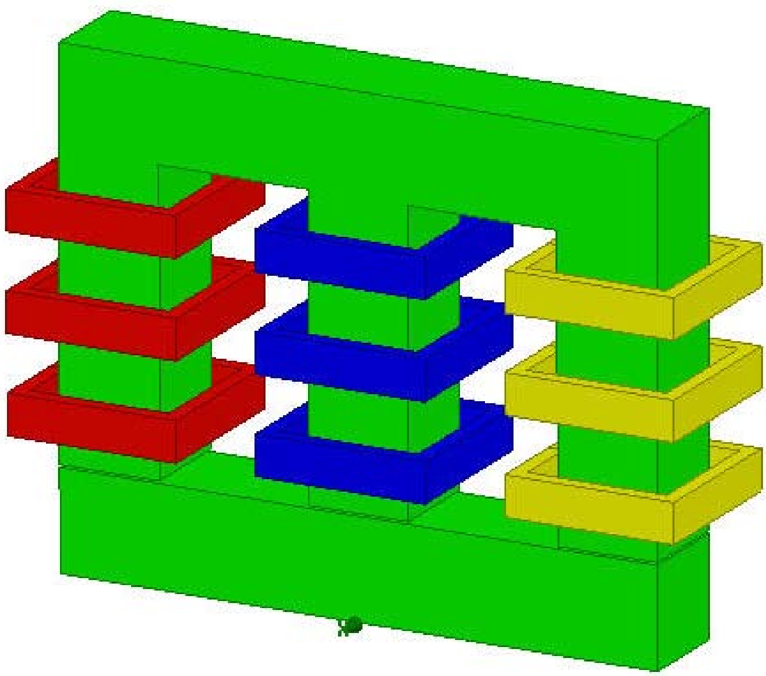
  - The previous B_vector plot automatically updates to show the fields for ***c1 = 500A.*** The flux density is higher as expected

Example (Magnetostatic) - Magnetic Force



Example (Magnetostatic) – Inductance Calculation

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



ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Box, Rectangle, Region**
    - Surface Operations: **Section**
    - Boolean Operations: **Subtract, Separate Bodies, Split**
    - 3D Modeler: **Delete Last Operation**
    - Sweep: **Along Path**
    - Duplicate: **Along Line**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Inductance Matrix**

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

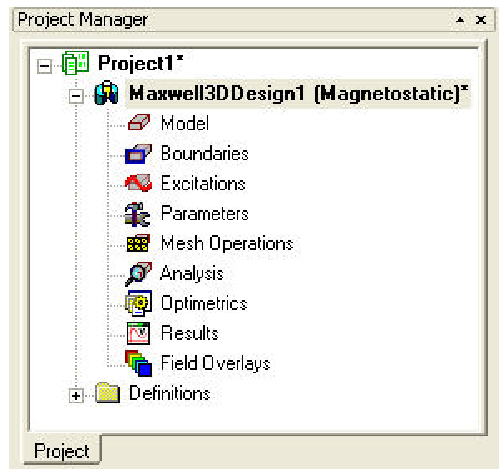
Setting Tool Options

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

Example (Magnetostatic) – Inductance Calculation

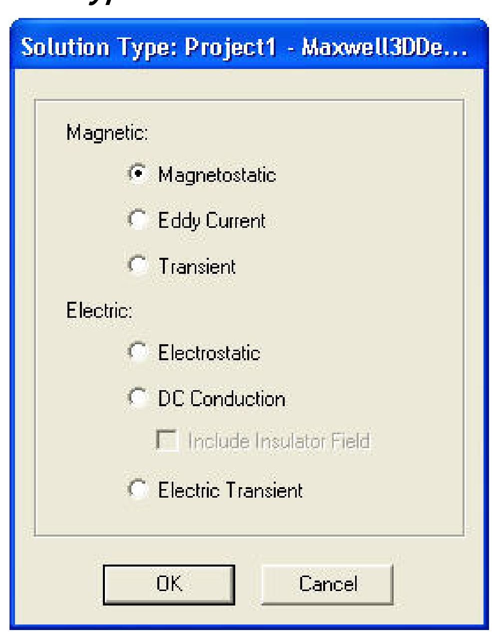
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



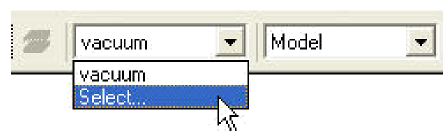
Example (Magnetostatic) – Inductance Calculation

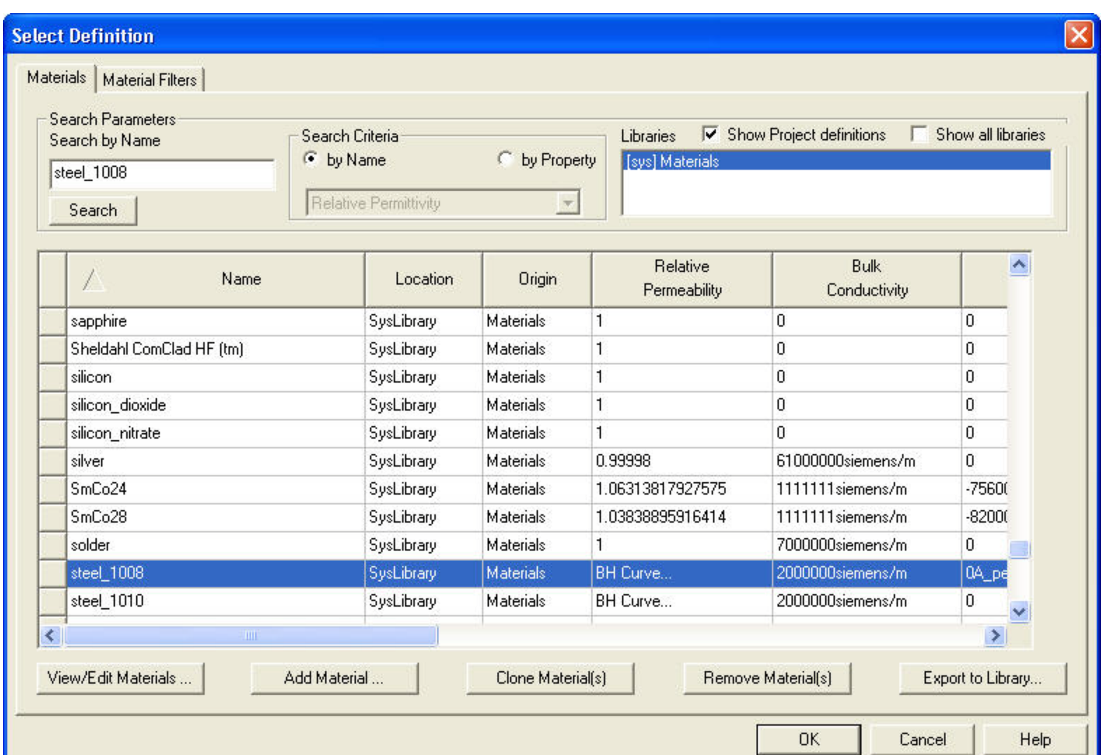
Set Model Units

- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **in** ***(inches)***
    2. Click the **OK** button

Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **steel_1008** in the **Search by Name** field
    2. Click the **OK** button





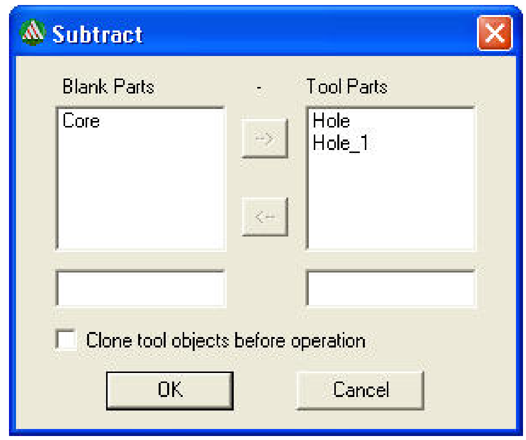
Example (Magnetostatic) – Inductance Calculation

Create Core

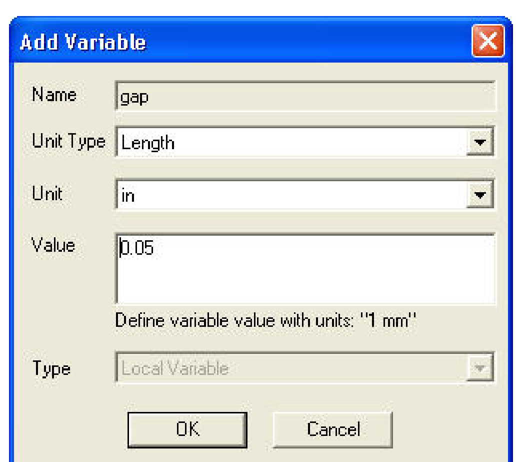
- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-1**, Y: **-6**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **2**, dY: **12**, dZ: **10**, Press the **Enter** key
  - Select the menu item ***View > Fit All > Active View.***
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Core**
    2. Change its color to **Green**
- **Create Another Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-1**, Y: **1**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **2**, dY: **3**, dZ: **6**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Hole**
- **Duplicate Hole**
  - Select the object **Hole** from the history tree
  - Select the menu item ***Edit > Duplicate > Around Axis***
    1. Axis: **Z**
    2. Angle: **180**
    3. Total Number: **2**
    4. Click the **OK** button

Example (Magnetostatic) - Inductance Calculation

- **Subtract Objects**
  - Select the menu item ***Edit > Select All***
  - Select the menu item, ***Modeler > Boolean > Subtract***
  - In Subtract Window
    1. Blank Parts: **Core**
    2. Tool Parts: **Hole** and **Hole_1**
    3. Click the **OK** button

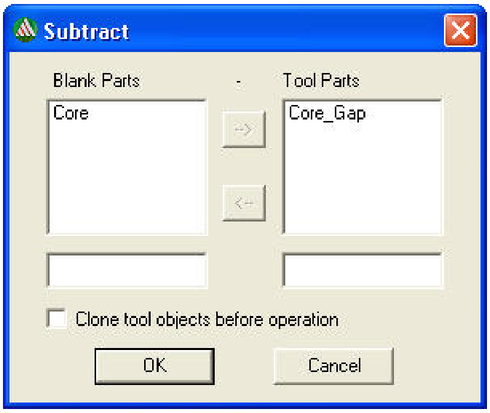


- **Create Core_Gap**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-1**, Y: **-6**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **2**, dY: **12**, dZ: **0.05**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Core_Gap**
- **Parameterize Gap**
  - Expand the history tree of the object **Core_Gap**
  - Double click on the command **CreateBox** from the tree
  - For **ZSize** type: **gap** and press **Enter**
  - In Add variable window,
    1. Unit Type: **Length**
    2. Unit: **inch**
    3. Value: **0.05**
    4. Press **OK**
  - Press **OK** to exit

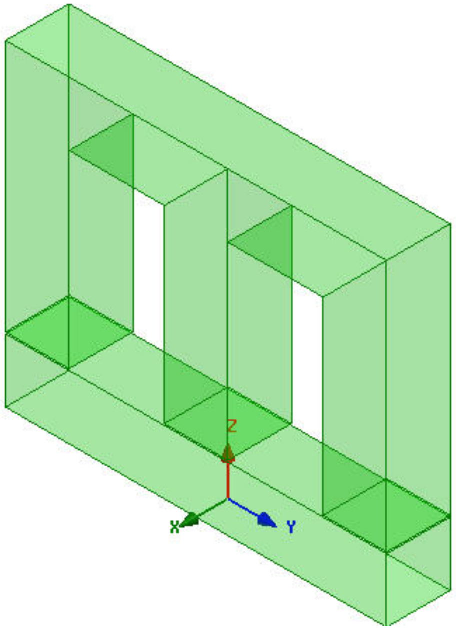


Example (Magnetostatic) – Inductance Calculation

- **Subtract Core**
  - Press **Ctrl** and select the objects **Core** and **Core_Gap** from the history tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Core**
    2. Tool Parts: **Core_Gap**
    3. Click the **OK** button



- **Separate Core**
  - Select the object **Core** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Change Attributes**
  - Select the object **Core** from the tree and goto Properties window
    1. Change the name of the object to **Core_E**
  - Select the object **Core_Separate1** from the tree and goto Properties window
    1. Change the name of the object to **Core_I**



Example (Magnetostatic) – Inductance Calculation

Set Default Material

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

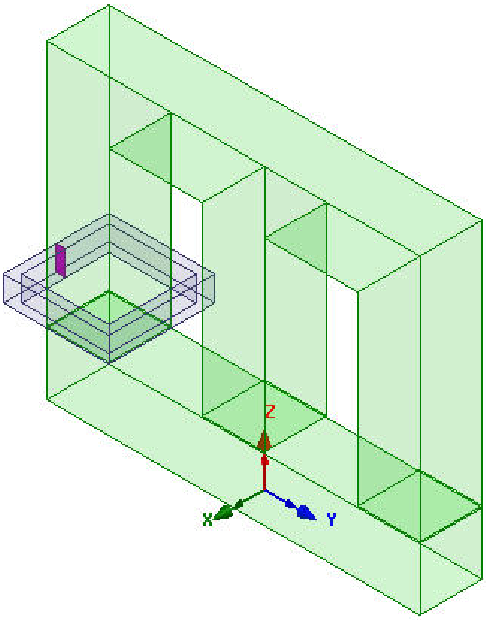
Create Coils and Terminals

- **Create Rectangle**
  - Select menu item ***Modeler > Grid Plane > YZ***
  - Select menu item ***Draw > Rectangle***.
    1. Using the coordinate entry fields, enter the rectangle position
       - X: **0**, Y: **-3.6**, Z: **3.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **0**, dY: **0.3**, dZ: -**0.8**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **CoilA_1**
- **Create Guide**
  - Select menu item ***Modeler > Grid Plane > XY***
  - Select menu item ***Draw > Rectangle***.
    1. Using the coordinate entry fields, enter the rectangle position
       - X: **1.5**, Y: **-6.5**, Z: **3**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **-3**, dY: **3**, dZ: **0**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Guide**
  - Select **Guide** from the tree and goto menu item ***Modeler > Delete Last Operation***
    - Note: This operation will remove the cover closed polygon operation that is performed automatically. The results will be a rectangular polyline

Example (Magnetostatic) – Inductance Calculation

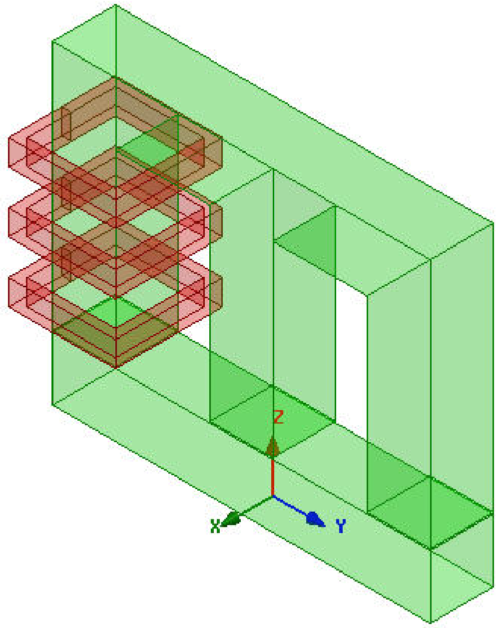
- **Sweep Object**
  - Press **Ctrl** and select the objects **CoilA_1** and **Guide** from the history tree
  - Select menu item ***Draw > Sweep > Along Path***
  - In Sweep along path window
    1. Angle of twist: **0 deg**
    2. Draft Angle: **0 deg**
    3. Draft type: **Round**
    4. Select **OK**
- **Create Section for CoilA_1**
  - Select the object **CoilA_1** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button

  - Rename CoilA_1_Section1 to **TerminalA_1**
- **Separate Sheets**
  - Select the sheet **TerminalA_1** resulted in last operation
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - Delete the extra sheet **TerminalA_1_Seperate1**



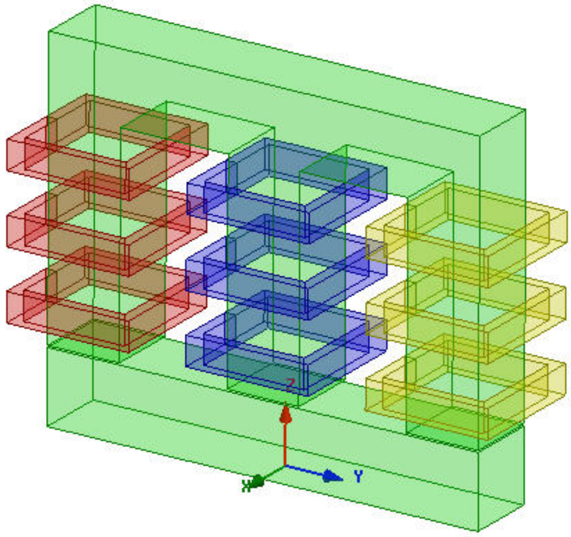
Example (Magnetostatic) – Inductance Calculation

- **Duplicate Objects**
  - Press **Ctrl** and select the objects **CoilA_1** and **TerminalA_1** from the tree
  - Select menu item ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **0**, dY: **0**, dZ: **1.925**, Press the **Enter** key
    3. Total Number : **3**
    4. Press **OK**
- **Change Attributes of Coils**
  - Select the resulting objects from the tree and goto Properties window
    1. Change the name of the object **CoilA_1_1** to **CoilA_2**
    2. Change the name of the object **CoilA_1_2** to **CoilA_3**
    3. Change color of **CoilA_1,CoilA_2** and **CoilA_3** to **Red**
- **Change Attributes of Terminals**
  - Select the resulting objects from the tree and goto Properties window
    1. Change the name of the object **TerminalA_1_1** to **TerminalA_2**
    2. Change the name of the object **TerminalA_1_2** to **TerminalA_3**
    3. Change color of **TerminalA_1,TerminalA_2** and **TerminalA_3** to **Red**



Example (Magnetostatic) – Inductance Calculation

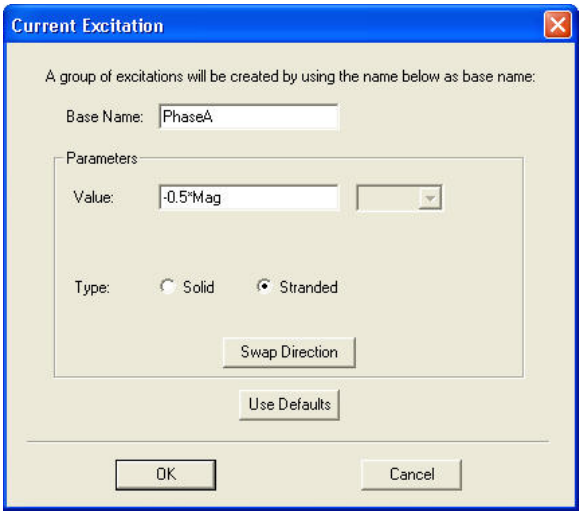
- **Duplicate Coils**
  - Press **Ctrl** and select the object **CoilA_1, CoilA_2 ,CoilA_3,TerminalA_1, TerminalA_2** and **TerminalA_3** from the tree
  - Select menu item ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **0**, dY: **5**, dZ: **0**, Press the **Enter** key
    3. Total Number : **3**
    4. Press **OK**
- **Change Attributes**
  - Select the resulting objects from the tree and goto Properties window
    1. Change the name of the objects **CoilA_1_1, CoilA_2_1** and **CoilA_3_1** to **CoilB_1, CoilB_2** and **CoilB_3**
    2. Change the name of the objects **CoilA_1_2, CoilA_2_2** and **CoilA_3_2** to **CoilC_1, CoilC_2** and **CoilC_3**
    3. Change the name of the terminals **TerminalA_1_1, TerminalA_2_1, TerminalA_3_1, TerminalA_1_2, TerminalA_2_2** and **TerminalA_3_2** to **TerminalB_1, TerminalB_2, TerminalB_3, TerminalC_1, TerminalC_2** and **TerminalC_3**
    4. Change color of all CoilB and TerminalB objects to **Blue**
    5. Change color of all CoilC and TerminalC objects to **Yellow**



Example (Magnetostatic) – Inductance Calculation

Create Excitations

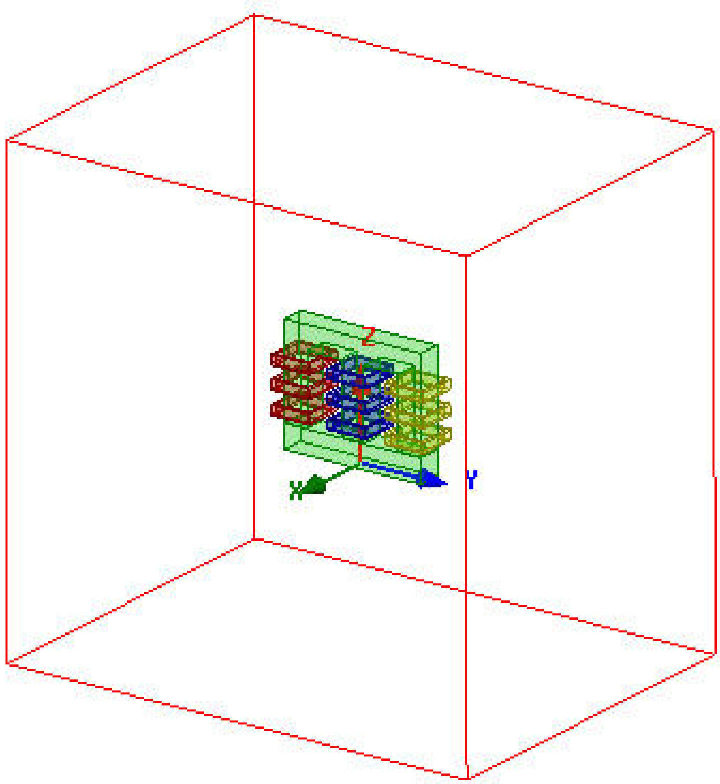
- **Assign Excitation to TerminalA**
  - Press **Ctrl** and select objects **TerminalA_1, TerminalA_2** and **TerminalA_3**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseA**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Press **OK**
  - In Add variable window,
    1. Unit Type: **Current**
    2. Unit: **A**
    3. Value: **30**
    4. Press **OK**



- **Assign Excitation to TerminalB**
  - Press **Ctrl** and select objects **TerminalB_1, TerminalB_2** and **TerminalB_3**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseB**
    2. Value: **Mag**
    3. Type: **Stranded**
    4. Press **OK**
- **Assign Excitation to TerminalC**
  - Press **Ctrl** and select objects **TerminalC_1, TerminalC_2** and **TerminalC_3**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseC**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Press **OK**

Define Region

- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Padding Data: **Pad individual directions**
    2. Padding Type: **Percentage Offset**
    3. Directions:
       - +X = **400**
       - -X = **400**
       - +Y = **100**
       - -Y = **100**
       - +Z = **150**
       - -Z = **150**
    4. Press **OK**



Inductance Calculation Defining Equations

- Inductance in a magnetostatic simulation can be defined as

$$\lambda_1 = L_{11} I_1 + L_{12} I_2 + \cdots$$
$$\lambda_2 = L_{21} I_1 + L_{22} I_2 + \cdots$$
$$\cdots$$

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

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

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

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

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

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

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

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

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

- If only linear materials are used, the value of the inductance will be valid for any value of source currents.

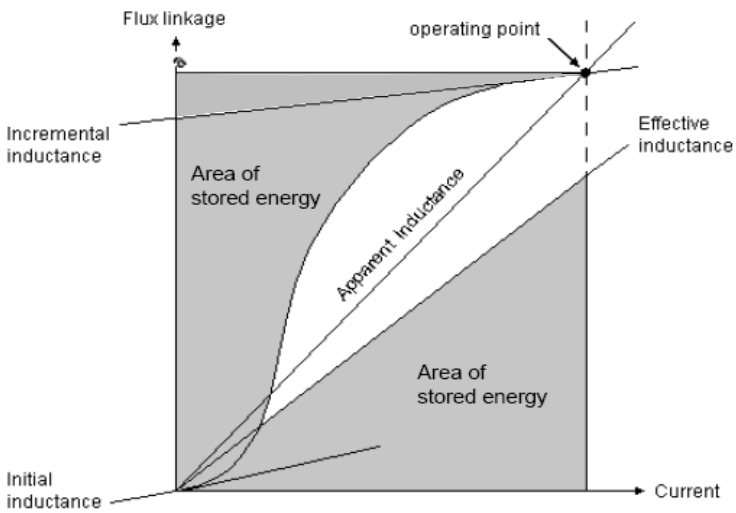
Example (Magnetostatic) – Inductance Calculation

Inductance Calculation Methodology

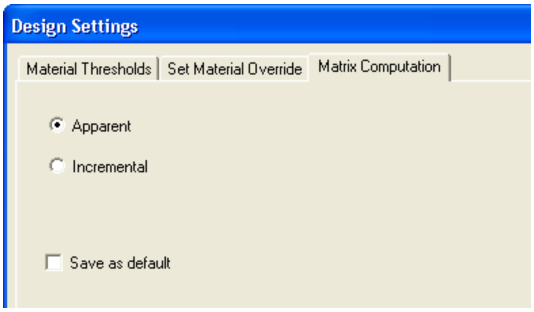
- The inductance calculation is performed after the field solution and depends on the values of the field solution at every point in the solution space because it requires an energy integral in the entire solution space.

- After the field solution is completed, the relative permeability values are "frozen". Then, one ampere is sequentially excited in each coil, and a field solution is performed with the frozen permeabilities. With these field solutions, the energy integral is performed and the inductance is obtained as described previously.

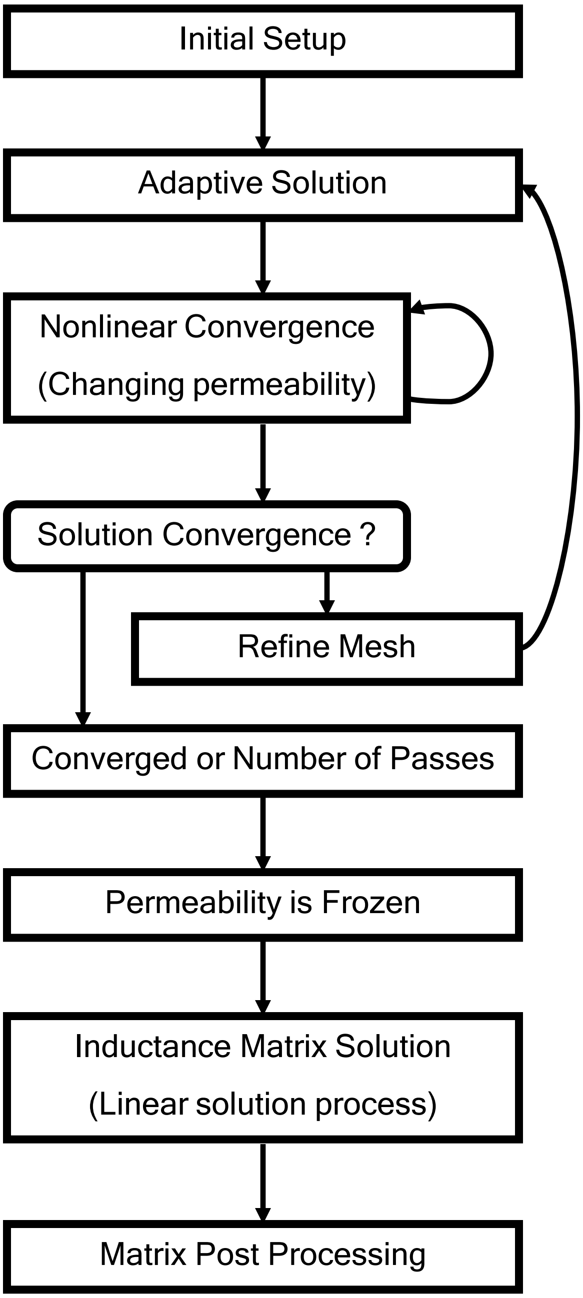
- The inductance obtained in this manner is the inductance per turn^2 (referred to as the nominal inductance in the software) and is the apparent inductance (determined by the flux linkage/current operating point in the graphic below).



- Incremental inductance can also be obtained using Maxwell by setting the parameter ***Maxwell 3D > Design Settings > Matrix Computation*** to the value **Incremental**



Inductance Calculation Process



Inductance Calculation Post Processing

- The nominal inductances that are computed by the solver are returned in units of Henries/turns^2. In order to return each inductance in units of Henries, the number of turns for each coil must be specified at some point. Also, the inductance matrix can be grouped to represent windings in series, and a factor can be included if the winding is made up of parallel branches. These are all considered post-processing, and they can help to return meaningful quantities.

- The post processed inductances are calculated as follows:

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

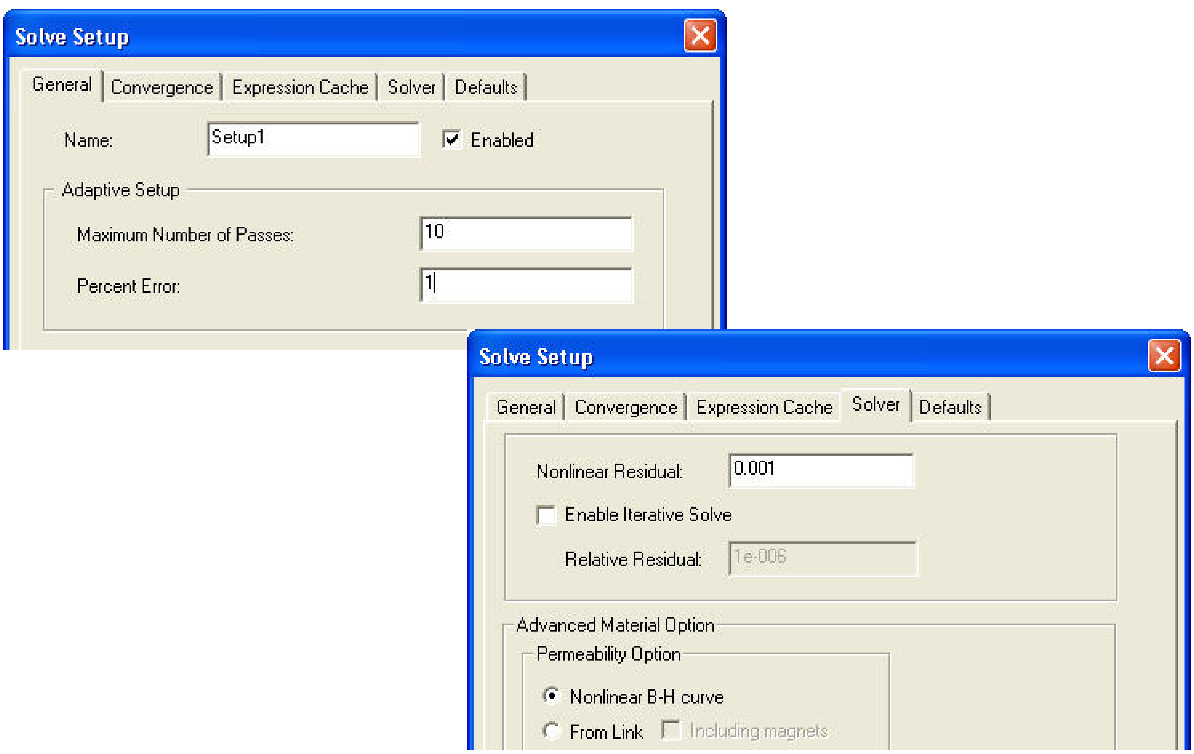
$$L_{group} = \sum_{i\_group} \sum_{j\_group} L_{ij}$$

$$L_{branched} = L_{group} \big/ b^2$$

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

Analysis Setup

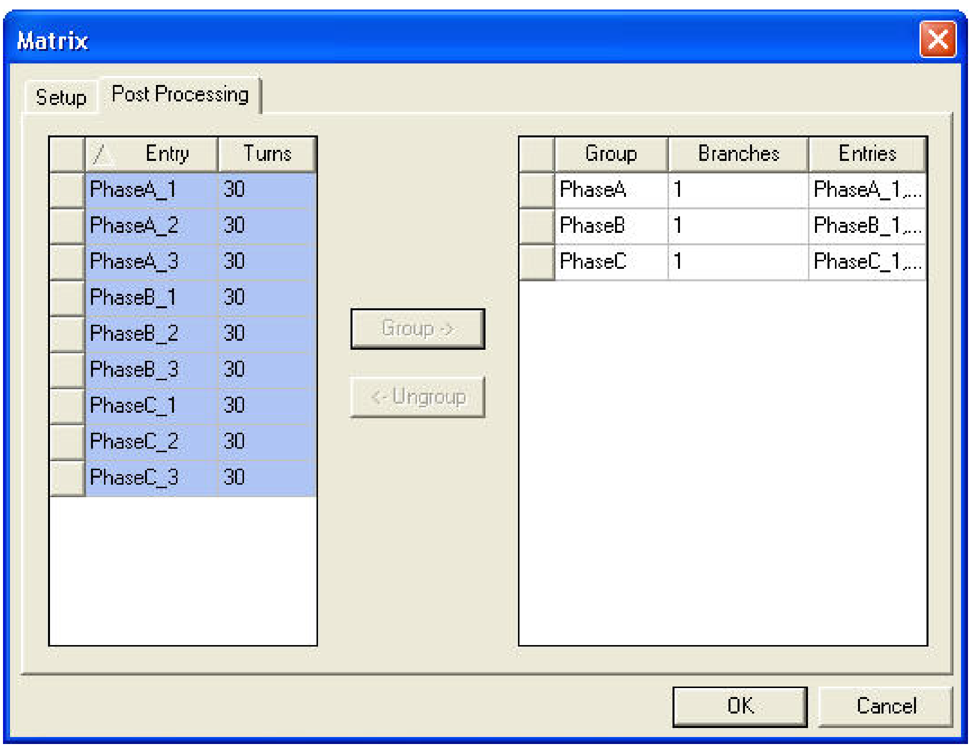
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. General Tab
       - Percentage Error: **1**
    2. Solver Tab
       - Nonlinear Residuals : **0.001**
    3. Click the **OK** button



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

Create Matrix Parameters

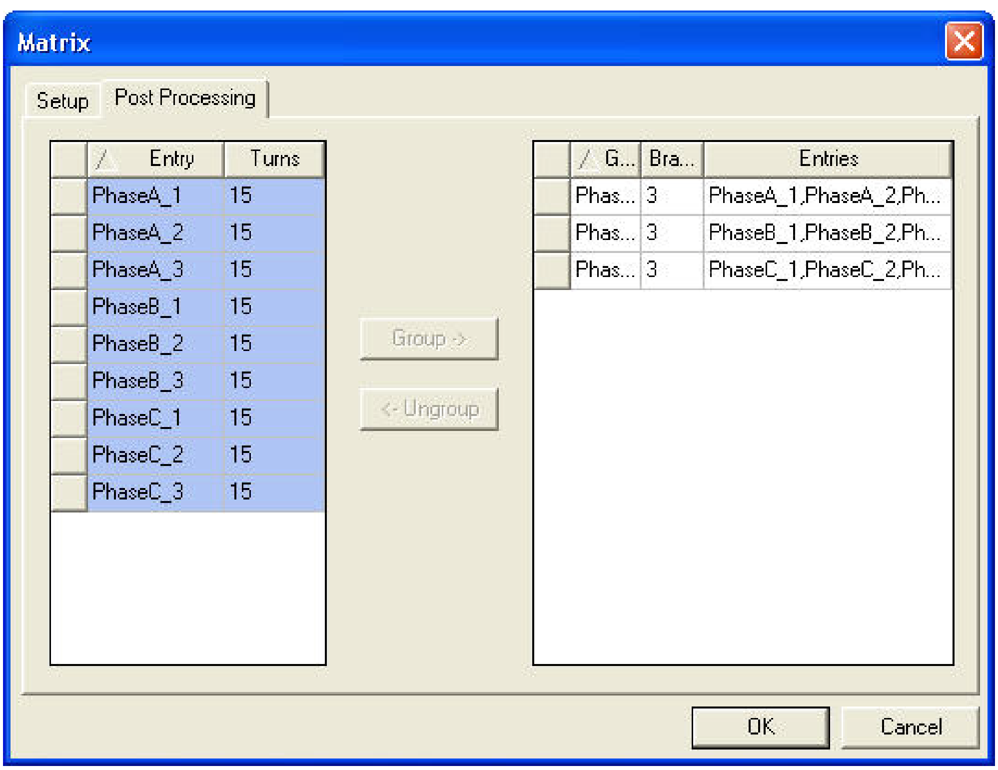
- **Create parameters for 30 Turns of Coil**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Solver Tab:
       - Include : ☑ **Checked** for all sources
    2. In Post Processing Tab
       1. Turns : Specify **30** for each of the sources
       2. Press **Ctrl** and select **PhaseA_1, PhaseA_2** and **PhaseA_3**
       3. Select **Group**
       4. Rename the group created as **PhaseA**
       5. Leave Branches to **1**
       6. In similar way, create groups **PhaseB** and **PhaseC**
    3. Press **OK**



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

Example (Magnetostatic) – Inductance Calculation

- **Create parameters for 15 Turns of Coil**
  - We will create a second Matrix Parameter to show what the settings signify.
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Solver Tab:
       - Include : ☑ **Checked** for all sources
    2. In Post Processing Tab
       1. Turns : Specify **15** for each of the sources
       2. Press **Ctrl** and select **PhaseA_1, PhaseA_2** and **PhaseA_3**
       3. Select **Group**
       4. Rename the group created as **PhaseA**
       5. Change Branches to **3**
       6. In similar way, create groups **PhaseB** and **PhaseC**



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

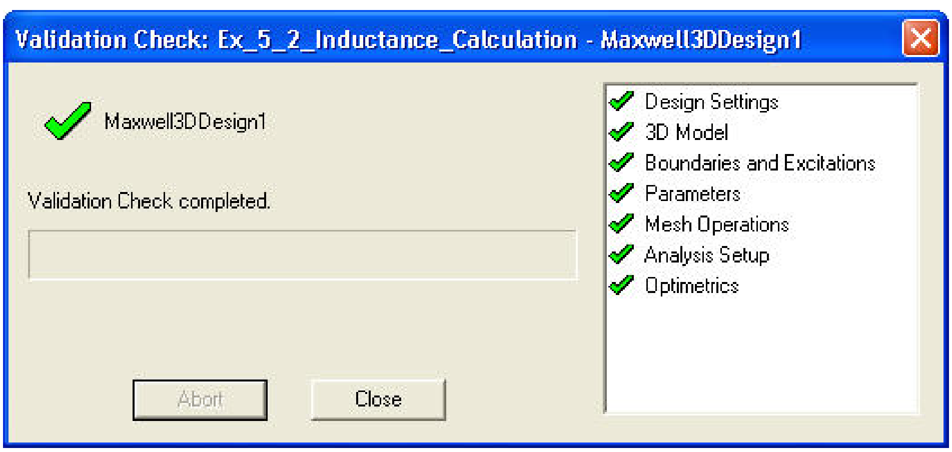
Example (Magnetostatic) – Inductance Calculation

Save 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: **Ex_5_2_Inductance_Calculation**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

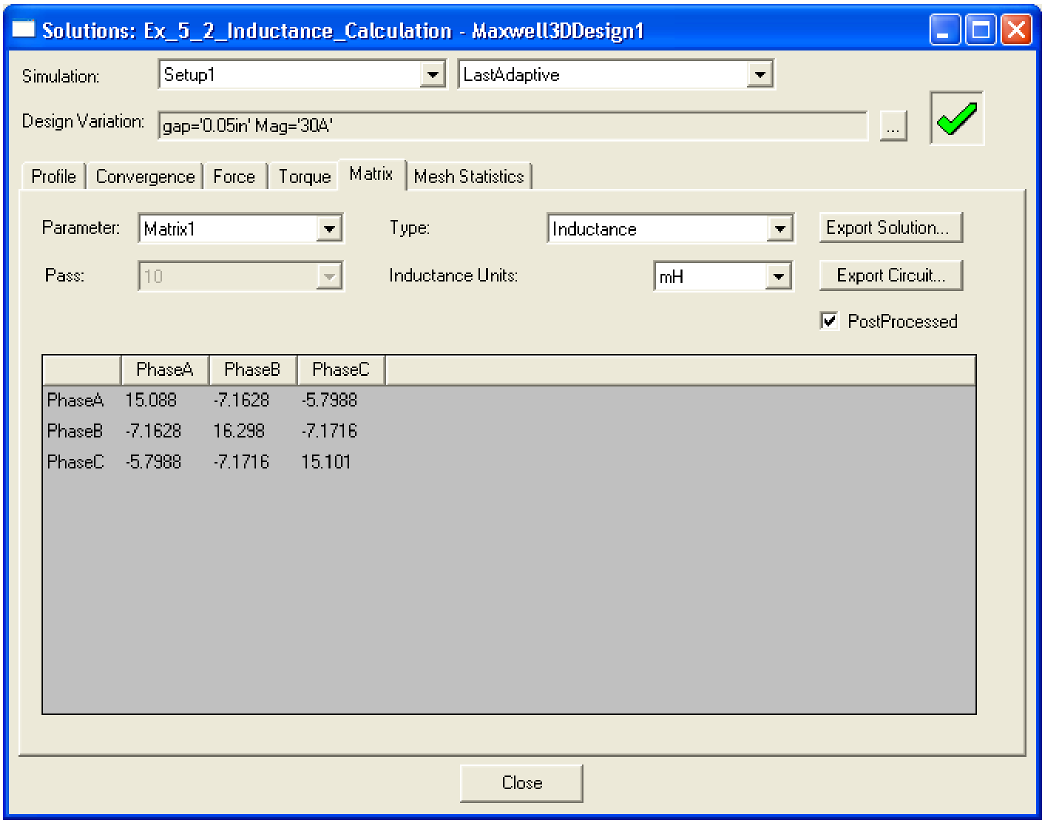
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Example (Magnetostatic) – Inductance Calculation

Solution Data

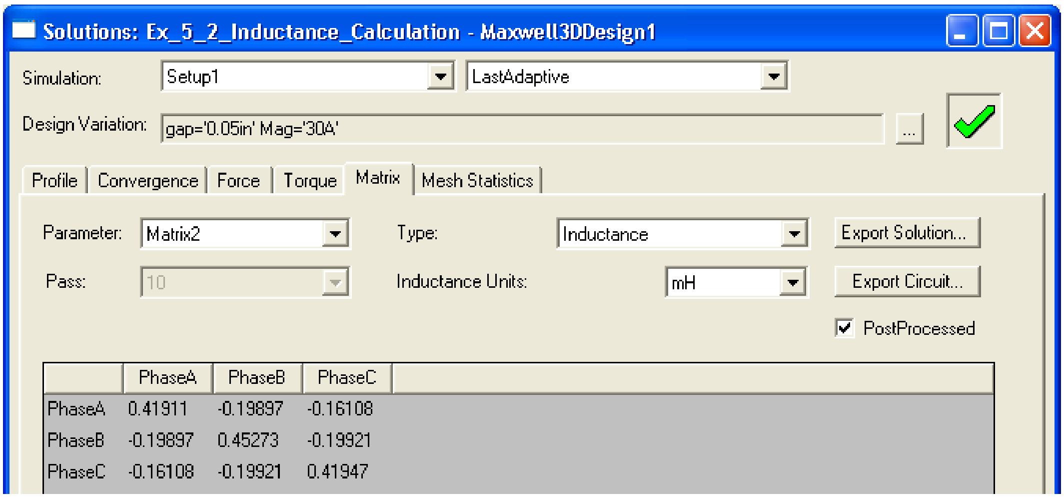
- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
  - Matrix Tab
    1. By default, non-postprocessed data is shown for Matrix1
    2. Change the option PostProcessed: ☑ **Checked**



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

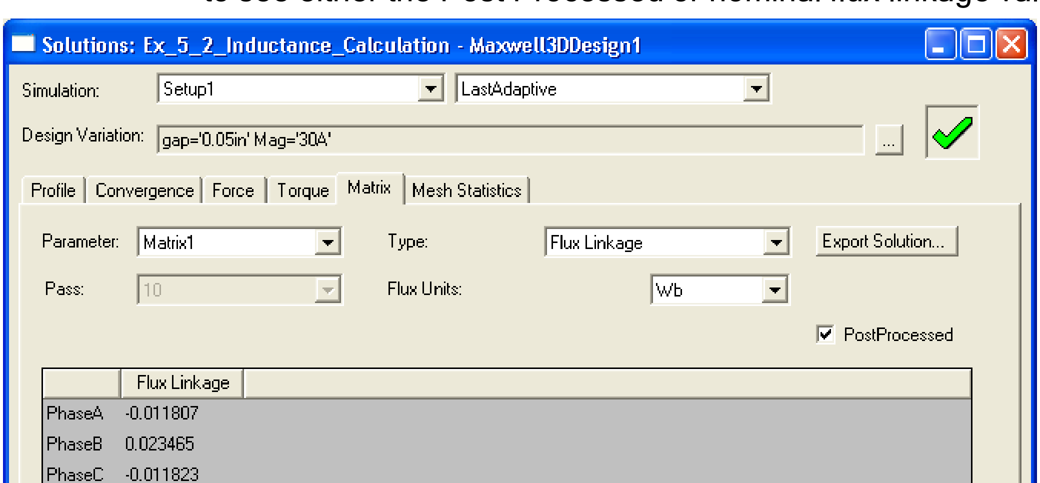
Example (Magnetostatic) – Inductance Calculation

3. Change Parameter to **Matrix2**



- **Note:**
  - Notice that the results change to reflect twice the turns and one-third the branches (2^2 * 3^2 = 36 times the previous inductance value).
  - Remember that by increasing the turns, the current in each turn is decreased to keep the total ampere-turns constant – this is very important for nonlinear materials. The solutions apply only for this value of current.
  - If you were to compare the non-post processed results for both matrices, you would see that these values are the same.

4. Change Parameter to **Matrix1** and the Solution Type to **Flux Linkage** to see either the Post Processed or nominal flux linkage values.



Example (Magnetostatic) – Inductance Calculation

Results and Interpretations

- These results are being obtained from a magnetostatic simulation with arbitrary constant current inputs defined for each leg. The notation used was in terms of 3 phases, which might imply that this is an AC inductor of some sort. The modeling of AC currents in a magnetostatic simulation can sometimes be valid for instantaneous values (of force and inductance), but the requirements are that eddy currents do not effect the solution (either a low frequency source or low loss materials). The other two options available in the Maxwell software are eddy current simulations (which allows AC sources, but does not allow nonlinear materials), and transient simulations (which allows the study of nonlinear and AC phenomena).

- This type of magnetostatic simulation would also be valuable for an Equivalent Circuit Extraction (again, eddy currents should be negligible) - this would require each phase to be excited as a variable, and run as a full parametric simulation including all phases across all possible current values (at least 5^3 variations to see any nonlinear behavior). Notice that if you do an equivalent circuit extraction of the inductance, the flux values automatically take into account the grouping and other post-processing values specified in the matrix parameter.

- Other magnetostatic results can be obtained from this simulation, such as force on the Core_I object and magnetic field in the steel (is it saturating?).

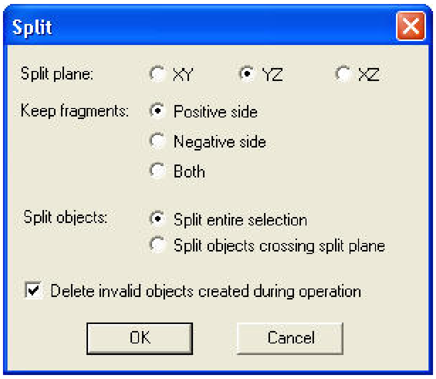
- Remember that this simulation uses nonlinear materials, so all of the solutions are only valid for this specific set of excitations. If you were to change the input currents, then you will have to solve again to see how the inductance and other results change.

- **Another important note is that although we included 2 matrix parameters, this is not necessary to see differently post-processed results.** You can edit the post-processing characteristics of the matrix after having solved the problem, and the results will adjust to reflect the changes without needing to resolve. The only time that you would need multiple matrices is if one matrix or the other did not include all the coils. **By using two matrices, the inductance calculation was performed twice, lengthening the simulation.** The point of this example was to show that multiple matrix parameters can be defined – not to suggest using multiple matrix parameters.

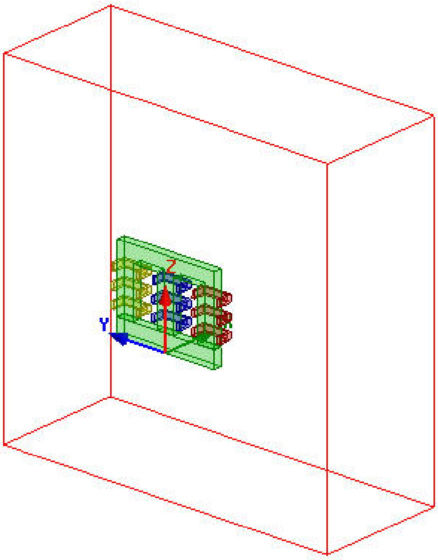
Example 2: Solving Symmetry Model

Create Symmetry Design

- **Copy Design**
  1. Select **Maxwell3D Design1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_2_Inductance_Calculation** in Project Manager window and select **Paste**
- **Split Geometry**
  - Select **Region** from the history tree
  - Select the menu item ***View > Visibility> Hide Selection > Active View***
  - Select the menu item ***Edit > Select All Visible***
  - Select menu item ***Modeler > Boolean > Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



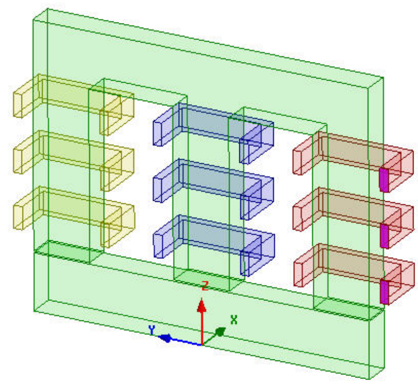
- **Modify Region**
  - Select menu item ***Draw > Region***
  - In Properties window,
    1. +X Padding Data: **800**
    2. -X padding Data: **0**
    3. Press **OK**
  - Select menu item ***View > Visibility > Show All > Active View*** to view Region



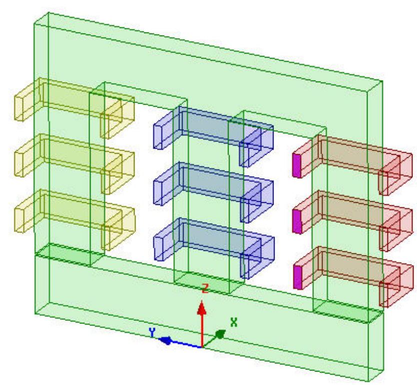
Example (Magnetostatic) – Inductance Calculation

Specify Excitations

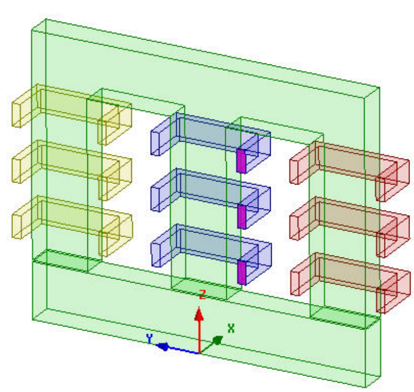
- **Create Excitation for CoilA**
  - Press **Ctrl** and select faces of CoilA as shown in image
  - Select menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseA_in**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Direction: **Positive X**
    5. Press **OK**



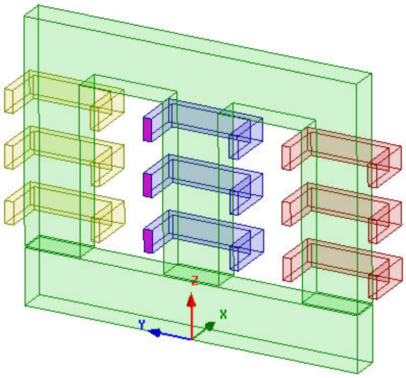
  - Press **Ctrl** and select faces of CoilA as shown in below image
  - Select menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseA_out**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Direction: **Negative X**
    5. Press **OK**



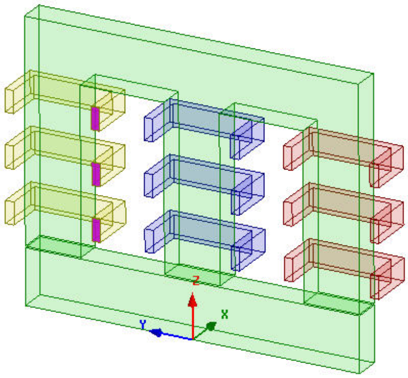
- **Create Excitations for CoilB**
  - Press **Ctrl** and select faces of CoilB as shown in below image
  - Select menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseB_in**
    2. Value: **Mag**
    3. Type: **Stranded**
    4. Direction: **Positive X**
    5. Press **OK**



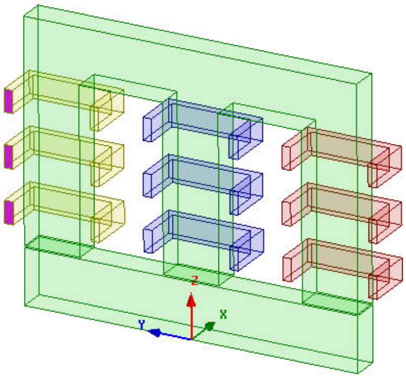
- Press **Ctrl** and select faces of CoilA as shown in below image
- Select menu item ***Maxwell 3D > Excitations > Assign > Current***
- In Current Excitation window,
  1. Name: **PhaseB_out**
  2. Value: **Mag**
  3. Type: **Stranded**
  4. Direction: **Negative X**
  5. Press **OK**



- **Create Excitation for CoilC**
  - Press **Ctrl** and select faces of CoilC as shown in below image
  - Select menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseC_in**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Direction: **Positive X**
    5. Press **OK**



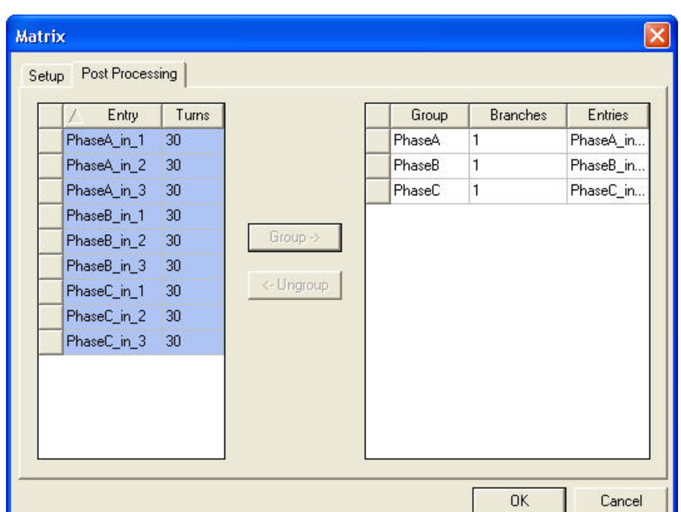
  - Press **Ctrl** and select faces of CoilC as shown in below image
  - Select menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseC_out**
    2. Value: **-0.5*Mag**
    3. Type: **Stranded**
    4. Direction: **Negative X**
    5. Press **OK**



Example (Magnetostatic) – Inductance Calculation

Create Matrix Parameters

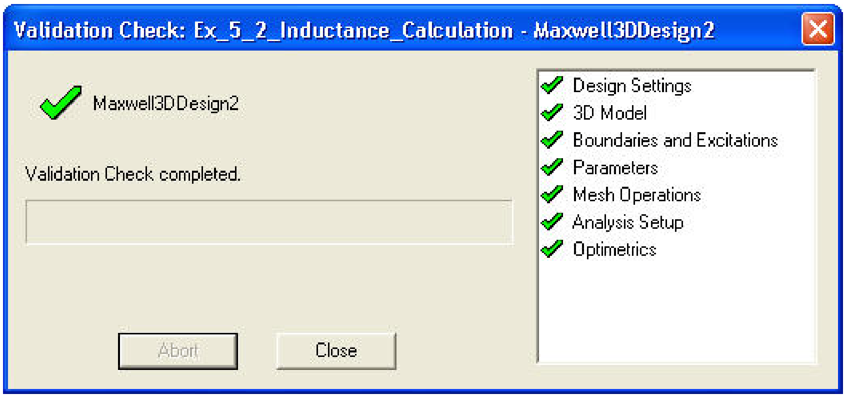
- **Note:**
  - When matrix parameters are defined, a check will be performed for proper conduction paths and for terminals correctly defined across a cross-section of the conduction paths. Several errors can appear when defining a matrix, and these often have to do with the definition of the conducting objects or the excitation.
  - The excitations that appear in the matrix setup, are those excitations that are pointing into the conductor. So, for a symmetry model that requires two excitations per conductor, only one excitation per conductor will be listed in the matrix setup – this excitation will be the one pointing into the modeled conductor.
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Solver Tab:
       - Include : ☑ **Checked** for all sources
    2. In Post Processing Tab
       1. Turns : Specify **30** for each of the sources
       2. Press **Ctrl** and select **PhaseA_in_1, PhaseA_in_2** and **PhaseA_in_3**
       3. Select **Group**
       4. Rename the group created as **PhaseA**
       5. Leave Branches to **1**
       6. In similar way, create groups **PhaseB** and **PhaseC**
    3. Press **OK**



Example (Magnetostatic) – Inductance Calculation

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



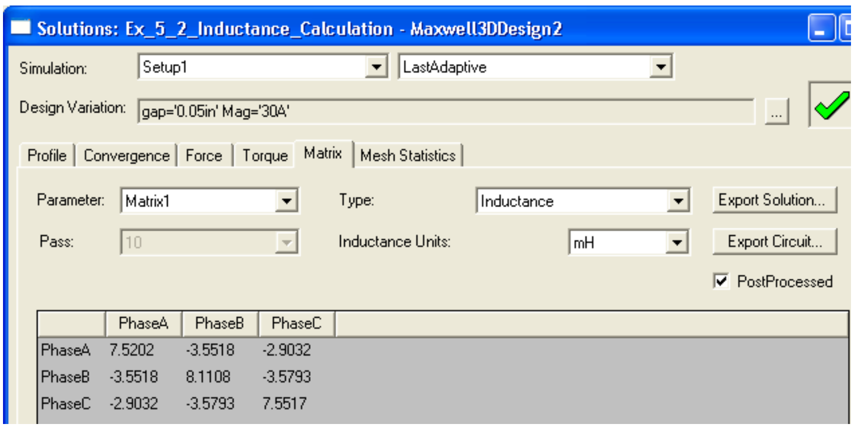
Note: To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

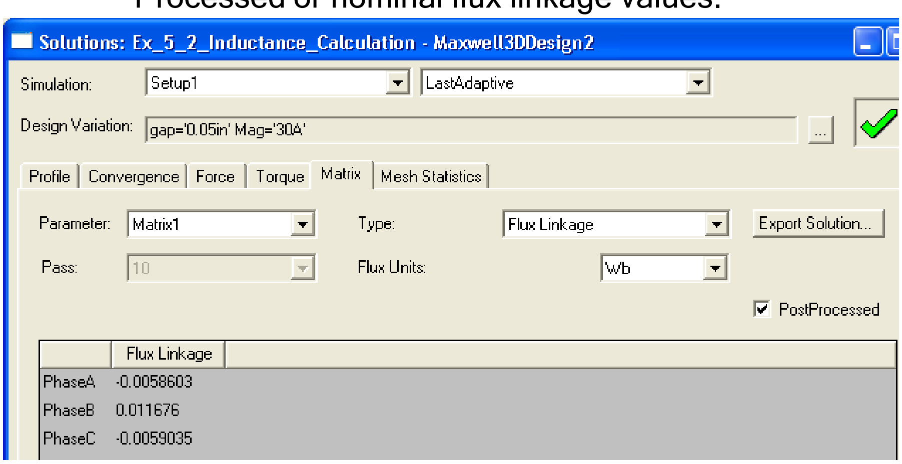
Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
  - Matrix Tab
    1. By default, non-postprocessed data is shown for Matrix1
    2. Change the option PostProcessed: ☑ **Checked**



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

    3. Change the Solution Type to **Flux Linkage** to see either the Post Processed or nominal flux linkage values.



Example 2: Incremental Inductance Calculation

Incremental Inductance Calculation

- Apparent inductance calculation works fine when material operation is within the linear region of BH Curve. When material is operating in non-linear region, inductance calculation with apparent method may not correct.
- In Such cases Incremental inductance gives better results compared to apparent inductance.
- Incremental inductance value is calculated using slope of the Flux linkage vs Current graph at material operating point.

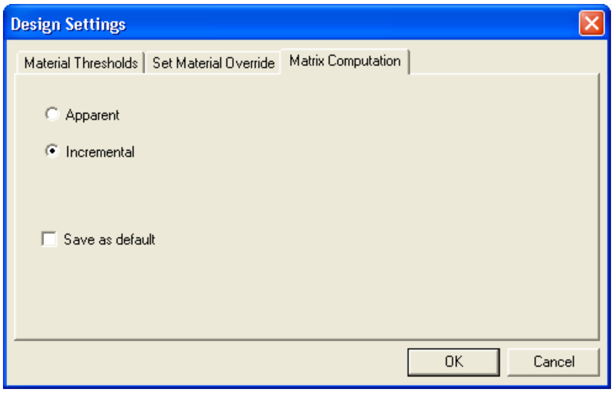
$$L_{jk}^{d} = \frac{\partial \lambda_j}{\partial i_k}$$

Create Symmetry Design

- **Copy Design**
  1. Select **Maxwell3D Design1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_2_Inductance_Calculation** in Project Manager window and select **Paste**

Change Inductance Calculation Method

- **To Change inductance Calculation method**
  - Select the menu item ***Maxwell 3D > Design Settings***
  - In Design Settings window,
    - Select the tab Matrix Computation
    - Press **OK**



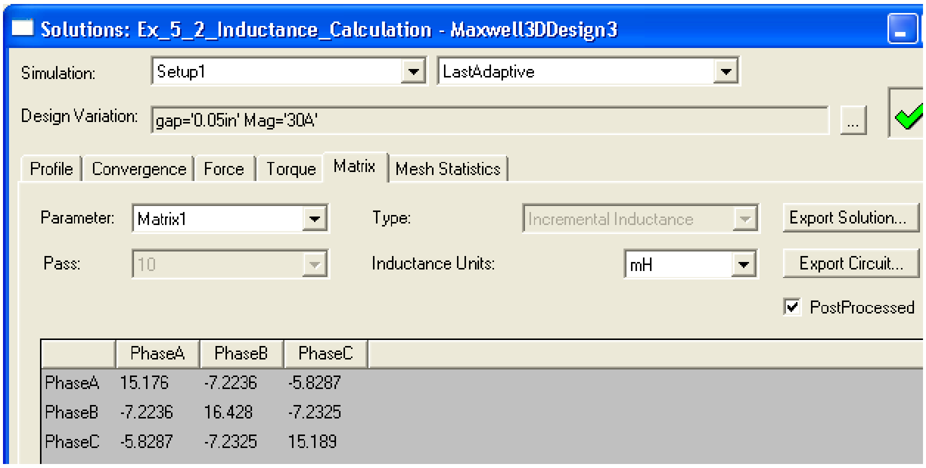
Example (Magnetostatic) – Inductance Calculation

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
  - Matrix Tab
    1. PostProcessed: ☑ **Checked**

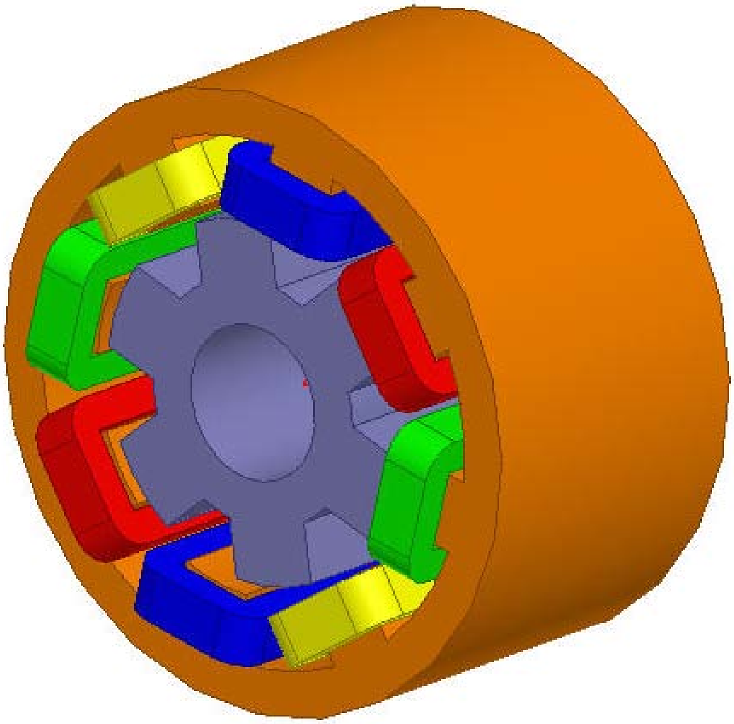


- It can be seen that the results are almost the same as well got with apparent inductance calculation method.
- Now we will drive the core into nonlinear region of operation and check the impact

Example (Magnetostatic) – Stranded Conductors

Stranded Conductors

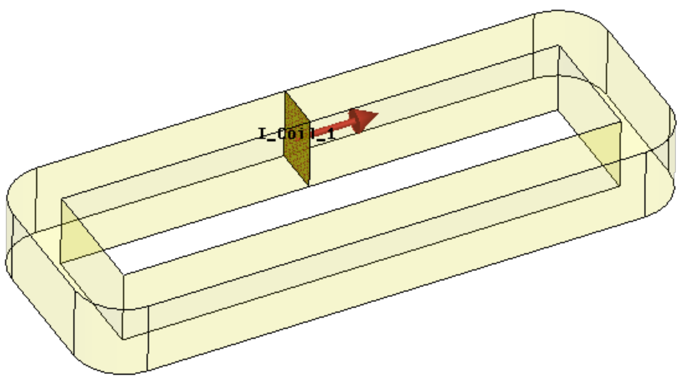
- This example is intended to show you how to create and analyze a magnetostatic problem with stranded conductors for a Switched Reluctance Motor geometry using Magnetostatic solver in the ANSYS Maxwell 3D Design Environment.
- Within the Maxwell 3D Design Environment, complete coils can be modeled as Stranded Conductors. There are many advantages to using Stranded Conductors when modeling coils that have multiple turns. The first obvious advantage is that a coil with multiple wires, say 2500, can be modeled as a single object as opposed to modeling each wire which would be impracticable. Secondly, defining a Stranded Conductor means that the current density will be uniform throughout the cross section of the conductor which is physically correct for a coil with many turns.
- The example that will be used to demonstrate how Stranded Conductors are implemented in a switched Reluctance Motor. This switched reluctance motor will have four phases and two coils per phase.
- Note: There is no fundamental difference between how stranded conductors are treated in the Eddy Current solver as compared to the Magnetostatic Solver. When using the Eddy Current Solver and specifying Stranded for the terminal excitation the Eddy Effect is automatically turned off



Example (Magnetostatic) – Stranded Conductors

Theory – Magnetostatic Solver

- When using the Magnetostatic Solver, with a Stranded Current Source, the solver assumes the following conditions:
  1. The current density is uniformly distributed over the cross section of the terminal as well as through the entire conductor.
  2. The direction of the current is indicated by the arrow as seen on the terminal. Change the direction of the current by clicking on ***Swap Direction*** in the Current Excitation window to reverse the flow.



  3. The solver does not know how many turns are represented by the coil, thus the value of current that is being applied represents the total ampere-turns. For this example, the value 3570 could represent any of the following:
     1. 1 amp thru 3570 turns
     2. 5 amps thru 750 turns
     3. 25 amps thru 150 turns
     4. 3570 amps thru 1 turn

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

Theory – Magnetostatic Solver (Continued)

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

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

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

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

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

ANSYS Maxwell Design Environment

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

Launching Maxwell

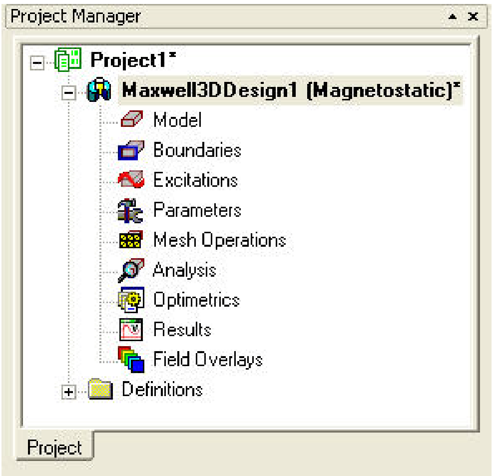
- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

Setting Tool Options

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

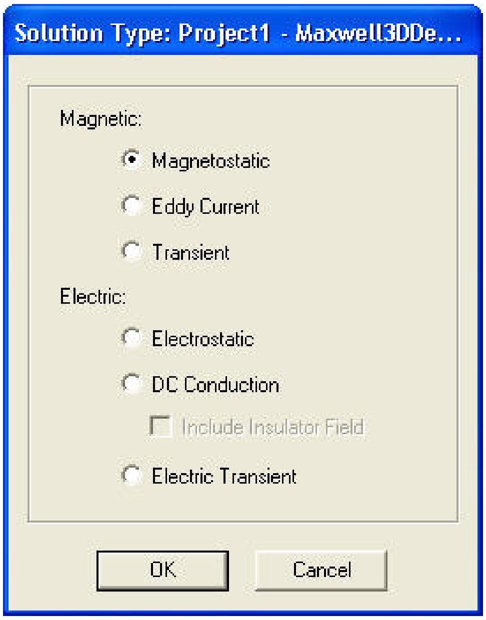
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - Select the menu item *Project > Insert Maxwell 3D Design*, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 3D > Solution Type*
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



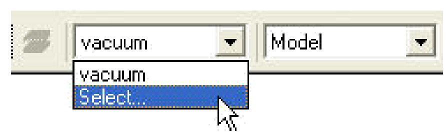
Example (Magnetostatic) - Stranded Conductors

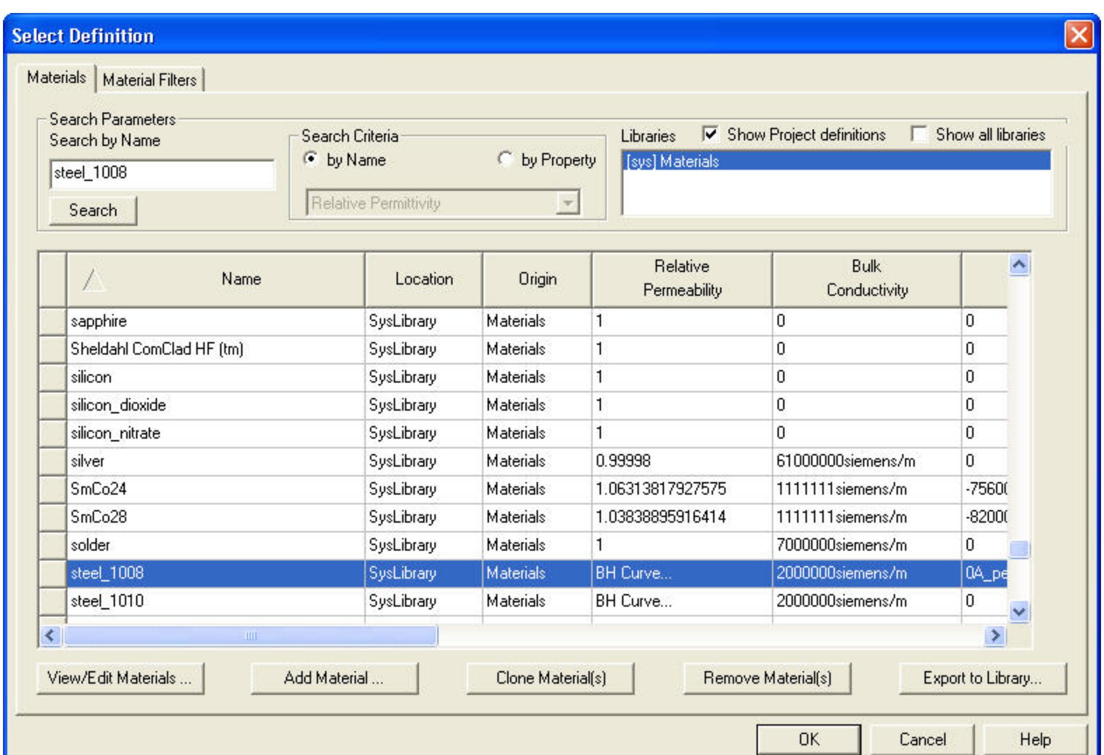
Set Model Units

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

Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **steel_1008** in the **Search by Name** field
    2. Click the **OK** button





Create Rotor

To Create Rotor

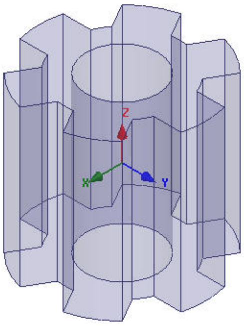
- Select the menu item ***Draw > User Defined Primitive > SysLib > RMxprt > SRMCore***
- In User Defined Primitive Operation window
  1. For the value of **DiaGap,** type: **70**, Click the **Tab** key to accept
  2. For the value of **DiaYoke**, type: **30**, Click the **Tab** key to accept
  3. For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

Parameters | Info

Name	Value	Unit	Evaluate...	Description
Command	CreateUse...			
Coordinate System	Global			
DLL Name	RMxprt/S...			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	70	mm	70mm	Core diameter on gap side, DiaGap<DiaYo...
DiaYoke	30	mm	30mm	Core diameter on yoke side, DiaYoke<DiaG...
Length	65	mm	65mm	Core length
Poles	6		6	Number of poles
ThkYoke	9	mm	9mm	Yoke thickness
Embrace	0.5		0.5	Pole embrace (the ratio of pole arc to pole ...
EndExt	0	mm	0mm	Coil one-side end extended length
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 1: core & coils; 2: coil; 3: terminal1; ...

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

- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Rotor**



Create Stator and Coil

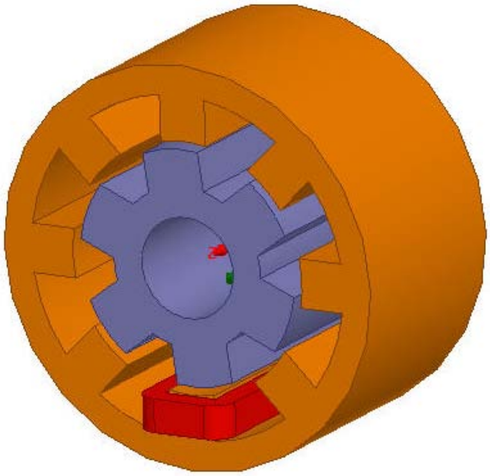
- **To Create Stator and Coils**
  - Select the menu item ***Draw > User Defined Primitive > SysLib > RMxprt > SRMCore***
  - In User Defined Primitive Operation window
    1. For the value of **DiaGap,** type: **75**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **120**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **65**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **8**, Click the **Tab** key to accept
    5. For the value of **ThkYoke**, type: **9**, Click the **Tab** key to accept
    6. For the value of **Embrace**, type: **0.5**, Click the **Tab** key to accept
    7. For the value of **EndExt**, type: **1**, Click the **Tab** key to accept
    8. For the value of **InfoCore**, type: **1**, Click the **Tab** key to accept
    9. Click the **OK** button

Parameters | Info

Name	Value	Unit	Evaluate...	Description
Command	CreateUse...			
Coordinate System	Global			
DLL Name	RMxprt/S...			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	75	mm	75mm	Core diameter on gap side, DiaGap<DiaYo...
DiaYoke	120	mm	120mm	Core diameter on yoke side, DiaYoke<DiaG..
Length	65	mm	65mm	Core length
Poles	8		8	Number of poles
ThkYoke	9	mm	9mm	Yoke thickness
Embrace	0.5		0.5	Pole embrace (the ratio of pole arc to pole ...
EndExt	1	mm	1mm	Coil one-side end extended length
LenRegion	200	mm	200mm	Region length
InfoCore	1		1	0: core; 1: core & coils; 2: coil; 3: terminal1; ..

Example (Magnetostatic) – Stranded Conductors

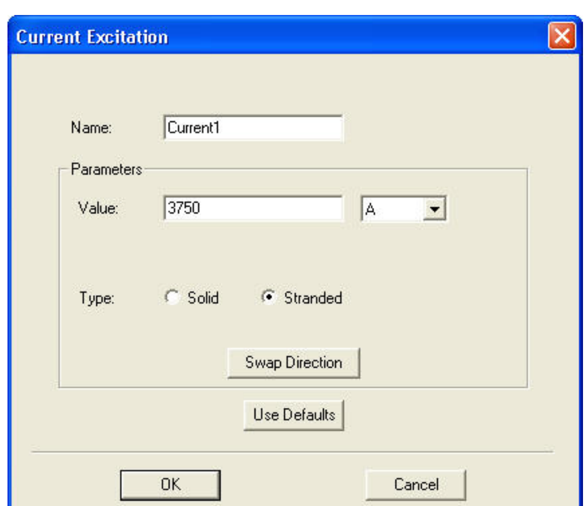
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Stator**
    2. Change its color to **Orange**
- **Separate Coils**
  - Select the object **Stator** from the tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Unwanted Objects**
  - Select the objects **Stator_Separate2, Stator_Separate3, Stator_Separate4, Stator_Separate5, Stator_Separate6, Stator_Separate7** and **Stator_Separate8** from the tree
  - Select the menu item ***Edit > Delete***
  - **Note:** We have kept only one coil and deleted others. We will recreate other coils after excitations are specified for one coil. This way we can copy coil terminal and excitation from one coil to all other coils. Thus we will can save time needed to specify excitation for each coil.
- **Change Attributes**
  - Select the object **Stator_Separate1** from the tree and goto Properties window
    1. Change the name of the object to **Coil_A1**
    2. Change its color to **Red**
    3. Goto Material tab and select **Edit**
    4. In Select Definition window,
       1. Type **copper** in the **Search by Name** field
       2. Click the **OK** button



Example (Magnetostatic) - Stranded Conductors

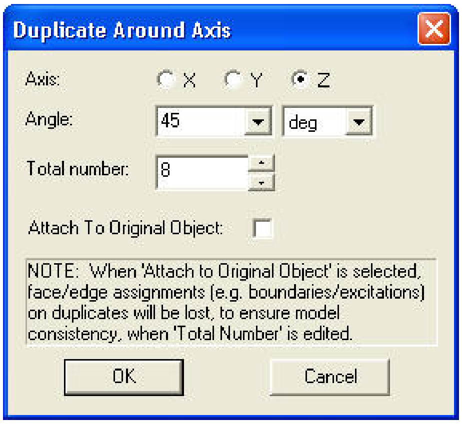
Create Excitations

- **Create Section**
  - Select the object **Coil_A1** from the tree
  - Select the menu item ***Modeler > Surface >Section***
  - In Section window,
    1. Section Plane: ***XY***
    2. Click the **OK** button
- **Change Attributes**
  - Select the object **Coil_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Terminal_A1**
    2. Change its color to **Red**
- **Separate Sections**
  - Select the object **Terminal_A1** from the tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the object **Terminal_A1_Separate1** from the tree
  - Select the menu item ***Edit > Delete***
- **Assign Excitation**
  - Select the object **Terminal_A1**
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **3750 A**
    3. Type: **Stranded**
    4. Press **OK**

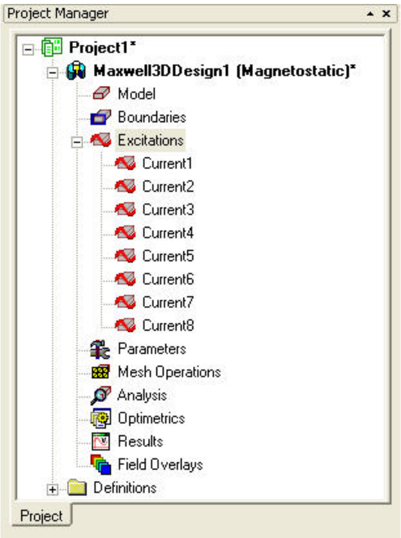


Duplicate Coil with Excitations

- **Verify Duplicate Boundary Option is Set**
  - Select the menu item *Tools > Options > Maxwell 3D Options*
  - Maxwell Options Window:
    1. Click the **General Options** tab
       - Duplicate boundaries/mesh operations with geometry: ☑ **Checked**
    2. Click the **OK** button
- **Duplicate Objects**
  - Select the object **Coil_A1** and sheet **Terminal_A1** from the tree
  - Select the menu item *Edit > Duplicate > Around Axis*
  - In Duplicate Around Axis window,
    1. Axis: **Z**
    2. Angle: **45deg**
    3. Total Number: **8**
    4. Click the **OK** button

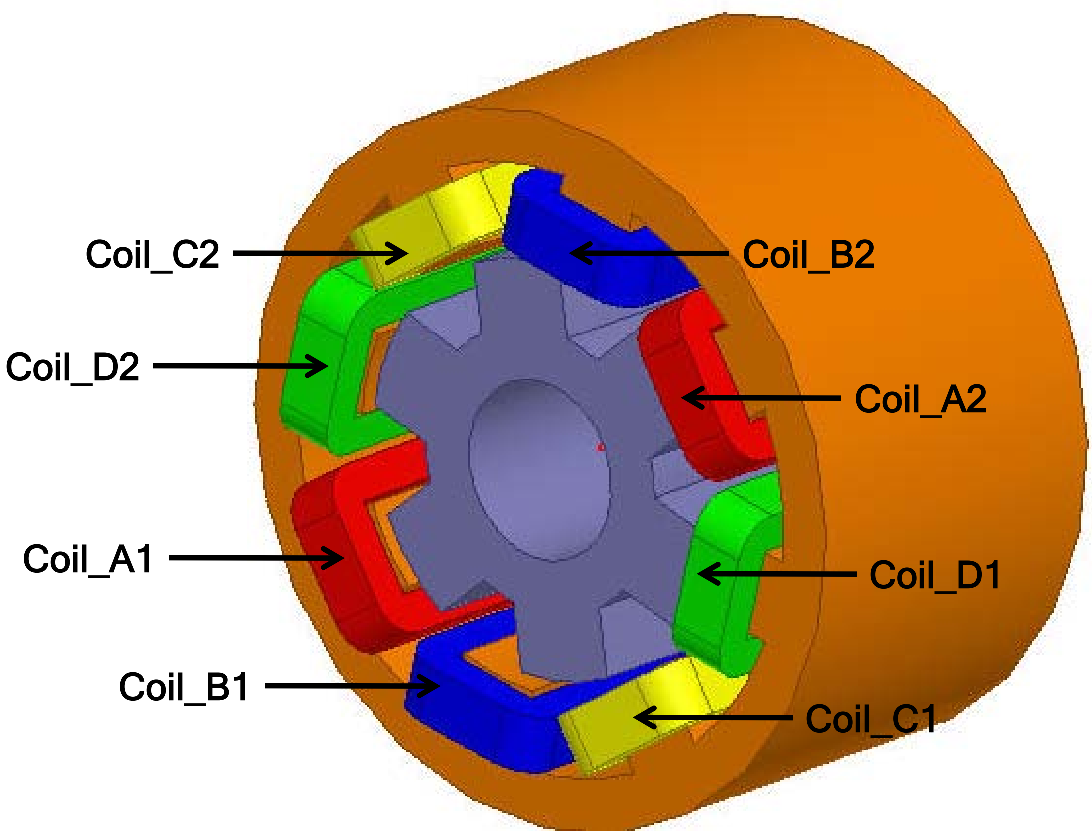


  - **Note:** As we have copies Terminals with the above option set, excitations are also copied for each terminal. In the tree now total eight excitations can be seen.



Example (Magnetostatic) - Stranded Conductors

- **Change Attributes for Coils**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the objects **Coil_A1_1, Coil_A1_2, Coil_A1_3, Coil_A1_4, Coil_A1_5, Coil_A1_6** and **Coil_A1_7** to **Coil_B1, Coil_C1, Coil_D1, Coil_A2, Coil_B2, Coil_C2** and **Coil_D2**
    2. Change the color of the objects **Coil_B1** and **Coil_B2** to **Blue**
    3. Change the color of the objects **Coil_C1** and **Coil_C2** to **Yellow**
    4. Change the color of the objects **Coil_D1** and **Coil_D2** to **Green**
- **Change Attributes for Terminals**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the sheets **Terminal_A1_1, Terminal_A1_2, Terminal_A1_3, Terminal_A1_4, Terminal_A1_5, Terminal_A1_6** and **Terminal_A1_7** to **Terminal_B1, Terminal_C1, Terminal_D1, Terminal_A2, Terminal_B2, Terminal_C2** and **Terminal_D2**
    2. Change color of the sheets **Terminal_B1** and **Terminal_B2** to **Blue**
    3. Change color of the sheets **Terminal_C1** and **Terminal_C2** to **Yellow**
    4. Change color of the sheets **Terminal_D1** and **Terminal_D2** to **Green**



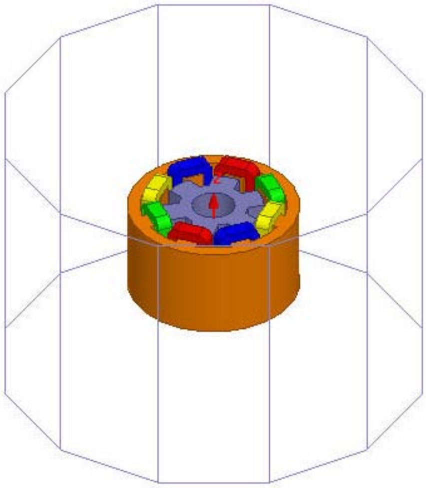
Example (Magnetostatic) – Stranded Conductors

Set Default Material

- **To Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

Create Region

- **To create the region:**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center position
       - X: **0.0**, Y: **0.0**, Z: **-100.0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius and height:
       - dX: **150.0** dY: **0.0**, dZ: **200.0**, Press the **Enter** key
    3. Segment Number Window
       - Number of Segments: **12**
       - Click the **OK** button
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Display Wireframe: ☑ **Checked**



- **Hide Region**
  - Select **Region** from the history tree
  - Select the menu item ***View >Visibility > Hide Selection > Active View***
  - Select the menu item ***View > Fit All > Active View.***

Analysis Setup

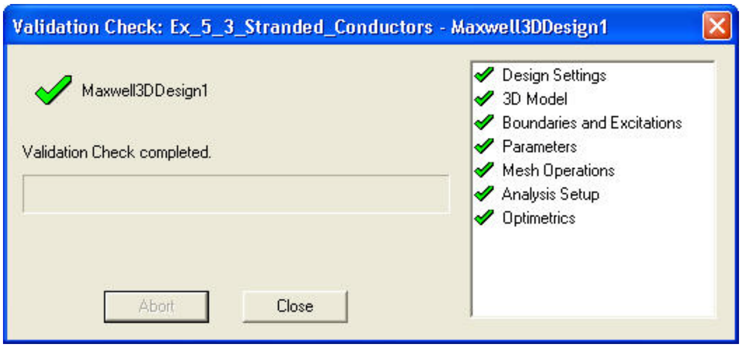
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From **Save As** window, type the Filename: **Ex_5_3_Stranded_Conductors**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

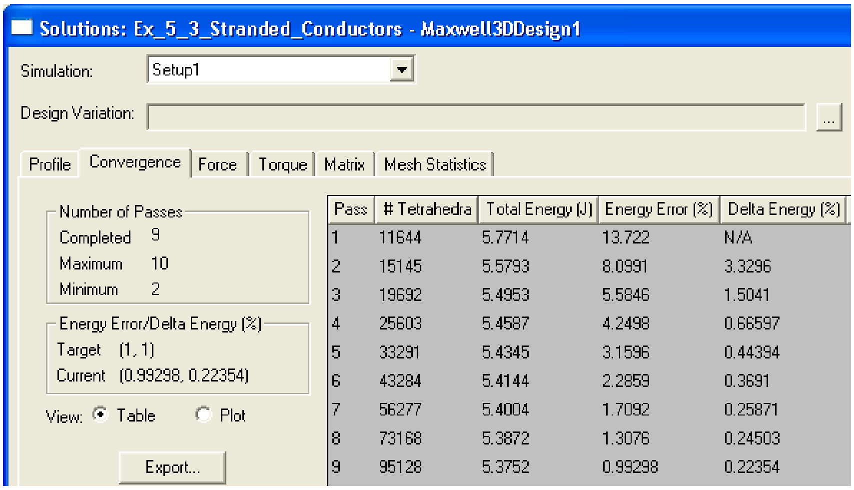
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

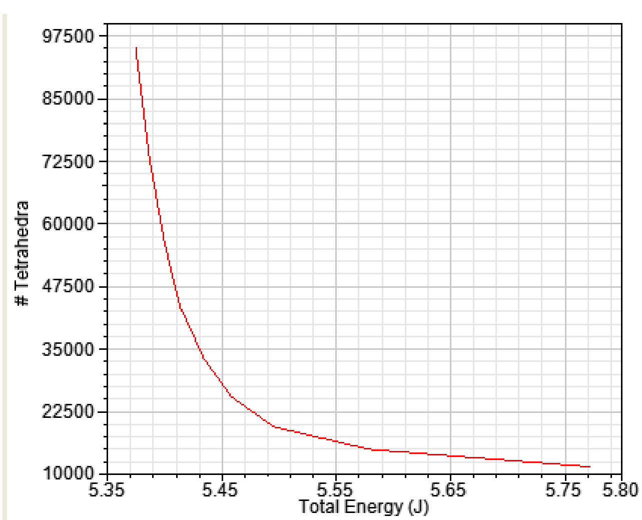
Example (Magnetostatic) - Stranded Conductors

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
  - Click the **Close** button

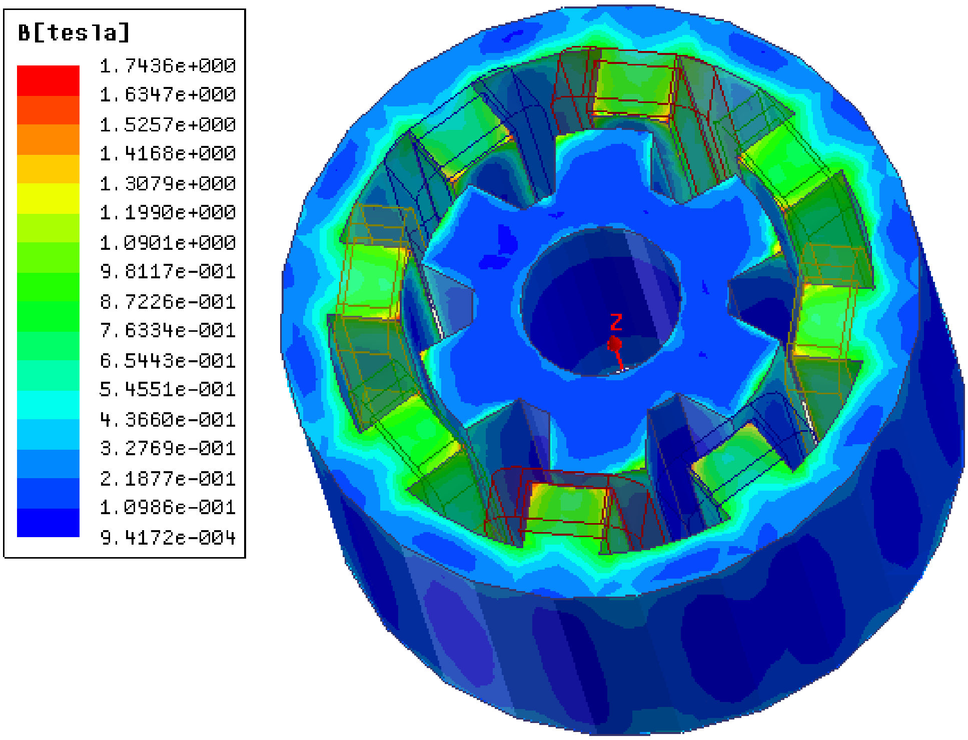


Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	11644	5.7714	13.722	N/A
2	15145	5.5793	8.0991	3.3296
3	19692	5.4953	5.5846	1.5041
4	25603	5.4587	4.2498	0.66597
5	33291	5.4345	3.1596	0.44394
6	43284	5.4144	2.2859	0.3691
7	56277	5.4004	1.7092	0.25871
8	73168	5.3872	1.3076	0.24503
9	95128	5.3752	0.99298	0.22354



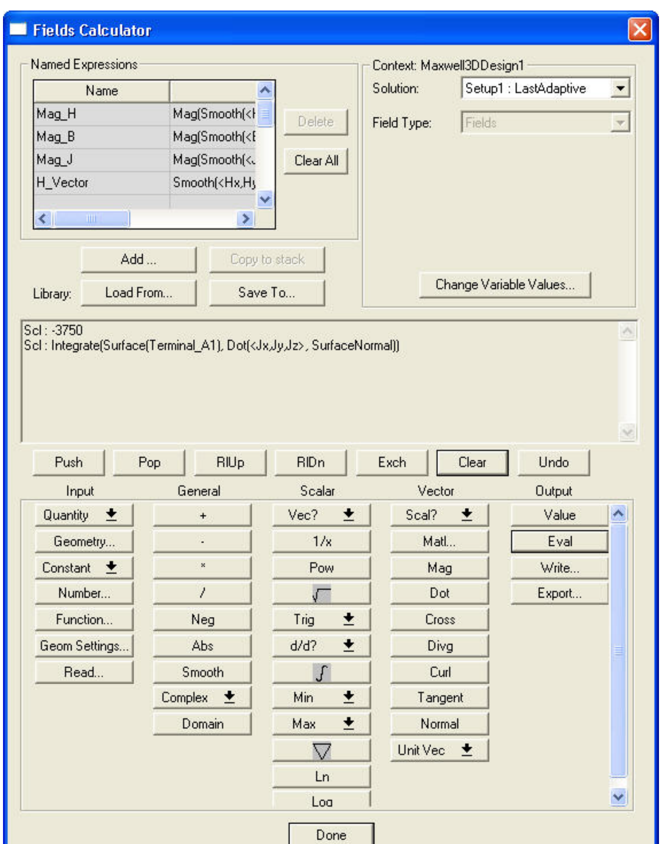
Create Field Overlay

- **To create a field plot:**
  - Select the objects **Stator** and **Rotor** from the tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In the Create Field Plot window:
    1. Solution: **Setup1 : LastAdaptive**
    2. Quantity: **Mag_B**
    3. Plot on Surface Only: ☑ **Checked**
    4. Click the **Done** button



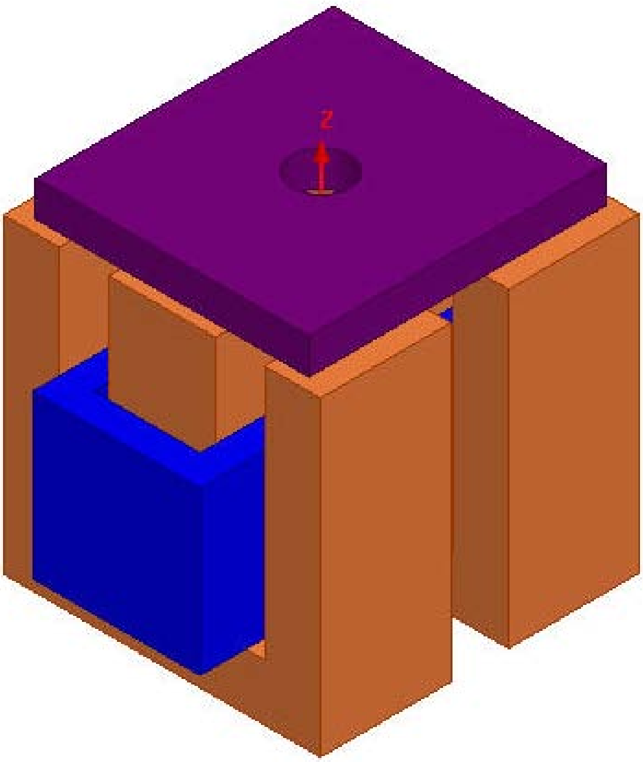
Calculate Current

- **Use the field calculator to verify the total amp-turns flowing in the winding:**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - In Fields Calculator window,
    - Select **Input > Quantity > J**
    - Select **Input > Geometry**
      1. Select the radio button **Surface**
      2. From the list select: **Terminal_A1**
      3. Click the **OK** button
    - Select **Vector > Normal** to determine the current normal to the terminal
    - Select ∫ (Integrate)
    - Click the **Eval** button to evaluate the results
    - **Note:** Value reported by calculator is current in Ampere-Turns flowing through the coil which is equal to **3750**
    - Press **Done** to exit calculator



Example (Magnetostatic) – ECE: Linear Movement

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



ANSYS Maxwell Design Environment

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

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

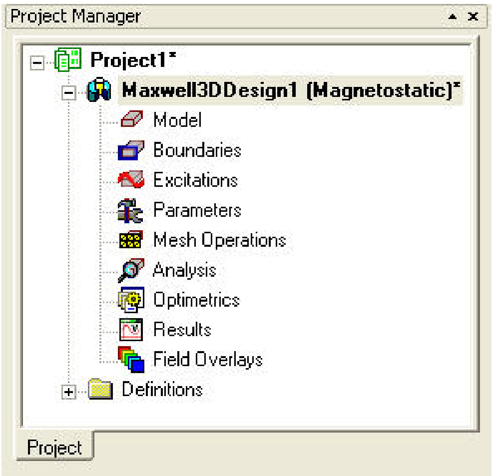
Setting Tool Options

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

Example (Magnetostatic) – ECE: Linear Movement

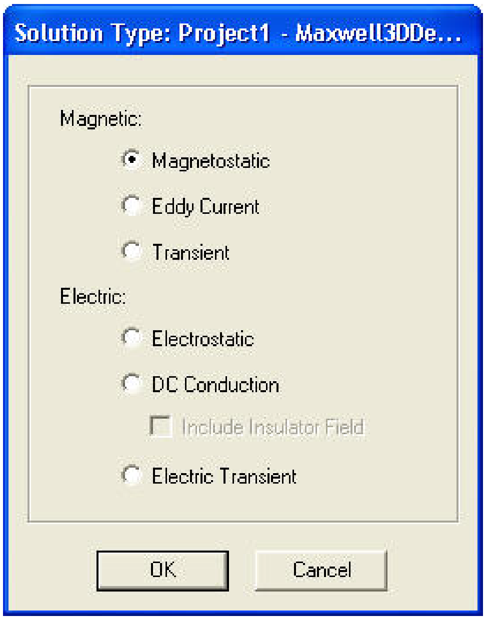
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



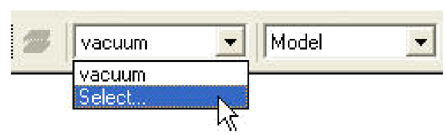
Example (Magnetostatic) – ECE: Linear Movement

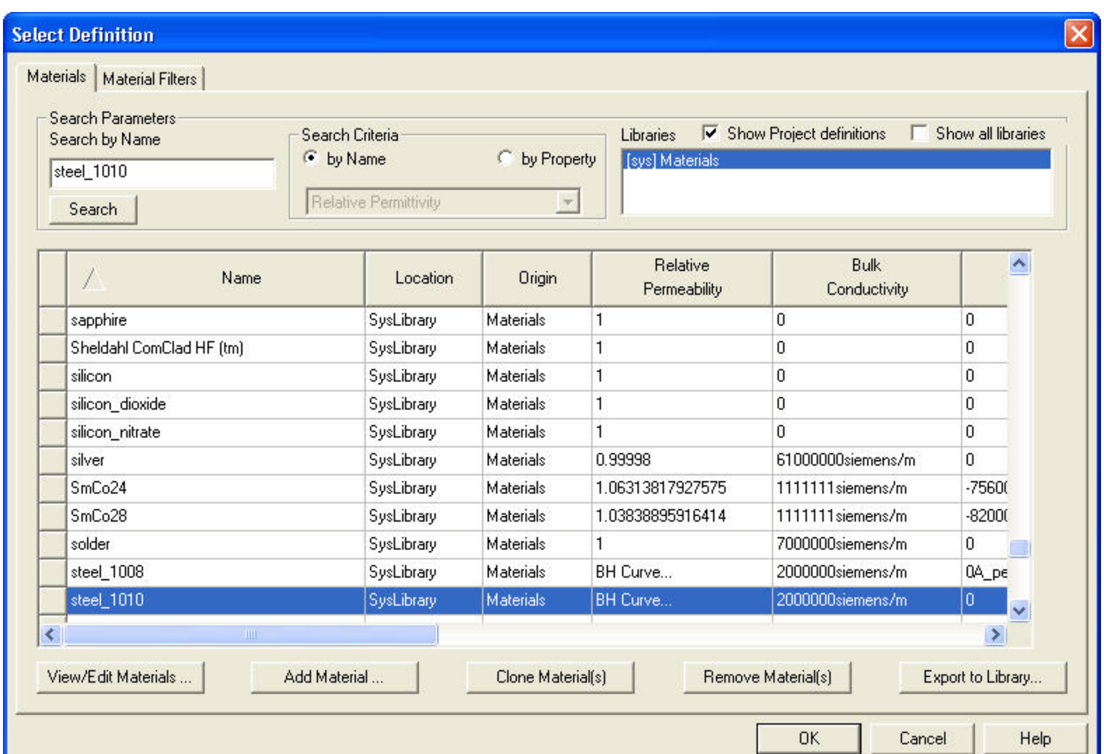
Set Model Units

- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **in** ***(inches)***
    2. Click the **OK** button

Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **steel_1010** in the **Search by Name** field
    2. Click the **OK** button

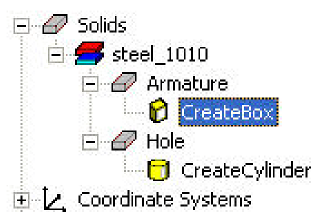




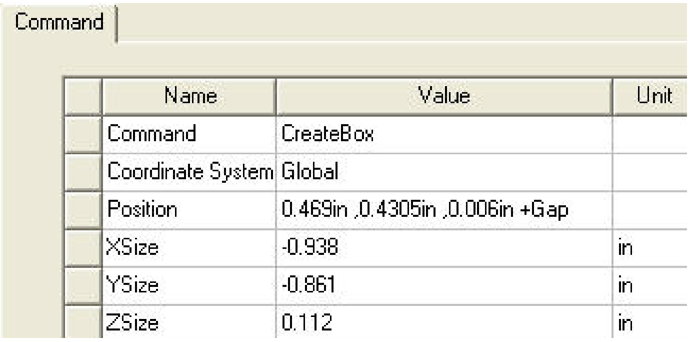
Example (Magnetostatic) – ECE: Linear Movement

Create Armature

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.469**, Y: **0.4305**, Z: **0.006**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **-0.938**, dY: **-0.861**, dZ: **0.112**, Press the **Enter** key
  - Select the menu item ***View > Fit All > Active View***.
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Armature**
    2. Change its color to **Purple**



- **Parameterize Armature**
  - Expand the history tree of the object **Armature**
  - Double click on the command **CreateBox** from the tree
  - For **Position**, type: 0.469, 0.4305, **0.006in +Gap**, Click the **Tab** key to accept
  - In Add variable window,
    1. Unit Type: **Length**
    2. Unit: **in** ***(inches)***
    3. Value: **0**
    4. Press **OK**
  - Press **OK** to exit



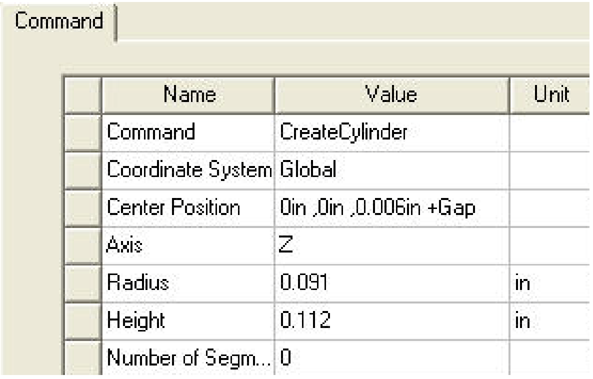
Command

Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	0.469in ,0.4305in ,0.006in +Gap	
XSize	-0.938	in
YSize	-0.861	in
ZSize	0.112	in

- **Note:** We will first run a nominal calculation with value of Gap=0. Then we will setup a parametric sweep to very Gap value over a range and check its effect on output parameters.

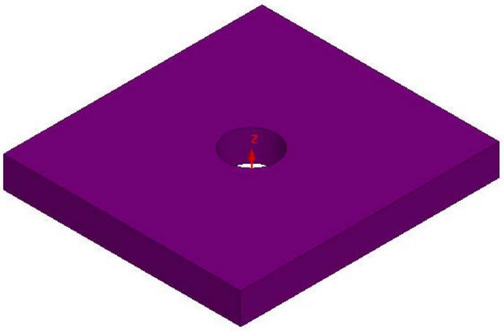
Example (Magnetostatic) – ECE: Linear Movement

- **Create Hole**
  - Select the menu item ***Draw > Cylinder***
    1. Using the coordinate entry fields, enter the centre of base
       - X: **0.0**, Y: **0.0**, Z: **0.006**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius and height of cylinder
       - dX: **0.091,** dY: **0.0**, dZ: **0.112**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Hole**
- **Parameterize Hole**
  - Expand the history tree of the object **Hole**
  - Double click on the command **CreateCylinder** from the tree
  - For **Position**, type: **0.469, 0.4305, 0.006in +Gap**, Click the **Tab** key to accept
  - Press **OK** to exit


Command

Name	Value	Unit
Command	CreateCylinder	
Coordinate System	Global	
Center Position	0in ,0in ,0.006in +Gap	
Axis	Z	
Radius	0.091	in
Height	0.112	in
Number of Segm...	0	

- **Subtract Hole**
  - Press **Ctrl** and select the objects **Armature** and **Hole** from the tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Armature**
    2. Tool Parts: **Hole**
    3. Click the **OK** button



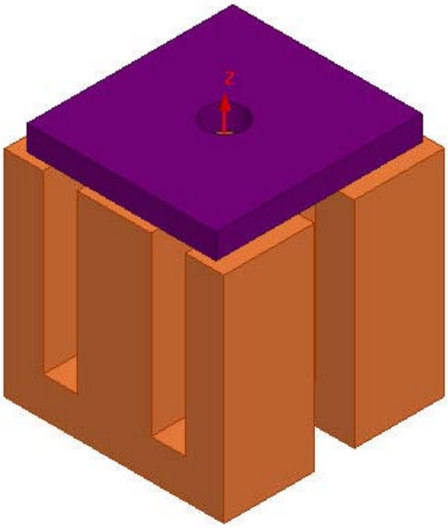
Create Stator

- **Create Profile for Sweep**
  - Select the menu item *Draw > Line*
    1. Using the coordinate entry field, enter the first point of the line
       - X: **0.5025,** Y: **0.0,** Z: **0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the second point of the line
       - X: **0.5025,** Y: **0.168,** Z:**0.0**, Press the **Enter** key
    3. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **0.168,** Z:**-0.803**, Press the **Enter** key
    4. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **0.324,** Z:**-0.803**, Press the **Enter** key
    5. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **0.324,** Z:**0.0**, Press the **Enter** key
    6. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **0.5,** Z:**0.0**, Press the **Enter** key
    7. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **0.5,** Z:**-0.98**, Press the **Enter** key
    8. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **-0.5,** Z:**-0.98**, Press the **Enter** key
    9. Using the coordinate entry field, enter the next point of the line
       - X: **0.5025,** Y: **-0.5,** Z:**0.0**, Press the **Enter** key
    10. Using the coordinate entry field, enter the next point of the line
        - X: **0.5025,** Y: **-0.324,** Z:**0.0**, Press the **Enter** key
    11. Using the coordinate entry field, enter the next point of the line
        - X: **0.5025,** Y: **-0.324,** Z:**-0.803**, Press the **Enter** key
    12. Using the coordinate entry field, enter the next point of the line
        - X: **0.5025,** Y: **-0.168,** Z:**-0.803**, Press the **Enter** key
    13. Using the coordinate entry field, enter the next point of the line
        - X: **0.5025,** Y: **-0.168,** Z:**0.0**, Press the **Enter** key
    14. Using the coordinate entry field, enter the last point of the line
        - X: **0.5025,** Y: **0.0,** Z:**0.0**, Press the **Enter** key
    15. Press **Enter** to exit
- **Change Attributes**
  - Select the object **Polyline1** from the tree and goto Properties window
    1. Change the name of the object to **Stator**
    2. Change its color to **Orange**

Example (Magnetostatic) – ECE: Linear Movement

- **Sweep Profile**
  - Select the object **Stator** from the history tree
  - Select the menu item ***Draw > Sweep >Along Vector***
    1. Using the coordinate entry field, enter the first point of the vector
       - X: **0.5025,** Y: **0.0,** Z:**0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the last point of the vector (Note that the following coordinates are ABSOLUTE)
       - X: **0.091,** Y: **0.0, Z:0.0**, Press the **Enter** key.

- **Create Mirror**
  - Select the object **Stator** from the history tree
  - Select the menu item ***Edit > Duplicate > Mirror***
    1. Using the coordinate entry field, enter the anchor point of mirror plane
       - X: **0.0,** Y: **0.0,** Z:**0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the target point of vector normal to the mirror plane
       - dX: **-1,** dY: **0.0,** dZ:**0.0**, Press the **Enter** key



Set Default Material

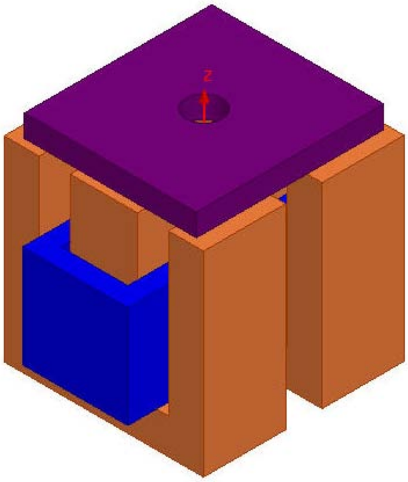
- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **copper** in the **Search by Name** field
    2. Click the **OK** button

Create Coil

- **Create Box**
  - Select the menu item ***Draw >Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.646**, Y: **-0.264**, Z: **-0.803,** Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **1.292**, dY: **0.528**, dZ: **0.511**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Coil**
    2. Change its color to **Blue**
- **Create another Box**
  - Select the menu item ***Draw >Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.55**, Y: **-0.168**, Z: **-0.803,** Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **1.1**, dY: **0.336**, dZ: **0.511**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **BaseCoil**

Example (Magnetostatic) – ECE: Linear Movement

- **Subtract Hole**
  - Press **Ctrl** and select the objects **Coil** and **BaseCoil** from the tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Coil**
    2. Tool Parts: **BaseCoil**
    3. Click the **OK** button



Set Default Material

- **To Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

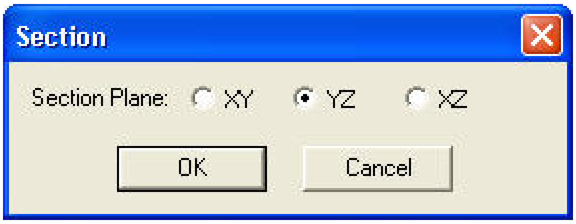
Create Band Region

- **Note:** During parametric sweep, we will very the location of armature, thus moving it in positive or negative Z direction. In such cases, it is advisable to create a Box (Band Region) around the moving object to help mesh creation. (Band Region is usually created with Transient solver to specify Object motion. But it is good to follow same process even for Magnetostatic simulation).
- **To Create Band Region**
  - Select the menu item ***Draw >Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.55,** Y: **-0.55**, Z: **0.003,** Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **1.1,** dY: **1.1,** dZ: **0.147,** Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Band**
    2. Display Wireframe: ☑ **Checked**

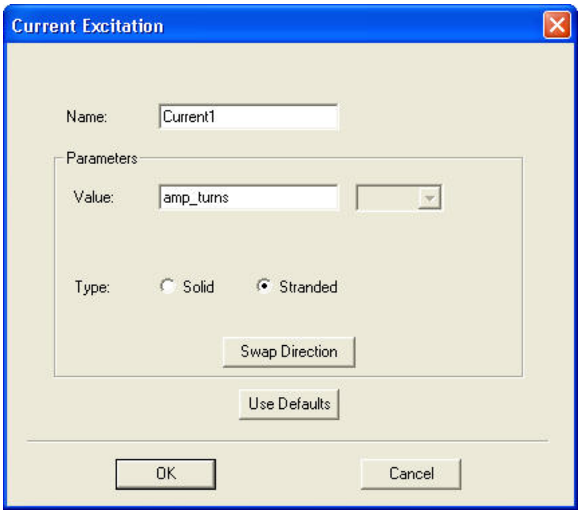
Example (Magnetostatic) – ECE: Linear Movement

Create Excitations

- **Create Section of coil for assigning Current**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button

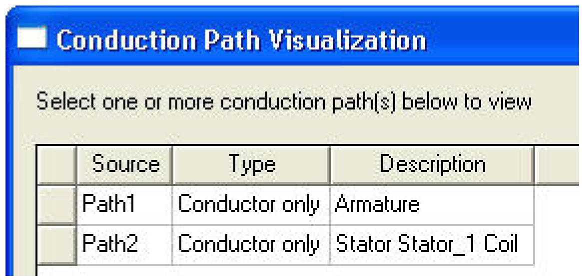


- **Change Attributes**
  - Select the object **Coil_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Coil_Terminal**
- **Separate Sheets**
  - Select the sheet **Coil_Terminal**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Coil_Terminal_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***
- **Assign Excitations**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **amp_turns**
    3. Type: **Stranded**
    4. Press **OK**
  - In Add variable window,
    1. Unit Type: **Current**
    2. Unit: **A**
    3. Value: **576**
    4. Press **OK**



Assign Boundary Conditions

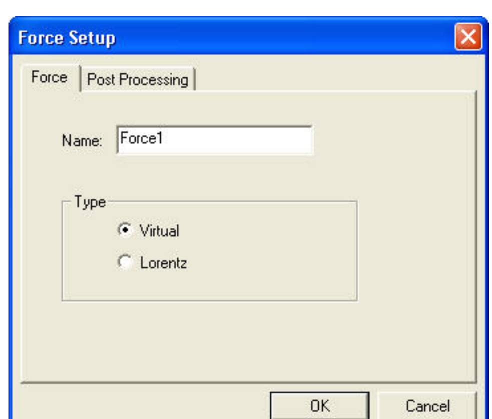
- **Check Conduction path**
  - Select the menu item ***Maxwell 3D > Excitations > Conduction Path > Show Conduction Paths***



- **Note:** We assume that no current leaks out of the coil and into the stators. However, the stators and the coil are touching which forms a conduction path from coil to both stators. Thus an insulating boundary must be applied to the either the coil or to both stators to avoid current leakage.
- **Assign Boundary**
  - Press **Ctrl** and select the objects **Stator** and **Stator_1** from the tree
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Insulating***
  - In Insulating Boundary window,
    1. Press **OK**

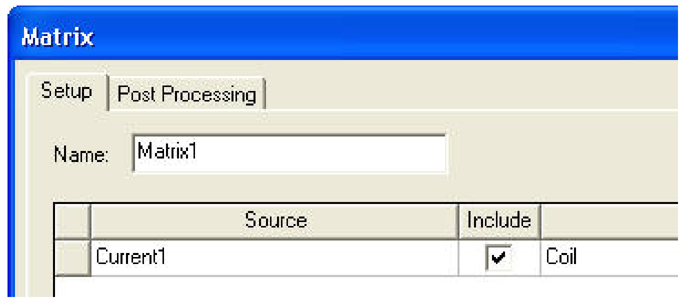
Assign Executive Parameters

- **Assign Force Parameter**
  - Select the object **Armature** from the history tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Force***
  - In Force Setup window,
    1. Name: **Force1**
    2. Type: **Virtual**
    3. Press **OK**



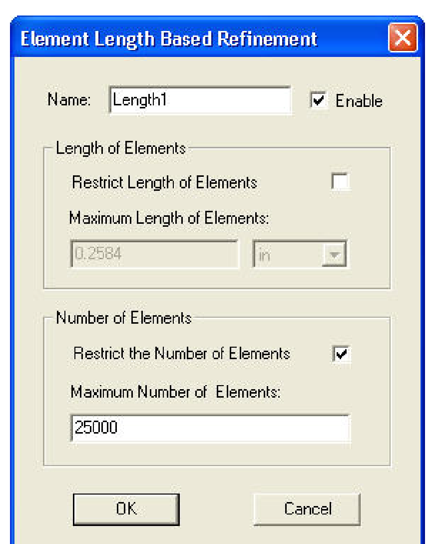
Example (Magnetostatic) – ECE: Linear Movement

- **Assign Matrix**
  - Select menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Solver Tab:
       - Include : ☑ **Checked** for source Current1
    2. Press **OK**



Assign Mesh Operations

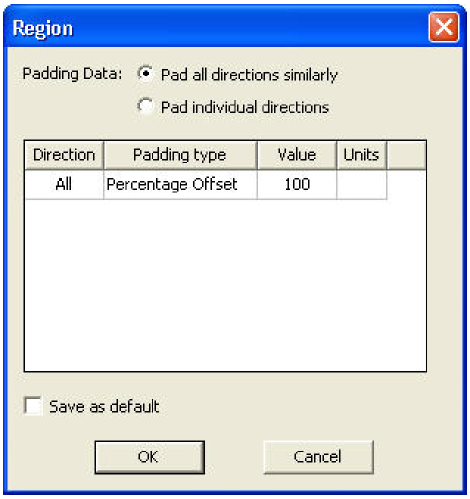
- **To Apply Mesh Operations**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Maxwell 3D > Mesh Operations >Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict Length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **25000**

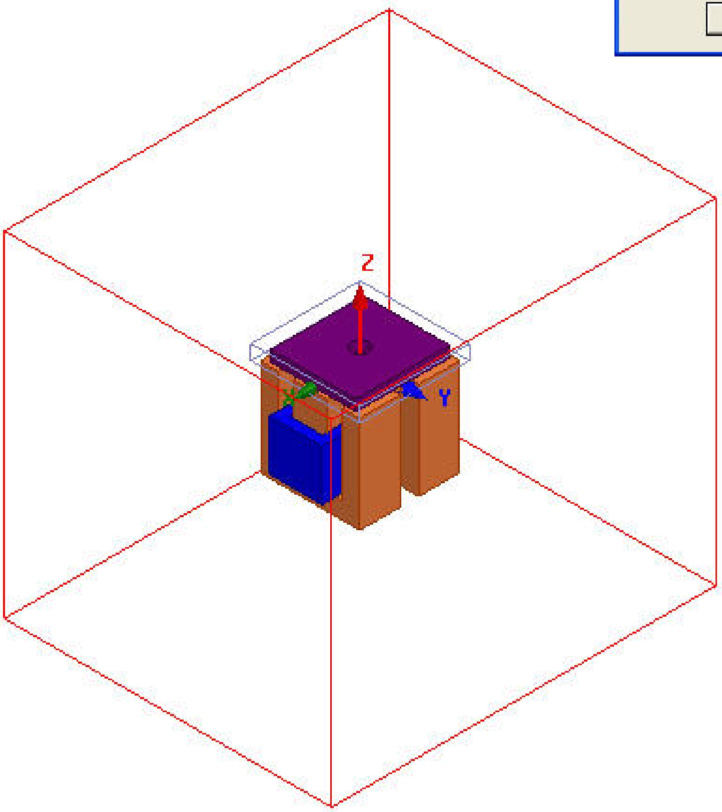


Define Region

Create Simulation Region

- Select the menu item *Draw > Region*
- In Region window,
  1. Pad all directions similarly: ☑ **Checked**
  2. Padding Type: **Percentage Offset**
  3. Value: **100**
  4. Press **OK**





Example (Magnetostatic) – ECE: Linear Movement

Analysis Setup

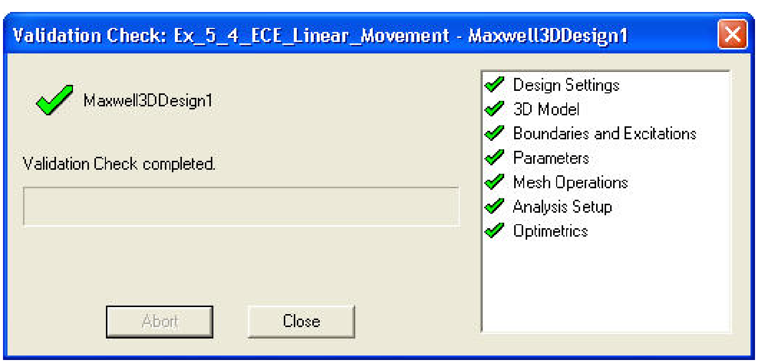
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From **Save As** window, type the Filename: **Ex_5_4_ECE_Linear_Movement**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



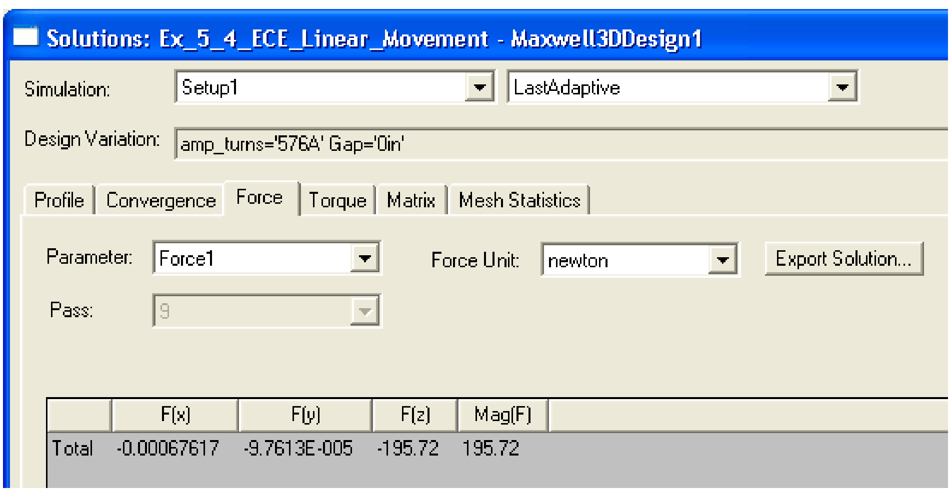
Note: To view any errors or warning messages, use the Message Manager.

Analyze

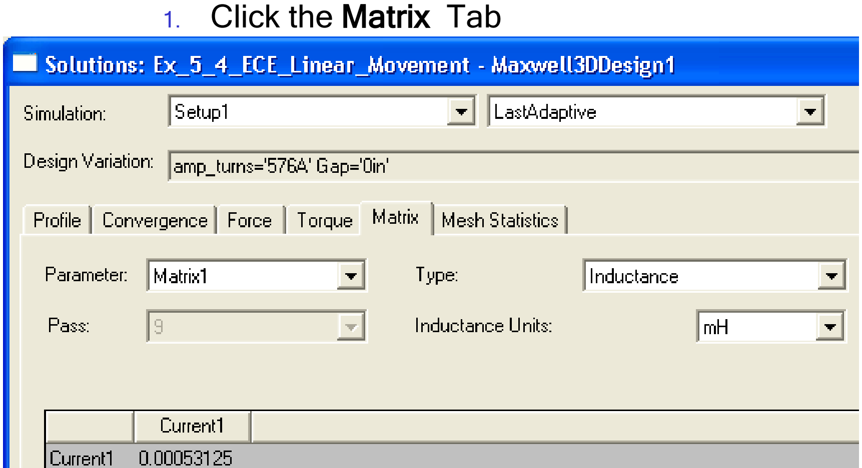
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To view the Force values:
      1. Click the **Force** Tab



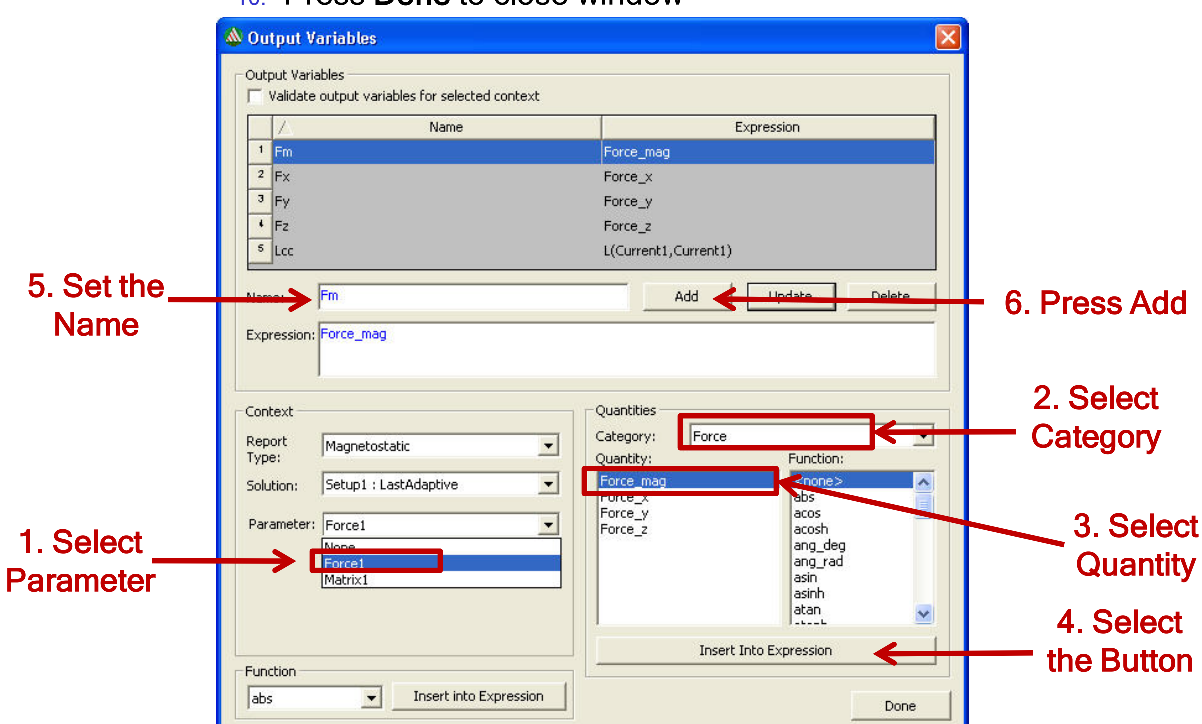
- To View Inductance values
  1. Click the **Matrix** Tab



Example (Magnetostatic) – ECE: Linear Movement

Setup the Parametric Analysis for ECE

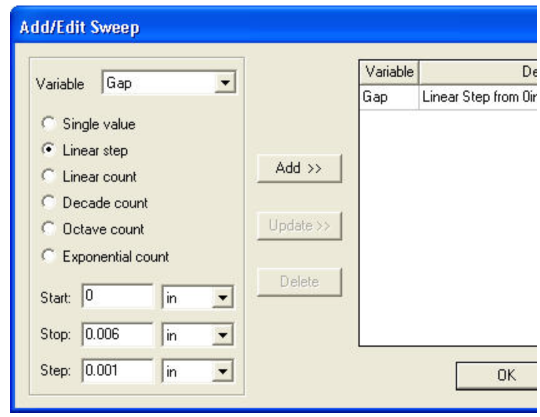
- The ECE model will have two input and two output variables
  1. Input: **amp_turns** and **Gap** (variables already defined)
  2. Output: Armature Force and Coil Inductance (need to create variables)
- **To Create Output Variables**
  - Select the menu item ***Maxwell 3D > Results > Output Variables***
  - In Output Variables window,
    1. Parameter: **Force1**
    2. Category: **Force**
    3. Qunatity: **Force_mag**
    4. Select the buttom **Insert Into Expression**
    5. Specify the name as **Fm** and select **Add**
    6. Similarly create variables **Fx, Fy** and **Fz** for Force_x, Force_y and Force_z respectively
    7. Change the Parameter to **Matrix**
    8. Change Category to **L** and Quantity to **L(Current1, Current1)** and select the button **Insert Into Expression**
    9. Specify the name as **Lcc** and select **Add**
    10. Press **Done** to close window

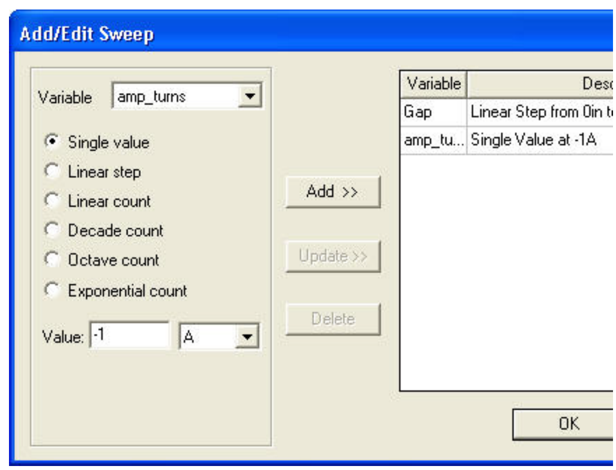


Example (Magnetostatic) – ECE: Linear Movement

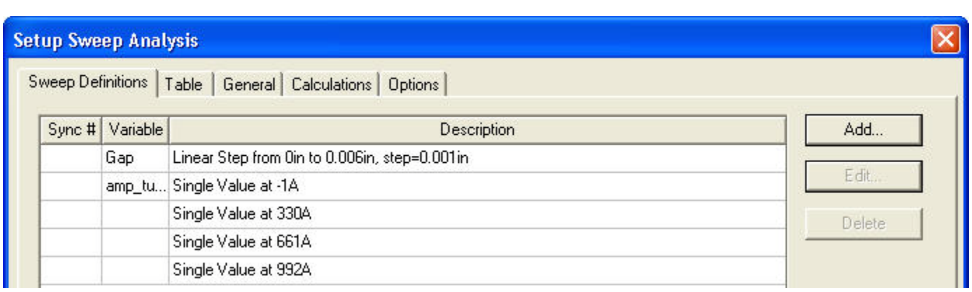
Create Parametric Sweep

- **To Create Parametric Sweep**
  - Select the menu item *Maxwell 3D > Optimetrics Analysis > Add Parametric*
  - In Setup Sweep Analysis window, on the **Sweep Definitions** tab, click the **Add** button
  - Add/Edit Sweep Dialog
    1. Select Variable: **Gap**
    2. Select **Linear Step**
    3. Start: **0**
    4. Stop: **0.006**
    5. Step: **0.001**
    6. Click **Add**
    7. Change Variable to **amp_turns**
    8. Change the option to **Single Value**
    9. Specify Value of **-1**
    10. Click **Add**



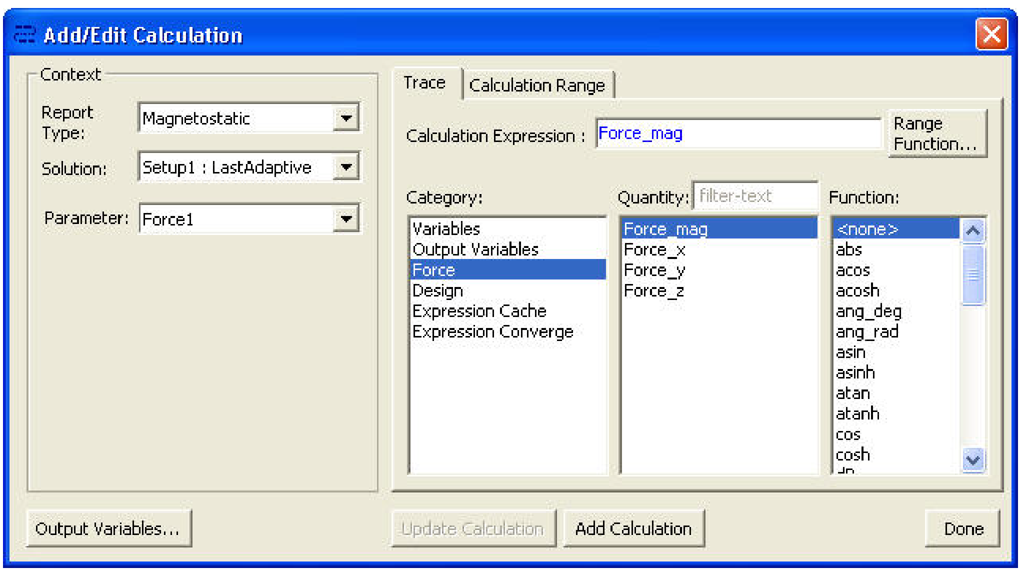


    11. Follow previous three steps again for values of **330A, 661A** and **992 A**
    12. Press **OK** to return to Setup Sweep Analysis window

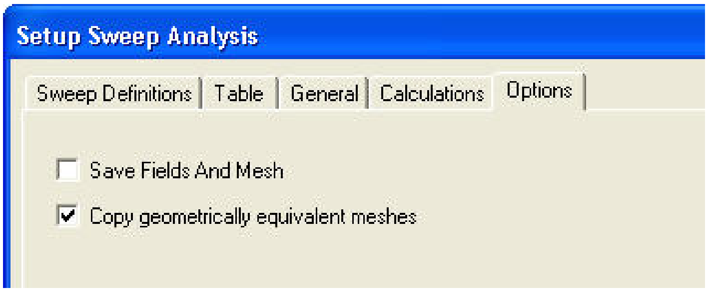


Example (Magnetostatic) – ECE: Linear Movement

- Select **Calculations** tab, select **Setup Calculations**
- In Add/Edit Calculation window,
    1. Parameter: **Force1**
    2. Category: **Output Variables**
    3. Quantity: **Fm**
    4. Click on **Add Calculation**
    5. Repeat same steps for **Fx, Fy** and **Fz**
    6. Change Parameter to **Matrix1**
    7. Category: **Output Variables**
    8. Quantity: **Lcc**
    9. Click on **Add Calculation**
    10. Press **Done**



- On **Options** tab
    1. Copy geometrically equivalent meshes: ☑ **Checked**
- Press **OK** to close window

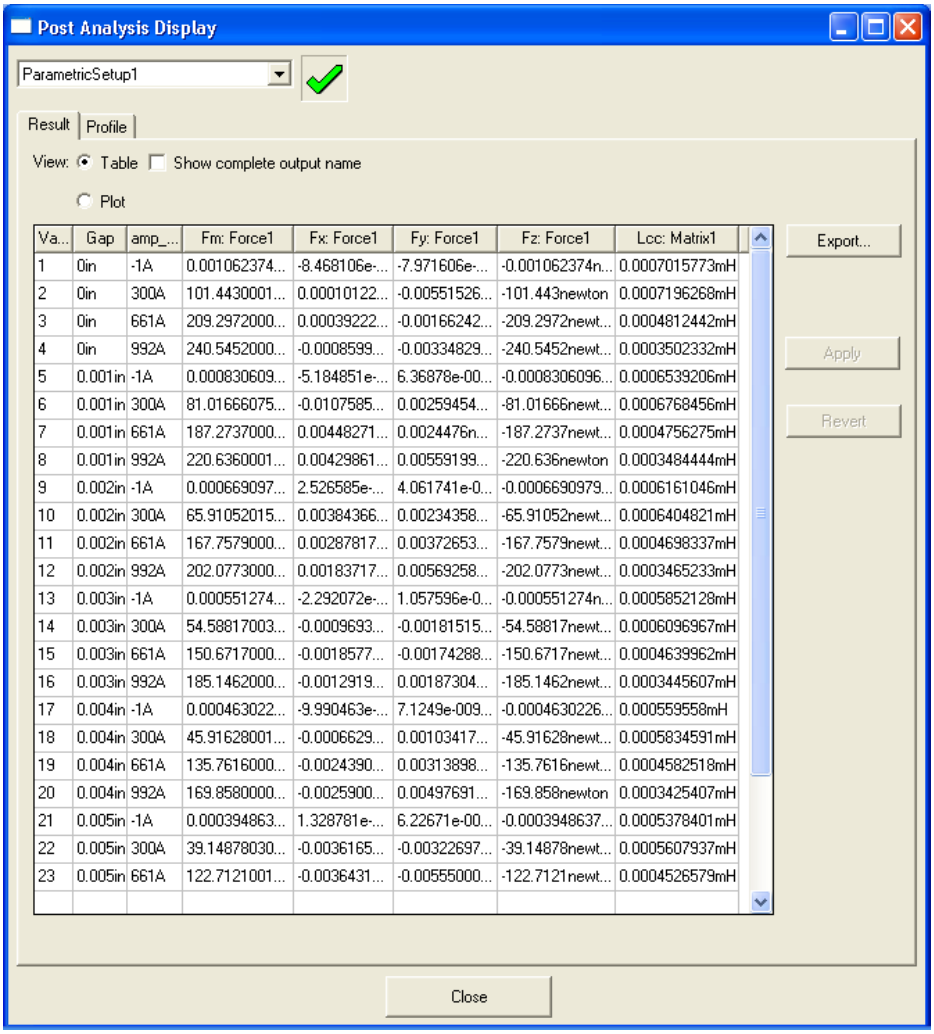


Analyze Parametric Sweep

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

Solution Data for Parametric Sweep

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

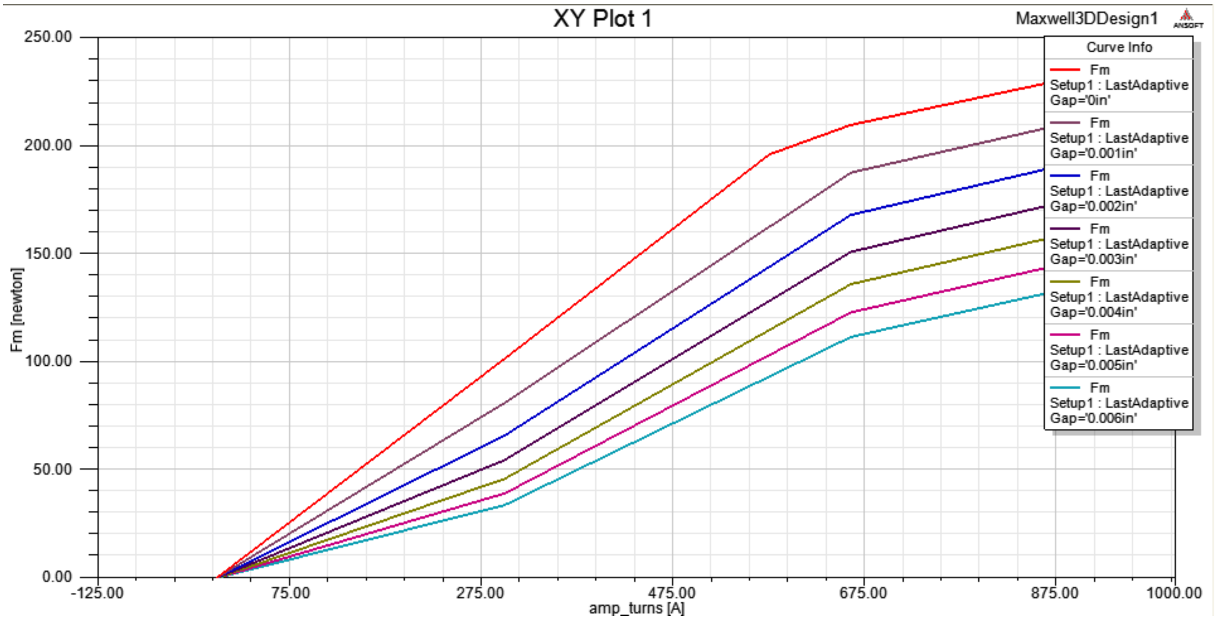


Va...	Gap	amp_...	Fm: Force1	Fx: Force1	Fy: Force1	Fz: Force1	Lcc: Matrix1
1	0in	-1A	0.001062374...	-8.468106e-...	-7.971606e-...	-0.001062374n...	0.0007015773mH
2	0in	300A	101.4430001...	0.00010122...	-0.00551526...	-101.443newton	0.0007196268mH
3	0in	661A	209.2972000...	0.00039222...	-0.00166242...	-209.2972newt...	0.0004812442mH
4	0in	992A	240.5452000...	-0.0008599...	-0.00334829...	-240.5452newt...	0.0003502332mH
5	0.001in	-1A	0.000830609...	-5.184851e-...	6.36878e-00...	-0.0008306096...	0.0006539206mH
6	0.001in	300A	81.01666075...	-0.0107585...	0.00259454...	-81.01666newt...	0.0006768456mH
7	0.001in	661A	187.2737000...	0.00448271...	0.0024476n...	-187.2737newt...	0.0004756275mH
8	0.001in	992A	220.6360001...	0.00429861...	0.00559199...	-220.636newton	0.0003484444mH
9	0.002in	-1A	0.000669097...	2.526585e-...	4.061741e-0...	-0.0006690979...	0.0006161046mH
10	0.002in	300A	65.91052015...	0.00384366...	0.00234358...	-65.91052newt...	0.0006404821mH
11	0.002in	661A	167.7579000...	0.00287817...	0.00372653...	-167.7579newt...	0.0004698337mH
12	0.002in	992A	202.0773000...	0.00183717...	0.00569258...	-202.0773newt...	0.0003465233mH
13	0.003in	-1A	0.000551274...	-2.292072e-...	1.057596e-0...	-0.000551274n...	0.0005852128mH
14	0.003in	300A	54.58817003...	-0.0009693...	-0.00181515...	-54.58817newt...	0.0006096967mH
15	0.003in	661A	150.6717000...	-0.0018577...	-0.00174288...	-150.6717newt...	0.0004639962mH
16	0.003in	992A	185.1462000...	-0.0012919...	0.00187304...	-185.1462newt...	0.0003445607mH
17	0.004in	-1A	0.000463022...	-9.990463e-...	7.1249e-009...	-0.0004630226...	0.000559558mH
18	0.004in	300A	45.91628001...	-0.0006629...	0.00103417...	-45.91628newt...	0.0005834591mH
19	0.004in	661A	135.7616000...	-0.0024390...	0.00313898...	-135.7616newt...	0.0004582518mH
20	0.004in	992A	169.8580000...	-0.0025900...	0.00497691...	-169.858newton	0.0003425407mH
21	0.005in	-1A	0.000394863...	1.328781e-...	6.22671e-00...	-0.0003948637...	0.0005378401mH
22	0.005in	300A	39.14878030...	-0.0036165...	-0.00322697...	-39.14878newt...	0.0005607937mH
23	0.005in	661A	122.7121001...	-0.0036431...	-0.00555000...	-122.7121newt...	0.0004526579mH

Example (Magnetostatic) – ECE: Linear Movement

Create Plot of Force Vs Current for each Gap

- **To create a report:**
  - Select the menu item ***Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot***
  - In Report window,
    1. Solution: **Setup1: LastAdaptive**
    2. Parameter: **Force1**
    3. Primary Sweep: **amp_turns**
    4. X: **Default**
    5. Category: **Output Variables**
    6. Quantity: **Fm**
    7. Press **New Report**
    8. Press **Close**



Save and Exit

- **To Save**
  - Select the menu item ***File > Save***
  - Select the menu item ***File > Exit***

This concludes Maxwell part.

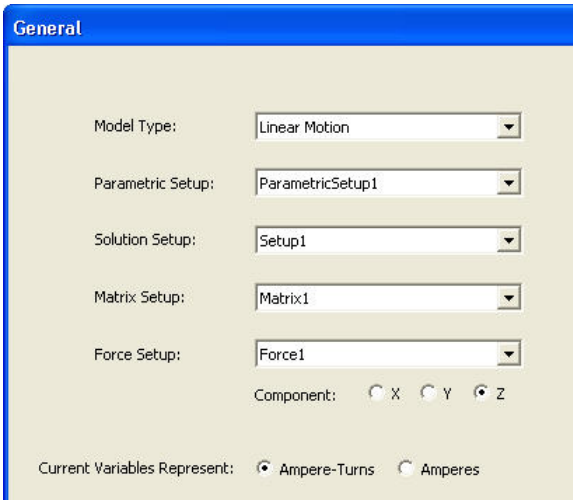
Example (Magnetostatic) – ECE: Linear Movement

ECE from the Parametric Analysis

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

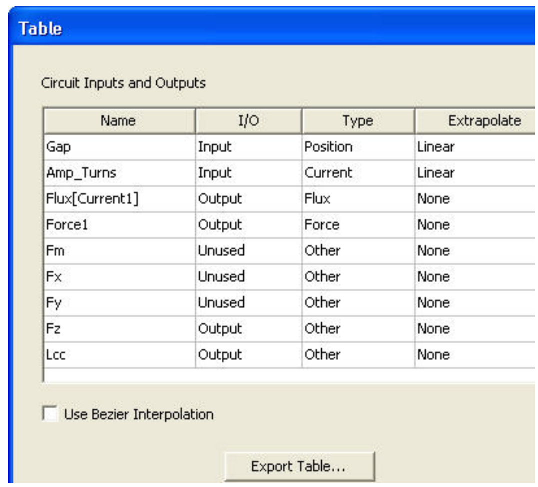
Extract Equivalent Circuit

- **To Extract Equivalent Circuit**
  - Launch Simplorer and in Simplorer, select the menu item ***Simplorer Circuit > SubCircuit > Maxwell Component > Add Equivalent Circuit***
  - In Maxwell Equivalent Circuit Model window,
    1. Source Project File: Browse to the location of saved **Maxwell Project** file and select it
    2. Design Type: Select **3D**
    3. Select the Button **Extract Equivalent Circuit**
  - Maxwell will be launched and source file will open. In Maxwell window,
    1. Ensure Model Type is set to **Linear Motion**
    2. Under Parametric Setup, select the parametric setup for which circuit needs to extracted
    3. Select the component **Z** as only Z component of force will be transferred
    4. Select Current Variable as **Ampere-Turns** as we have specified ampere-turns value for current (as we specified stranded Conductor)
    5. Press **Next**



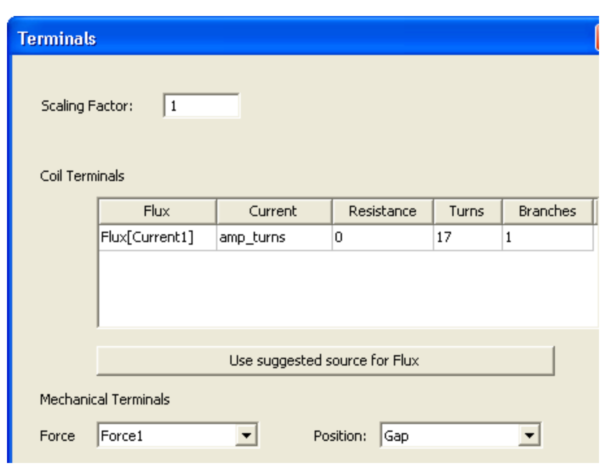
Example (Magnetostatic) – ECE: Linear Movement

- In the next window
  1. As the parameters **Fm, Fx** and **Fy** will not be used in simplorer circuit, change the **I/O** column for these parameters to **Unused**
  2. Make sure that the data types are correct: Gap corresponds to **Position** and Amp_Turns corresponds to **Current**.
  3. Optionally if data needs to be exported in tabular format, select **Export Table** button
  4. Press **Next**



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

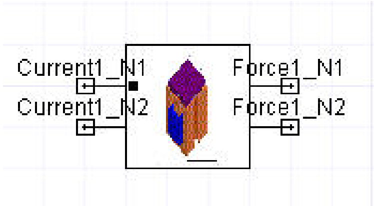
- Next window defines the scaling factor and the Terminals of the conservative nodes in Simplorer. The conservative nodes will have their Across and Through quantities solved by Simplorer, ensuring the physical meaning of the simulation. The Flux (and therefore the EMF) is the electrical Across quantity in this case. The current is the Through quantity. In the Mechanical domain, gap is the Across quantity whereas Force1 is the Through quantity.
- In Maxwell, we specified the conductor as stranded but we have not specified number of Turns. Set Turns column for Terminal as **17** and select **Finish**



Flux	Current	Resistance	Turns	Branches
Flux[Current1]	amp_turns	0	17	1

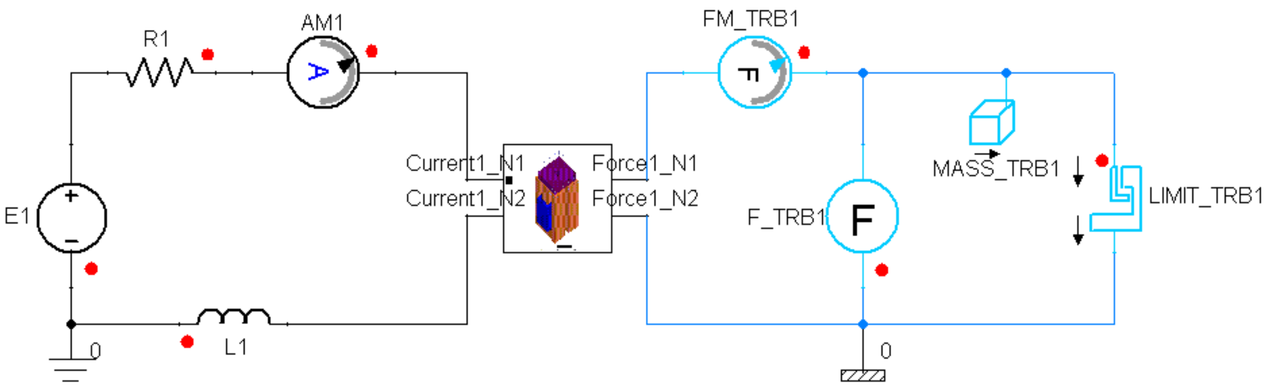
Example (Magnetostatic) – ECE: Linear Movement

- Return to Simplorer window, and select **OK** in the Maxwell Equivalent Circuit Model window
- Place the resulting component on the Project Page

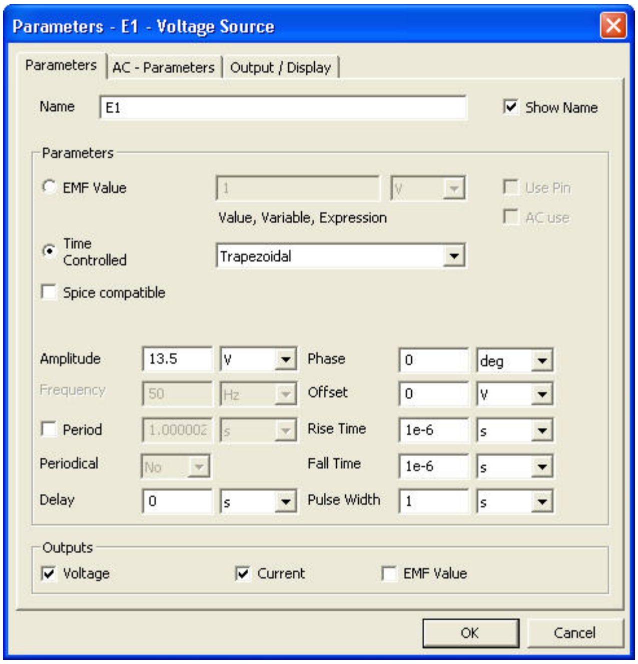


Simplorer Circuit

- **Build Simplorer Circuit**
  - Build the circuit as shown in below image

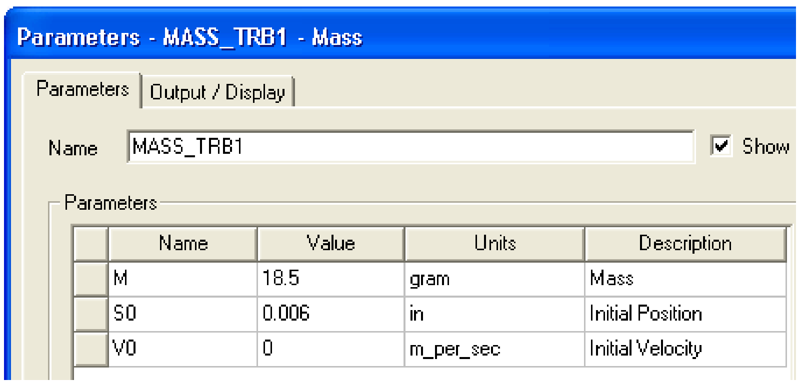


- Specifications of components is as below
  1. Voltage Source **E1**



Example (Magnetostatic) – ECE: Linear Movement

2. Resistor **R1, with R= 215 mOhm**
3. Inductor **L1,** with **L= 10pH**
4. Ammeter **AM1,** for current measurement
5. Force meter **FM_TRB1,** for force measurement
6. Force Source **F_TRB1,** with **F = 125 N**
7. A mass **MASS_TRB1** that describes the mass of the plunger of the actuator


Parameters - MASS_TRB1 - Mass

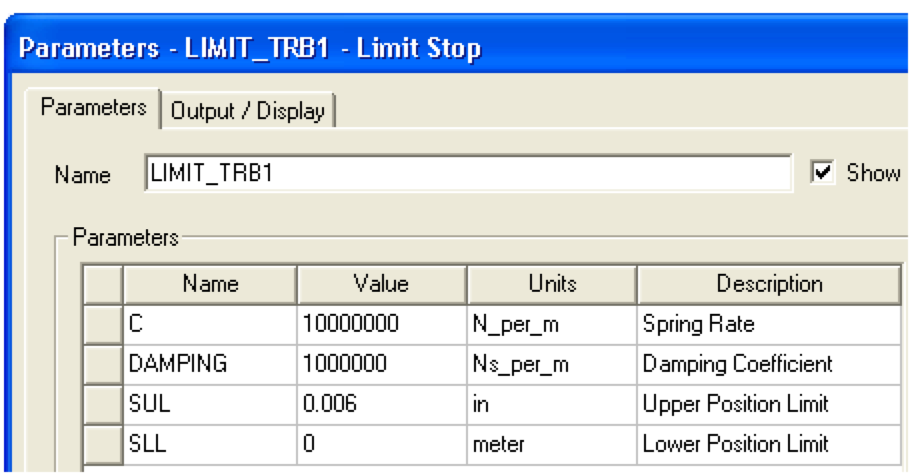
Parameters | Output / Display

Name: MASS_TRB1 ☑ Show

Parameters

Name	Value	Units	Description
M	18.5	gram	Mass
S0	0.006	in	Initial Position
V0	0	m_per_sec	Initial Velocity

8. A Limit stop **LIMIT_TRB1** that represents a parallel damper and spring combination:


Parameters - LIMIT_TRB1 - Limit Stop

Parameters | Output / Display

Name: LIMIT_TRB1 ☑ Show

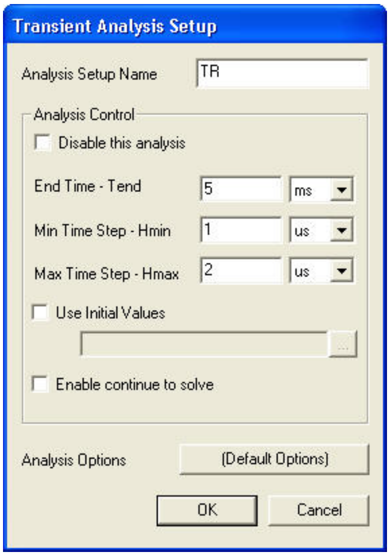
Parameters

Name	Value	Units	Description
C	10000000	N_per_m	Spring Rate
DAMPING	1000000	Ns_per_m	Damping Coefficient
SUL	0.006	in	Upper Position Limit
SLL	0	meter	Lower Position Limit

Example (Magnetostatic) – ECE: Linear Movement

Create Plots

- **Create ECE Parameters for Plot**
  - Double click on ECE component from the project page
  - In the window, goto **Quantities** tab
  - Goto the qunatities **Fz_OUT** and **ICurrent1** and check the box in the colum corresponding to **SDB**
  - Press **OK**
- **Create Plots**
  - Select the menu item ***Simplorer Circuit > Results > Create Standard Report > Rectangular Plot***
  - In Report window,
    1. Category: **All**
    2. Quantity: **ECE1.Fz_OUT**
    3. Press the button **New Report**
    4. Change Quantity to **ECE1.ICurrent1**
    5. Press the button **New Report**
    6. Change the Quantity to **MASS.S**
    7. For Y type **39.79*MASS.S**
    8. Press the button **New Report**
    9. Press the button **Close**



Create a Transient Analysis

- **To Create Transient Setup**
  - Select menu item ***Simplorer Circuit > Edit Setup***
  - In Transient Analysis Setup window,
    1. End Time: **5 ms**
    2. Min. Time Step: **1 us**
    3. Max Time Step: **2 us**
    4. Press **OK**

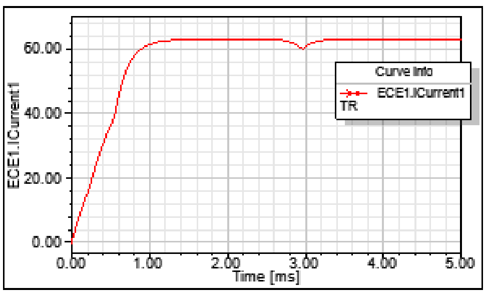
Run Solution

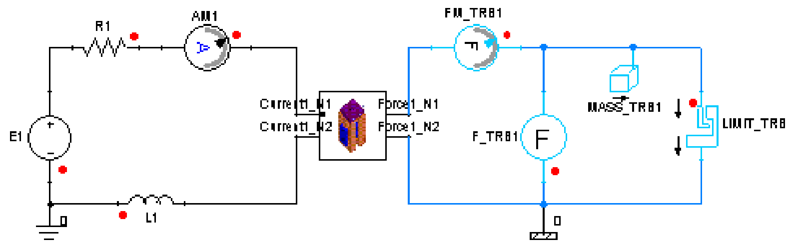
- **To Run Solution**
  - Select the menu item ***Simplorer Circuit > Analyze***

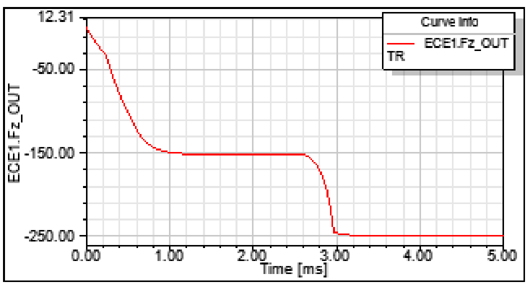
Example (Magnetostatic) – ECE: Linear Movement

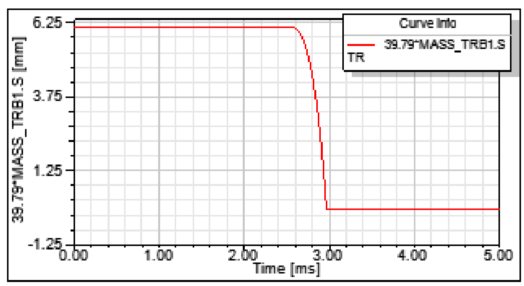
Results

View Result Plots





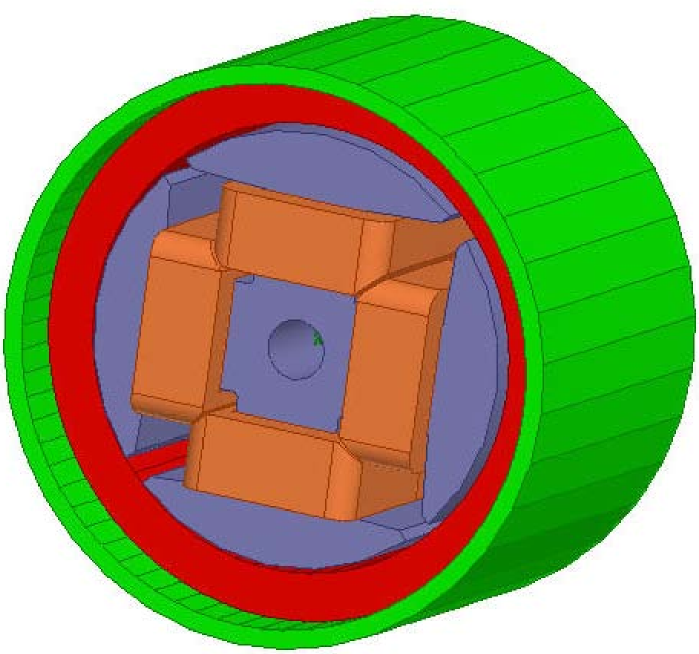




Example (Magnetostatic) – Anisotropic Material

Implementation and Application of Nonlinear Anisotropic Material

- In Maxwell, Nonlinear Anisotropic Material can be used with the Magnetostatic, and Transient Solvers for simulating soft magnetic materials. The advantage is significant because
  - *Non-linear lamination* is extensively used in low frequency electromagnetic devices for having high induction but with significant reduction of eddy current loss in the rolling direction.
  - *Non-linear anisotropy* is widely used magnetization, magnetic recording, power transformers and large size electrical machines. Oriented materials will have higher magnetic field like magnetic flux density in special directions as users want but lower magnetic field in other directions.
- In this note a magnetostatic model as one step of magnetization will be introduced including how to set up Non-linear lamination and Non-linear anisotropic materials.
- The example that will be used to demonstrate how anisotropic non-linear material & non-linear lamination are implemented in a Magnetization model is of a four-poles out-rotor. The intent of this write up is not how to simulate the out-rotor motors, it is rather to demonstrate how anisotropic materials are implemented within the Magnetostatic solver. This magnet ring is an non-linear anisotropic one (nonlinear in R & Z direction with low permeability in Theta direction) and the field core is a laminated one. (lamination direction is Global X) Thus we can show how these materials can be set up for create an four poles out-rotor.



Example (Magnetostatic) – Anisotropic Material

ANSYS Maxwell Design Environment

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

Example (Magnetostatic) - Anisotropic Material

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

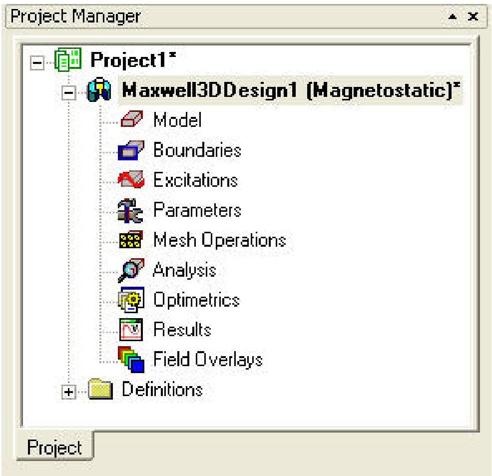
Setting Tool Options

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

Example (Magnetostatic) - Anisotropic Material

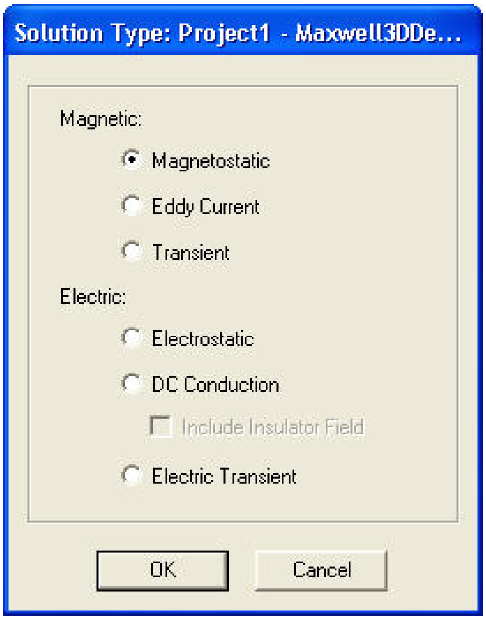
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



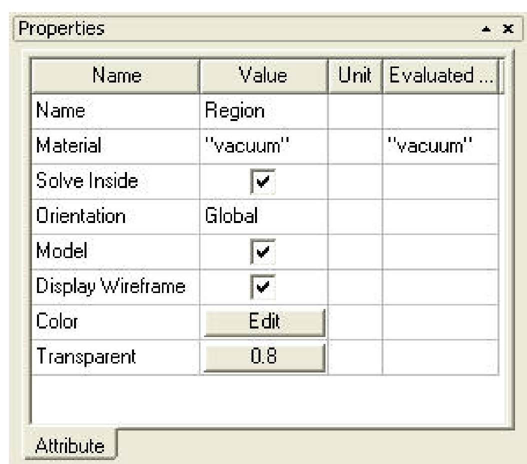
Example (Magnetostatic) - Anisotropic Material

Set Model Units

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

Create Region

- **Create a Box region**
  - Select the menu item *Draw > Box* or click on the icon.
    1. Using the coordinate entry field, enter the box position
       - **X: -50, Y: -50, Z: -50** , Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - **dX: 100, dY: 100, dZ: 100**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Display Wireframe: ☑ **Checked**



- **Hide Region**
  - Select **Region** from the history tree
  - Select the menu item *View > Visibility> Hide Selection > Active View*

Example (Magnetostatic) - Anisotropic Material

Create Field Core

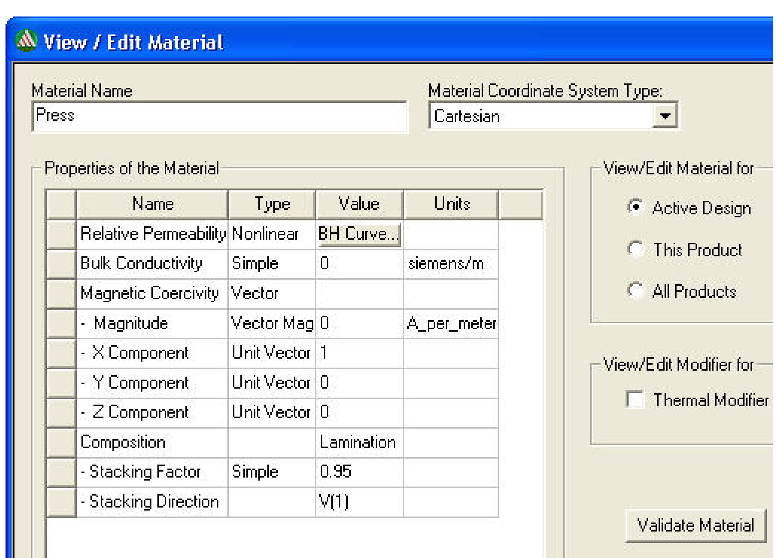
- **To Create Core**
  - Select *Draw > User Defined Primitive > SysLib > RMxprt > SlotCore*
  - In User Defined Primitive Operation window
    1. For the value of **DiaGap,** type: **36.6**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **5**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **22**, Click the **Tab** key to accept
    4. For the value of **Slots**, type: **4**, Click the **Tab** key to accept
    5. For the value of **SlotType**, type: **3**, Click the **Tab** key to accept
    6. For the value of **Hs1**, type: **5**, Click the **Tab** key to accept
    7. For the value of **Hs2**, type: **5.5**, Click the **Tab** key to accept
    8. For the value of **Bs1**, type: **11.5**, Click the **Tab** key to accept
    9. For the value of **Bs2**, type: **0.5**, Click the **Tab** key to accept
    10. Click the **OK** button

Name	Value	Unit	Evaluated Value	Description
DiaGap	36.6	mm	36.6mm	Core diameter on gap si...
DiaYoke	5	mm	5mm	Core diameter on yoke ...
Length	22	mm	22mm	Core length
Skew	0	deg	0deg	Skew angle in core len...
Slots	4		4	Number of slots
SlotType	3		3	Slot type: 1 to 6
Hs0	1	mm	1mm	Slot opening height
Hs01	0	mm	0mm	Slot closed bridge height
Hs1	5	mm	5mm	Slot wedge height
Hs2	5.5	mm	5.5mm	Slot body height
Bs0	2.5	mm	2.5mm	Slot opening width
Bs1	11.5	mm	11.5mm	Slot wedge maximum w...
Bs2	0.5	mm	0.5mm	Slot body bottom width,...
Rs	0	mm	0mm	Slot body bottom fillet

- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **FieldCore**
- **Rotate FieldCore**
  - Select the object **FieldCore** from the history tree
  - Select the menu item *Edit > Arrange > Rotate*
  - In Rotate window,
    1. Set Axis to **Y**
    2. Set Angle to **90**
    3. Press **OK**

Define Material for FieldCore

- **To Create Material**
  - Select the object **FieldCore** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Select option **Add Material**
    2. In View/Edit Material window
       1. Material Name : **Press**
       2. Relative Permeability
          - Change Type to **Nonlinear**
          - Click the **BH Curve** tab that will appear in Value field
          - In BH Curve window,
            1. Select **Import Dataset**
            2. Change File type to **Tab Delimited data file**
            3. Locate the file **Press.tab** and **Open** it
            4. Press **OK** to close BH Curve window
       3. Composition: **Change to Lamination**
          - Stacking Factor: **0.95**
          - Stacking Direction: **V(1)** [X Direction]
       4. Press **Validate Material**
       5. Press **OK**
    3. Press **OK** to exit Select Definition window



Create Housing

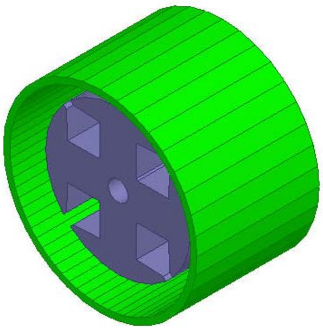
- **Change Grid Plane**
  - Select the menu item *Modeler > Grid Plane > YZ*
- **Create Regular Polyhedra**
  - Select the menu item *Draw > Regular Polyhedron*
    1. Using the coordinate entry field, enter the center position
       - X: **-15.3,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**24.4**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **30.8**, Press the **Enter** key
    4. Number of Segments: **36**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Housing**
    2. Change its color to **Green**
- **Create Housing1**
  - Select the menu item *Draw > Regular Polyhedron*
    1. Using the coordinate entry field, enter the center position
       - X: **-15.3,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**3.25**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **-1.7**, Press the **Enter** key
    4. Number of Segments: **16**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Housing1**

Example (Magnetostatic) – Anisotropic Material

- **Unite Objects**
  - Press **Ctrl** and select the objects **Housing** and **Housing1** from the tree
  - Select the menu item ***Modeler > Boolean > Unite***
- **Create Housing2**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry field, enter the center position
       - X: **-14.5,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**22.8**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **30.8**, Press the **Enter** key
    4. Number of Segments: **72**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Housing2**
- **Create Housing3**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry field, enter the center position
       - X: **0,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**2.45**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **-20**, Press the **Enter** key
    4. Number of Segments: **16**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Housing3**

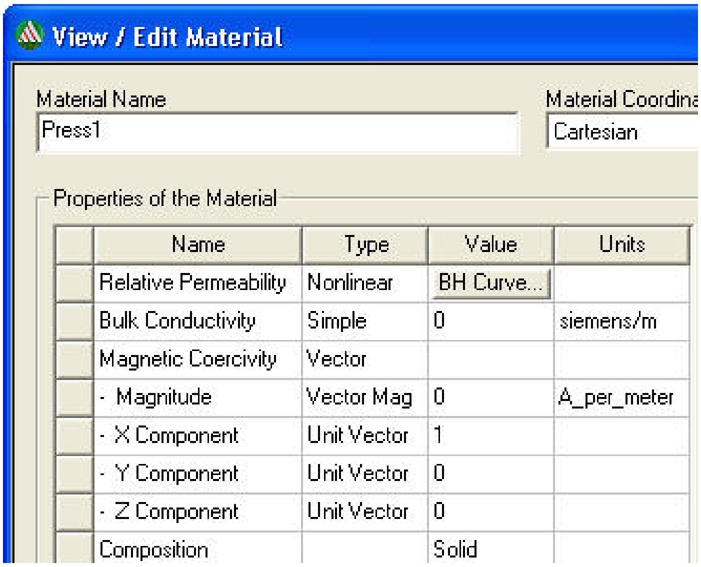
Example (Magnetostatic) - Anisotropic Material

- **Subtract Objects**
  - Press **Ctrl** and select objects **Housing, Housing2** and **Housing3** from tree
  - Select the menu it ***Modeler > Boolean > Subtract***
  - In Subtract window
    1. Set **Housing** as Blank Part
    2. Set **Housing2** and **Housing3** as Tool Parts
    3. Press **OK**



Define Material for Housing

- **To Create Material**
  - Select the object **Housing** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **Press** in the **Search by Name** field
    2. Select option **Clone Material**
    3. In View/Edit Material window
       1. Material Name : **Press1**
       2. Composition: **Change to Solid**
       3. Press **Validate Material**
       4. Press **OK**
    4. Press **OK** to exit Select Definition window



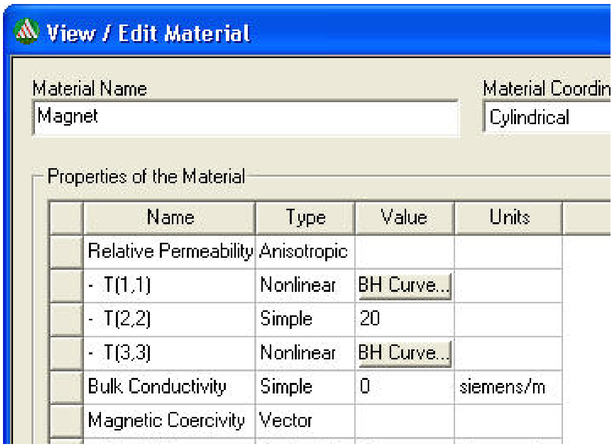
Name	Type	Value	Units
Relative Permeability	Nonlinear	BH Curve...	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_meter
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Composition		Solid	

Create RingMagnet

- **Create Regular Polyhedra**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry field, enter the center position
       - X: **-10,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**22.8**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **20**, Press the **Enter** key
    4. Number of Segments: **72**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **RingMagnet** and color to **Red**
- **Create Hole**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry field, enter the center position
       - X: **-10,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius
       - dZ:**18.8**, Press the **Enter** key
    3. Using the coordinate entry field, enter the height
       - dX: **20**, Press the **Enter** key
    4. Number of Segments: **72**
    5. Press **OK**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Hole**
- **Subtract Objects**
  - Press **Ctrl** and select objects **RingMagnet** and **Hole** from tree
  - Select the menu it ***Modeler > Boolean > Subtract***
  - In Subtract window
    1. Set **RingMagnet** as Blank Part
    2. Set **Hole** as Tool Parts
    3. Press **OK**

Define Material for RingMagnet

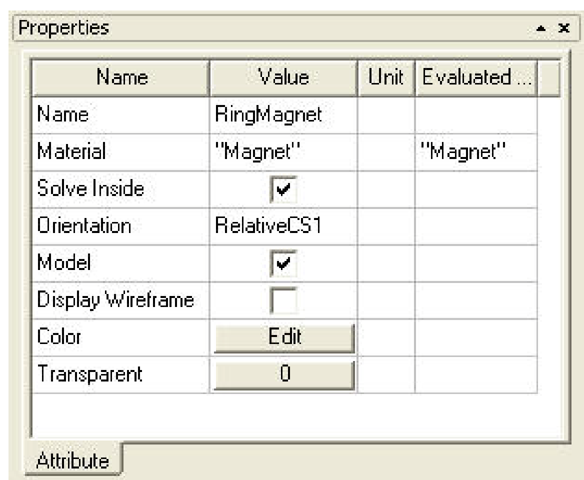
- **To Create Material**
  - Select the object **RingMagnet** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Select option **Add Material**
    2. In View/Edit Material window
       1. Material Name : **Magnet**
       2. Material Coordinate System Type: **Cylindrical**
       3. Relative Permeability
          - Change Type to **Anisotropic**
          - Change T(1,1) to **Nonlinear**
          - Click the **BH Curve** tab that will appear in Value field
          - In BH Curve window,
            1. Select **Import Dataset**
            2. Change File type to **Tab Delimited data file**
            3. Locate the file **Mag.tab** and **Open** it
            4. Press **OK** to close BH Curve window
          - Change value for T(2,2) to **20**
          - Change T(3,3) to **Nonlinear**
          - Click the **BH Curve** tab that will appear in Value field
          - In BH Curve window,
            1. Import the same **Mag.tab** file
       4. Press **OK**
    3. Press **OK** to exit



Orient RingMagnet

- **Note:** Material created for RingMagnet is with reference to cylindrical coordinate system. In Cylindrical Coordinate System, Theta direction is fixed as T(2,2), the Global CS can't be used for RingMagnet directly as its orientation is not along required direction. **Relative Coordinate System** must be created for it.
- **To Create Relative Coordinate System**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Rotated***
    1. Using the coordinate entry field, enter the X axis position
       - X: **0,** Y: **1,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the XY Plane position in Absolute values
       - X: **0,** Y: **0,** Z: **1**, Press the **Enter** key

- **Change Orientation of Magnet**
  - Select **RingMagnet** from the tree and goto Properties window
  - Change Orientation to **RelativeCS1**



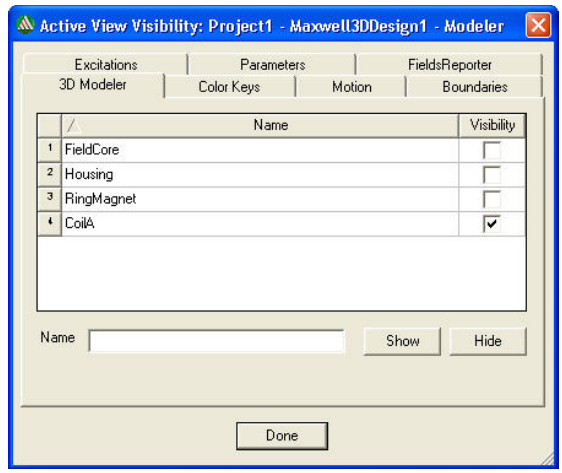
Example (Magnetostatic) – Anisotropic Material

Create Coil

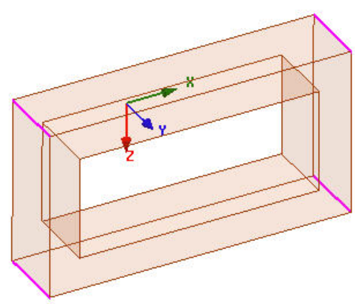
- **Change Work Coordinate System**
  - Goto ***Modeler > Coordinate System > Set Working CS***
  - In Select Coordinate System Window
    1. Select **Global**
    2. Press **Select**
- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **14.2**, Y: **6.2**, Z: **-7.8**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **-28.2**, dY: **6.5**, dZ: **15.6**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **CoilA**
    2. Change its color to **Orange**
- **Create CoilA_1**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **11.2**, Y: **6.2**, Z: **-4.8**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **-22.4**, dY: **6.5**, dZ: **9.6**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **CoilA_1**
- **Subtract Objects**
  - Press **Ctrl** and select objects **CoilA** and **CoilA_1** from tree
  - Select the menu it ***Modeler > Boolean > Subtract***
  - In Subtract window
    1. Set **CoilA** as Blank Part
    2. Set **CoilA_1** as Tool Parts
    3. Press **OK**

Example (Magnetostatic) - Anisotropic Material

- **Hide All other Objects**
  - Select the menu item ***View > Visibility > Active View Visibility***
  - In Active View Visibility window
    1. Change the visibility for all the objects to ☐**Unchecked**
    2. Keep the visibility for CoilA ☑**Checked**
    3. Press **Done**



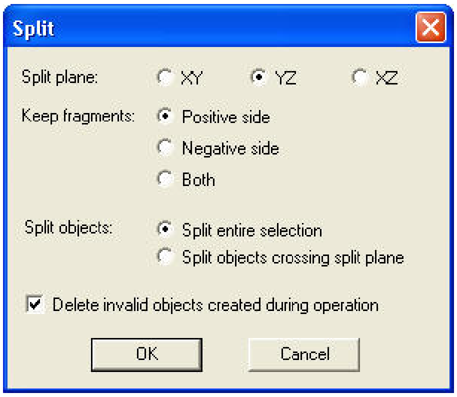
- **Add Fillets**
  - Select the menu item ***Edit > Select >Edges*** or press the key **E** from keyboard
  - Press **Ctrl** and select edges of coil as shown in image
  - Select the menu item ***Modeler > Fillet***
  - In Fillet Properties window,
    1. Set Fillet Radius to **2**
    2. Press **OK**



- **Create Coordinate System to Split Coil**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Both***
    1. Using the coordinate entry fields, enter the origin
       - X: **0**, Y: **0.2**, Z: **0.2**, Press the **Enter** key
    2. Using the coordinate entry field, enter the X axis position
       - dX: **0,** dY: **1,** dZ: **-1**, Press the **Enter** key
    3. Using the coordinate entry field, enter the XY Plane position
       - dX: **0,** dY: **1,** dZ: **1**, Press the **Enter** key

Example (Magnetostatic) - Anisotropic Material

- **Split Coil**
  - Select the menu item ***Edit > Select >Objects*** or press the key **O** from keyboard to change selection to Objects
  - Select the object **CoilA** from the tree
  - Select the menu item ***Modeler > Boolean > Split***
  - In Split window,
    1. Split Plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split Objects: **Split entire selection**
    4. Press **OK**



  - Similarly perform Split with **XZ** Plane
    1. Split Plane: **XZ**
    2. Keep fragments: **Positive side**
    3. Split Objects: **Split entire selection**
    4. Press **OK**
  - **Note:** Ensure RelativeCS2 its work Coordinate system while performing Split operation

Assign Material to Coil

- **To assign material:**
  - Select the object **Coil** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **copper** in the **Search by Name** field
    2. Click the **OK** button

Create Coil terminal

- **Change Work Coordinate System**
  - Goto ***Modeler > Coordinate System > Set Working CS***
  - In Select Coordinate System Window
    1. Select **Global**
    2. Press **Select**

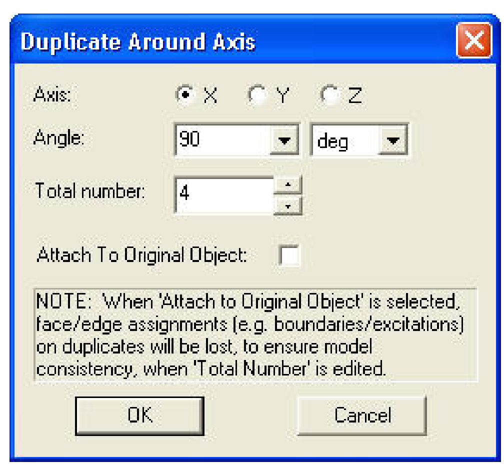
Example (Magnetostatic) - Anisotropic Material

- **Create Coil Terminal**
  - Select the object **CoilA** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button

- **Change Attributes**
  - Select the object **CoilA_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Terminal_A**
- **Separate Sheets**
  - Select the sheet **Terminal_A**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Terminal_A_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***

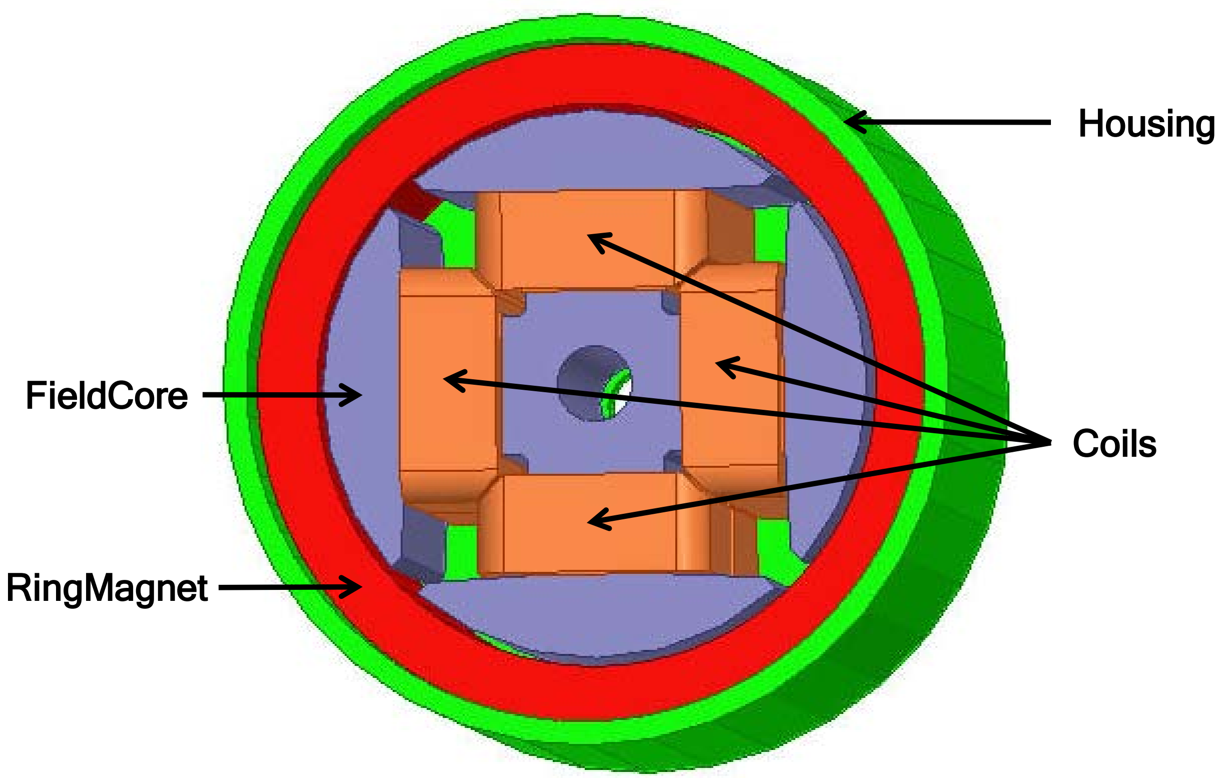
Duplicate Coil and Terminal

- **To Duplicate Coil and Terminal**
  - Press **Ctrl** and select the objects **CoilA** and **Terminal_A**
  - Select the menu item ***Edit > Duplicate > Around Axis***
  - In Duplicate Around Axis window
    1. Axis: **X**
    2. Angle: **90**
    3. Total number : **4**
    4. Press **OK**



Example (Magnetostatic) - Anisotropic Material

- **Change Attributes**
  - Select the objects from the tree and goto Properties window
    1. Change the name of the objects **CoilA_2, CoilA_3** and **CoilA_4** to **CoilB, CoilC** and **CoilD**
  - Select the sheets from the tree and goto Properties window,
    1. Change the name of the sheets **Terminal_A_1, Terminal_A_2** and **Terminal_A_3** to **Terminal_B, Terminal_C** and **Terminal_D**
- **View all Objects**
  - Select the menu item ***View > Visibility > Show All > Active View***
  - Select **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***



Example (Magnetostatic) - Anisotropic Material

Specify Excitations

- **Assign Excitations for CoilA and CoilC**
  - Select the sheet **Terminal_A** and **Terminal_C** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current_A_C**
    2. Value: **4000**
    3. Type: **Stranded**
    4. Current Direction: **negative X** (Use **Swap Direction** if needed)
    5. Press **OK**
- **Assign Excitations for CoilB and CoilD**
  - Select the sheet **Terminal_B** and **Terminal_D** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current_B_D**
    2. Value: **4000**
    3. Type: **Stranded**
    4. Current Direction: **positive X** (Use **Swap Direction** if needed)
    5. Press **OK**

Analysis Setup

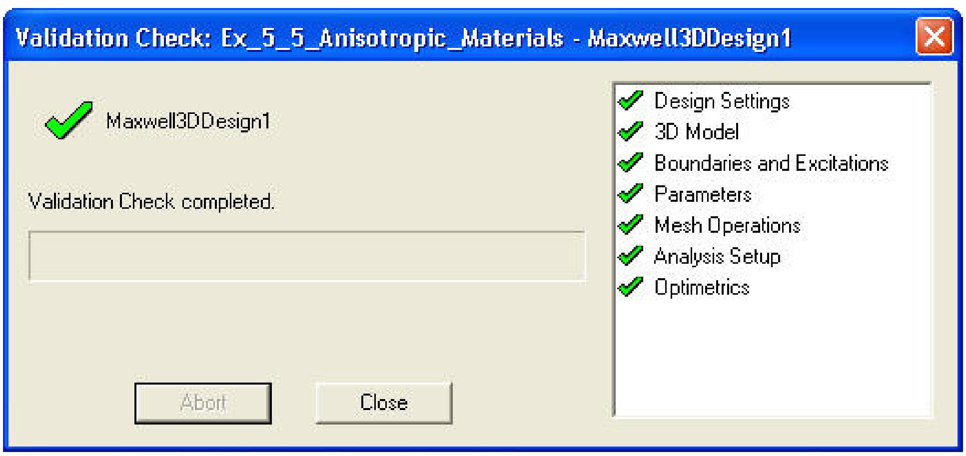
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. General Tab
       - Percentage Error: **5**
    2. Convergence Tab:
       - Refinement Per Pass: **30**
    3. Solver Tab
       - Nonlinear Residuals : **0.01**
    4. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **Ex_5_5_Anisotropic_Materials**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - 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 3D > Analyze All***

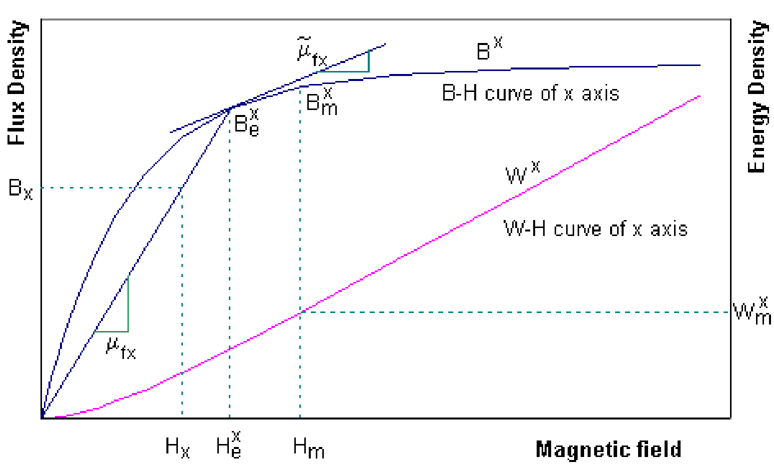
Theory of the Nonlinear Anisotropic Material for soft magnetic materials

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

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

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

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



Example (Magnetostatic) - Anisotropic Material

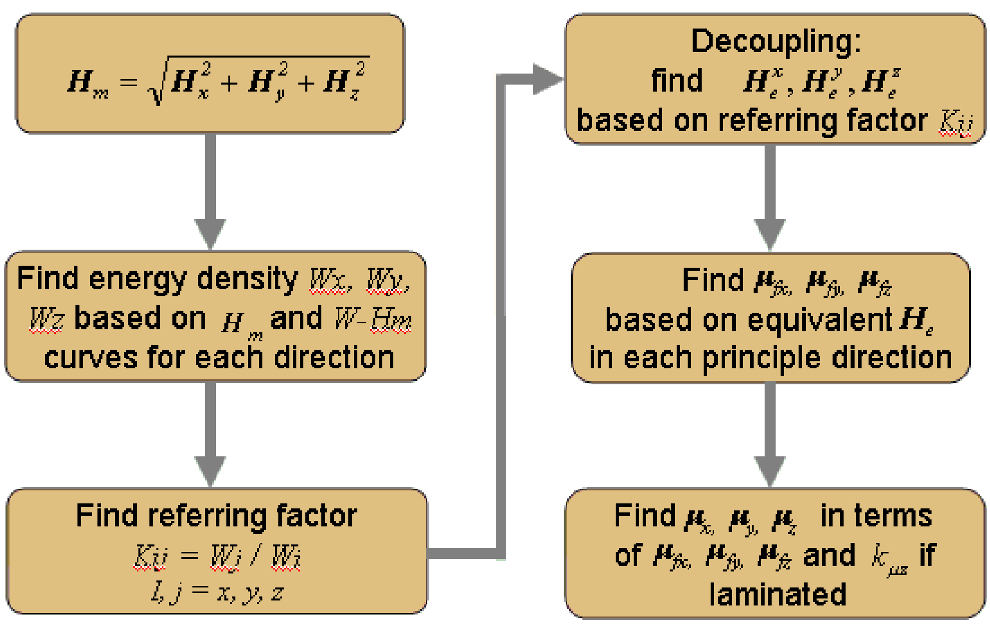
- Taking the lamination into account, the three components of permeability, based on the derived permeability (that is, μ_{fx}, μ_{fy}, μ_{fz}) in iron part, become

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

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

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

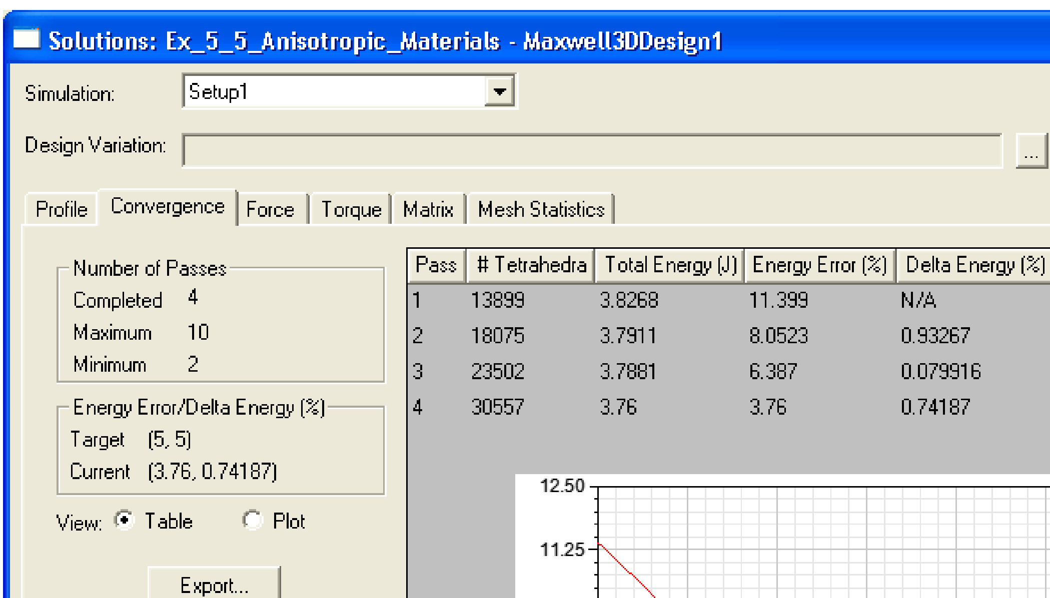
- **The Modeling Sequence will be as follows:**

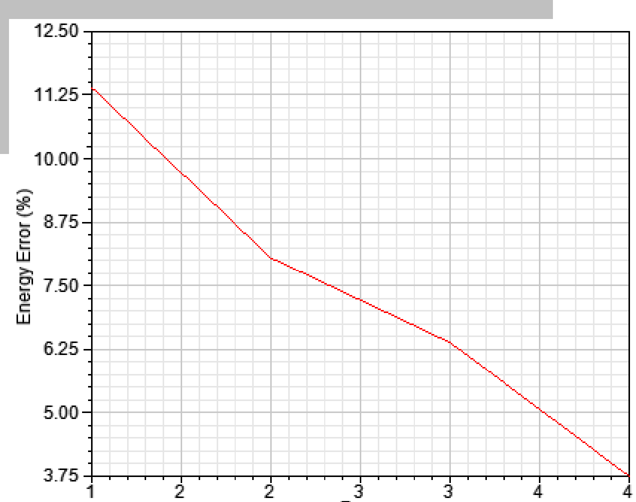


Reference:
[1] D. Lin, P. Zhou, Z. Badics, W. N. Fu, Q. M. Chen and Z. J. Cendes, "**A New Nonlinear Anisotropic Model for Soft Magnetic Materials**", **CSY0084 of COMPUMAG'2005**

Solution Data

- **To view the Solution Data:**
  - Select the menu item *Maxwell 3D > Results > Solution Data*
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To View Mesh information
      1. Click **Mesh Statistics** Tab
    - Select **Close** to close the window

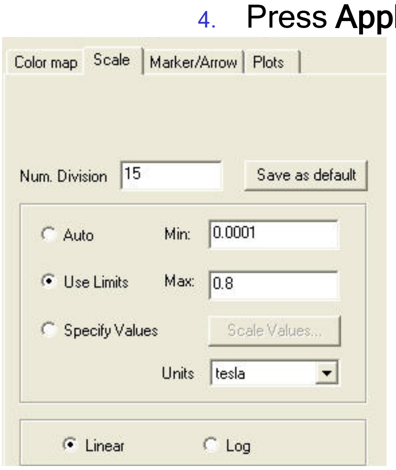


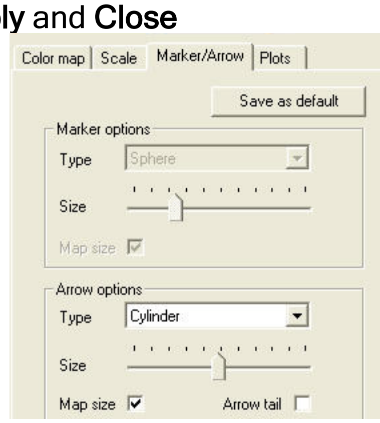


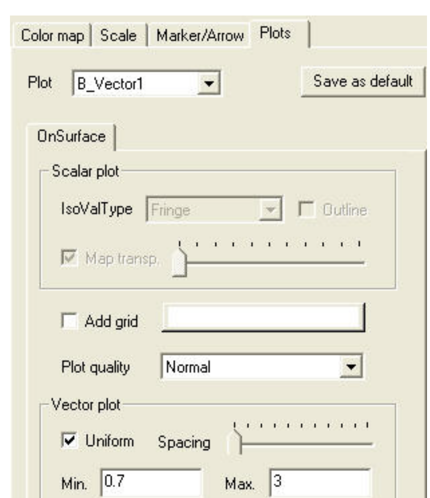
Example (Magnetostatic) - Anisotropic Material

Vector Plot

- **Create Vector Plot**
  - Select **Global:YZ** plane from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > B > B_Vector***
  - In Create Field Plot window
    1. In Volume: **RingMagnet**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - User Limits: ☑ **Checked**
         1. Min: **0.0001**
         2. Max: **0.8**
    2. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Six spaces from left**
         2. Arrow tail: ☐ **Unchecked**
    3. **Plots** tab
       - Vector Plot
         1. Spacing: **Set to Minimum**
         2. Min: **0.7**
         3. Max: **3**
    4. Press **Apply** and **Close**

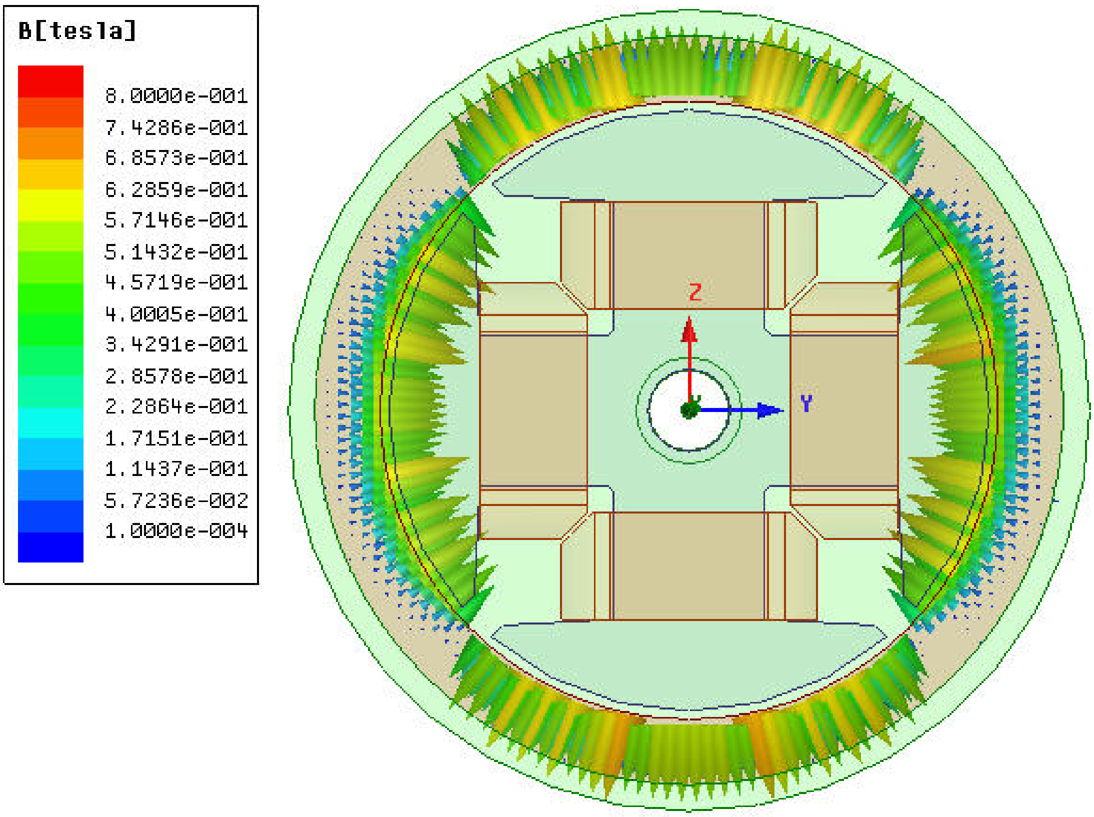






Example (Magnetostatic) - Anisotropic Material

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

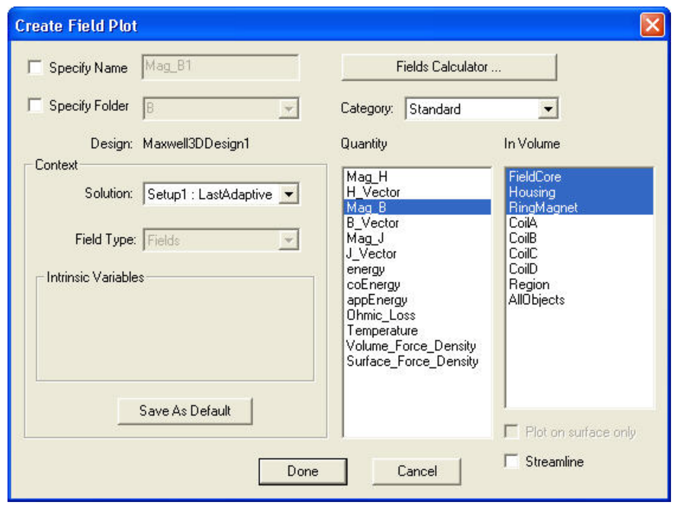


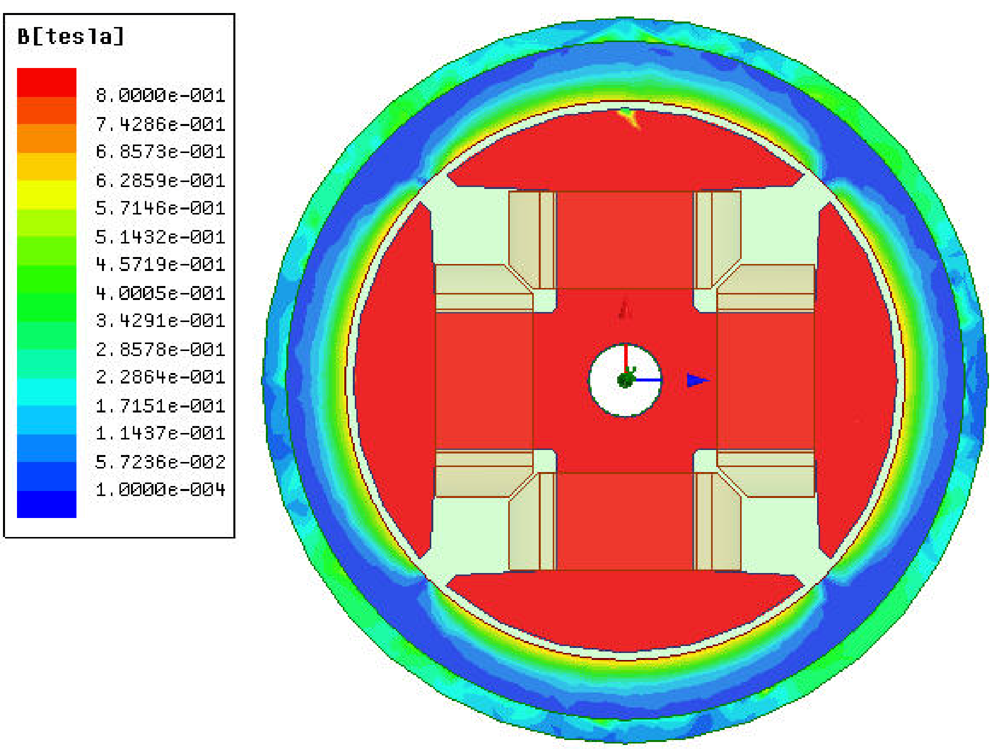
- **Note:** Please note that if the RingMagnet is not set up as anisotropic one (i.e. uniform Nonlinear one) the B field between two magnetic poles will be very large in the theta directions, which will be not as good as user wanted.

Example (Magnetostatic) - Anisotropic Material

Plot Flux Density Contours

- **To Plot contours of Magnetic Flux Density**
  - Select **Global:YZ** plane from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window
    1. In Volume: Press **Ctrl** and select objects **FieldCore, Housing** and **RingMagnet**
    2. Select **Done**



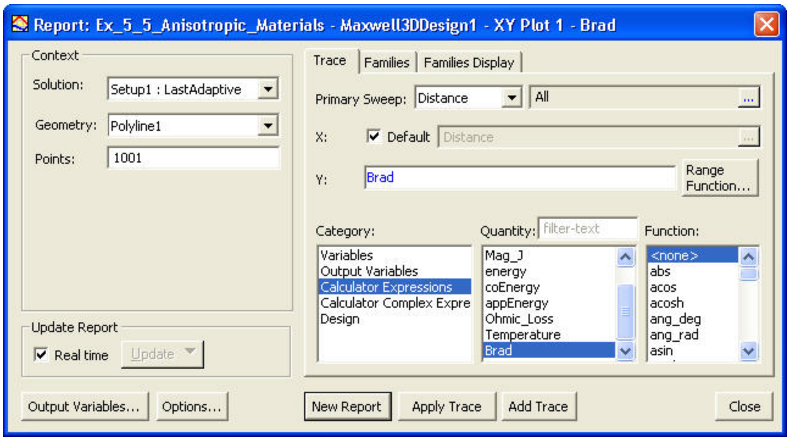


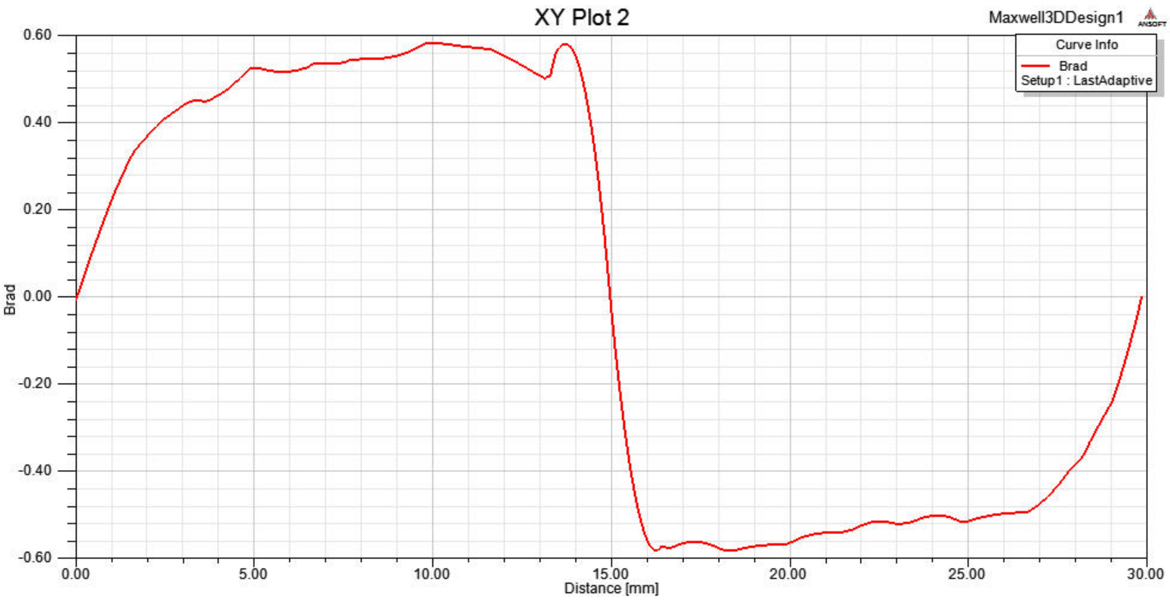
Create Rectangular Plot

- **Create Arc for Rectangular plot**
  - Select menu item ***Draw > Arc > 3 Point***
  - A massage will pop up asking if the entity needs to be created as non model object. Select **Yes** to it.
    1. Using Co-ordinate entry field Enter the first end point
       - **X = 0, Y = 13.435, Z = 13.435,** Press **Enter**
    2. Using Co-ordinate entry field Enter the arc intercept (middle point)
       - **X = 0, Y = 0, Z = 19,** Press **Enter**
    3. Using Co-ordinate entry field Enter the second end point
       - **X = 0, Y = -13.435, Z = 13.435,** Press **Enter**
    4. Press **Enter** to exit
- **Create parameter for radial magnetic filed strength**
  - Goto ***Maxwell 3D > Fileds > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **General > Smooth**
    3. Select **Vector > Scal? > ScalarX**
    4. Select **Input > Function > Scalar > PHI**
    5. Select **Scalar > Trig > Cos**
    6. Select **General > ***
    7. Select **Input > Quantity > B**
    8. Select **General > Smooth**
    9. Select **Vector > Scal? > ScalarY**
    10. Select **Input > Function > Scalar > PHI**
    11. Select **Scalar > Trig > Sin**
    12. Select **General > ***
    13. Select **General > +**
    14. Select **Add** and set the name of expression as **Brad**
    15. Press **Done**

Example (Magnetostatic) - Anisotropic Material

- **To Create Plot:**
  - Select ***Maxwell 3D > Results > Create Fields Report > Rectangular Plot***
  - In Reports window
    1. Geometry: **Polyline1**
    2. **Trace** Tab
       1. X axis: **Default**
       2. Y axis
          - Category: **Calculator Expressions**
          - Quantity: **Brad**
          - Function: **None**
    3. Select **New Report**



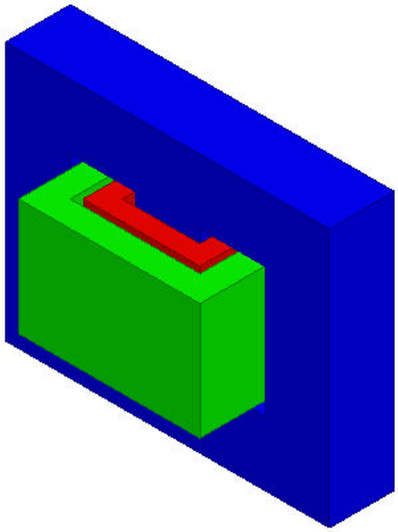


Example (Magnetostatic) – Symmetry Boundaries

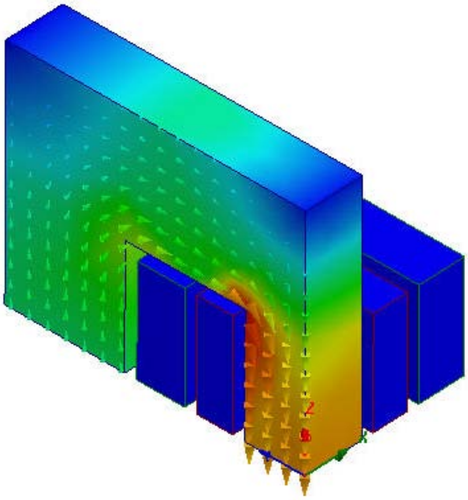
The application of Symmetry Boundaries

- There are many advantages to using Symmetry Boundaries when modeling. Meshes can be decreased greatly by using Symmetry Boundaries when you model the shape of an analytical object . As a result, the saving of the calculation resource and shortening analytical time become possible.
- In this note, the model of very simple transformer will be analyzed using Magnetostatic solver. At this time, the model size is reduced by using the Symmetry boundaries. The intent of this write up is not how to simulate the Transformer, it is rather to demonstrate how Symmetry boundaries is implemented within the Magnetostatic solver. A very simple transformer that rolls the winding of Coil_In and Coil_Out in core is used.

Complete Geometry



Symmetry Model



Example (Magnetostatic) – Symmetry Boundaries

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Line, Rectangle, Box**
    - Surface Operations: **Section, Sweep Along Path**
    - Boolean Operations: **Subtract, Separate Bodies, Split, Duplicate (Mirror)**
  - Boundaries/Excitations
    - Current: **Stranded**
    - **Symmetry**
  - Analysis
    - **Magnetostatic**
  - Results
    - **Solution Data**
  - Field Overlays:
    - **Flux Density Plots**

Example (Magnetostatic) – Symmetry Boundaries

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

Setting Tool Options

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

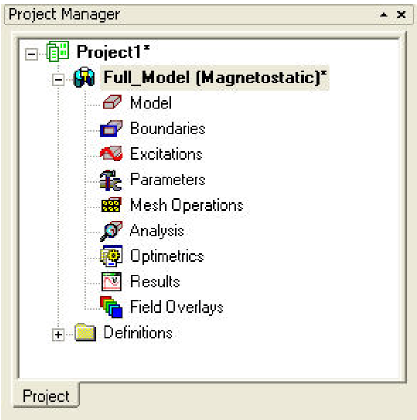
Example (Magnetostatic) - Symmetry Boundaries

Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

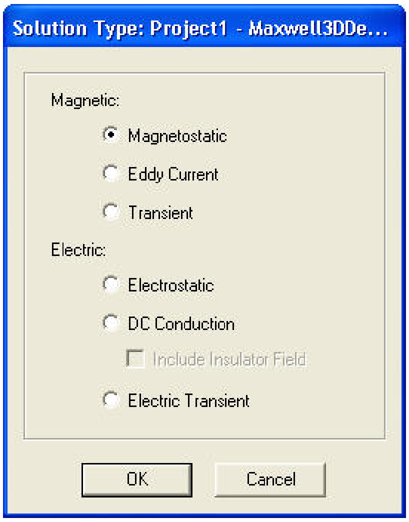
Change Design name

- **To Change Design Name:**
  - Right-click **Maxwell3D Design1** at the project manager window and select **Rename.**
  - Change the Design Name to **Full_Model**



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button

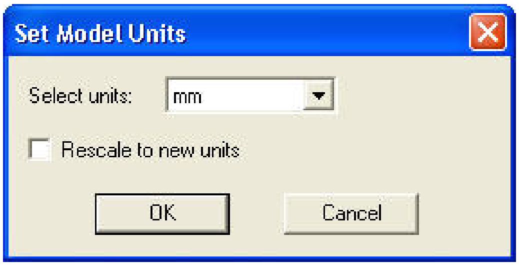


Example (Magnetostatic) – Symmetry Boundaries

Example 1: Solve Full Model

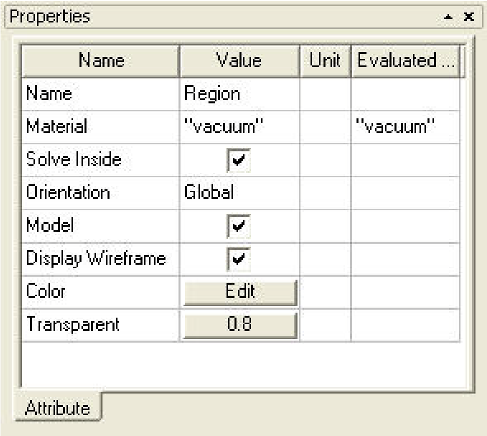
Set Model Units

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



Create Region

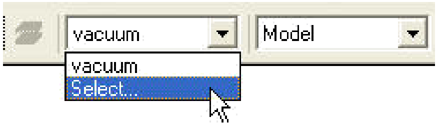
- **Create a Box region**
  - Select the menu item ***Draw > Box*** or click on the icon.
    1. Using the coordinate entry field, enter the box position
       - **X: -75, Y: -75, Z: -75** , Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - **dX: 150, dY: 150, dZ: 150**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Display Wireframe: ☑ **Checked**

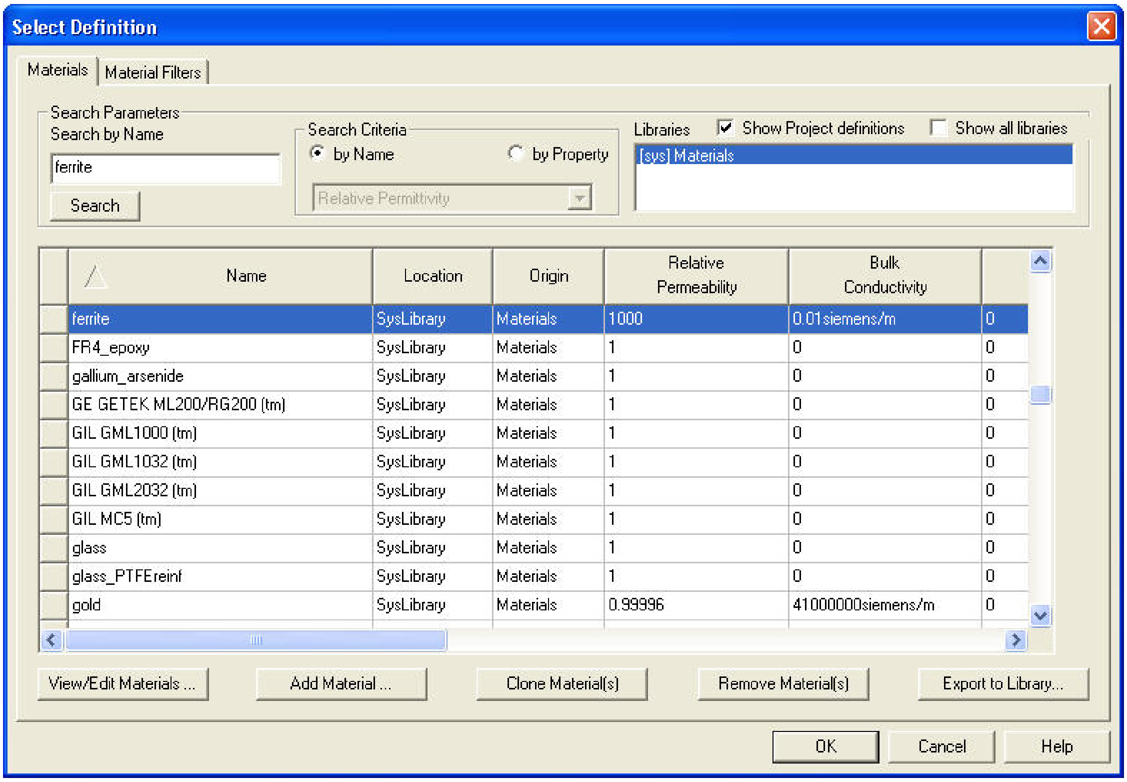


- **Hide Region**
  - Select **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***

Set Default Material

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





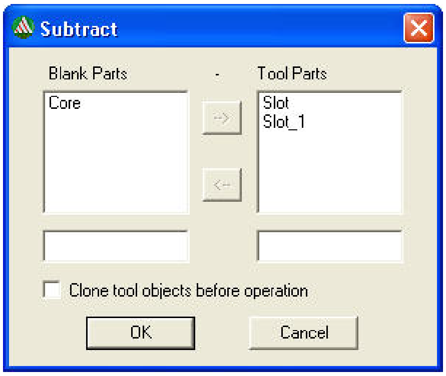
Example (Magnetostatic) – Symmetry Boundaries

Create Core

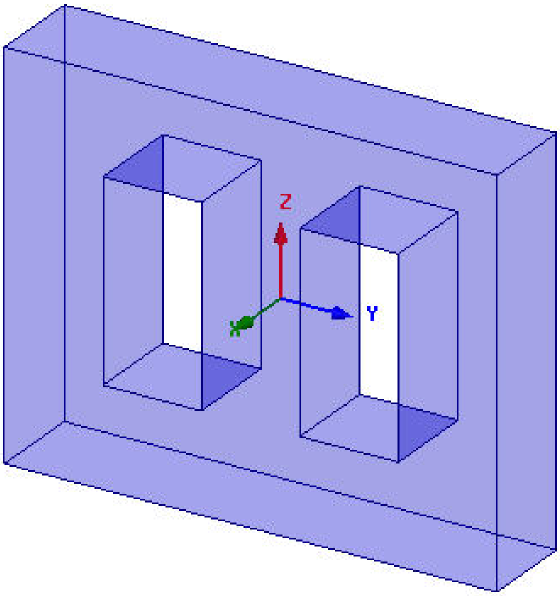
- **Change Grid Plane**
  - Select the menu item ***Modeler > Grid Plane > YZ***
- **Create Rectangle**
  - Select menu item ***Draw > Rectangle***.
    1. Using the coordinate entry fields, enter the rectangle position
       - X: **5**, Y: **-25**, Z: **-20**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **0**, dY: **50**, dZ: **40**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Core**
    2. Change the color of the object to **Blue**
- **Create Slot**
  - Select menu item ***Draw > Rectangle***.
    1. Using the coordinate entry fields, enter the rectangle position
       - X: **5**, Y: **-15**, Z: **10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **0**, dY: **10**, dZ: **-20**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Slot**
- **Duplicate Slot**
  - Select the object **Slot** from the history tree
  - Select the menu item ***Edit > Duplicate > Mirror***
    1. Using the coordinate entry field, enter the anchor point of mirror plane
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the target point of vector normal to the mirror plane
       - dX: **0**, dY: **1**, dZ: **0**, Press the **Enter** key

Example (Magnetostatic) - Symmetry Boundaries

- **Subtract Slot from Core**
  - Press **Ctrl** and select the objects **Core, Slot** and **Slot_1**
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Core**
    2. Tool Parts: **Slot, Slot_1**
    3. Click the **OK** button



- **Sweep Object**
  - Select the object **Core** from the history tree
  - Select the menu item ***Draw > Sweep >Along Vector***
    1. Using the coordinate entry field, enter the first point of the vector
       - X: **0,** Y: **0,** Z:**0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the last point of the vector
       - dX: **-10,** dY: **0,** dZ:**0**, Press the **Enter** key
    3. Press **OK**



Set Default Material

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

Create Coil_In

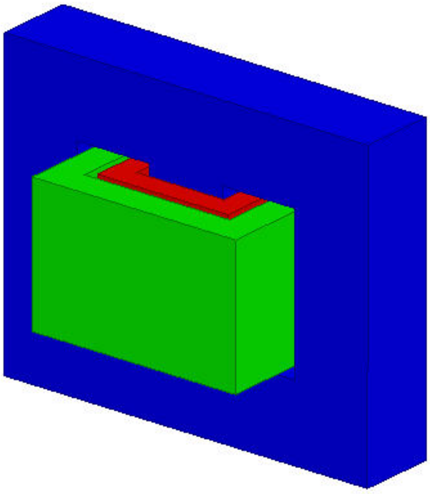
- Create Box
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-9**, Y: **-9**, Z: **-9**, Press the **Enter** key
    2. Using the coordinate entry fields, enter opposite corner of the box:
       - dX: **18**, dY: **18**, dZ: **18**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Coil_In**
    2. Change its color to **Red**
- **Create another Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-6**, Y: **-6**, Z: **-9**, Press the **Enter** key
    2. Using the coordinate entry fields, enter opposite corner of the box:
       - dX: **12**, dY: **12**, dZ: **18**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Coil_In_1**
- **Subtract Objects**
  - Press **Ctrl** and select the objects **Coil_In** and **Coil_In_1**
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract Window,
    1. Blank Parts: **Coil_In**
    2. Tool Parts: **Coil_In_1**
    3. Click the **OK** button

Create Coil_Out

- **Create Profile for Sweep**
  - Select menu item ***Draw > Rectangle***.
    1. Using the coordinate entry fields, enter the rectangle position
       - X: **0**, Y: **10**, Z: **-9**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **0**, dY: **4**, dZ: **18**, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Coil_Out**
    2. Change the color of the object to **Green**
- **Create Path for Sweep**
  - Select the menu item ***Draw > Line***
    1. Using the coordinate entry field, enter the first point of the line
       - X: **10,** Y: **10,** Z: **9**, Press the **Enter** key
    2. Using the coordinate entry field, enter the second point of the line
       - X: **-10,** Y: **10,** Z: **9**, Press the **Enter** key
    3. Using the coordinate entry field, enter the next point of the line
       - X: **-10,** Y: **-10,** Z: **9**, Press the **Enter** key
    4. Using the coordinate entry field, enter the next point of the line
       - X: **10,** Y: **-10,** Z: **9**, Press the **Enter** key
    5. Using the coordinate entry field, enter the first point of the line
       - X: **10,** Y: **10,** Z: **9**, Press the **Enter** key
    6. Press **Enter**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Coil_Out_Path**
  - Select the object **Coil_Out_Path** and select the menu item ***Modeler > Delete Last Operation***
    - **Note:** The object created in the last step results into a sheet. In order to get a polyline, we delete the Coverlines command that is automatically performed.

Example (Magnetostatic) – Symmetry Boundaries

- **Create Sweep**
  - Press **Ctrl** and select the objects **Coil_Out** and **Coil_Out_Path** from the tree
  - Select the menu item ***Draw > Sweep >Along Path***
  - Press **OK**



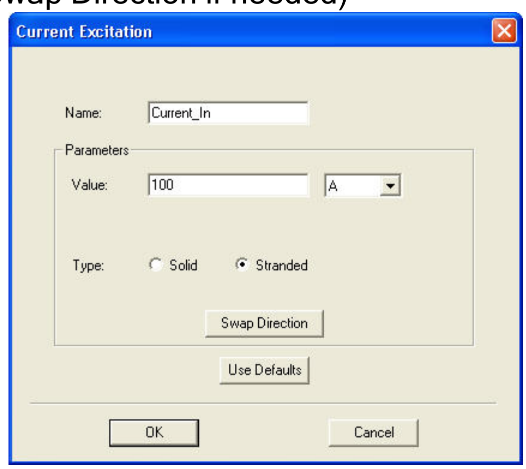
Create Coil Terminals

- **Create Section**
  - Press **Ctrl** and select the objects **Coil_In** and **Coil_Out** from the tree
  - Select the menu item ***Modeler > Surface >Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button
- **Change Attributes**
  - Select the object **Coil_In_Section1** and **Coil_Out_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Term_In** and **Term_Out**
- **Separate Sections**
  - Press **Ctrl** and select the objects **Term_In** and **Term_Out** from the tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Press **Ctrl** and select the object **Term_In_Separate1** and **Term_Out_Separate1** from the tree
  - Select the menu item ***Edit > Delete***

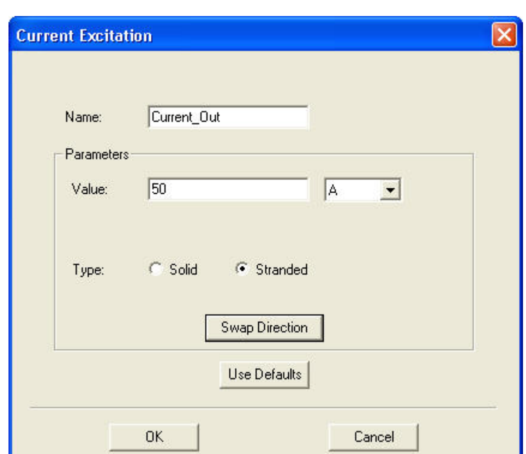
Example (Magnetostatic) - Symmetry Boundaries

Create Excitations

- **Assign Excitation for Coil_In**
  - Select the object **Term_In**
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **Current_In**
    2. Value: **100 A**
    3. Type: **Stranded**
    4. Direction: **Positive X** (Use Swap Direction if needed)
    5. Press **OK**



- **Assign Excitation for Coil_Out**
  - Select the object **Term_Out**
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **Current_Out**
    2. Value: **50 A**
    3. Type: **Stranded**
    4. Direction: **Negative X** (Use Swap Direction if needed)
    5. Press **OK**



Example (Magnetostatic) – Symmetry Boundaries

Analysis Setup

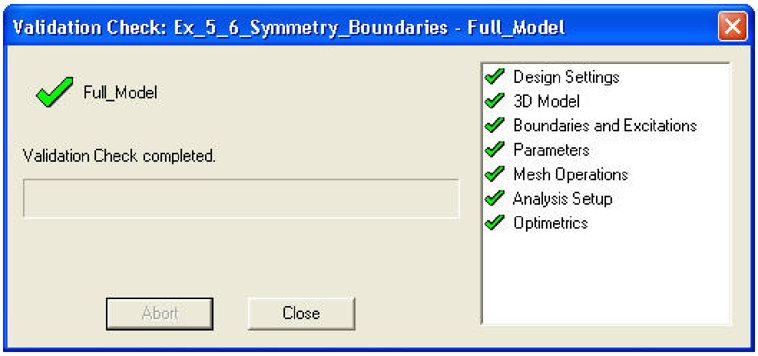
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From **Save As** window, type the Filename: **Ex_5_6_Symmetry_Boundaries**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



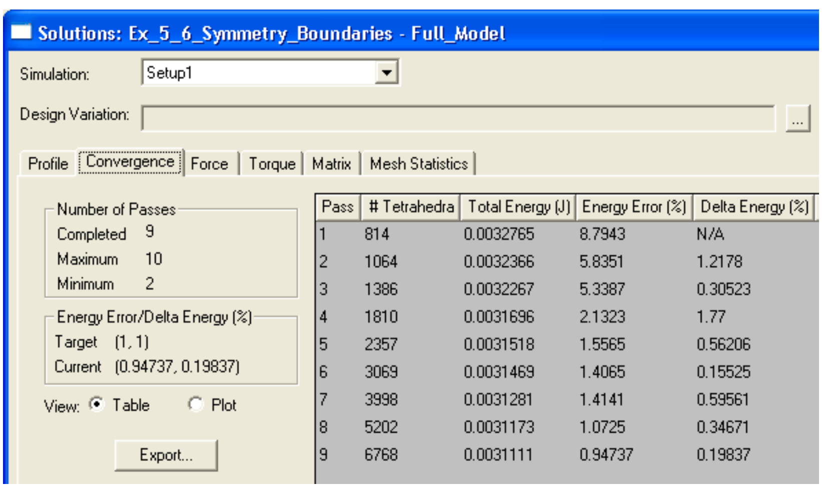
Note: To view any errors or warning messages, use the Message Manager.

Analyze

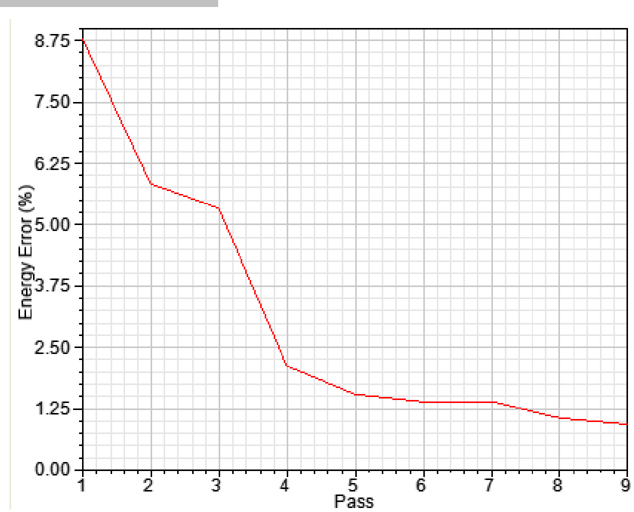
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item *Maxwell 3D > Results > Solution Data*
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To View Mesh information
      1. Click **Mesh Statistics** Tab
    - Select **Close** to close the window



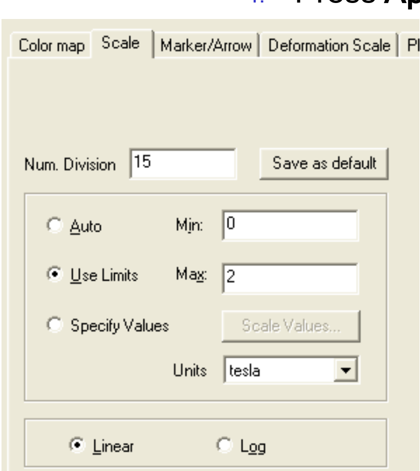
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	814	0.0032765	8.7943	N/A
2	1064	0.0032366	5.8351	1.2178
3	1386	0.0032267	5.3387	0.30523
4	1810	0.0031696	2.1323	1.77
5	2357	0.0031518	1.5565	0.56206
6	3069	0.0031469	1.4065	0.15525
7	3998	0.0031281	1.4141	0.59561
8	5202	0.0031173	1.0725	0.34671
9	6768	0.0031111	0.94737	0.19837

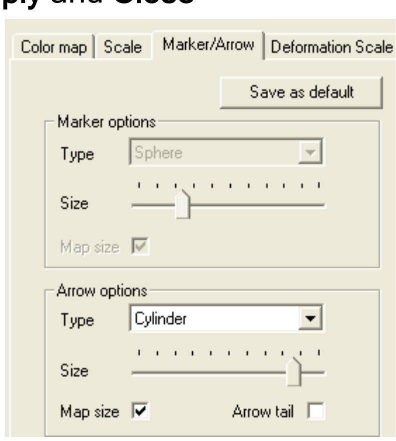


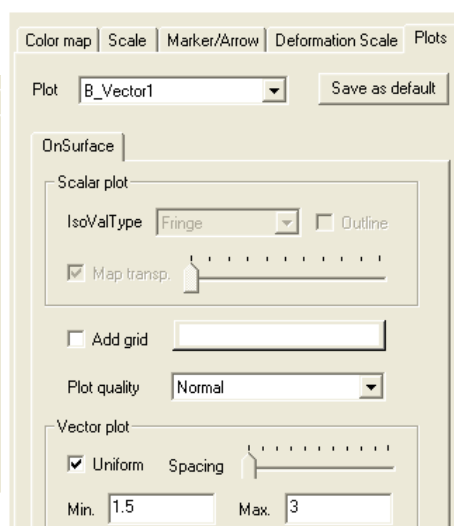
Example (Magnetostatic) – Symmetry Boundaries

Vector Plot

- **Create Vector Plot**
  - Select **Global:YZ** plane from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > B > B_Vector***
  - In Create Field Plot window
    1. In Volume: **AllObjects**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - User Limits: ☑ **Checked**
         1. Min: **0**
         2. Max: **2**
    2. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Set to appropriate value**
         2. Arrow tail: ☐ **Unchecked**
    3. **Plots** tab
       - Vector Plot
         1. Spacing: **Set to Minimum**
         2. Min: **1.5**
         3. Max: **3**
    4. Press **Apply** and **Close**

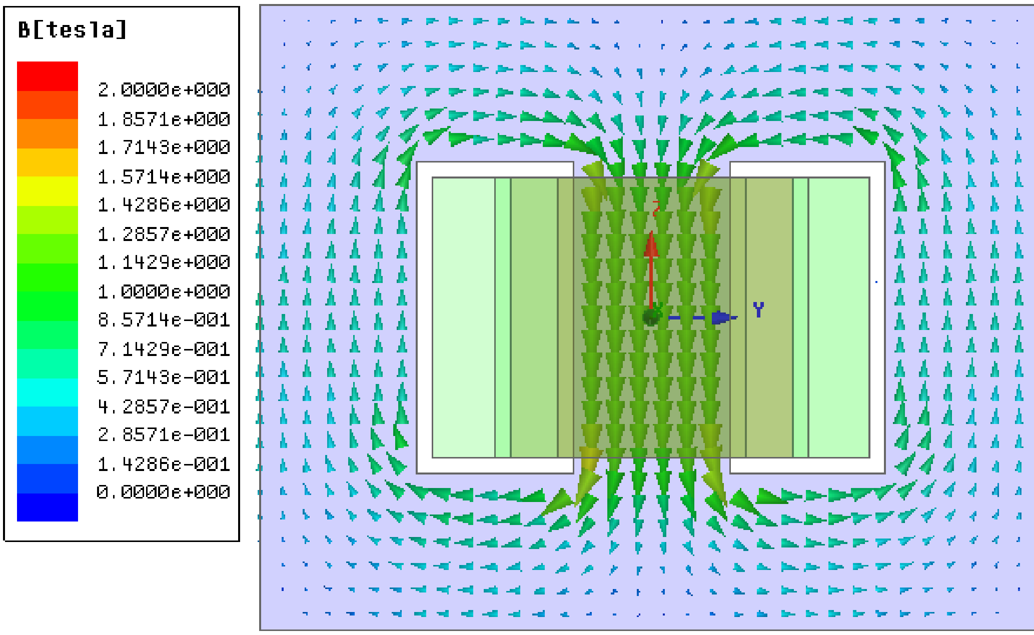






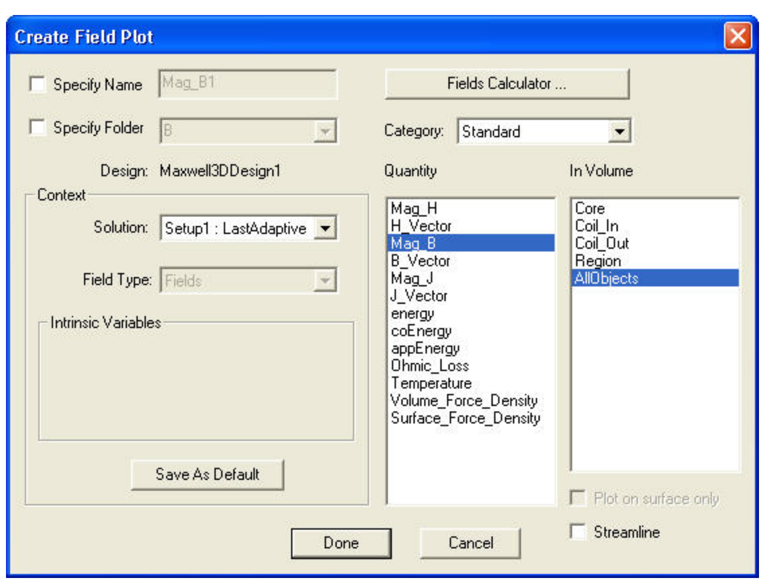
Example (Magnetostatic) - Symmetry Boundaries

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



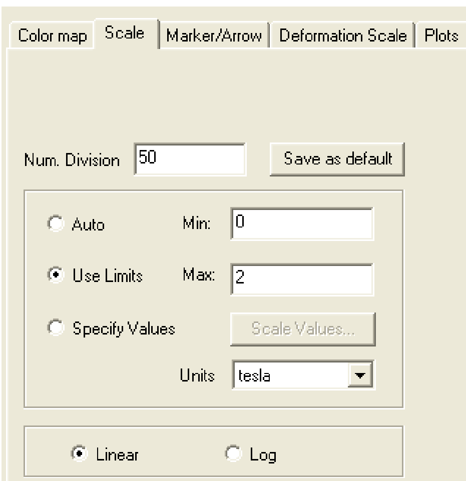
Plot Flux Density Contours

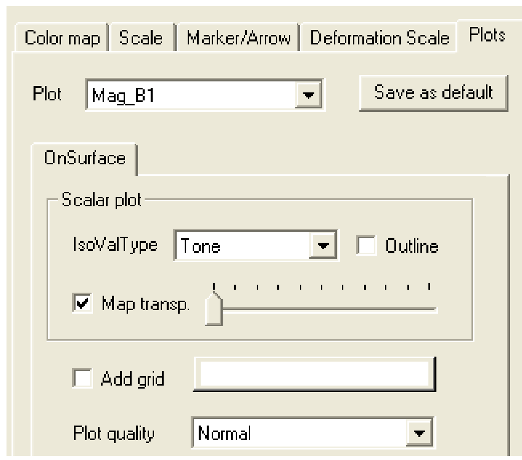
- **To Plot contours of Magnetic Flux Density**
  - Select **Global:YZ** plane from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window
    1. In Volume: **AllObjects**
    2. Select **Done**

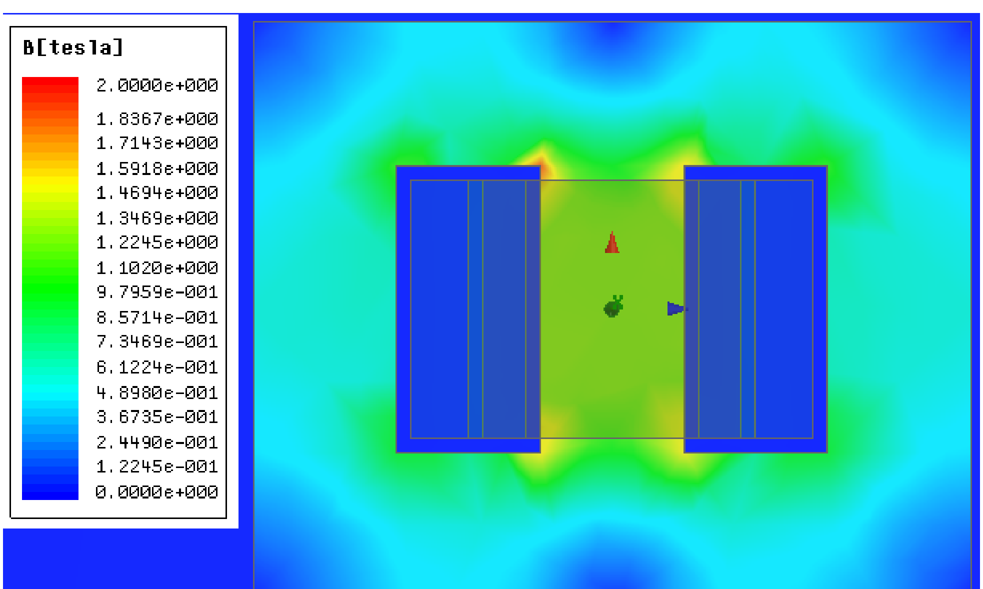


Example (Magnetostatic) – Symmetry Boundaries

- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - Num. Divisions: **50**
    2. **Plots** tab
       - Plot: **Mag_B1**
       - Scalar Plot
         - IsoValType: **Tone**
    3. Press **Apply** and **Close**







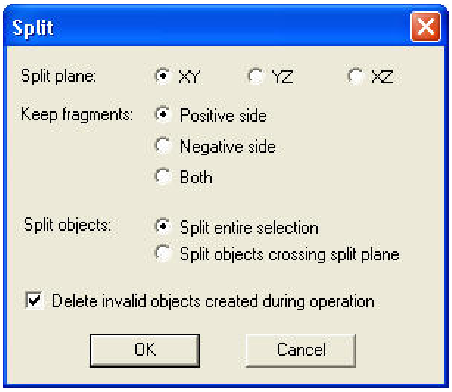
Example 2: Solve 1/8th of Model using Symmetry

Create Symmetry Design

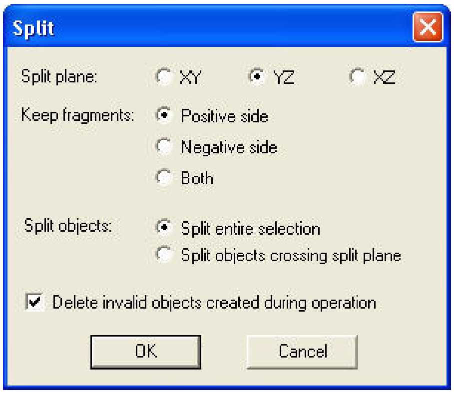
- **Copy Design**
  1. Select the design **Full_Model** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_6_Symmetry_Boundaries** in Project Manager window and select **Paste**
  3. Change the name of the design to **1_8_Model**

Split Model for 1/8th Section

- **Divide by XY Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XY**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

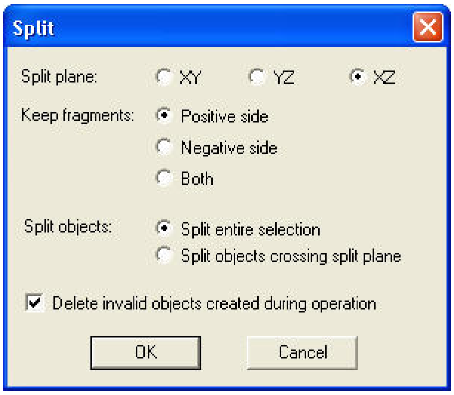


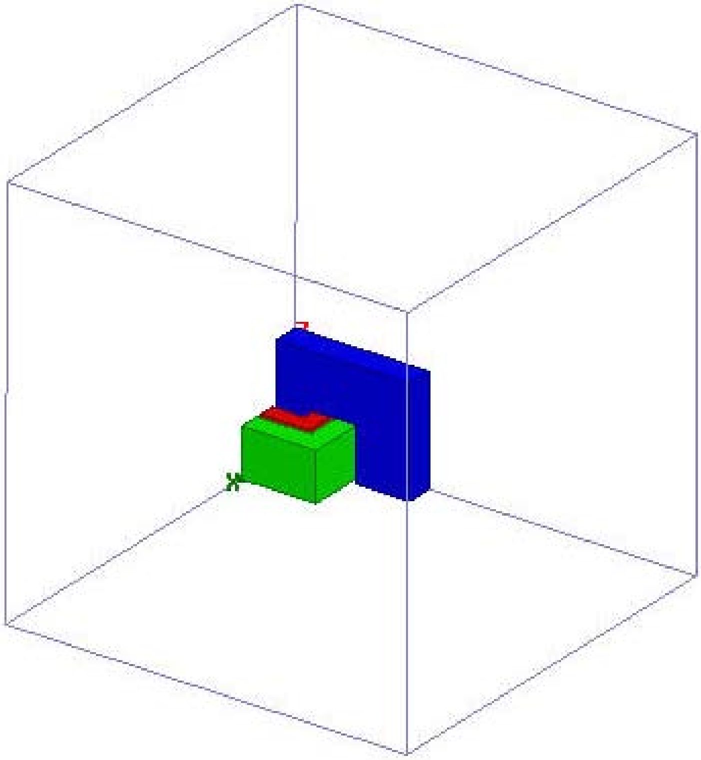
- **Divide by YZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



Example (Magnetostatic) – Symmetry Boundaries

- **Divide by XZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

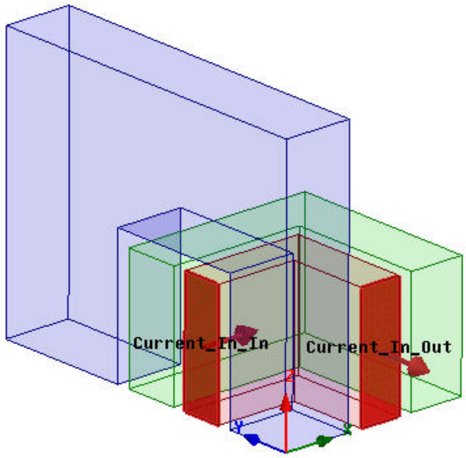




Example (Magnetostatic) – Symmetry Boundaries

Create Excitations

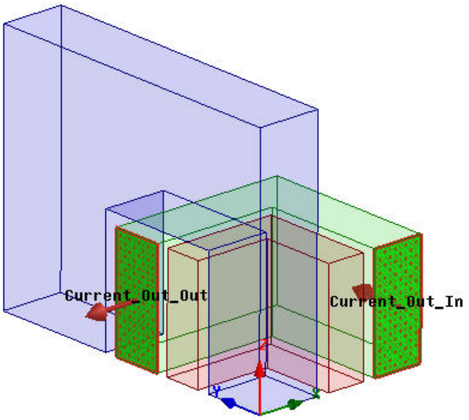
- **Assign Excitation for Coil_In**
  - Select the menu item *Edit > Select > Faces* or press **F** from the keyboard to change selection to faces
  - Select the face of **Coil_In** which touches the YZ Plane
  - Select the menu item *Maxwell 3D > Excitations > Assign >Current*
  - In Current Excitation window,
    1. Name: **Current_In_In**
    2. Value: **50 A**
    3. Type: **Stranded**
    4. Direction: **Positive X**
    5. Press **OK**
  - Select the face of **Coil_In** which touches the XZ Plane
  - Select the menu item *Maxwell 3D > Excitations > Assign >Current*
  - In Current Excitation window,
    1. Name: **Current_In_Out**
    2. Value: **50 A**
    3. Type: **Stranded**
    4. Direction: **Negative Y**
    5. Press **OK**



- **Assign Excitation for Coil_Out**
  - Select the face of **Coil_Out** which touches the YZ Plane
  - Select the menu item *Maxwell 3D > Excitations > Assign >Current*
  - In Current Excitation window,
    1. Name: **Current_Out_Out**
    2. Value: **25 A**
    3. Type: **Stranded**
    4. Direction: **Negative X**
    5. Press **OK**

Example (Magnetostatic) – Symmetry Boundaries

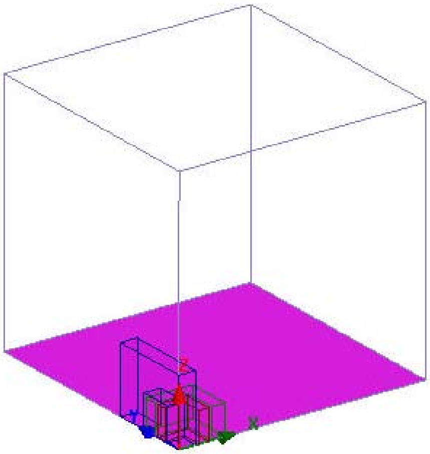
- Select the face of **Coil_Out** which touches the XZ Plane
- Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
- In Current Excitation window,
  1. Name: **Current_Out_In**
  2. Value: **25 A**
  3. Type: **Stranded**
  4. Direction: **Positive Y**
  5. Press **OK**



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

Assign Boundaries

- **To Assign Symmetry Boundaries**
  - Select the face of the Region with which touch with XY Plane
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Symmetry***
  - In Symmetry Boundary window,
    1. Symmetry: **Even (Flux Normal)**
    2. Press **OK**



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

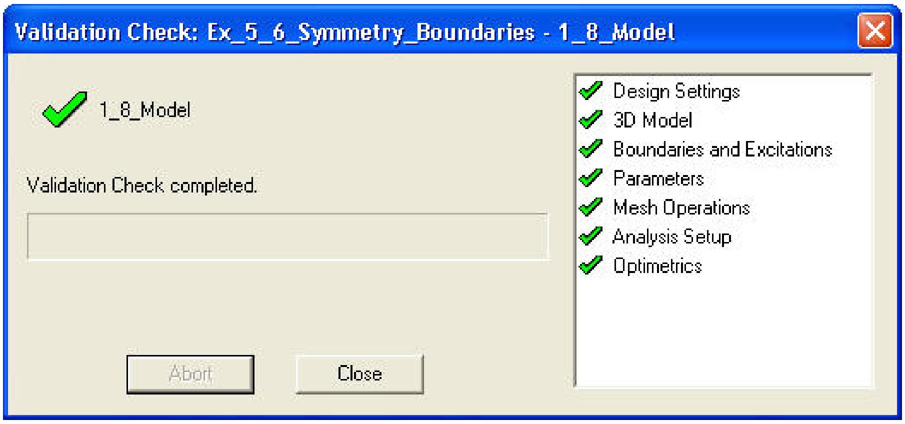
Example (Magnetostatic) – Symmetry Boundaries

Analysis Setup

- There is already **Setup1** because it copied from the **Full_Model** design. Analyze this design by the untouched setting.

Model Validation

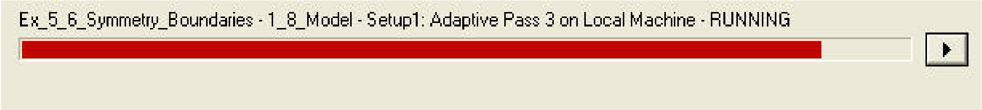
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

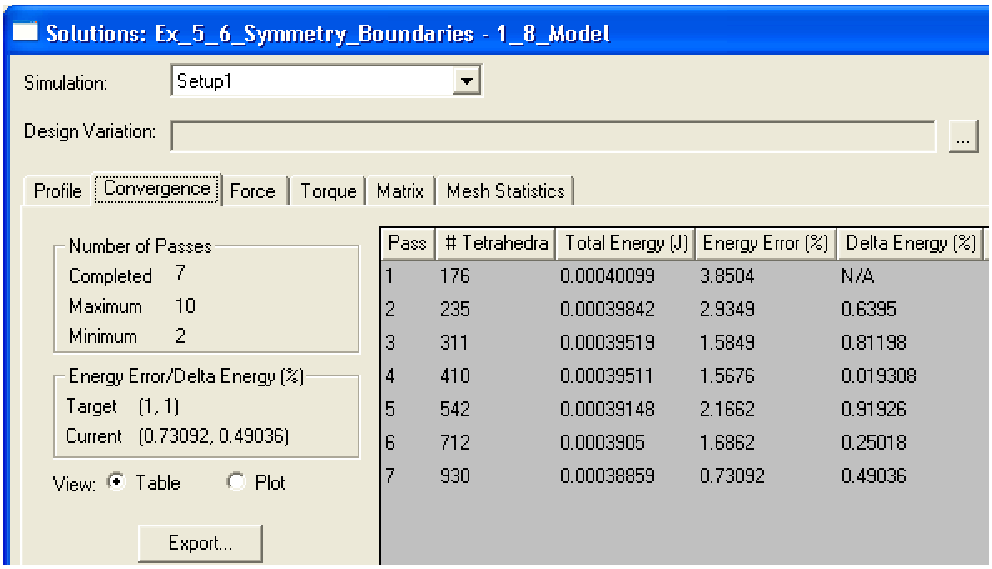
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

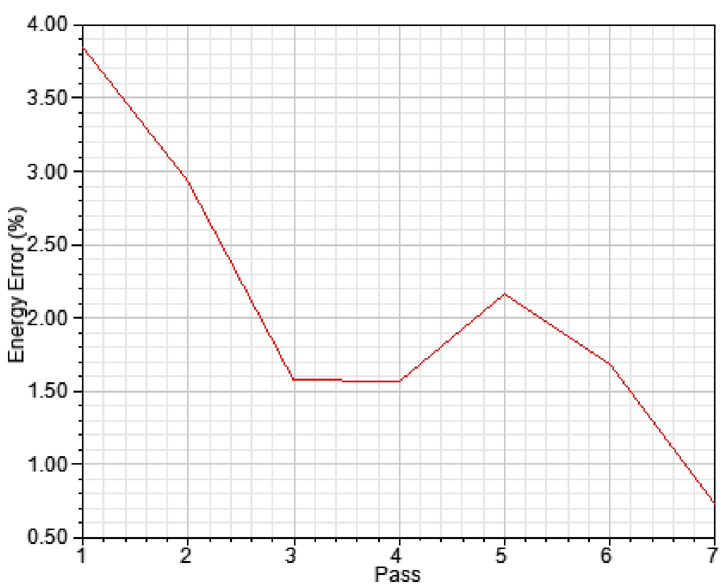


Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***



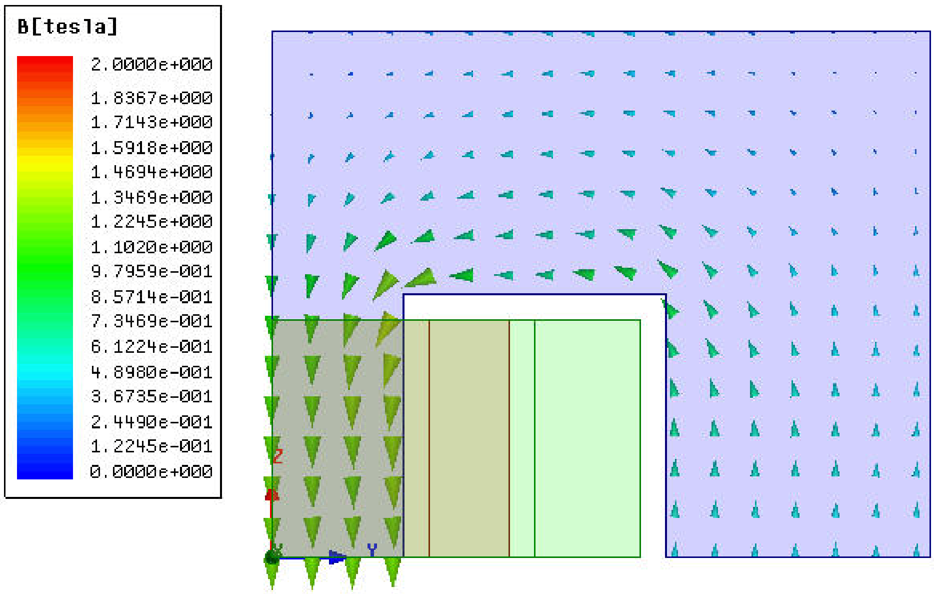
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	176	0.00040099	3.8504	N/A
2	235	0.00039842	2.9349	0.6395
3	311	0.00039519	1.5849	0.81198
4	410	0.00039511	1.5676	0.019308
5	542	0.00039148	2.1662	0.91926
6	712	0.0003905	1.6862	0.25018
7	930	0.00038859	0.73092	0.49036

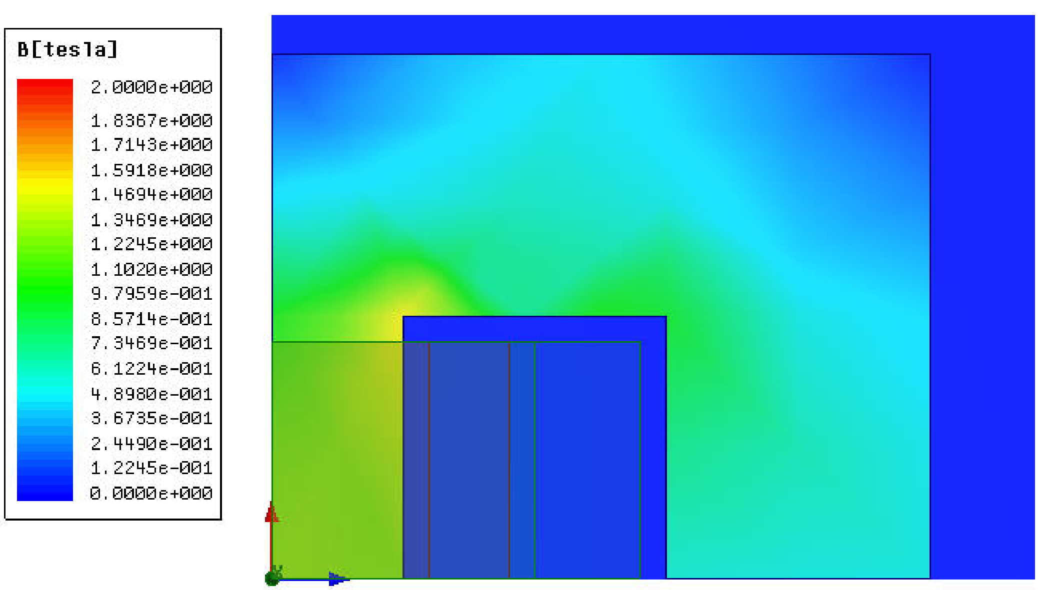


- In the full model, the number of mesh was 6768, now it becomes 930 meshes in 1/8 models. That means the number of mesh become about 1/7.

Field Plots

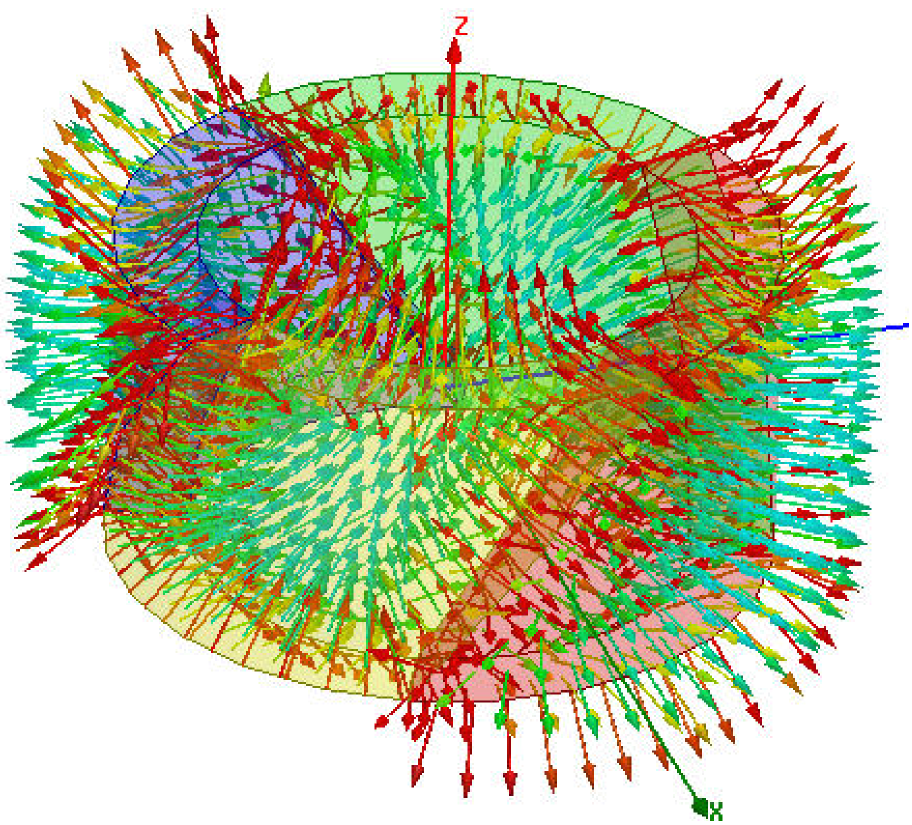
- Field plots are already available as design was copied from **Full_Model**





Permanent Magnet Magnetization

- Each object can have the coordinate system that is independent and is arbitrary in Maxwell. The magnetization setting can be done in an arbitrary direction for using this coordinate system.
- Moreover, since it is also possible to do the magnetization setting by using the equation and the numeric character data, the user can do a variety of magnetization settings.
- This chapter introduces the magnetization setting co-ordinate system, by equations for direction or by character data



ANSYS Maxwell Design Environment

- The following features of the Ansoft Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Circle, Line**
    - Modeler operations: **Sweep**
    - Boolean Operations: **Subtract**
  - Analysis
    - **Magnetostatic**
  - Field Overlays:
    - **B Vector Plots**
  - Results
    - **Calculator Expressions**
    - **Plots**

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

Setting Tool Options

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

Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the On the Standard toolbar, or select the menu item *File > New*.
  - Select the menu item *Project > Insert Maxwell 3D Design*, or click on the icon

Change Design name

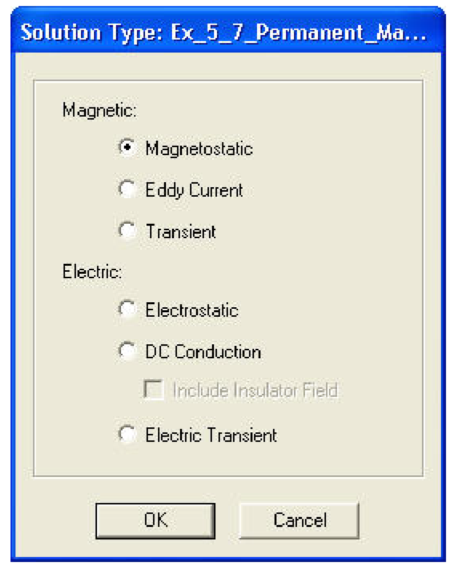
- **To Change Design Name:**
  - Right-click **Maxwell3D Design1** at the project manager window and select **Rename.**
  - Change the Design Name to **Ring01**.

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: **Ex_5_7_Permanent_Magnet**
  3. Click the **Save** button

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



Set Model Units

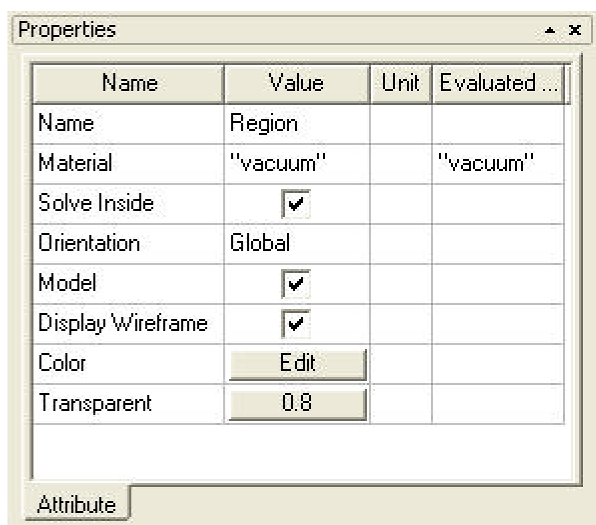
- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **mm**
    2. Click the **OK** button

Example 1 : Coordinate system of object and setting for each axis

In this example, it is introduced the magnetization using Coordinate system of object.
Material : **Mag_North** and **Mag_South** is newly made in this Example.

Create Region

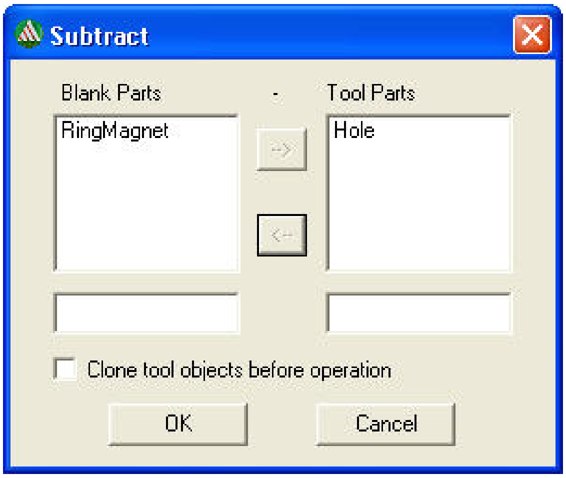
- **Create a Box region**
  - Select the menu item ***Draw > Box*** or click on the icon.
    1. Using the coordinate entry field, enter the box position
       - **X: -200, Y: -200, Z: -200** , Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - **dX: 400, dY: 400, dZ: 400**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Display Wireframe: ☑ **Checked**



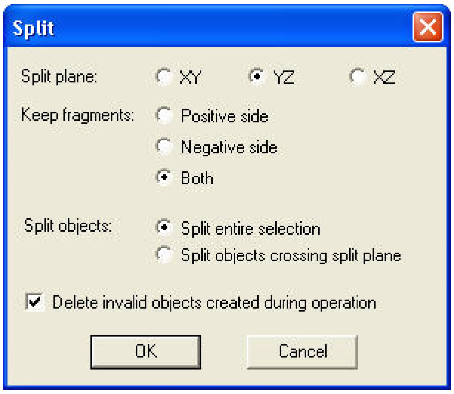
- **Hide Region**
  - Select **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***

Create Magnets

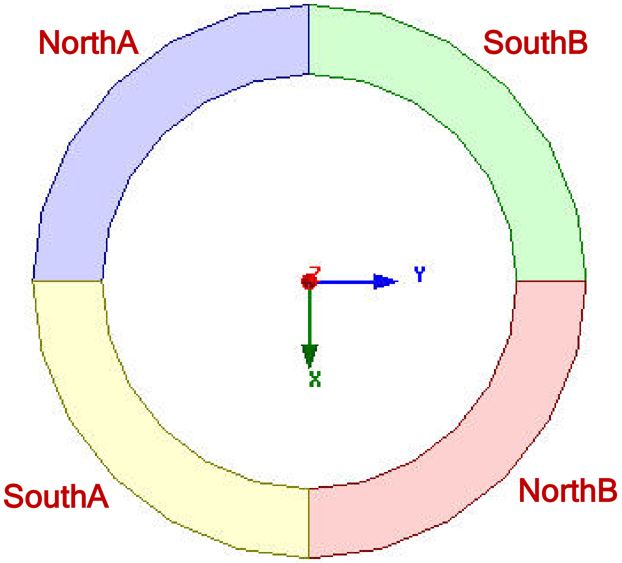
- **Create Circle**
  - Goto menu item ***Draw > Circle***
    1. Using the coordinate entry field, enter the center position
       - X: **0,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius of circle
       - dX:**40** for radius, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **RingMagnet**
    2. Change its color to **Blue**
- **Draw another circle**
  - Goto menu item ***Daw > Circle***
    1. Using the coordinate entry field, enter the center position
       - X: **0,** Y: **0,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the radius of circle
       - dX:**30** for radius, Press the **Enter** key
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Hole**
- **Subtract Hole from RingMagnet**
  - Select **RingMagnet** and **Hole** from the history tree
  - Goto menu item ***Modeler > Boolean > Subtract***
    1. Select **RingMagnet** in Blank Parts
    2. Select **Hole** in Tool Parts
    3. Press **OK**



- **Split with YZ Plane**
  - Select **RingMagnet** from the History Tree
  - Goto menu item ***Modeler > Boolean > Split***
    1. Split Plane: Select **YZ Plane**
    2. Keep Fragments: **Both**
    3. Press **OK**

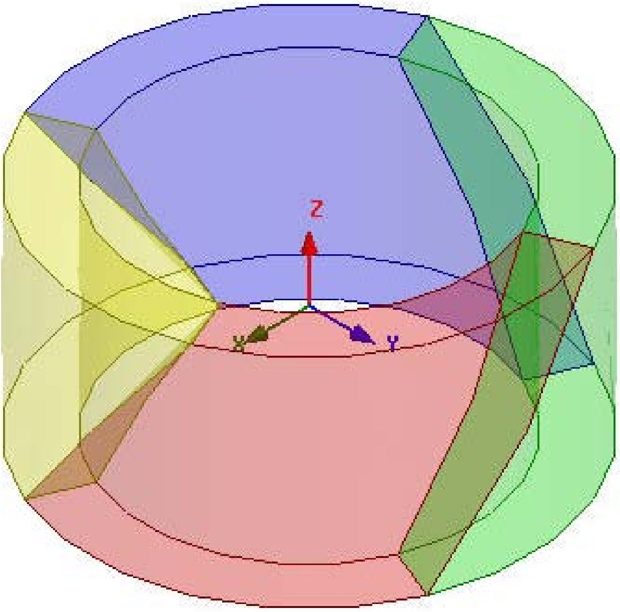


- **Split with XZ Plane**
  - Press **Ctrl** and select **RingMagnet** and **RingMagnet_Split1** from the tree
  - Goto menu item ***Modeler > Boolean > Split***
    1. Split Plane: Select **XZ Plane**
    2. Keep Fragments: **Both**
    3. Press **OK**
- **Change Attributes**
  - Change the attributes of the four objects resulted from split operation
  - RingMagnet
    1. Name: **NorthA**
    2. Color: **Blue**
  - RingMagnet_Split1
    1. Name: **SouthA**
    2. Color: **Yellow**
  - RingMagnet_Split1_Split1
    1. Name: **NorthB**
    2. Color: **Red**
  - Ringmagnet_Split2
    1. Name: **SouthB**
    2. Color: **Green**



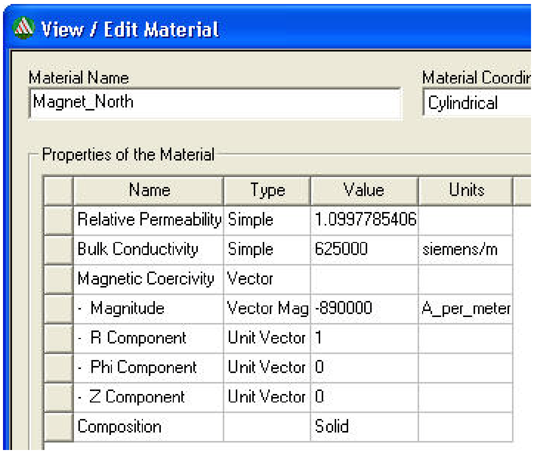
- **Create Guide for Sweep**
  - Goto menu item ***Draw > Line***
    1. Using the coordinate entry field, enter first vertex positions
       - X: **0,** Y: **0,** Z: **-20**, Press the **Enter** key
    2. Using the coordinate entry field, enter second vertex positions
       - X: **0,** Y: **0,** Z:**20**, Press the **Enter** key
    3. Press **Enter** key to Exit
- **Create Sweep**
  - Select **NorthA, NorthB, SouthA, SouthB** and **Polyline1** from the History Tree
  - Goto menu item ***Draw > Sweep > Along Path***
    1. Set Angle of Twist to **45 degrees**
    2. Press **OK**

  - **Note:** The 45 degree angle given in sweep command is the Skew Angle of the Magnets. This value will be give through equation in the next exercise of magnetization using equations.



Create Magnet Materials: Magnet_North, Magnet_South

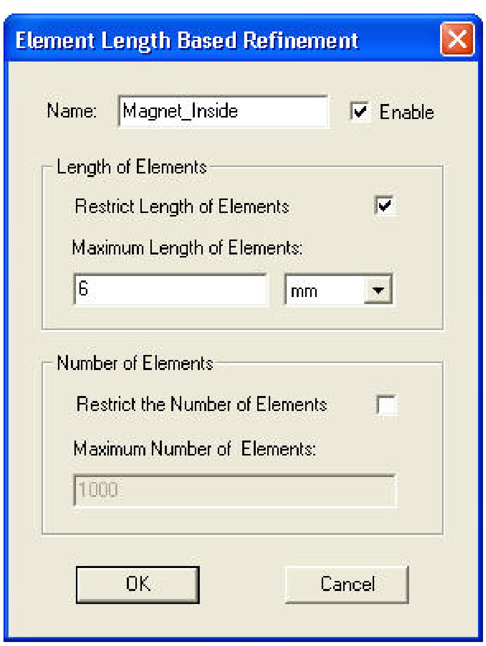
- **To Create Material Magnet_North:**
  - Select **NorthA** and **NorthB** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **NdFe35** in the **Search by Name** field
    2. Select option **Clone Material**
    3. In View/Edit Material window
       1. Material Name : **Magnet_North**
       2. Material Coordinate System : **Cylindrical**
       3. Magnitude : **-890000**
       4. R Component : **1**
       5. Phi Component: **0**
       6. Z Component : **0**
       7. Press **OK**
    4. select **OK** to assign the material.

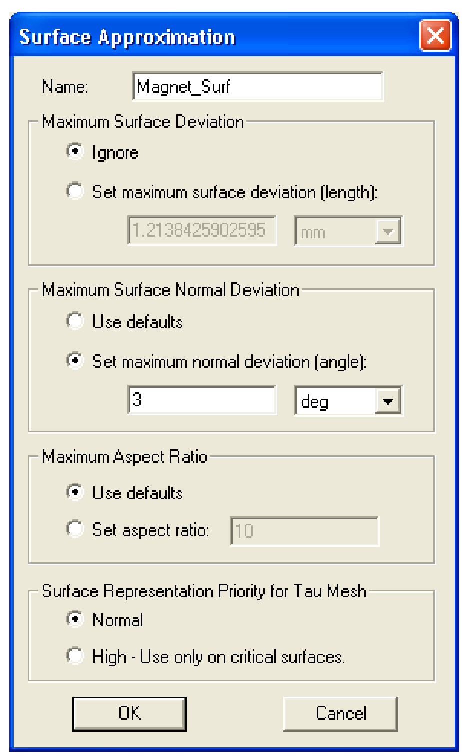


- **To Create Material Magnet_South:**
  - Select **SouthA** and **SouthB** from history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **NdFe35** in the **Search by Name** field
    2. Select option **Clone Material**
    3. In View/Edit Material window
       1. Material Name : **Magnet_South**
       2. Material Coordinate System : **Cylindrical**
       3. Magnitude : **-890000**
       4. R Component : **-1**
       5. Phi Component: **0**
       6. Z Component : **0**
       7. Press **OK**
    4. select **OK** to assign the material.

Apply Mesh Operations

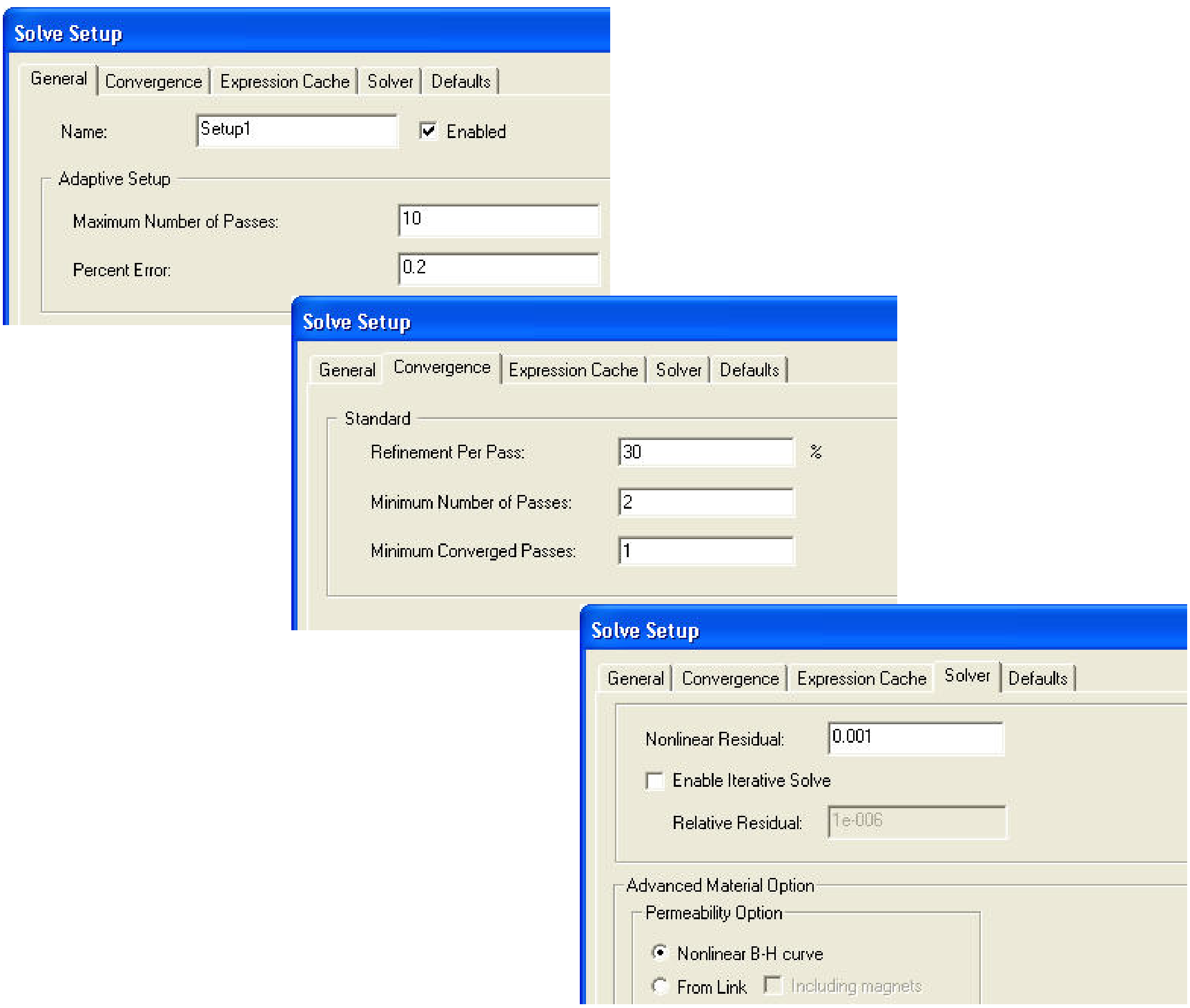
- **Apply Length Based Mesh Operations**
  - Select **NorthA, NorthB, SouthA** and **SouthB** from History Tree
  - Goto menu item ***Maxwell 3D > Mesh Operations >Assign > Inside Selection > Length Based…***
    1. Name: **Magnet_Inside**
    2. Restrict Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **6mm**
    4. Press **OK**
- **Apply Surface Approximation**
  - Select **NorthA, NorthB, SouthA** and **SouthB** from History Tree
  - Select menu item ***Maxwell 3D > Mesh Operations > Assign > Surface Approximation***
    1. Name: **Magnet_Surf**
    2. Maximum Surface Normal Deviation
       - Set maximum normal deviation (angle): **3 deg**
    3. Press **OK**





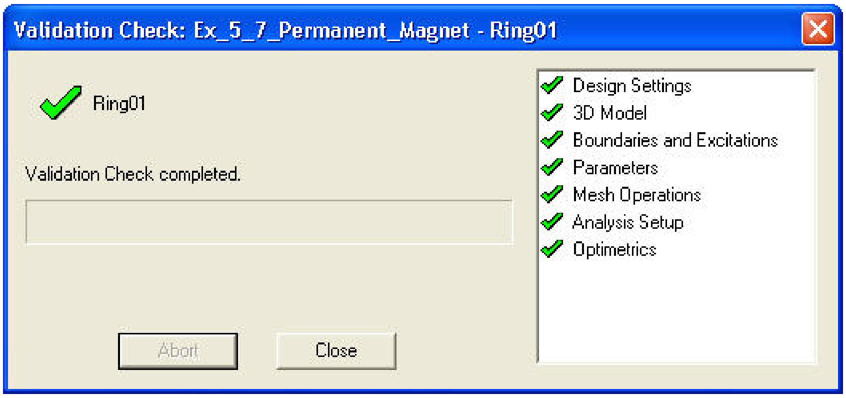
Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup*** In Solve Setup window
    1. **General** tab
       - Percentage Error: **0.2**
    2. Press **OK**



Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



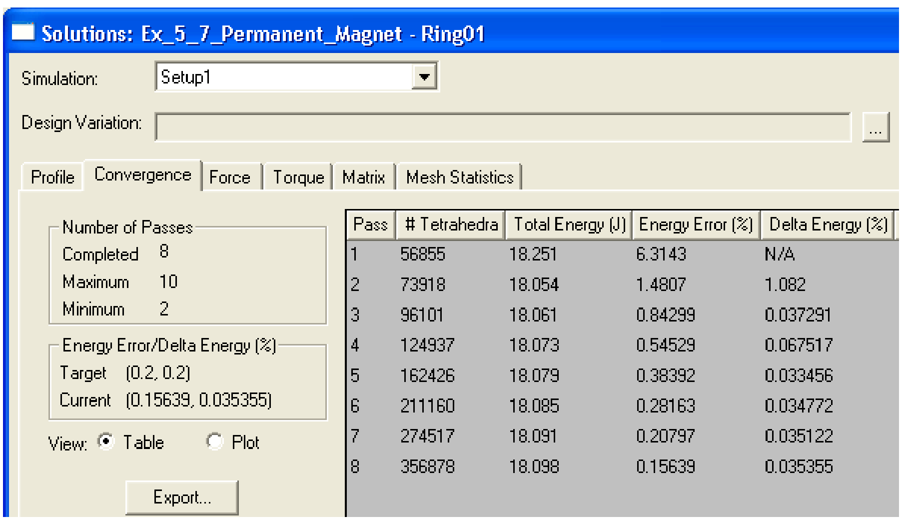
Note: To view any errors or warning messages, use the Message Manager.

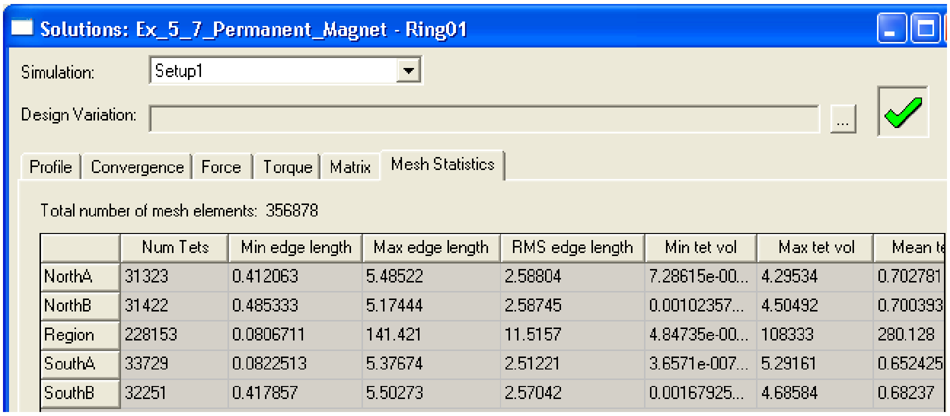
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

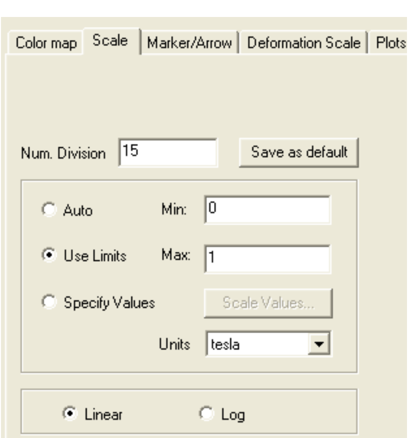
- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To View Mesh information
      1. Click **Mesh Statistics** Tab
    - Select **Close** to close the window

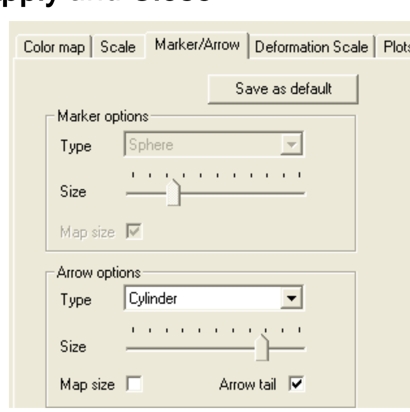


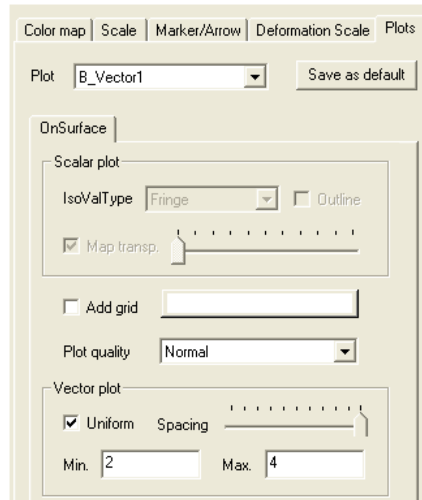


Vector Plot

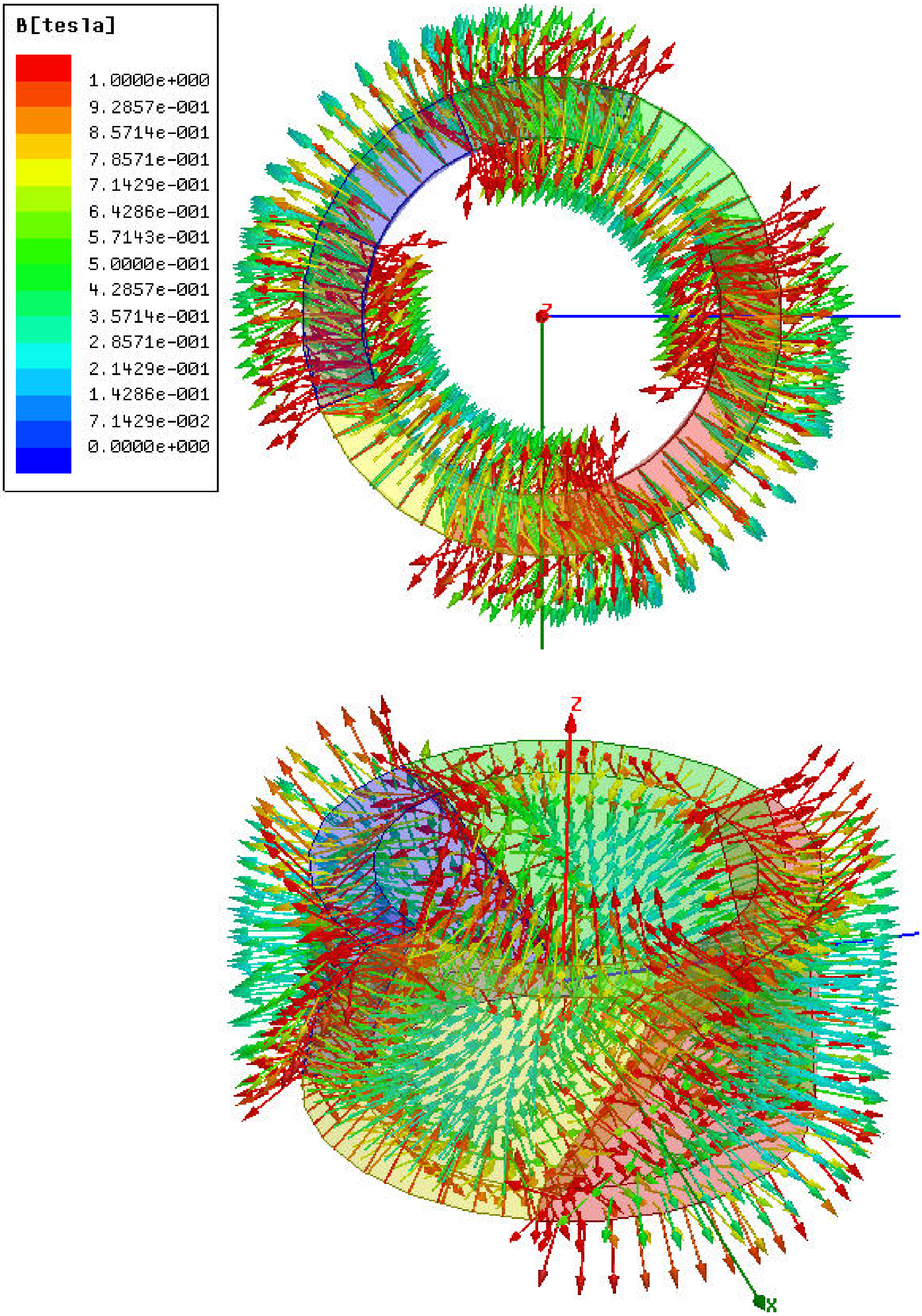
- **Create Vector Plot**
  - Select **NorthA, NorthB, SouthA** and **SouthB** from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > B > B_Vector***
  - In Create Field Plot window
    1. Plot on surface only : ☑ **Checked**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - User Limits: ☑ **Checked**
         1. Min: **0**
         2. Max: **1**
    2. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Set to appropriate value**
         2. Map Size: ☐ **Unchecked**
    3. **Plots** tab
       - Vector Plot
         1. Spacing: **Set to Maximum**
         2. Min: **2**
         3. Max: **4**
    4. Press **Apply** and **Close**







- Each of the magnets is magnetized in radial direction. The direction of magnetization of North magnets is opposite to that of South magnets.



Create Entities for Rectangular Plot

- **Turn off automatic covering of polylines**
  - Goto *Tools > Options > Modeler Options*
    1. Operation tab
       - Automatically cover closed polylines: ☐ **Unchecked**
    2. Press **OK**
- **Create Circle**
  - Select menu item *Draw > Circle*
  - A massage will pop up asking if the entity needs to be created as non model object. Select **Yes** to it.
    1. Using Co-ordinate entry field Enter the center of the circle
       - **X = 0, Y = 0, Z = -10,** Press **Enter**
    2. Using Co-ordinate entry field Enter the radius
       - **dX = 40, dY = 0, dZ = 0,** Press **Enter**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **CircleA**
- **Create Copies of CircleA along Z axis**
  - Select CircleA from history tree
  - Goto menu item *Edit > Duplicate > Along Line*
    1. Using Co-ordinate entry field Enter the first point of duplicate vector
       - **X = 0, Y = 0, Z = -10,** Press **Enter**
    2. Using Co-ordinate entry field Enter the second point
       - **dX = 0, dY = 0, dZ = 10,** Press **Enter**
    3. Set **Total Number** of Duplicates to **3**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **CircleB** and **CircleC**

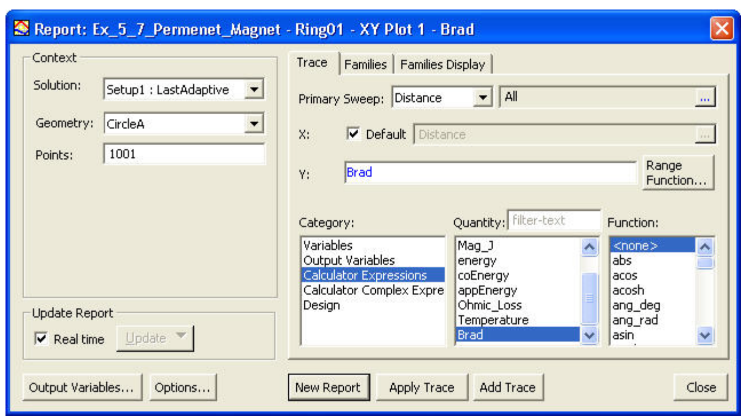
Create parameter Brad

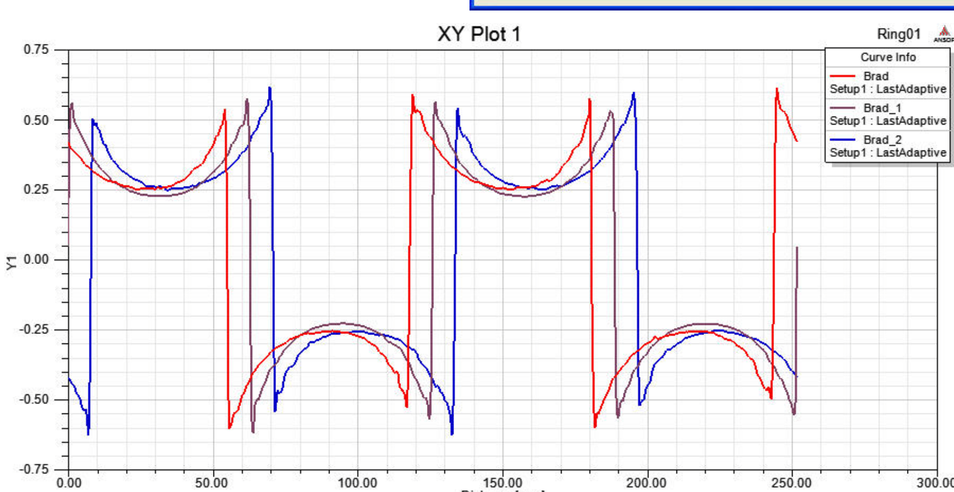
- **Create parameter for radial magnetic filed strength**
  - Goto ***Maxwell 3D > Fileds > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **General > Smooth**
    3. Select **Vector > Scal? > ScalarX**
    4. Select **Input > Function > Scalar > PHI**
    5. Select **Scalar > Trig > Cos**
    6. Select **General > ***
    7. Select **Input > Quantity > B**
    8. Select **General > Smooth**
    9. Select **Vector > Scal? > ScalarY**
    10. Select **Input > Function > Scalar > PHI**
    11. Select **Scalar > Trig > Sin**
    12. Select **General > ***
    13. Select **General > +**
    14. Select **Add** and set the name of expression as **Brad**
    15. Press **Done**

Rectangular Plot

To Create Plot:

- Select *Maxwell 3D > Results > Create Fields Report > Rectangular Plot*
- In Reports window
  1. Geometry: **CircleA**
  2. **Trace** Tab
     1. X axis: **Default**
     2. Y axis
        - Category: **Calculator Expressions**
        - Quantity: **Brad**
        - Function: **None**
  3. Select **New Report**
  4. Without closing window, change geometry to **CircleB**
  5. Select **Add Trace**
  6. Change Geomety to **CircleC**
  7. select **Add Trace**
  8. Select **Close**



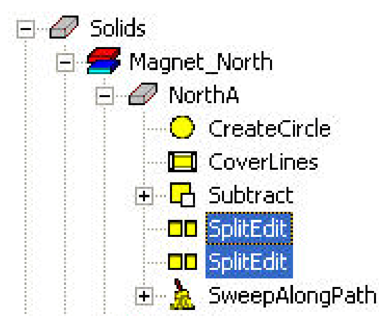


Example2 : Magnetization using the equation

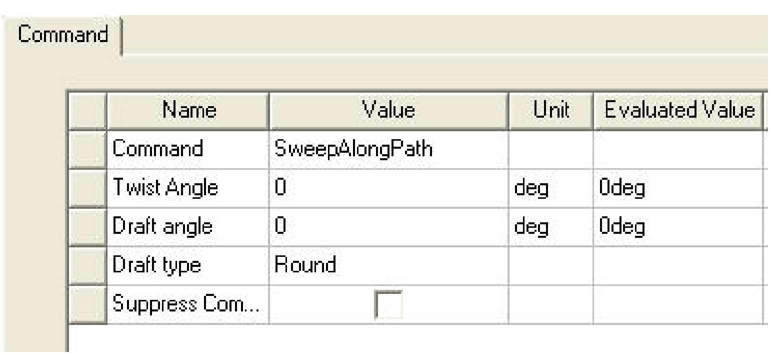
In this example, direction of Magnetization is changed using equation. We will learn to achieve same result as in last exercise without modeling four different poles of Magnet

Prepare Design

- **Copy Design**
  1. Select **Ring01** design in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_7_Permanent_Magnet** in Project Manager window and select **Paste**
  3. **Ring2** design is created
- **Edit Geometry**
  1. Open the history of **NorthA** in history tree
  2. Select the commands **SplitEdit** and delete them using **Delete** key on your keyboard



  3. Double click on the command **SweepAlongPath** from history tree
  4. Change Twist Angle from 45 to **0 degress**
  5. Press **OK**

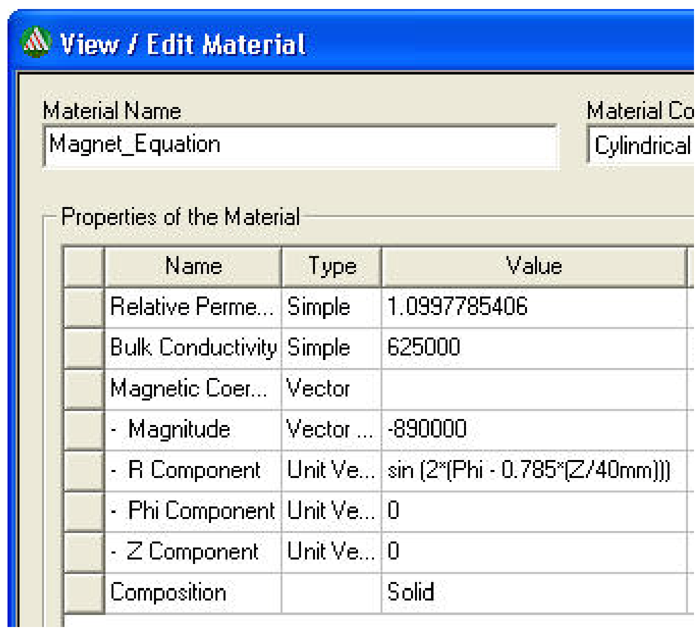

Command

Name	Value	Unit	Evaluated Value
Command	SweepAlongPath		
Twist Angle	0	deg	0deg
Draft angle	0	deg	0deg
Draft type	Round		
Suppress Com...	☐		

- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **RingMagnet**

Create New Material: Magnet_Equation

- **To Create material Magnet_Equation:**
  - Select **RingMagnet** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Select option **Clone Material**
    2. In View/Edit Material window
       1. Material Name : **Magnet_Equation**
       2. Magnitude : **-890000**
       3. R Component : **sin (2*(Phi - 0.785*(Z/40mm)))**
       4. Press **OK**
    3. select **OK.**



Note : About the equation

- The equations used in this case is a sine function which switches the value of R from positive to negative with Phi hence altering the direction of magnetization. Rest of the equation gives skew to the direction along Z direction. This gives the field similar to four pole magnet with 45 degree skew modeled in last exercise

Number of Poles / 2

sin (2*(Phi - 0.785*(Z/40mm)))

Skew angle in radians[45 deg]

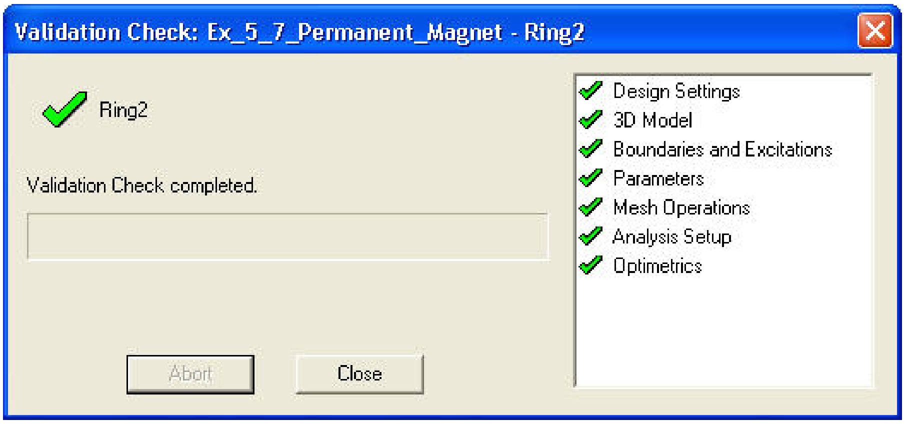
Length of magnet

Analysis Setup

- There is already **Setup1** because it copied from the **Ring01** design. Analyze this design by the untouched setting.

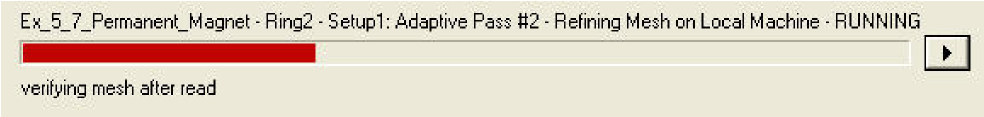
Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



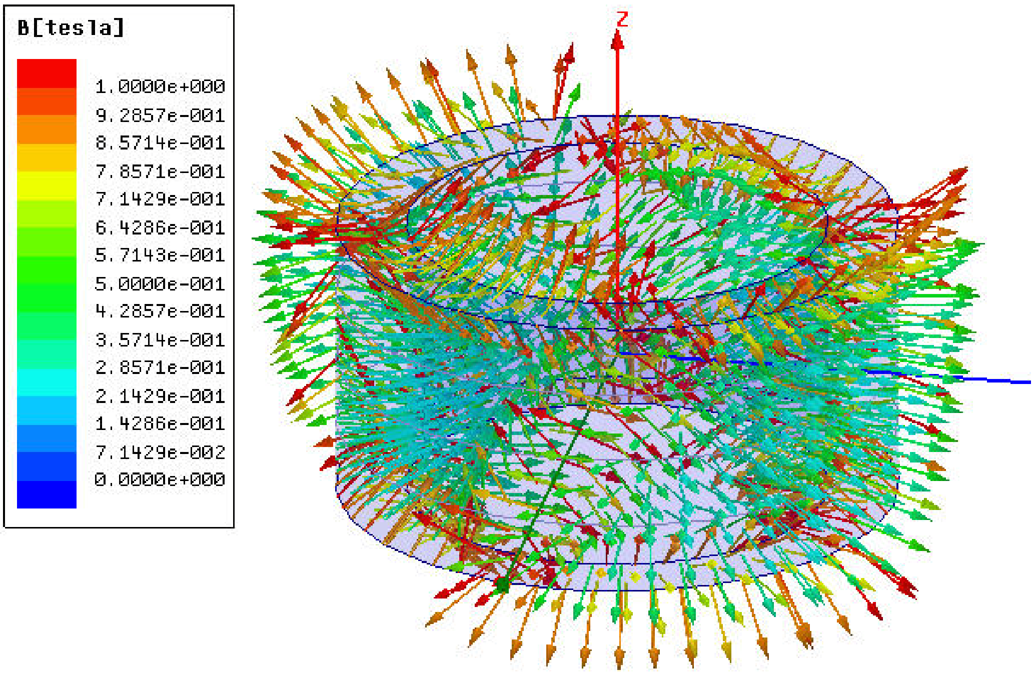
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

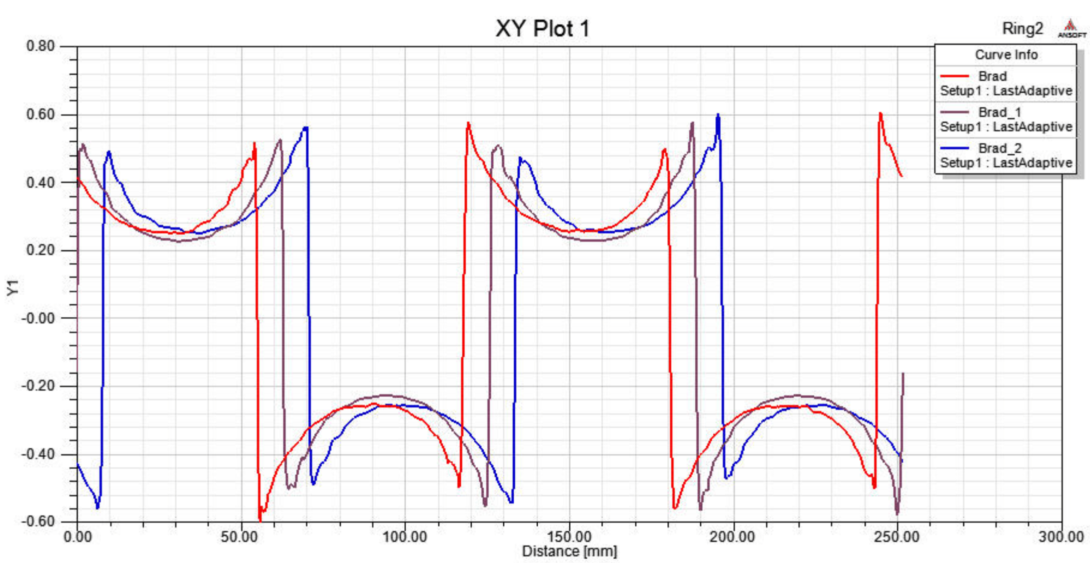


Results

- Double click ***Field Overlays > B > B_Vector*** in Project tree to view vector plot



- Check the Plots of **Brad** that we created in last exercise. The results in both cases should be nearly same



Example3 : Magnetization using Input Curve

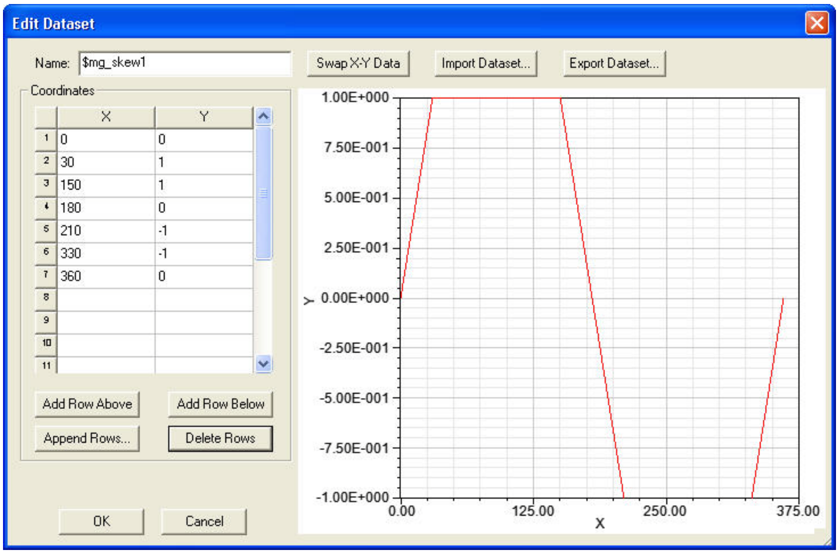
In this example, we will learn to vary the magnitude and direction of magnetization by specifying behavior curve to it

Create Design

- **To copy design**
  1. Select **Ring2** in Project Manager tree, right click and select **Copy**
  2. Select **Ex_5_7_Permanent_Magnet** in Project Manager window and select **Paste**

Create New Material : Magnet_Curve

- **To Create material Magnet_Curve:**
  - Select **RingMagnet** from tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Select option **Clone Material**
    2. In View/Edit Material window
       1. Material Name : **Magnet_Curve**
       2. Magnitude : **-890000 * pwl_periodic($mg_skew1, 2* (Phi * 180 / PI - 45 *(Z / 40 mm)))**
       3. **Add Dataset** window will pop up.
          - Enter data points as shown in image and Press **OK**
       4. R Component : **1**
       5. Press **OK**
    3. Select **OK**



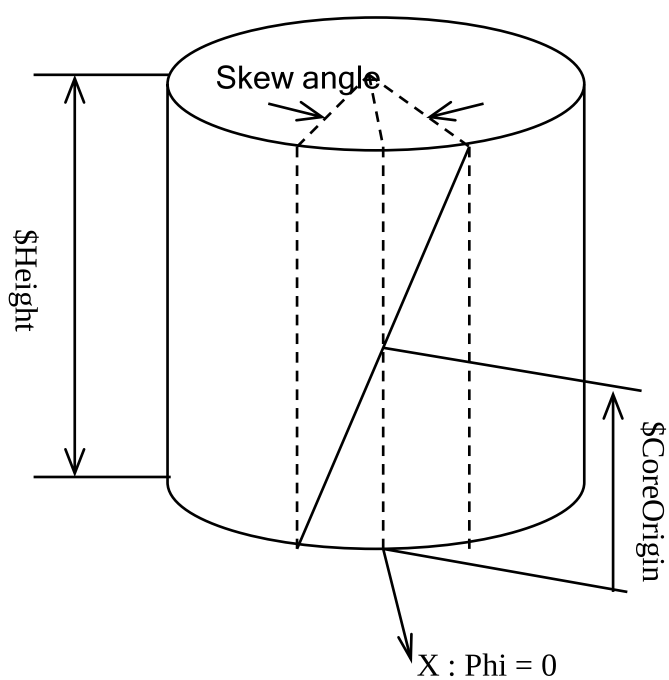
- Note : About the function

Magnetization | Number of Poles | 0 position

-837999 * pwl_periodic($mg_skew1, 2* (Phi*180/PI - 45 *((Z- $CoreOrigin / 2 mm)))

Magnetization wave form | Skew angle[deg] | Length of magnet

- The function which has been specified in the material definition sets a multiplier $mg_skew1 for the magnitude of magnetic coercivity.
- Input curve that we specified varies the multiplier $mg_Skew1 from **+1** to **-1** with the change in value of **Phi**. Hence it varies the direction of magnetization of magnets.
- Number of poles value will multiply the value of Phi. Hence ensuring value of $mag_skew1 cycles twice in single rotation of Phi. This results two peaks (North Pole) and two valleys (South Pole) giving a four pole magnet.
- Moreover, to apply the skew, magnetic data is rotated with Z axis coordinates.
- $CoreOrigin defines the Z position where value of $mg_skew becomes zero. In our case this value is 0mm. Hence it is not considered in the function specified.

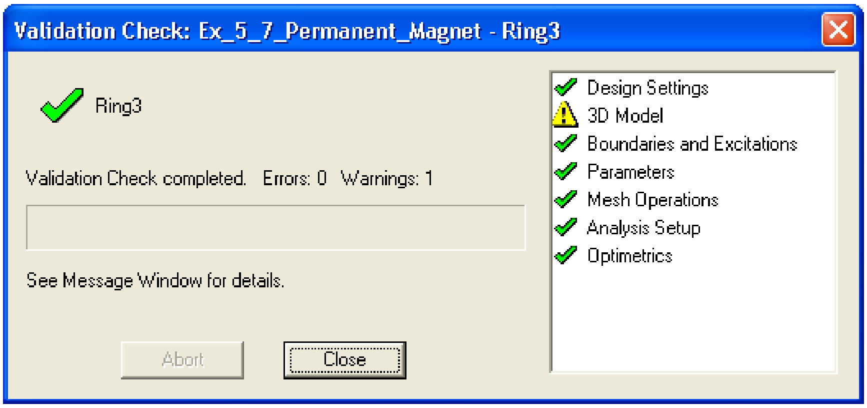


Analysis Setup

- There is already **Setup1** because it copied from the **Ring2** design. Analyze this design by the untouched setting.

Model Validation

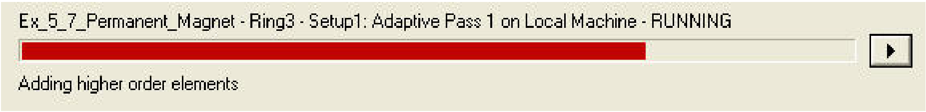
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



  - **Note:** A warning will be given by Maxwell informing if coercivity value is set to positive, the direction will be reversed.

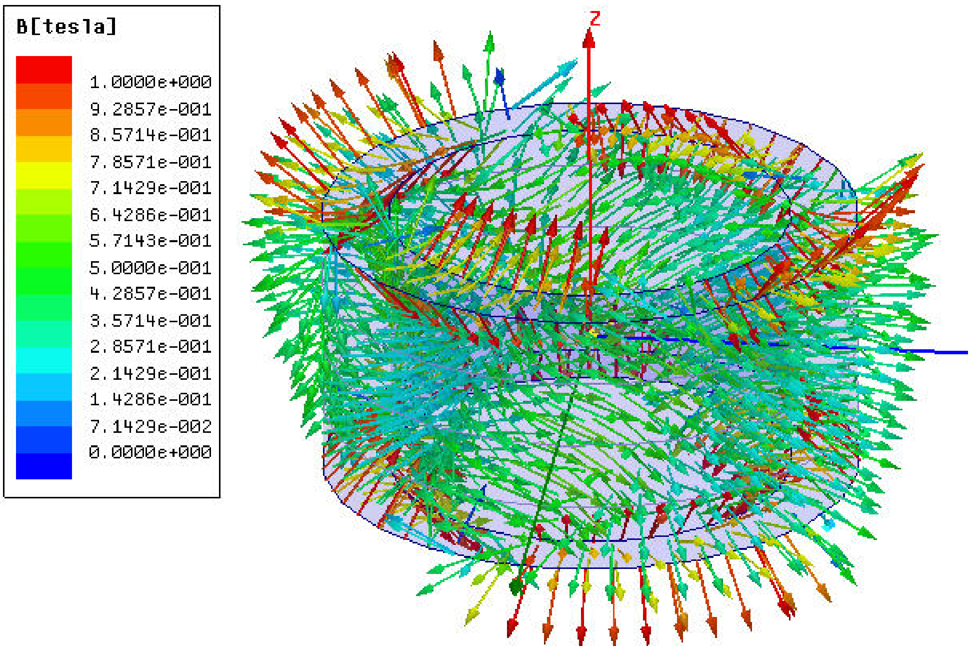
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

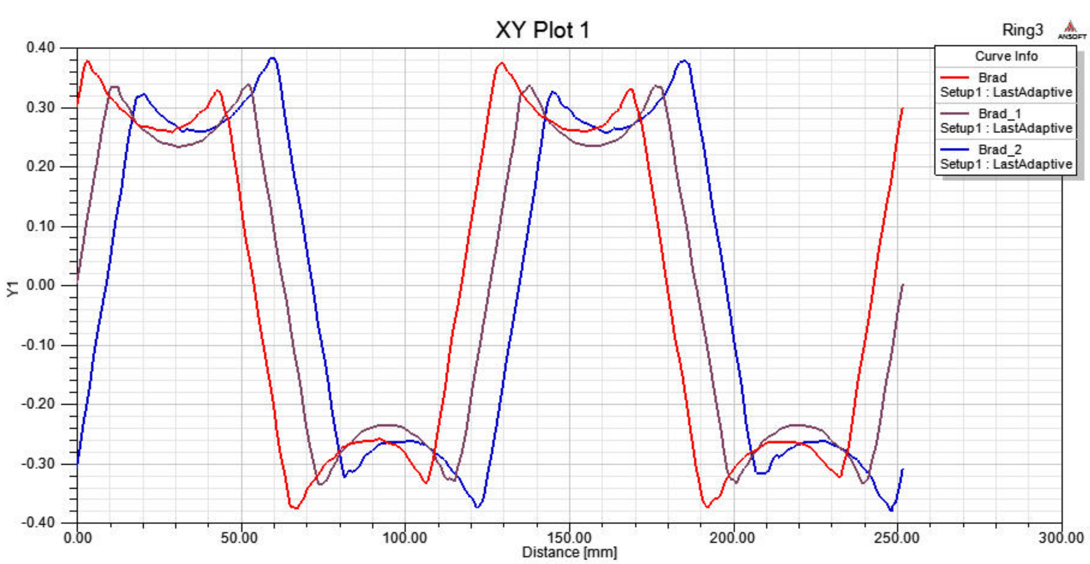


Results

- Double click ***Field Overlays > B > B_Vector*** in Project tree to view vector plot



- Check the Plots of **Brad** that we created in last exercise. The difference in the results can be seen. The strength of magnetic filed in radial direction varies with Phi. The function results in for pole magnet as expected.



Example (Magnetostatic) – Master/Slave Boundaries

Assigning Master/Slave Boundary Conditions

- **Boundary Conditions**
  - Boundary conditions enable you to control the characteristics of planes, faces, or interfaces between objects. Boundary conditions are important to understand and are fundamental to solution of Maxwell's equations.
- **Master/Slave**
  - Master and slave boundaries enable you to model planes of periodicity where the H-field (E-field for Electrostatic Solver) at every point on the slave boundary surface is forced to match the H-field of every corresponding point on the master boundary surface. The transformation used to map the H-field from the master to the slave is determined by specifying a coordinate system on both the master and slave boundaries.
- **Considerations**
  - Some considerations for Master/Slave boundaries:
    - They can only be assigned to planar surfaces.
    - The geometry of the surface on one boundary must match the geometry on the surface of the other boundary.
- **Why They are Important**
  - When used properly, Master/Slave boundary conditions can be successfully utilized to reduce the model complexity.
  - The model complexity usually is directly tied to the solution time and computer resources so it is a competitive advantage to utilize them whenever possible.

Example (Magnetostatic) – Master/Slave Boundaries

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - User Defined Primitives (UDP): **SRMCore**
    - Surface Operations: **Section**
    - Boolean Operations: **Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
    - Boundaries: **Master/Slave**
  - Analysis
    - **Magnetostatic**
  - Field Overlays:
    - **H Vector**

Example (Magnetostatic) – Master/Slave Boundaries

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

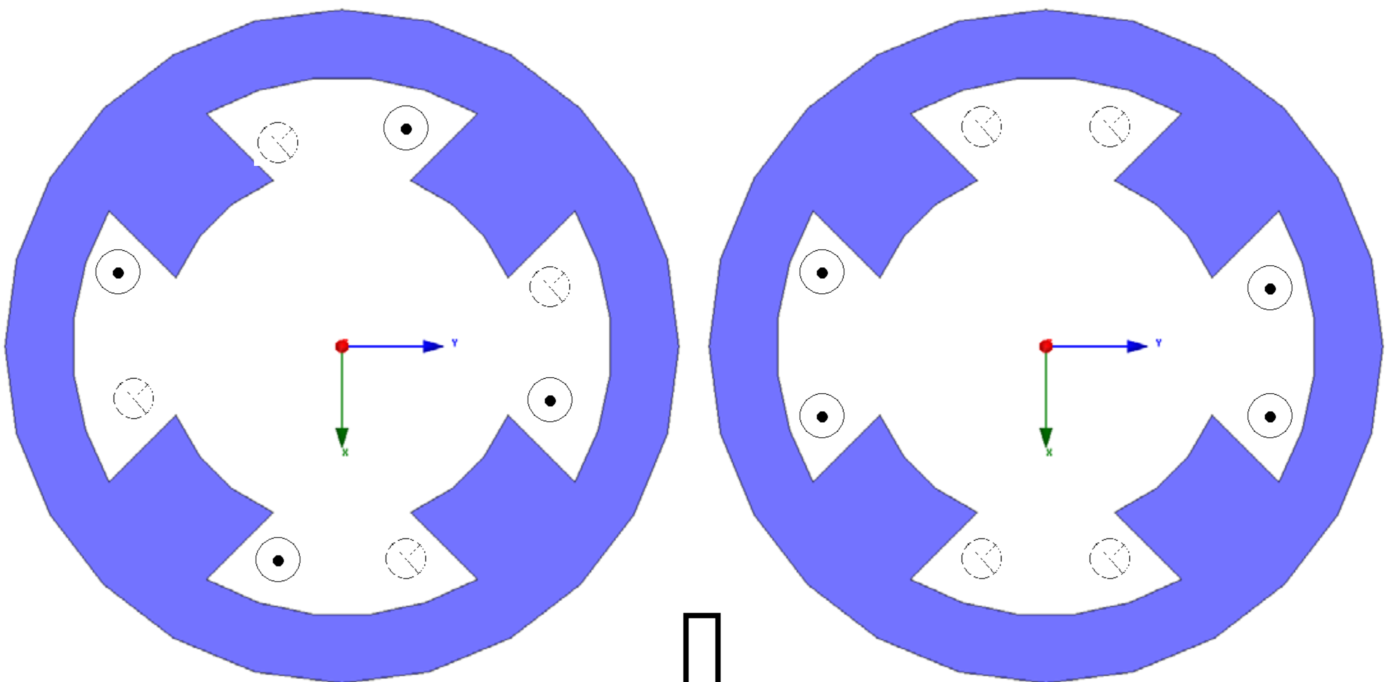
Setting Tool Options

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

Example (Magnetostatic) – Master/Slave Boundaries

Designs in the Tutorial

- There will be four designs in this example, as shown below. The objective is to demonstrate how to assign Master/Slave boundary conditions and how they can help reduce model complexity and hence solution time.

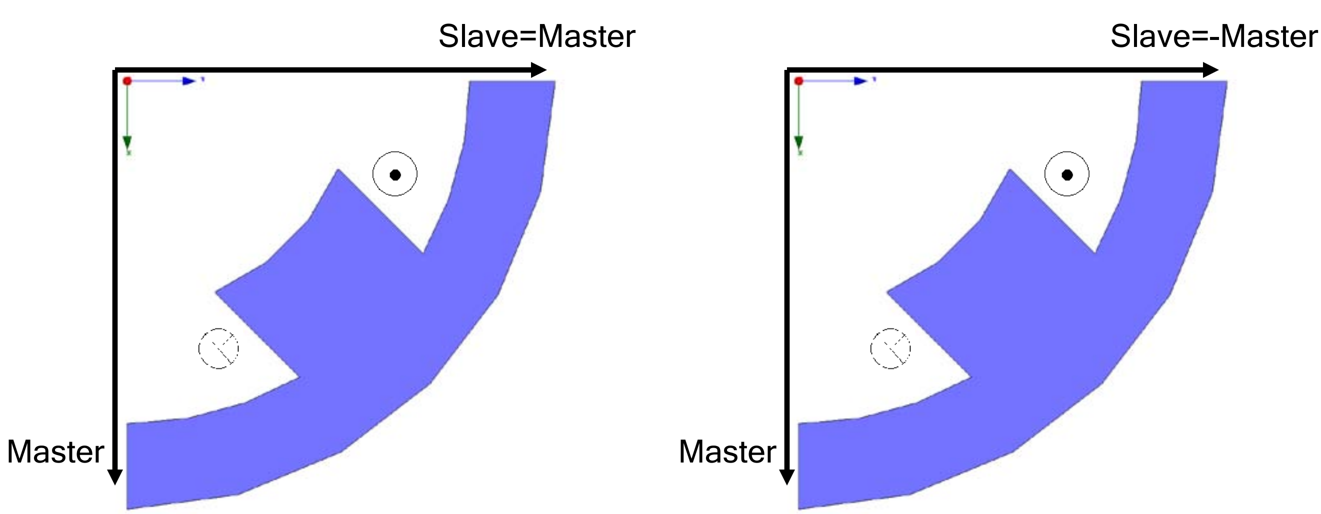


Design1: Full_Model_1

Design2: Full_Model_2

Design3: Quarter_Model_1, Slave=Master

Design4: Quarter_Model_2, Slave=-Master



Example (Magnetostatic) – Master/Slave Boundaries

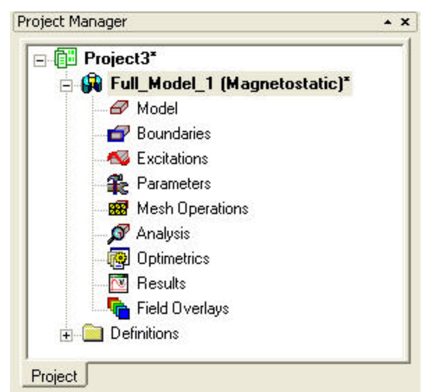
Problem1: Solve Full_Model_1

Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

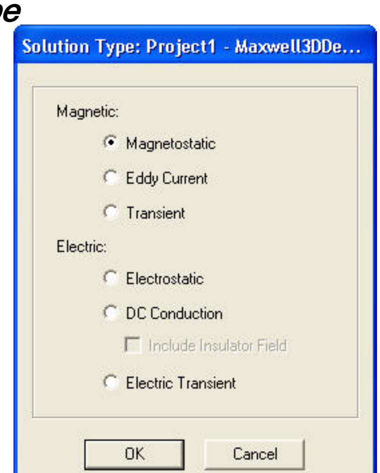
Change Design name

- **To Change Design Name:**
  - Right-click **Maxwell3D Design1** at the project manager window and select **Rename.**
  - Change the Design Name to **Full_Model_1**



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



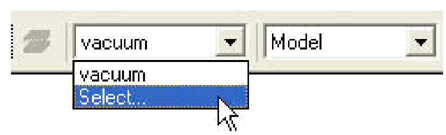
Example (Magnetostatic) – Master/Slave Boundaries

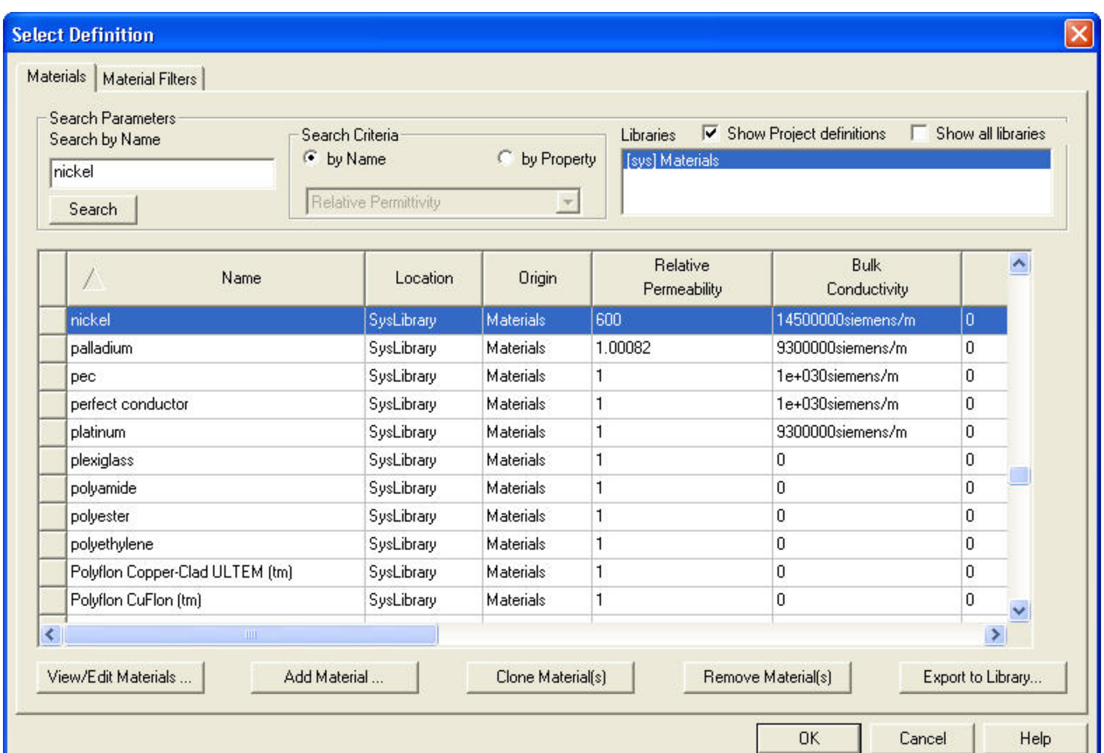
Set Model Units

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

Set Default Material

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





Create Stator and Coils

- **To create Stator and Coils**
  - Select the menu item ***Draw > User Defined Primitives > SysLib > RMxprt > SRMCore***
  - In User Defined Primitive Operation window
    1. For the value of **DiaGap,** type: **80**, Click the **Tab** key to accept
    2. For the value of **DiaYoke**, type: **150**, Click the **Tab** key to accept
    3. For the value of **Length**, type: **10**, Click the **Tab** key to accept
    4. For the value of **Poles**, type: **4**, Click the **Tab** key to accept
    5. For the value of **InfoCore**, type: **1**, Click the **Tab** key to accept
    6. Click the **OK** button

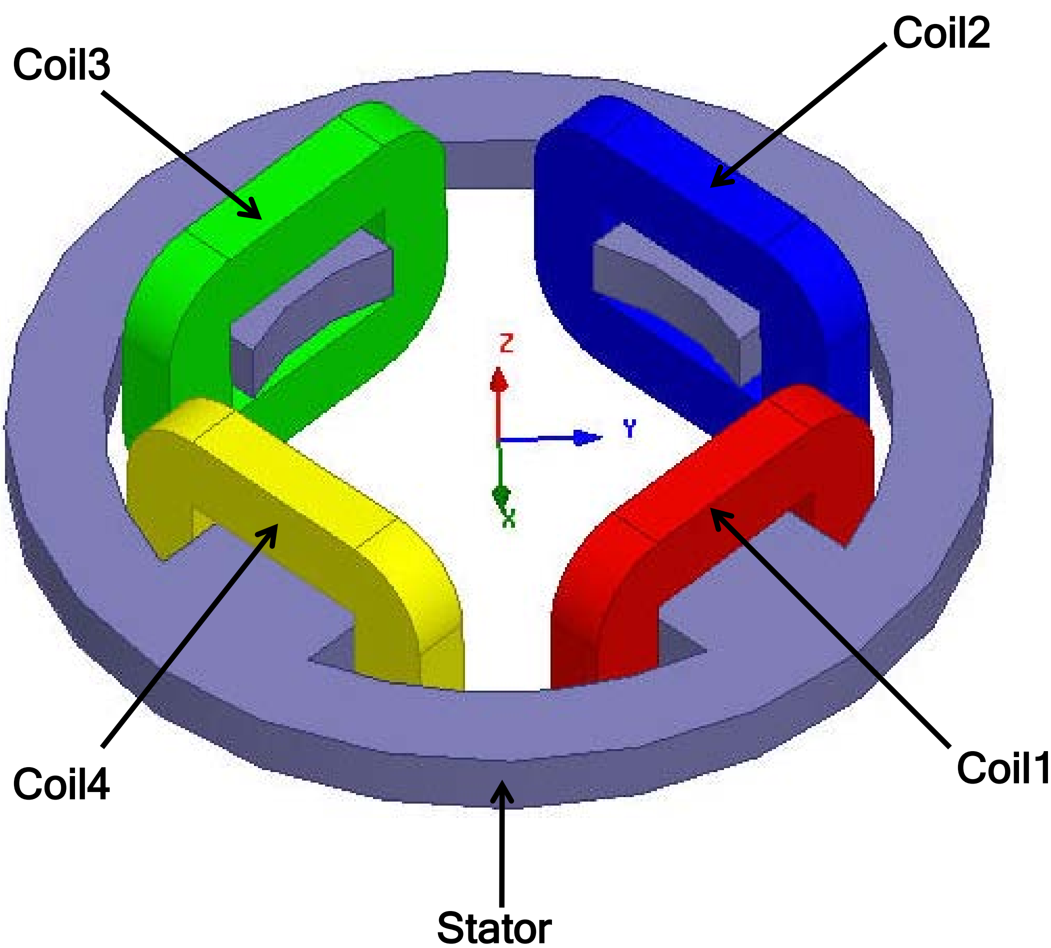
Parameters | Info

Name	Value	Unit	Evaluate...	Description
Command	CreateUserDefine...			
Coordinate Sys...	Global			
DLL Name	RMxprt/SRMCore			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	80	mm	80mm	Core diameter on gap side, DiaGap<...
DiaYoke	150	mm	150mm	Core diameter on yoke side, DiaYoke...
Length	10	mm	10mm	Core length
Poles	4		4	Number of poles
ThkYoke	15	mm	15mm	Yoke thickness
Embrace	0.5		0.5	Pole embrace (the ratio of pole arc to ...
EndExt	5	mm	5mm	Coil one-side end extended length
LenRegion	200	mm	200mm	Region length
InfoCore	1		1	0: core; 1: core & coils; 2: coil; 3: term...

- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Stator**
- **Separate Objects**
  - Select the object **Stator** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***

Example (Magnetostatic) – Master/Slave Boundaries

- **Change Attributes**
  - Select the objects from the tree and goto Properties window
    1. Change the name of the object **Stator_Separate1** to **Coil1** and Color to **Red**
    2. Change the name of the object **Stator_Separate2** to **Coil3** and Color to **Green**
    3. Change the name of the object **Stator_Separate3** to **Coil2** and Color to **Blue**
    4. Change the name of the object **Stator_Separate4** to **Coil4** and Color to **Yellow**

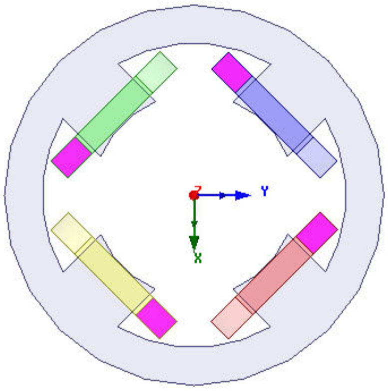


Assign Material for Coils

- **To Assign Material**
  - Press **Ctrl** and select the objects **Coil1, Coil2, Coil3** and **Coil4**, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **copper** in the **Search by Name** field
    2. Click the **OK** button

Create Coil Terminals

- **Create Section**
  - Press **Ctrl** and select the objects **Coil1, Coil2, Coil3** and **Coil4** from the tree
  - Select the menu item ***Modeler > Surface >Section***
  - In Section window,
    1. Section Plane: **XY**
    2. Click the **OK** button
- **Change Attributes**
  - Select the sheets from the tree and goto Properties window
    1. Change the name of the sheets **Coil1_Section1, Coil2_Section1, Coil3_Section1** and **Coil4_Section1** to **Term1, Term2, Term3** and **Term4** respectively
- **Separate Sections**
  - Press **Ctrl** and select sheets **Term1, Term2, Term3** and **Term4** from tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Press **Ctrl** and select the sheets **Term1_Separate1, Term2_Separate1, Term3_Separate1** and **Term4_Separate1** from the tree
  - Select the menu item ***Edit > Delete***

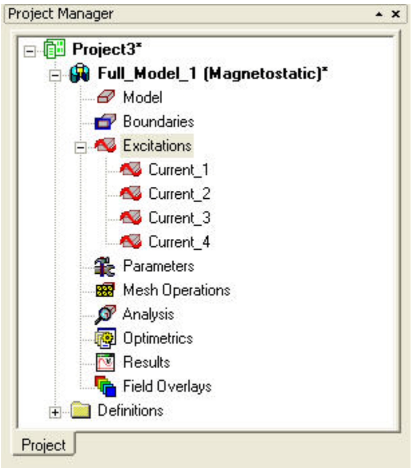


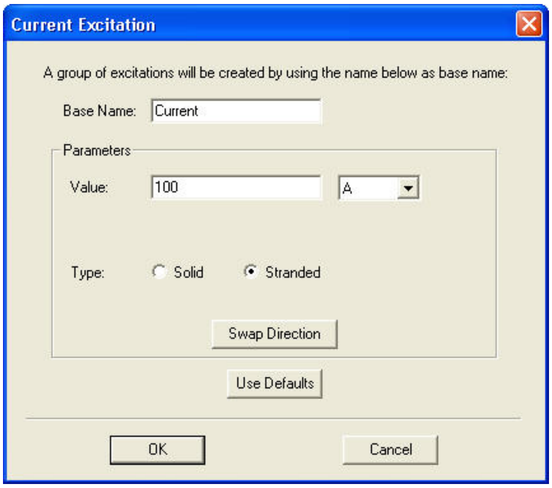
Example (Magnetostatic) – Master/Slave Boundaries

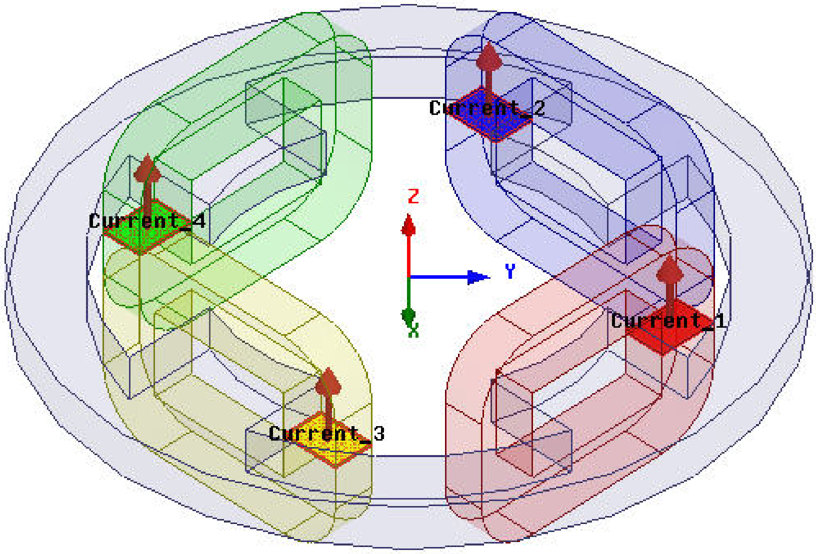
Create Excitations

To Assign Excitations

- Press **Ctrl** and select the sheets **Term1, Term2, Term3** and **Term4**
- Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
- In Current Excitation window,
  1. Name: **Current**
  2. Value: **100 A**
  3. Type: **Stranded**
  4. Direction: **Positive Z**(Use Swap Direction if needed)
  5. Press **OK**







Example (Magnetostatic) – Master/Slave Boundaries

Set Default Material

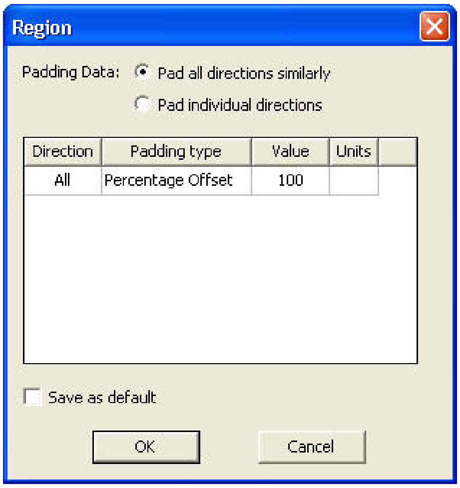
To Set Default Material

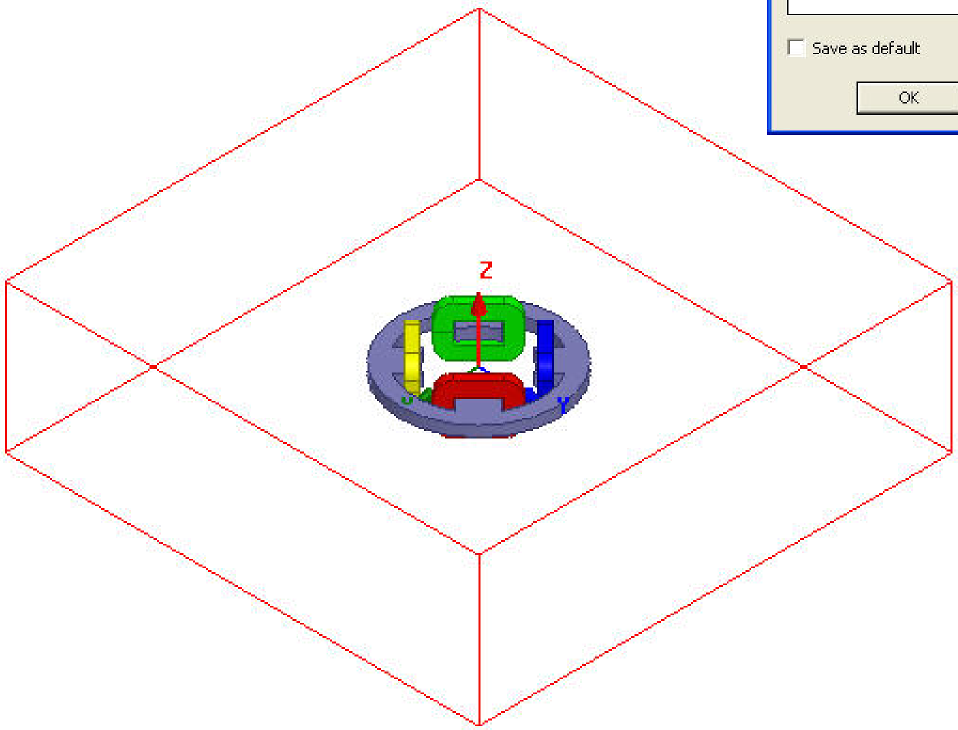
- Using the 3D Modeler Materials toolbar, choose **Vacuum**

Define Region

Create Simulation Region

- Select the menu item ***Draw > Region***
- In Region window,
  1. Pad all directions similarly: ☑ **Checked**
  2. Padding Type: **Percentage Offset**
  3. Value: **100**
  4. Press **OK**





Example (Magnetostatic) – Master/Slave Boundaries

Analysis Setup

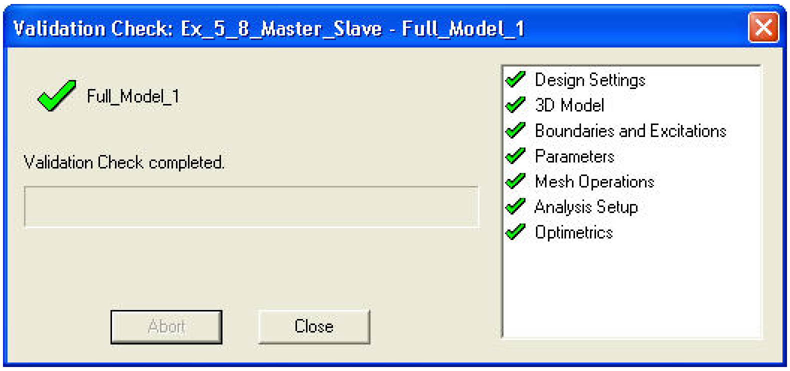
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **Ex_5_8_Master_Slave**
  3. Click the **Save** button

Model Validation

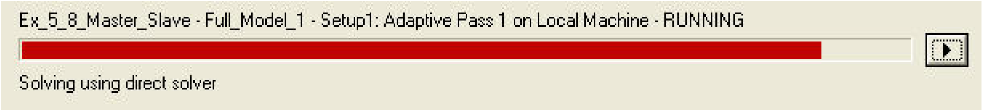
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

Analyze

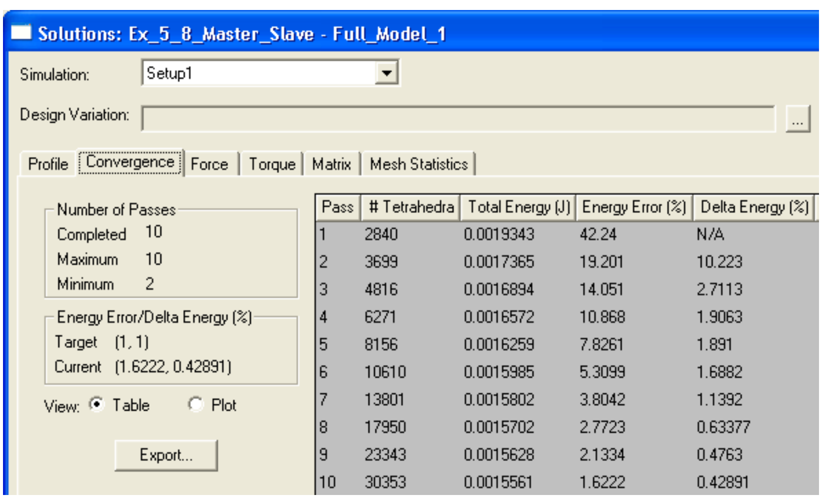
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***



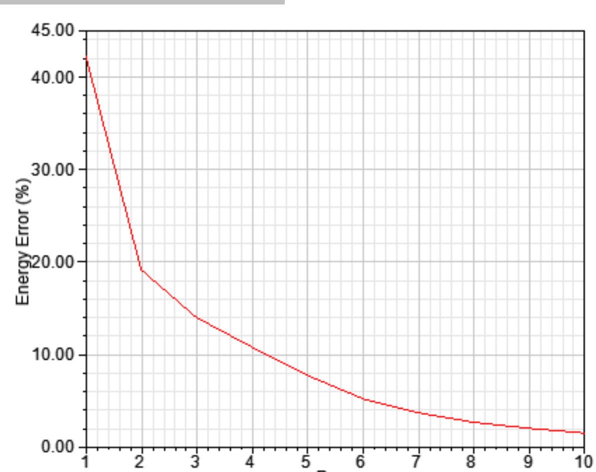
Example (Magnetostatic) – Master/Slave Boundaries

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To View Mesh information
      1. Click **Mesh Statistics** Tab
    - Select **Close** to close the window



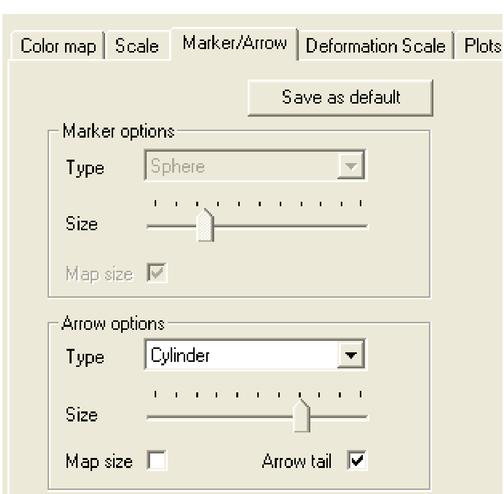
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	2840	0.0019343	42.24	N/A
2	3699	0.0017365	19.201	10.223
3	4816	0.0016894	14.051	2.7113
4	6271	0.0016572	10.868	1.9063
5	8156	0.0016259	7.8261	1.891
6	10610	0.0015985	5.3099	1.6882
7	13801	0.0015802	3.8042	1.1392
8	17950	0.0015702	2.7723	0.63377
9	23343	0.0015628	2.1334	0.4763
10	30353	0.0015561	1.6222	0.42891

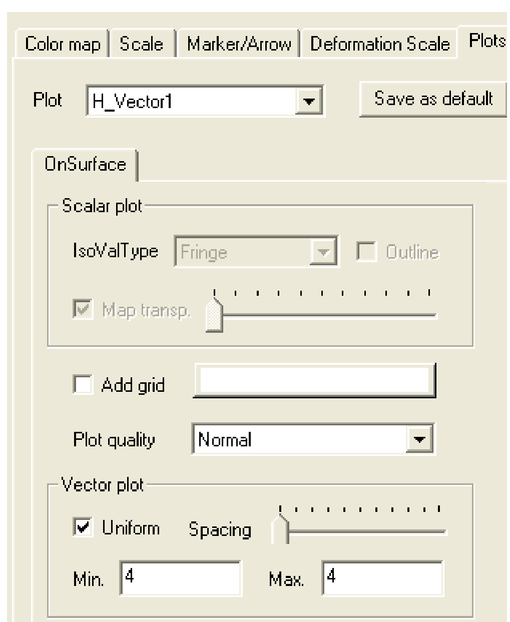


Example (Magnetostatic) – Master/Slave Boundaries

Vector Plot

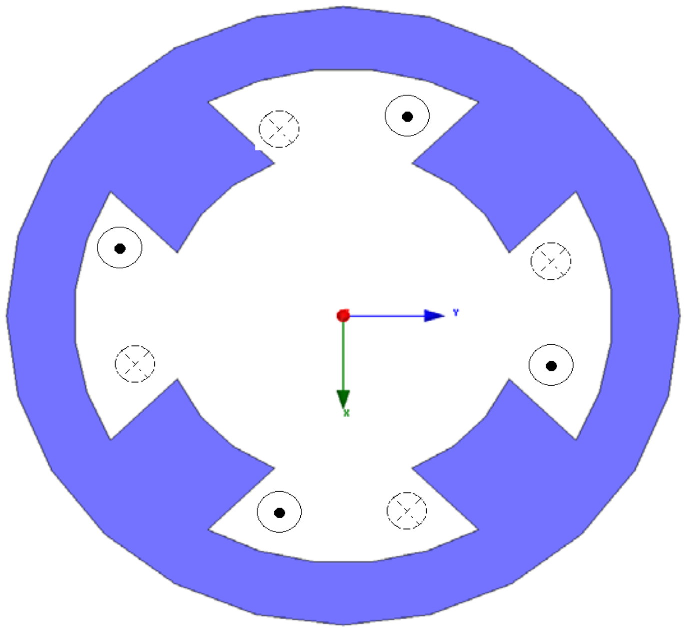
- **Create Vector Plot**
  - Select **Global:XY** plane from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > H > H_Vector***
  - In Create Field Plot window
    1. In Volume: **AllObjects**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Set to appropriate value**
         2. Map Size: ☐ **Unchecked**
    2. **Plots** tab
       - Vector Plot
         1. Spacing: **Set to Minimum**
         2. Min: **4**
         3. Max: **4**
    3. Press **Apply** and **Close**

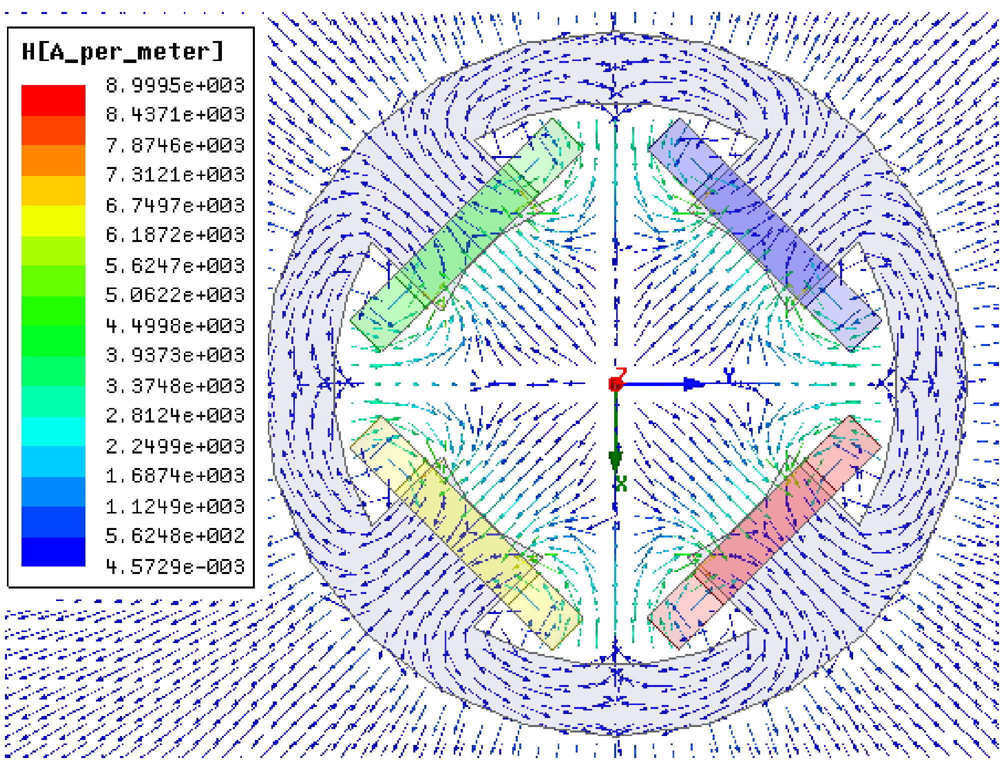




Example (Magnetostatic) – Master/Slave Boundaries

- H-field vector plot will appear, which is the result of the current excitation shown below





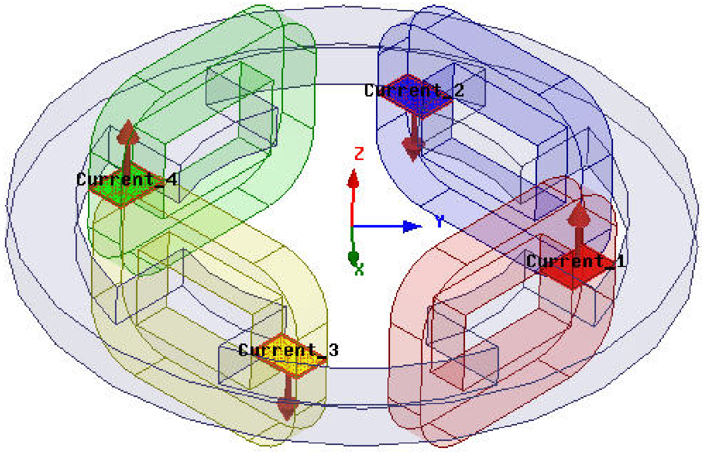
Problem 2: Solve Full_Model_2

Create Symmetry Design

- **Copy Design**
  1. Select the design **Full_Model_1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_8_Master_Slave** in Project Manager window and select **Paste**
  3. Change the name of the design to **Full_Model_2**

Modify Excitations

- **Modify Direction of Current in Coil2**
  - Expand the tree Excitations in Project Manager window
  - Double click on excitation corresponding to **Coil2** (in this case **Current2**)
  - In Current Excitation window,
    - Use **Swap Direction** (Set direction to Negative Z)
    - Press **Ok**
- **Modify Direction of Current in Coil4**
  - Expand the tree Excitations in Project Manager window
  - Double click on excitation corresponding to **Coil4** (in this case **Current3**)
  - In Current Excitation window,
    - Use **Swap Direction** (Set direction to Negative Z)
    - Press **Ok**



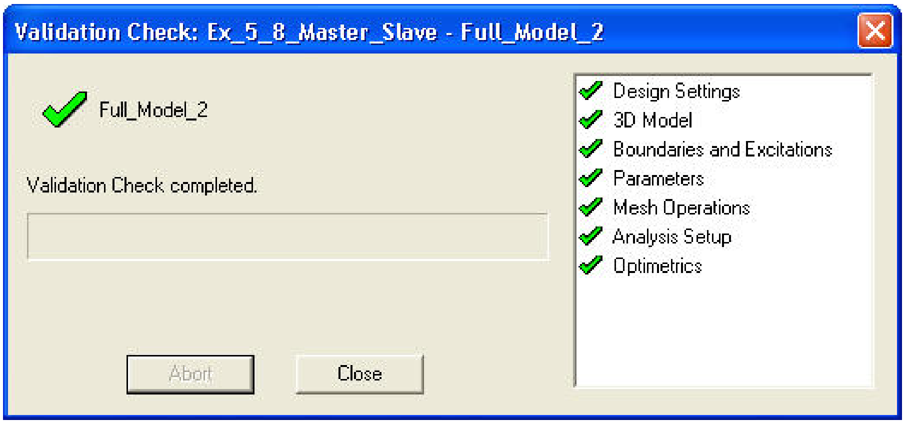
Example (Magnetostatic) – Master/Slave Boundaries

Analysis Setup

- There is already **Setup1** because it copied from the **Full_Model_2** design. Analyze this design by the untouched setting.

Model Validation

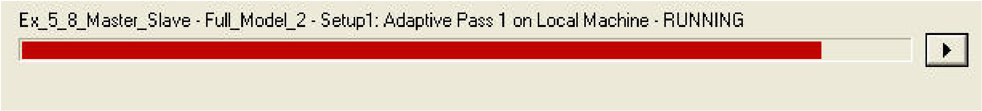
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

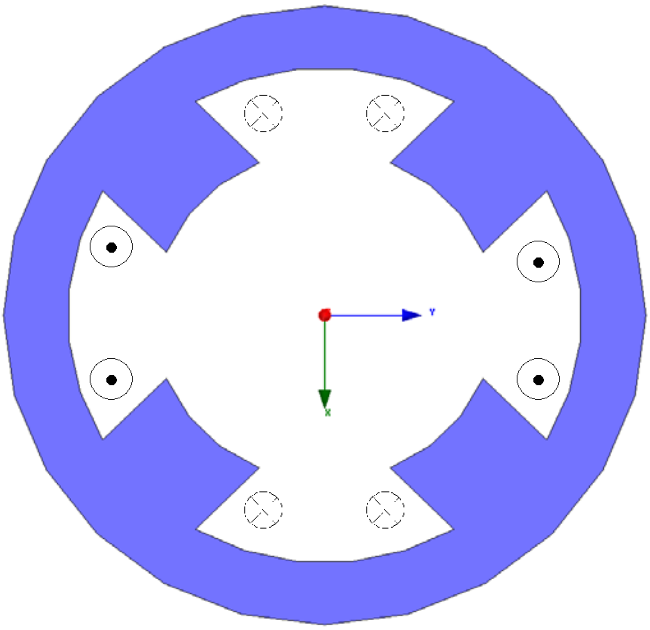
Analyze

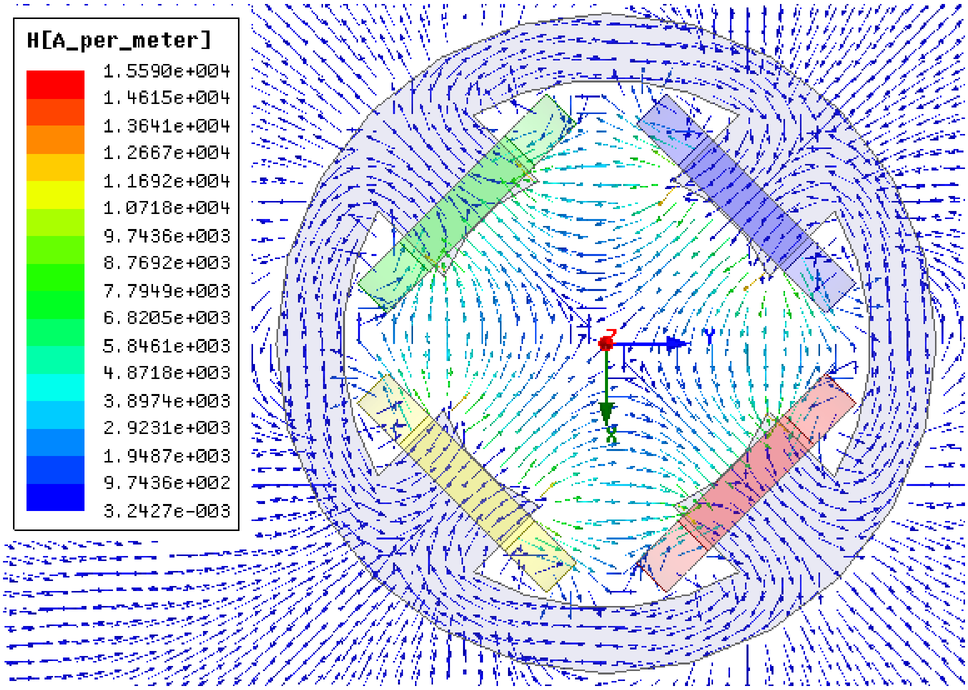
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***



Field Plots

- Field plots are already available as design was copied from **Full_Model_1**
- Double click ***Field Overlays > Field > H > H_Vector_1*** from Project Manager tree





Example (Magnetostatic) - Master/Slave Boundaries

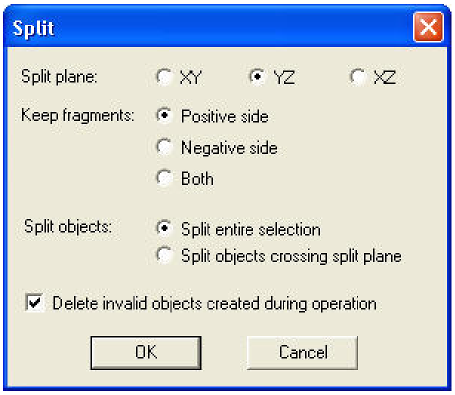
Problem 3: Solve Quarter Geometry with Master = Slave

Create Symmetry Design

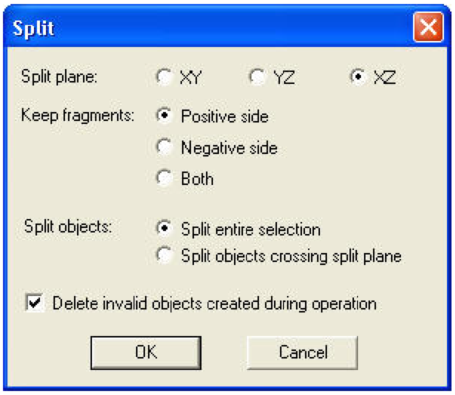
- **Copy Design**
  1. Select the design **Full_Model_1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_8_Master_Slave** in Project Manager window and select **Paste**
  3. Change the name of the design to **Quarter_Model_1**

Split Model for 1/4th Section

- **Divide by YZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



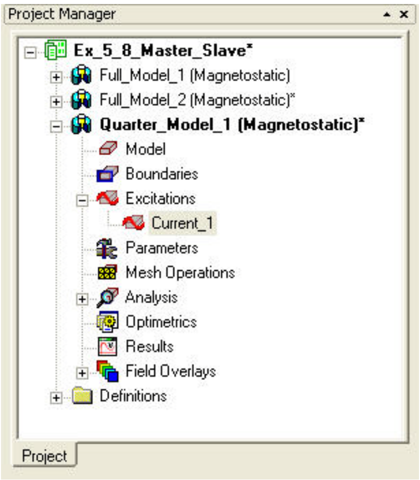
- **Divide by XZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

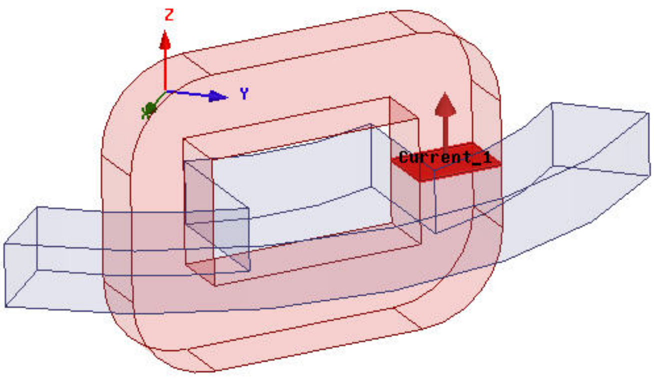


Example (Magnetostatic) – Master/Slave Boundaries

Excitations

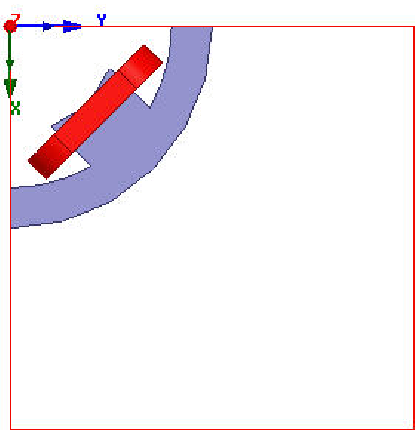
- As we are considering 1/4th section of geometry, apart from **Coil1**, all other coils have been removed. Excitation assigned to **Coil1** is available from the first simulation.

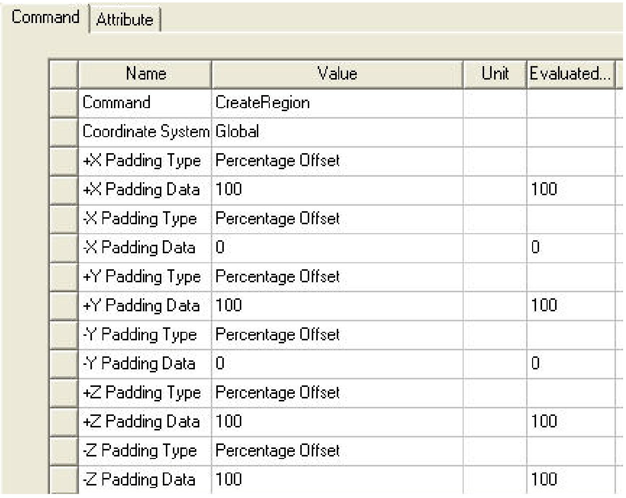




Modify Region

- **To Modify Region**
  - Select the menu item ***Draw > Region***
  - In Properties window of Region
    1. Change -X Padding Data to **0**
    2. Change -Y Padding Data to **0**
    3. Press **OK**



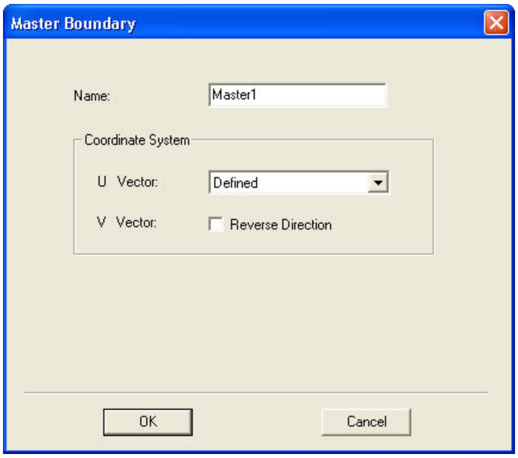


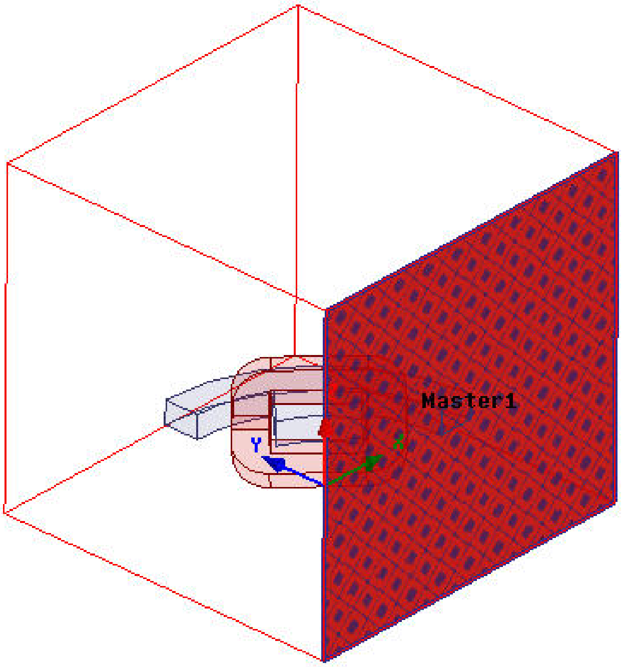
Name	Value	Unit	Evaluated...
Command	CreateRegion		
Coordinate System	Global		
+X Padding Type	Percentage Offset		
+X Padding Data	100		100
-X Padding Type	Percentage Offset		
-X Padding Data	0		0
+Y Padding Type	Percentage Offset		
+Y Padding Data	100		100
-Y Padding Type	Percentage Offset		
-Y Padding Data	0		0
+Z Padding Type	Percentage Offset		
+Z Padding Data	100		100
-Z Padding Type	Percentage Offset		
-Z Padding Data	100		100

Example (Magnetostatic) – Master/Slave Boundaries

Assign Boundaries

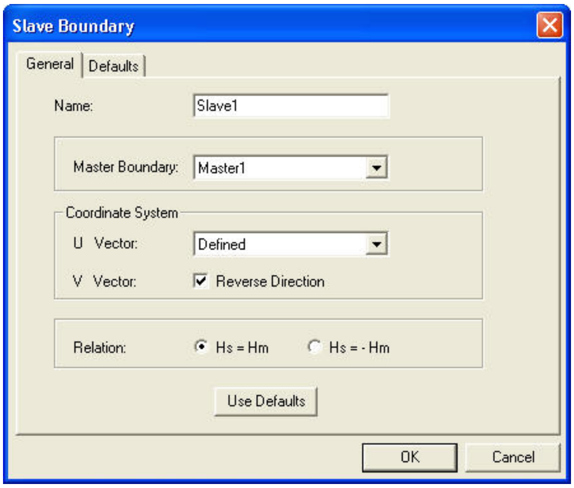
- **Assign Master Boundary**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard to change selection to faces
  - Select the face of the Region which touch with XZ Plane
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Master***
  - In Master Boundary window
    1. U Vector: Set the option to **New Vector**
        1. Using the coordinate entry fields, enter the first vertex
            - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
        2. Using the coordinate entry fields, enter the second vertex
            - dX: **0**, dY: **0**, dZ: **1**, Press the **Enter** key
    2. Press **OK**

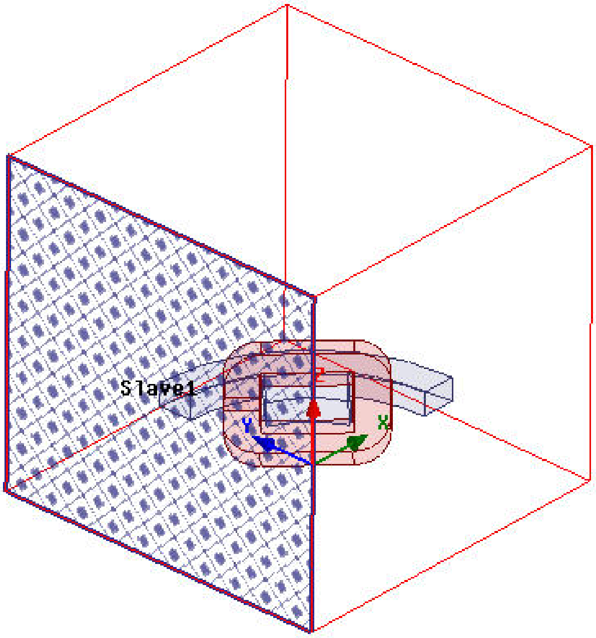




Example (Magnetostatic) – Master/Slave Boundaries

- **Assign Slave Boundary**
  - Select the face of the Region with which touch with YZ Plane
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Slave***
  - In Slave Boundary window
    1. Master boundary: Select **Master1**
    2. U Vector: Set the option to **New Vector**
        1. Using the coordinate entry fields, enter the first vertex
            - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
        2. Using the coordinate entry fields, enter the second vertex
            - dX: **0**, dY: **0**, dZ: **1**, Press the **Enter** key
    3. Relation: Select **Hs=Hm**
    4. Press **OK**





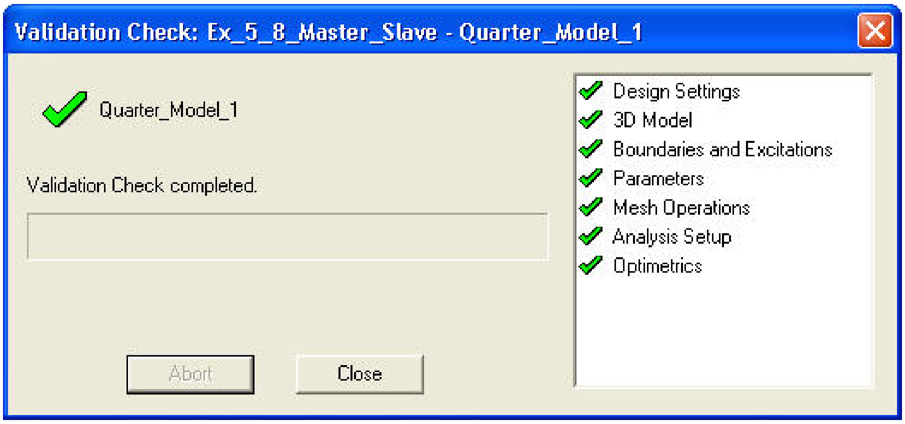
Example (Magnetostatic) – Master/Slave Boundaries

Analysis Setup

- There is already **Setup1** because it copied from the **Full_Model_1** design. Analyze this design by the untouched setting.

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

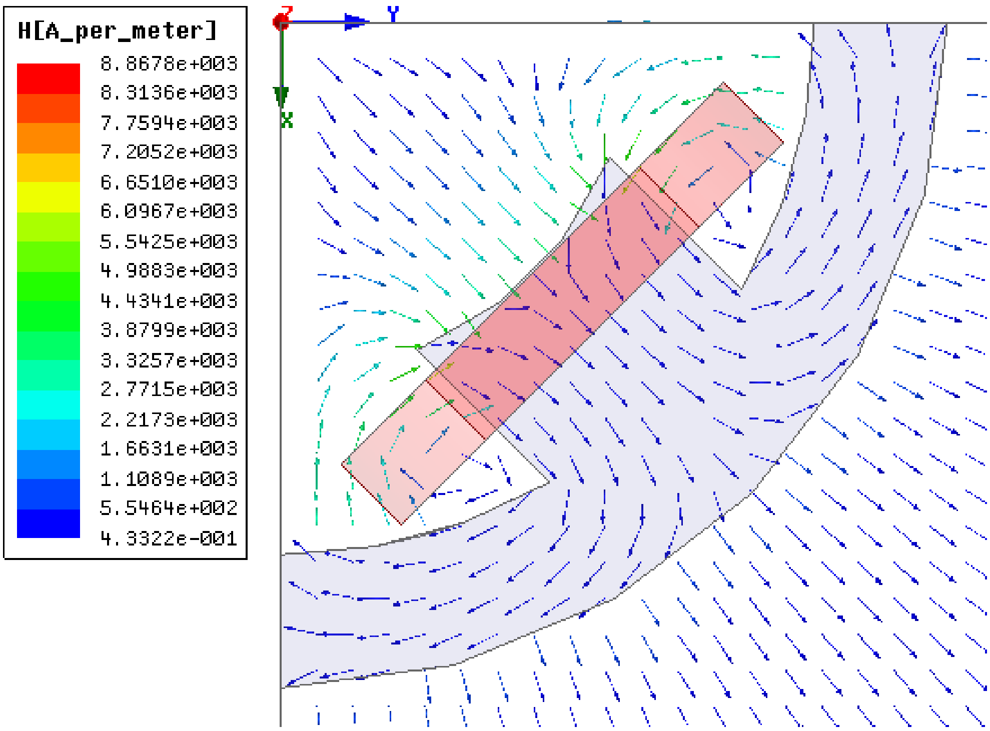
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Example (Magnetostatic) – Master/Slave Boundaries

Field Plots

- Field plots are already available as design was copied from **Full_Model_1**
- Double click ***Field Overlays > Field > H > H_Vector_1*** from Project Manager tree



Example (Magnetostatic) – Master/Slave Boundaries

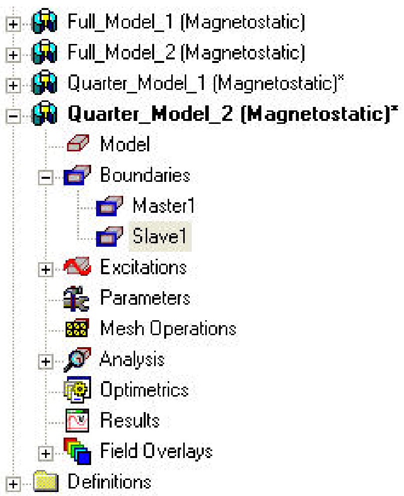
Problem 3: Solve Quarter Geometry with Salve= -Master

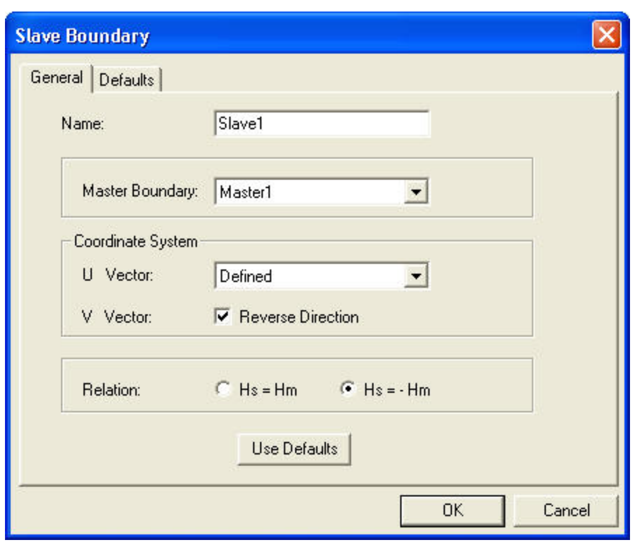
Create Design

- **Copy Design**
  1. Select the design **Quarter_Model_1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_5_8_Master_Slave** in Project Manager window and select **Paste**
  3. Change the name of the design to **Quarter_Model_2**

Modify Boundary

- **Modify Slave Boundary to achieve Slave = -Master**
  - Expand the tree Boundaries in Project Manager window
  - Double click on Slave1 from the tree
  - In Slave Boundary window
    1. Relation: Change to **Hs=-Hm**
    2. Press **OK**





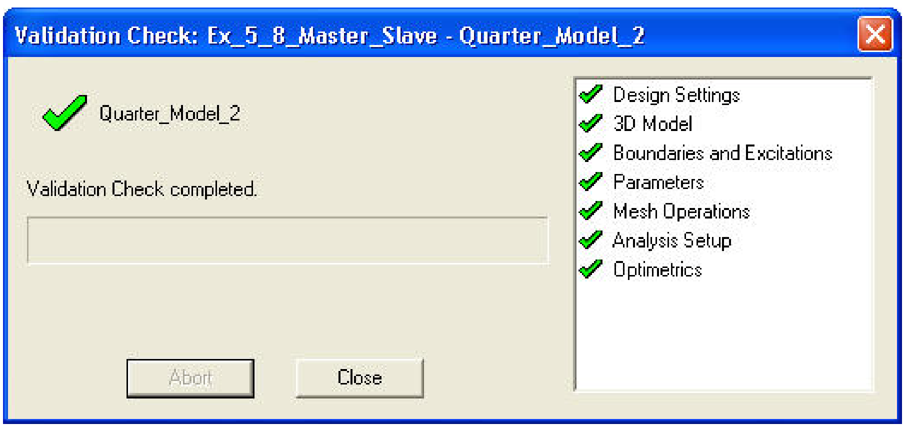
Example (Magnetostatic) – Master/Slave Boundaries

Analysis Setup

- There is already **Setup1** because it copied from the **Quarter_Model_1** design. Analyze this design by the untouched setting.

Model Validation

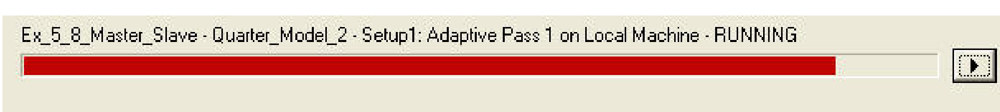
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

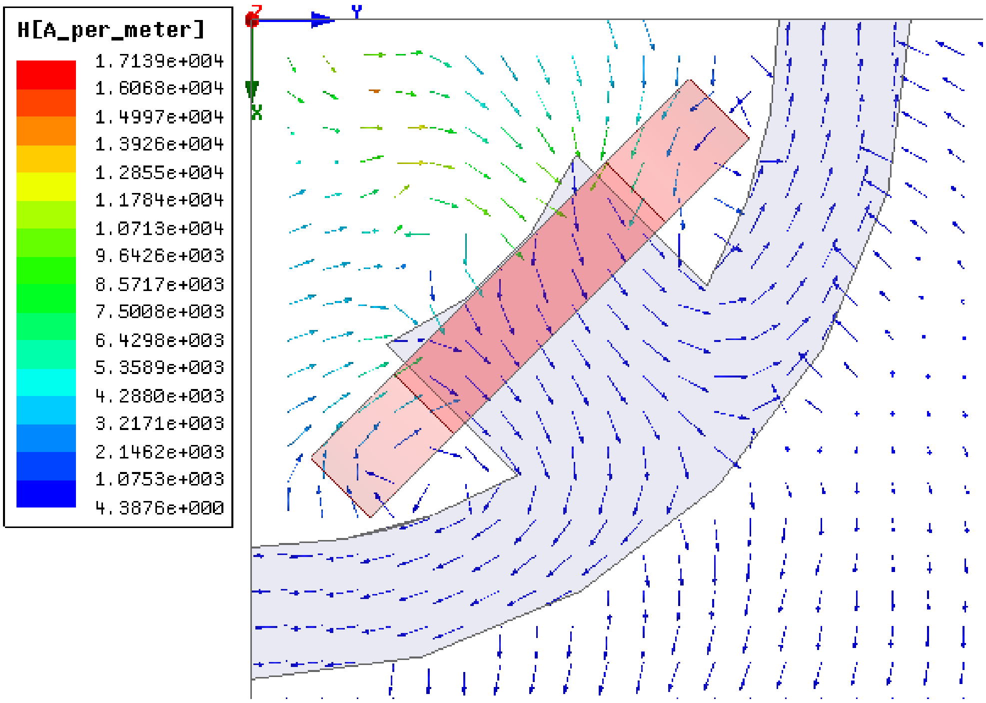
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***



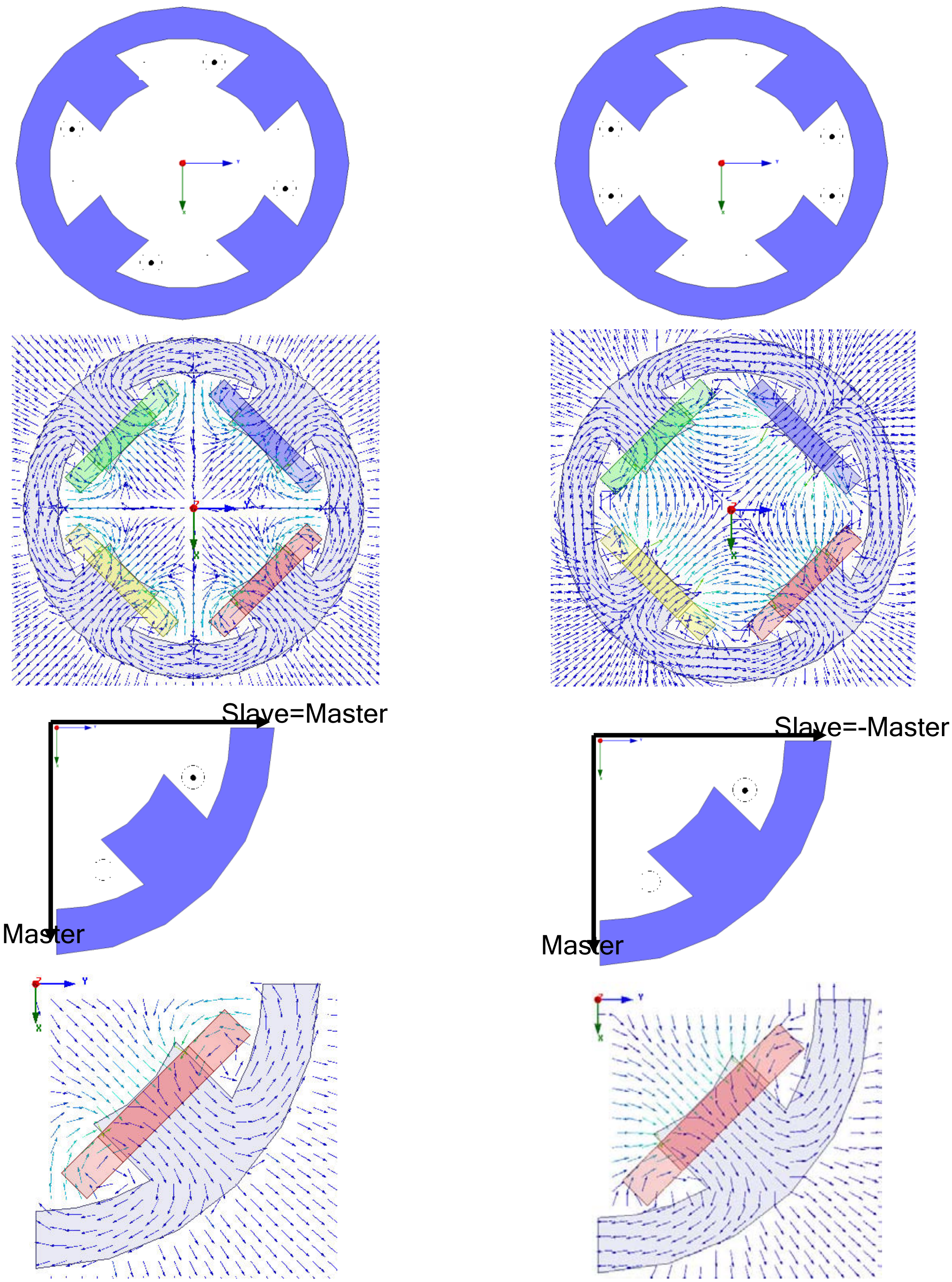
Field Plots

- Field plots are already available as design was copied from **Quarter_Model_1**
- Double click ***Field Overlays > Field > H > H_Vector_1*** from Project Manager tree



Example (Magnetostatic) – Master/Slave Boundaries

Topic Summary



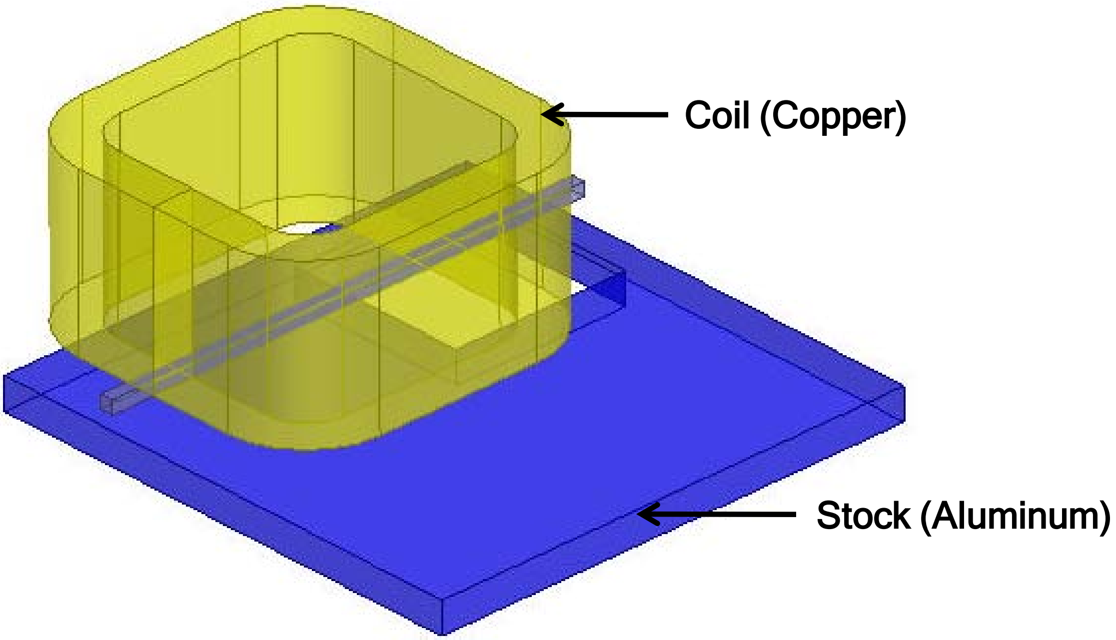
Chapter 6.0 – Eddy Current Examples

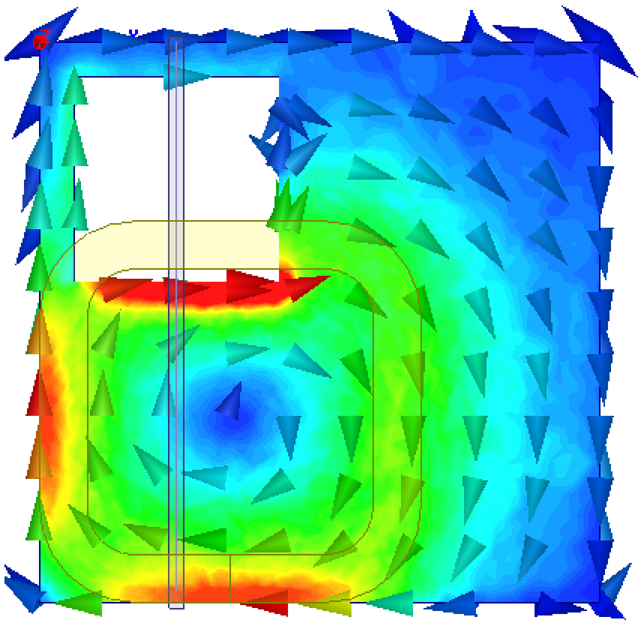
- 6.1 – Asymmetrical Conductor with a Hole
- 6.2 – Radiation Boundary
- 6.3 - Instantaneous Forces on Busbars

Example (Eddy Current) - Asymmetric Conductor

The Asymmetrical Conductor with a Hole

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





Example (Eddy Current) - Asymmetric Conductor

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Box**
    - Surface Operations: **Section**
    - Boolean Operations: **Subtract, Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Eddy Current**
  - Results
    - **Calculator Expressions**
    - **Plots**
  - Field Overlays:
    - **Mag_J**

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

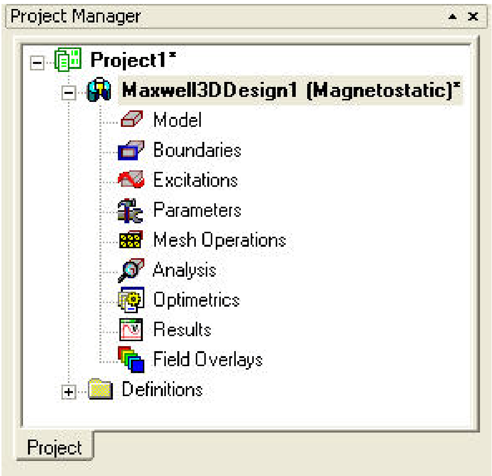
Setting Tool Options

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

Example (Eddy Current) - Asymmetric Conductor

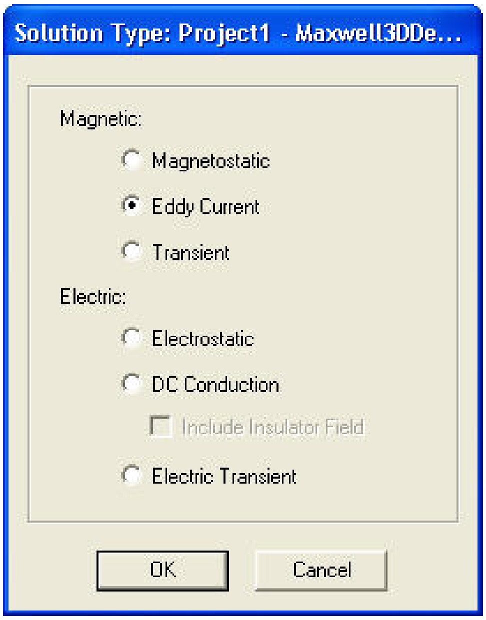
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Eddy Current**
    2. Click the **OK** button

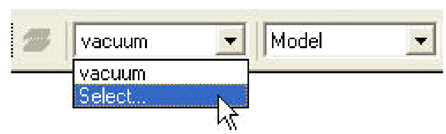


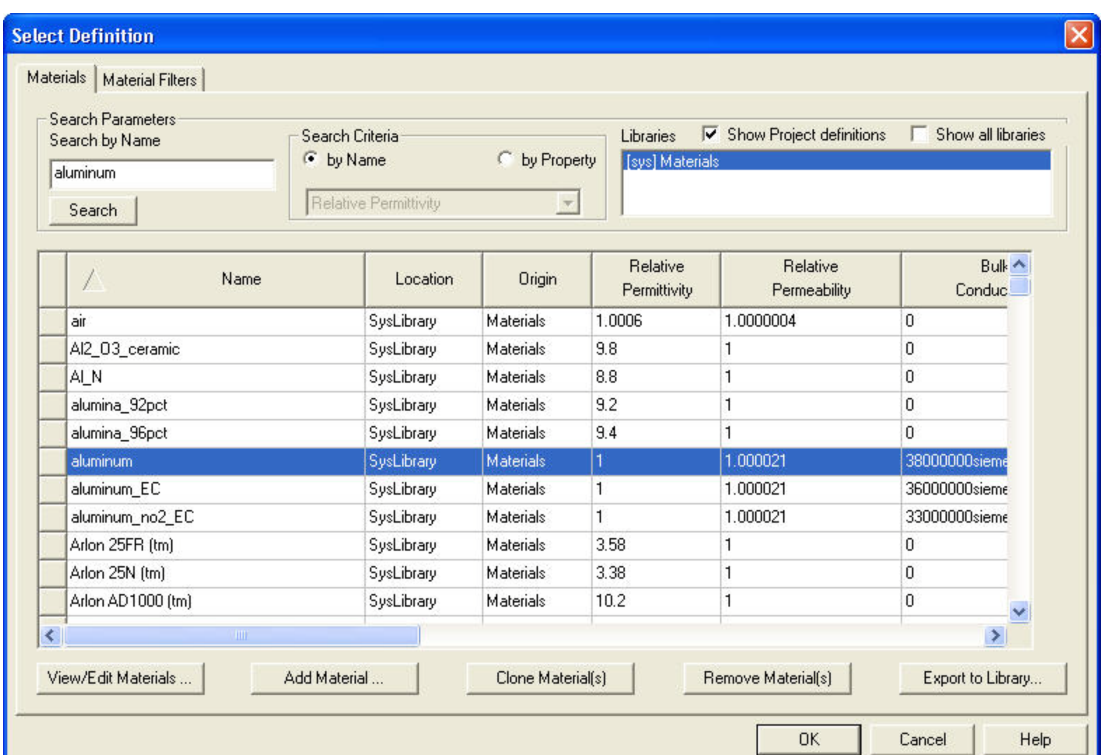
Set Model Units

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

Set Default Material

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

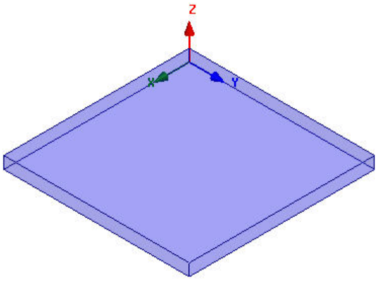




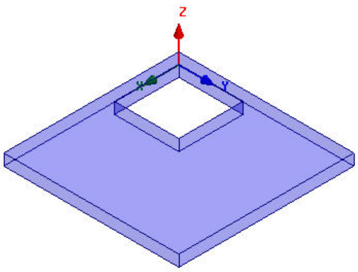
Example (Eddy Current) - Asymmetric Conductor

Create Stock

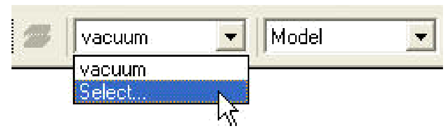
- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **294**, dY: **294**, dZ: **19**, Press the **Enter** key
  - Select the menu item ***View > Visibility > Fit All > Active View.***
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Stock**
    2. Change its color to **Blue**



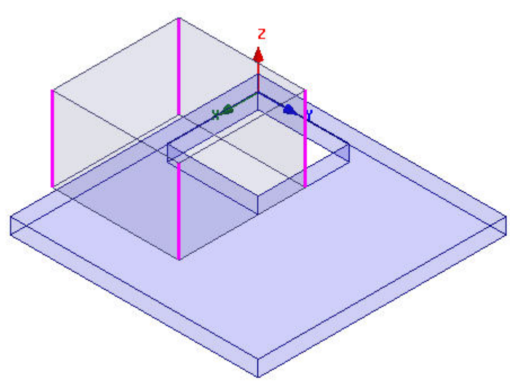
- **Create Hole**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **18**, Y: **18**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **108**, dY: **108**, dZ: **19**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Hole**
- **Subtract Objects**
  - Select the menu item ***Edit > Select All***
  - Select the menu item, ***Modeler > Boolean > Subtract***
    1. Blank Parts: **Stock**
    2. Tool Parts: **Hole**
    3. Click the **OK** button

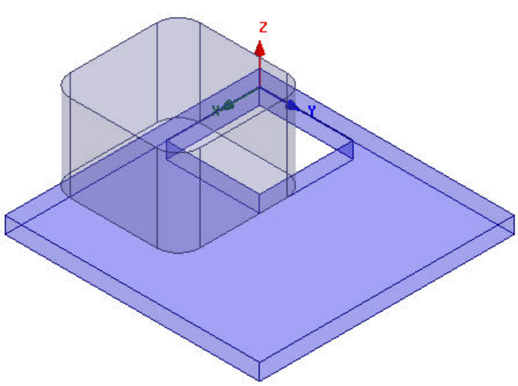


- ## Set Default Material
  - **To set the default material:**
    - Using the 3D Modeler Materials toolbar, choose **Select**
    - In Select Definition window,
      1. Type **Copper** in the **Search by Name** field
      2. Click the **OK** button



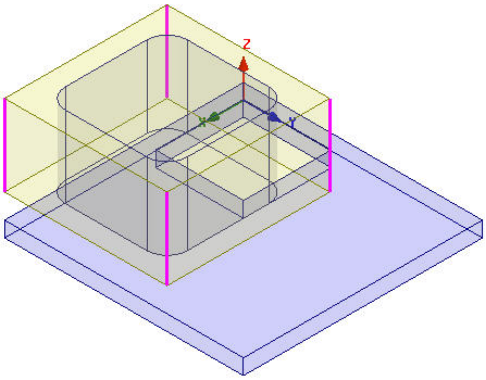
- ## Create Coil
  - **Create Coil_Hole**
    - Select the menu item ***Draw > Box***
      1. Using the coordinate entry fields, enter the box position
         - X: **119**, Y: **25**, Z: **49**, Press the **Enter** key
      2. Using the coordinate entry fields, enter the opposite corner of the box:
         - dX: **150**, dY: **150**, dZ: **100**, Press the **Enter** key
  - **Change Attributes**
    - Select the object from the tree and goto Properties window
      1. Change the name of the object to **Coil_Hole**
  - **Create Fillets**
    - Select the menu item ***Edit > Select > Edges*** or press **"E"** from keyboard
    - Press **Ctrl** and select the edges as shown in image
    - Select the menu item ***Modeler > Fillet***
    - In Fillet Properties Window,
      1. Fillet Radius: **25mm**
      2. Setback Distance: **0mm**
      3. Click the **OK** button



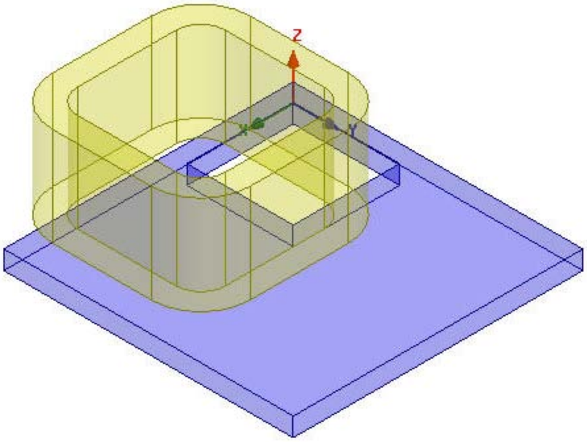


Example (Eddy Current) - Asymmetric Conductor

- **Create Coil**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **94**, Y: **0**, Z: **49**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **200**, dY: **200**, dZ: **100**, Press the **Enter** key
- **Change Attributes**
  - Select the object from the tree and goto Properties window
    1. Change the name of the object to **Coil**
    2. Change its color to **Yellow**
- **Create Fillets**
  - Press **Ctrl** and select the edges as shown in image
  - Select the menu item ***Modeler > Fillet***
  - In Fillet Properties Window,
    1. Fillet Radius: **50mm**
    2. Setback Distance: **0mm**
    3. Click the **OK** button

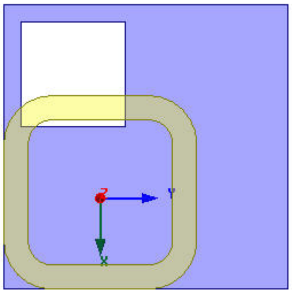


- **Subtract Objects**
  - Select the menu item ***Edit > Select > Objects*** or press **"O"** from keyboard
  - Pres **Ctrl** and select the objects **Coil** and **Coil_Hole** from history tree
  - Select the menu item, ***Modeler > Boolean > Subtract***
    1. Blank Parts: **Coil**
    2. Tool Parts: **Coil_Hole**
    3. Click the **OK** button



Create Offset Coordinate System

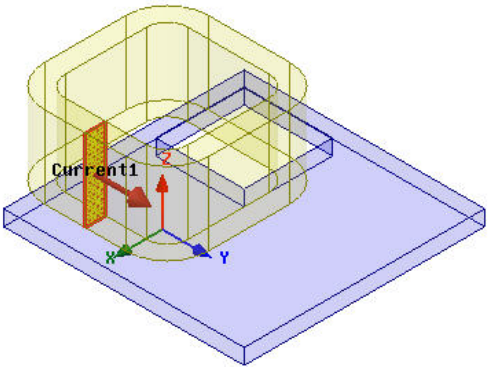
- **To Create Offset Coordinate System**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Offset***
    1. Using the coordinate entry fields, enter the origin
       - X: **200**, Y: **100**, Z: **0**, Press the **Enter** key



Create Excitations

- **Create Section of coil for assigning Current**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **XZ**
    2. Click the **OK** button

- **Change Attributes**
  - Select the object **Coil_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Coil_Terminal**
- **Separate Sheets**
  - Select the sheet **Coil_Terminal**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Coil_Terminal_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***
- **Assign Excitations**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **2742**
    3. Phase: **0**
    4. Type: **Stranded**
    5. Current Direction: **positive Y**
       (Use **Swap Direction** if needed)
    6. Press **OK**



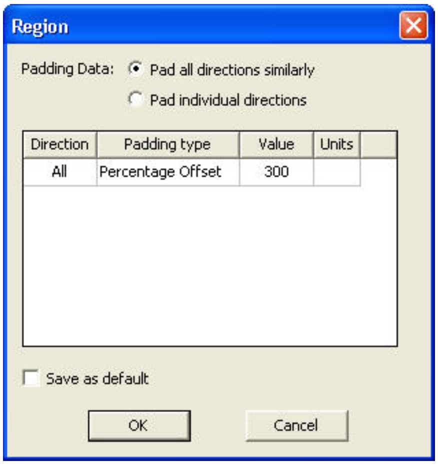
Example (Eddy Current) - Asymmetric Conductor

Set Default Material

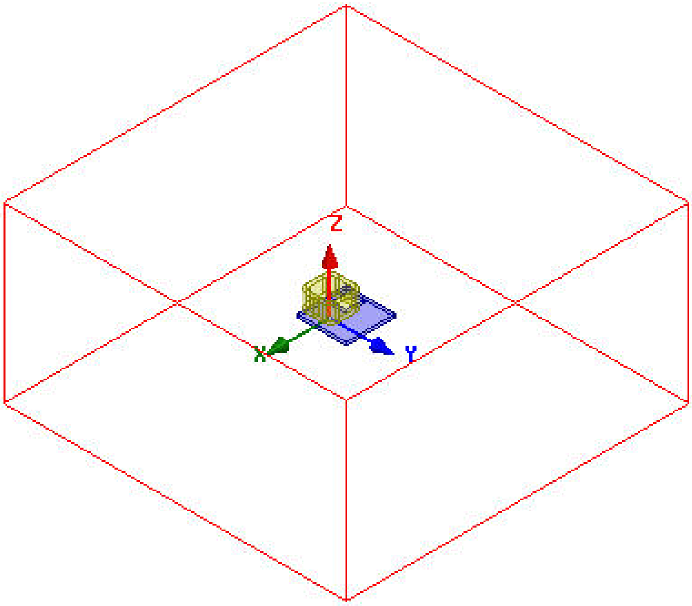
- **To Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **300**
    4. Press **OK**



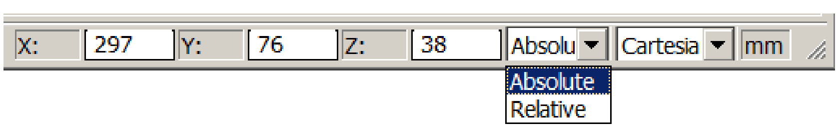
  - **Note:** For all Maxwell 3D projects a solution space must be defined. Unless a partial model (like a motor wedge) is used, a region is usually created as described above.



Example (Eddy Current) - Asymmetric Conductor

Create Dummy Object

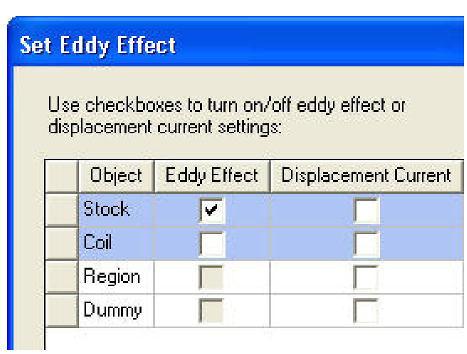
- **Note:** As a post-processing results we are interested to calculate the flux density on a line with the following end points: [0; 72; 34] and [288; 72; 34]. In order to get as accurate results as possible, the mesh around the line has to be fine enough. It is generally a good practice to create dummy objects around regions of interest so that the mesh quality can be better controlled in those regions. For this particular problem we create a box around the line of interest.
- **Change Work Coordinate System**
  - Goto ***Modeler > Coordinate System > Set Working CS***
  - In Select Coordinate System Window
    1. Select **Global**
    2. Press **Select**
- **To Create Dummy Object**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-3**, Y: **68**, Z: **30**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box: (in **Absolute** values)
       - X: **297**, Y: **76**, Z: **38**, Press the **Enter** key



- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Dummy**

Set Eddy Effects

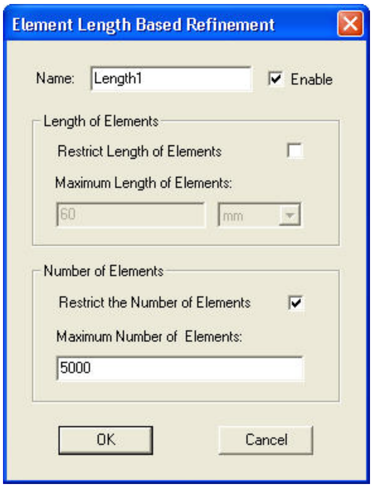
- **To Set Eddy Effects**
  - Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effect***
  - In Set Eddy Effects window
    1. Eddy Effects:
       - Stock: ☑ **Checked**
       - Coil: ☐ **UnChecked**
    2. Displacement Current:
       - Coil and Stock : ☐ **Unchecked**
    3. Press **OK**



Apply Mesh Operations

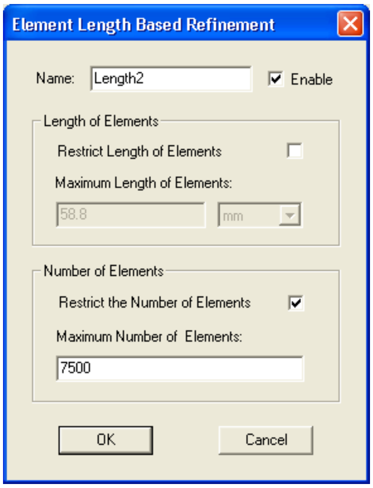
Apply Length Based Mesh Operations for Dummy

- Select the object **Dummy** from the history tree
- Goto menu item ***Maxwell 3D > Mesh Operations >Assign > Inside Selection > Length Based***
  1. Name: **Length1**
  2. Restrict Length of Elements: ☐ **Unchecked**
  3. Restrict the Number of Elements: ☑ **Checked**
  4. Maximum Number of Elements: **5000**
  5. Press **OK**



Apply Length Based Mesh Operations for Stock

- Select the object **Stock** from the history tree
- Goto menu item ***Maxwell 3D > Mesh Operations >Assign > Inside Selection > Length Based***
  1. Name: **Length2**
  2. Restrict Length of Elements: ☐ **Unchecked**
  3. Restrict the Number of Elements: ☑ **Checked**
  4. Maximum Number of Elements: **7500**
  5. Press **OK**



Analysis Setup

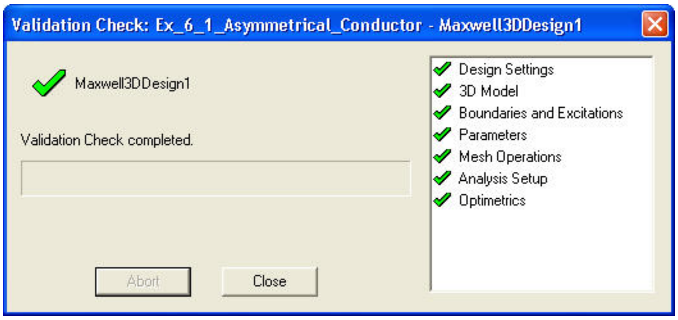
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.2**
    2. **Solver** Tab
       - Adaptive Frequency: **200 Hz**
    3. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **Ex_6_1_Asymmetrical_Conductor**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - 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 3D > Analyze All***

Create Objects for Rectangular Plot

- **To Create Line Object**
  - Select menu item ***Draw > Line***
  - A massage will pop up asking if the entity needs to be created as non model object. Select **Yes** to it.
    1. Using Co-ordinate entry field Enter the first vertex
       - **X = 0, Y = 72, Z = 34,** Press **Enter**
    2. Using Co-ordinate entry field Enter the second vertex
       - **X = 288, Y = 72, Z = 34,** Press **Enter**
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **FieldLine**

Create Parameters

- **Create parameter Bz_real**
  - Go to ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **Vector > Scal? > ScalarZ**
    3. Select **General > Complex > Real**
    4. Select **General > Smooth**

       Note: The flux density will be displayed by default in the units of Tesla [T]. If you wish to see the results in Gaussian units perform steps 5 and 6, otherwise jump to step 7
    5. Select **Input > Number**
       - Type: **Scalar**
       - Value: **10000**
       - Press **OK**
    6. Select **General > ***
    7. Select **Add** and set the name of expression as **Bz_real**

Example (Eddy Current) - Asymmetric Conductor

- **Create parameter Bz_imag**
  - Go to ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **Vector > Scal? > ScalarZ**
    3. Select **General > Complex > Imag**
    4. Select **General > Smooth**

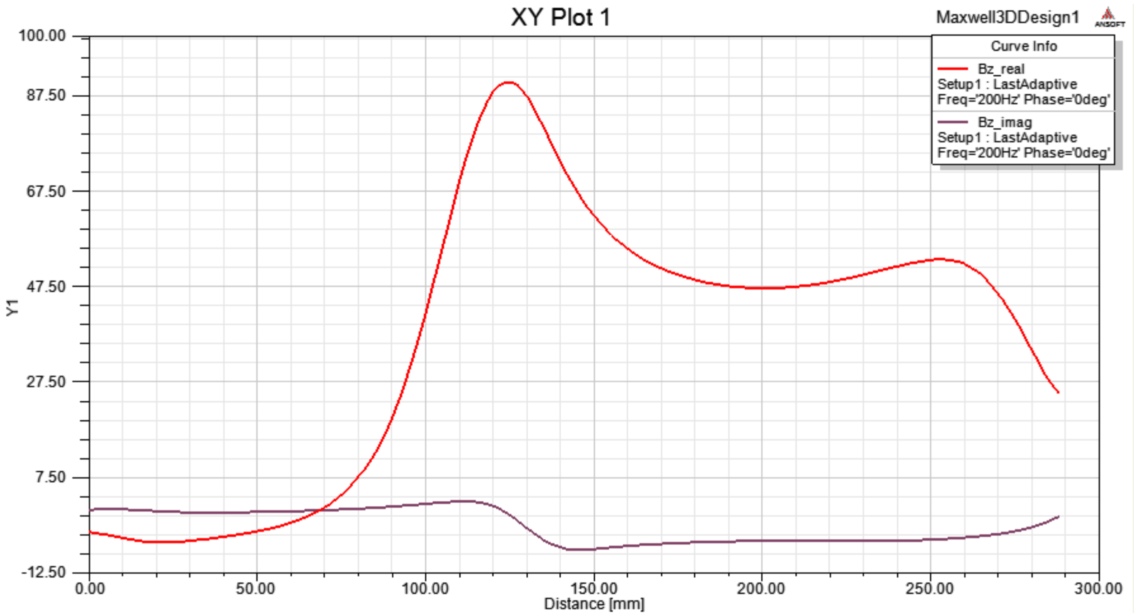
       Note: The flux density will be displayed by default in the units of Tesla [T]. If you wish to see the results in Gaussian units perform steps 5 and 6, otherwise jump to step 7
    5. Select **Input > Number**
       - Type: **Scalar**
       - Value: **10000**
       - Press **OK**
    6. Select **General > ***
    7. Select **Add** and set the name of expression as **Bz_imag**
    8. Press **Done**

Rectangular Plot

- **To Create Plot:**
  - Select ***Maxwell 3D > Results > Create Fields Report > Rectangular Plot***
  - In Reports window
    1. Geometry: **FieldLine**
    2. **Trace** Tab
       1. X axis: **Default**
       2. Y axis
          - Category: **Calculator Expressions**
          - Quantity: **Bz_real**
          - Function: **None**
    3. Select **New Report**
    4. Without closing window, change quantity to **Bz_imag**
    5. Select **Add Trace**
    6. Select **Close**

Example (Eddy Current) - Asymmetric Conductor

- **Rectangular Plot**

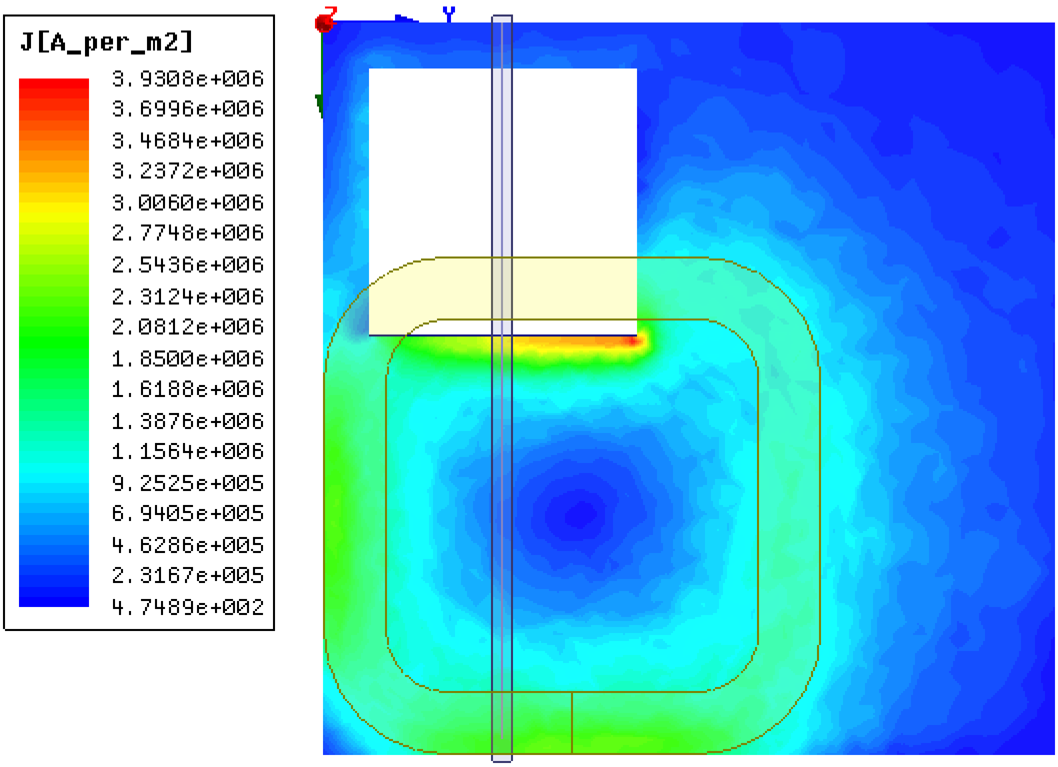


Field Overlays

- **Create Field Plots for Mag_J**
  - Double click on **Maxwell3D Design1** from Project Manager window to return to Maxwell Project
  - Select the object **Stock** from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > J > Mag_J***
  - In Create Field Plot window
    1. Plot on surface only : ☑ **Checked**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - Num. Divisions: **50**
    2. Press **Apply** and **Close**

Example (Eddy Current) - Asymmetric Conductor

- **Field Plots Mag_J**

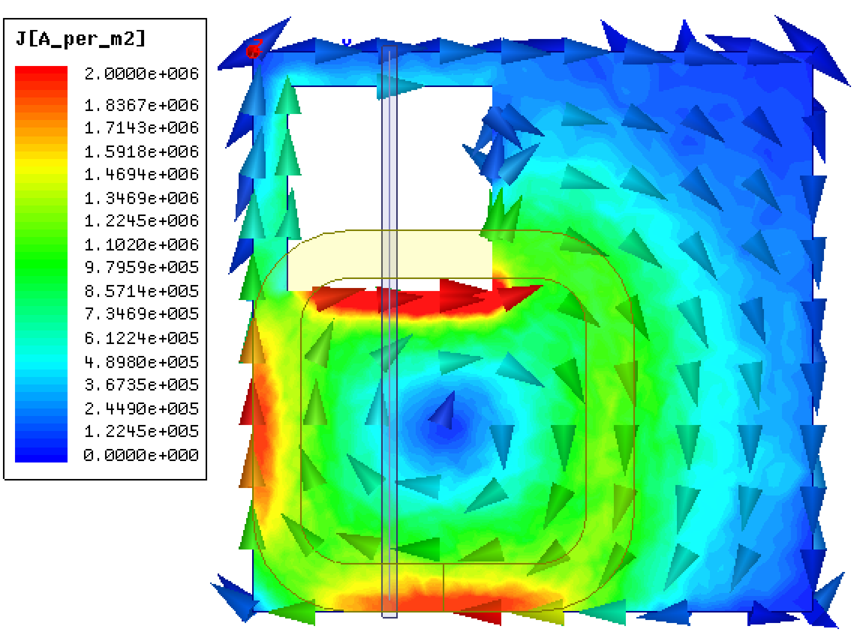


Vector Plot

- **Create Vector Plot**
  - Select the object **Stock** from history tree
  - Select menu item ***Maxwell 3D > Fields > Fields > J > Vector_J***
  - In Create Field Plot window
    1. In Volume: **AllObjects**
    2. Plot on surface only : ☑ **Checked**
    3. Select **Done**

Example (Eddy Current) - Asymmetric Conductor

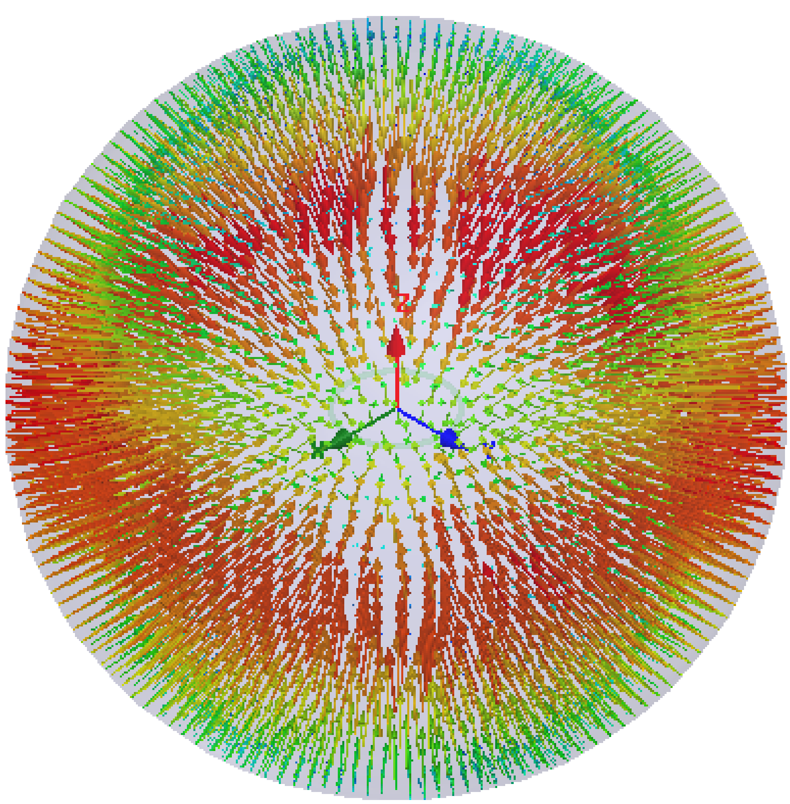
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - User Limits: ☑ **Checked**
         1. Min: **0**
         2. Max: **2e6**
    2. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Set to appropriate value**
         2. Map Size: ☐ **Unchecked**
         3. Arrow tail: ☐ **Unchecked**
    3. **Plots** tab
       - Plot: **Vector_J1**
       - Vector Plot
         1. Spacing: **Set to Minimum**
         2. Min: **30**
         3. Max: **60**
    4. Press **Apply** and **Close**



Example (Eddy Current) – Radiation Boundary

The implementation and application of a Radiation Boundary in the Eddy Current Solver

- Radiation Boundaries are used when one wants to simulate the free emission of fields into space. The main distinction of this boundary from all other boundary conditions is that the radiation boundary does not enforce any restrictions on the field behavior. This is used specifically to model unbounded eddy currents and propagating waves without reflection. This boundary has several specific uses which will be discussed in detail in this document.
- Note that Maxwell3D is a low-frequency simulator, so it is not recommended to model standard antennas and wave propagation with this tool. Maxwell is ideal for near-field simulations for the investigation of interactions between low-frequency inductors and RFID applications.
- The example that will be used to demonstrate how Radiation Boundaries are implemented is a Magnetic Dipole. The intent of this write up is not how to simulate a dipole antenna, it is rather to demonstrate how Radiation Boundaries are implemented within the Eddy Current solver.



Example (Eddy Current) – Radiation Boundary

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Torus, Sphere**
    - Surface Operations: **Section**
    - Boolean Operations: **Separate Bodies**
  - Boundaries/Excitations
    - Current: **Solid**
    - **Radiation Boundary**
    - **Symmetry Boundary**
  - Analysis
    - **Eddy Current**
  - Results
    - **Calculator Expressions**
  - Field Overlays:
    - **Field Plots**

Example (Eddy Current) – Radiation Boundary

Launching Maxwell

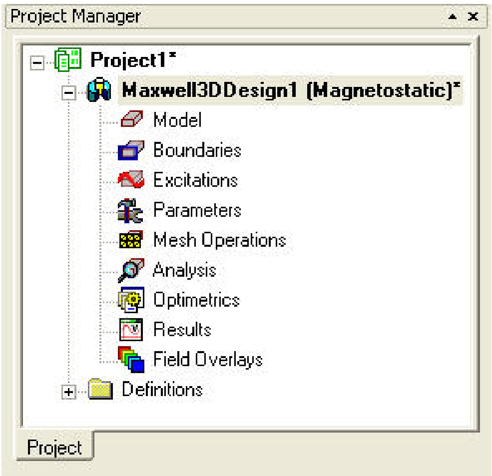
- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

Setting Tool Options

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

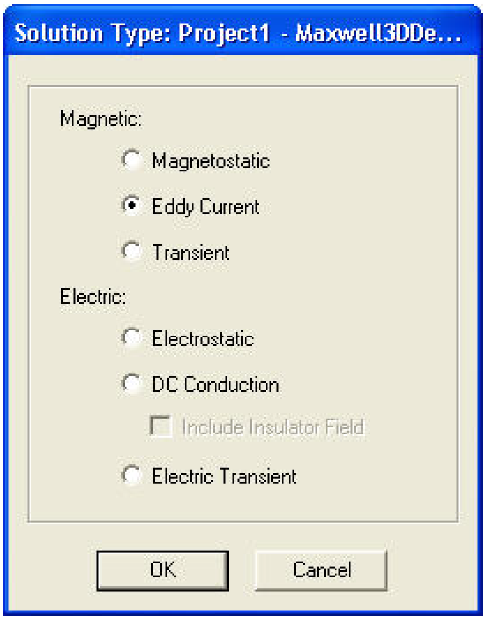
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



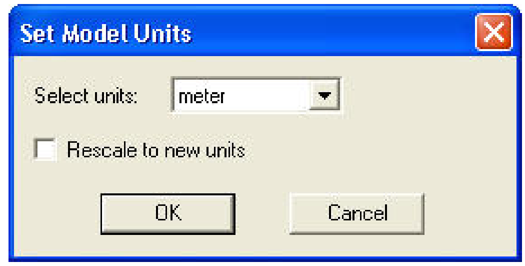
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Eddy Current**
    2. Click the **OK** button



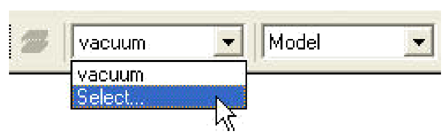
Set Model Units

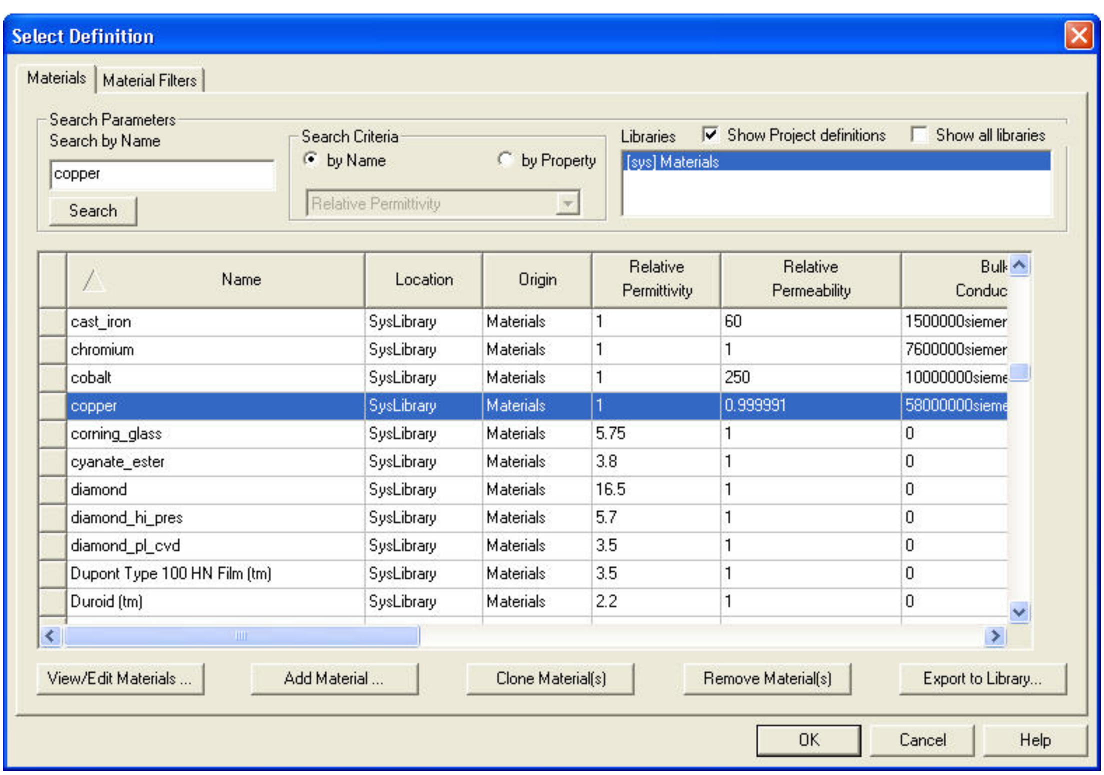
- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **meter**
    2. Click the **OK** button



Set Default Material

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





Create Dipole Ring

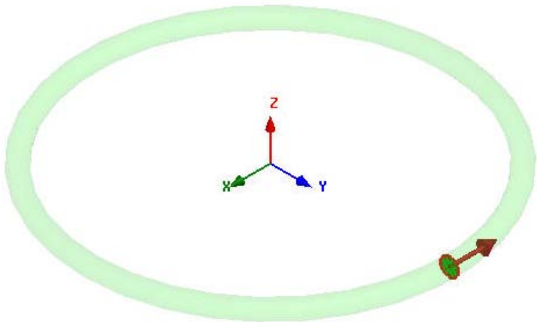
- **Create Torus**
  - Select the menu item ***Draw > Torus***
    1. Using the coordinate entry fields, enter the center point of torus
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the inner radius
       - dX: **0.0095**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the outer radius
       - dX: **0.001**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Select the menu item ***View > Fit All > Active View.***
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Ring**
    2. Change its color to **Green**

Create Excitations

- **Create Section of coil for assigning Current**
  - Select the object **Ring** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button
- **Change Attributes**
  - Select the object **Ring_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Terminal**
- **Separate Sheets**
  - Select the sheet **Terminal**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Terminal_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***

Example (Eddy Current) – Radiation Boundary

- **Assign Excitations**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **1.414**
    3. Phase: **0**
    4. Type: **Solid**
    6. Press **OK**



- **Note:** In Eddy Current Solver, peak current is always specified instead of RMS

Set Default Material

- **To Set Default Material**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

Create Region

- **To create the region:**
  - Select the menu item ***Draw > Sphere***
    1. Using the coordinate entry fields, enter the center of Sphere
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius:
       - dX: **0.06** dY: **0**, dZ: **0**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Region**

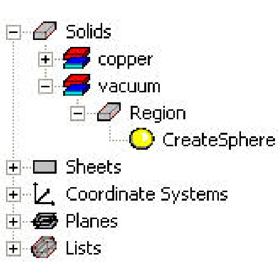
Example (Eddy Current) – Radiation Boundary

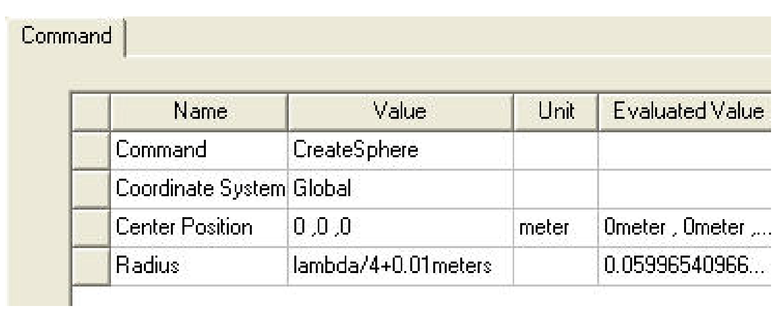
Radiation Boundary Overview

- The radiation boundary is only available in the Eddy Current solver – it is only applicable for frequency domain solutions.
- The radiation boundary should be used in conjunction with displacement current in the background object – implying that the fields in the background can propagate to the edge of the model. This means that if displacement currents are simulated in the background, then a radiation boundary should be used on the outside surface of the background.
- A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space, such as antenna designs. Maxwell3D absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure.
- A radiation surface does not have to be spherical, but it must be exposed to the background, convex with regard to the radiation source, and located at least a quarter wavelength from the radiating source. In some cases the radiation boundary may be located closer than one-quarter wavelength, such as portions of the radiation boundary where little radiated energy is expected.
- Remember that Maxwell3D is a near-field simulator, so the dimension of components should be much smaller than the wavelength (i.e. lambda/10)
- The wavelength at 1.5GHz is 0.2m. Therefore, our 0.06m radius sphere is more than one quarter wavelength from the ring. In order to make the boundary placement more automated, we can use variables and equations to define the size of the bounding object.

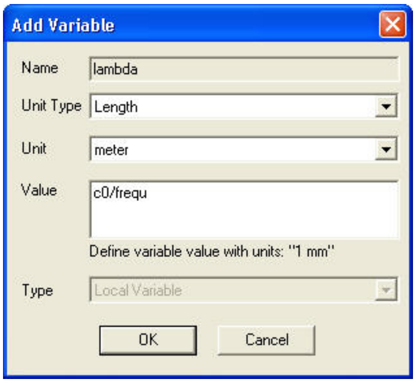
Assign Function to the Radius of the Region

- **Specify Radius of Region as Function of Speed of Light "c0" and Frequency**
  - Expand the history tree of the object **Region**
  - Double click on the command **CreateSphere** from the tree
  - For **Radius**, type: **lambda/4 + 0.01meters**, Click the **Tab** key

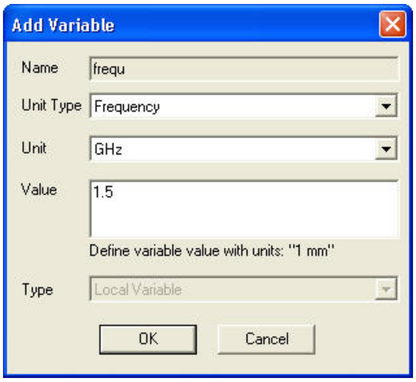




Example (Eddy Current) – Radiation Boundary



- In Add variable window for **lambda**
  1. Value: **c0/frequ**
  2. Press **OK**
- In Add variable window for **frequ**
  1. Unit Type: **Frequency**
  2. Unit: **GHz**
  3. Value: **1.5**
  4. Press **OK**
- Press **OK**



Assign Boundary

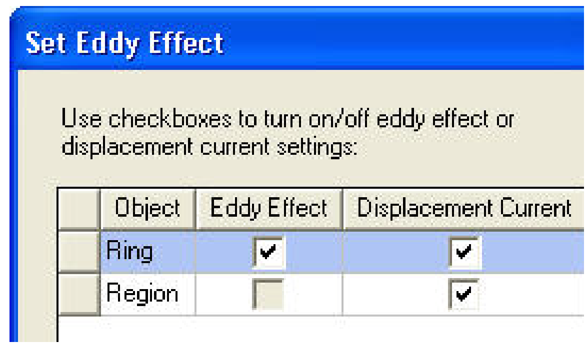
- **Assign Radiation Boundary**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard to change selection to faces
  - Select the face of the Region
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Radiation***
  - In Radiation Boundary window,
    1. Press **OK**

- **Note:** This assigns a radiation boundary to the outside of the sphere, allowing the waves to propagate outside of the sphere. If no radiation boundary is assigned, the interactions of the components may be correct, but the fields near the boundaries will certainly be incorrect. The Poynting vector on the outer surface is a clear indicator of this, which we will explore later.

Example (Eddy Current) - Radiation Boundary

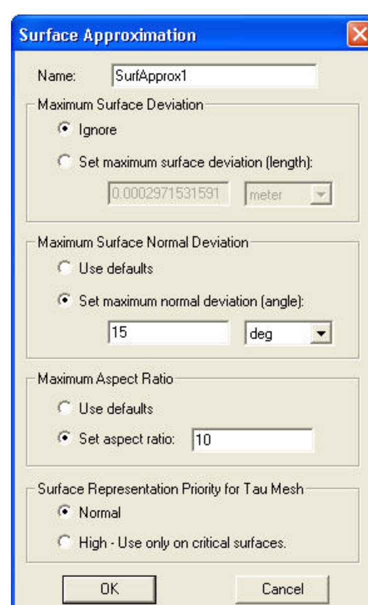
Set Eddy Effects

- **To Set Eddy Effects**
  - Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effect***
  - In Set Eddy Effects window
    1. Eddy Effects:
       - Ring: ☑ **Checked**
    2. Displacement Current:
       - Ring: ☑ **Checked**
       - Region: ☑ **Checked**
    3. Press **OK**



Apply Mesh Operations

- **Apply Length Based Mesh Operations for Ring**
  - Select the menu item ***Edit >Select > Objects*** or press **"O"** from keyboard
  - Select the object **Ring** from the history tree
  - Goto menu item ***Maxwell 3D > Mesh Operations >Assign > Surface Approximation***
  - In Surface Approximation window
    1. Maximum Surface Normal Deviation
       - Set maximum normal deviation (angle): **15 deg**
    2. Maximum Aspect Ratio
       - Set Aspect Ratio: **10**
    3. Press **OK**



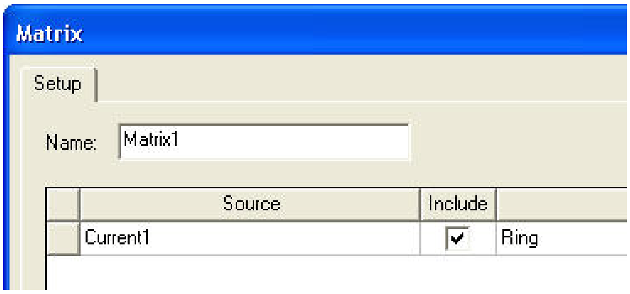
Example (Eddy Current) - Radiation Boundary

Create Outer Surface

- **Note:** One important part of creating a geometry is to consider the results needed in the post-processor. We will wish to use the outer surface of the sphere to compute the radiated power and the radiation resistance. Therefore, we will need to create a surface to integrate over that defines precisely the area that we need.
- **To Create Outer Surface**
  - Select the outer face of the Region
  - Select the menu item ***Modeler > Surface > Create Object from Face***
- **Change Attributes**
  - Select the resulting object from the tree and goto Properties window
    1. Change the name of the object to **Outside**

Assign Matrix Parameters

- **To Assign Parameters**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window
    1. Current1
       - Include: ☑ **Checked**
    2. Press **OK**



Analysis Setup

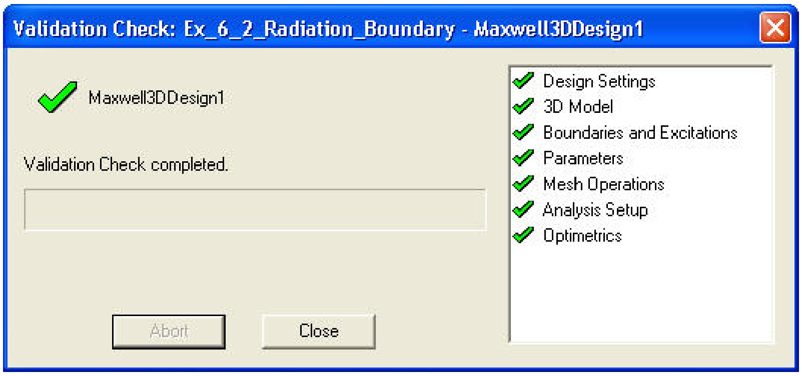
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.2**
    2. **Solver** Tab
       - Adaptive Frequency: **1.5 GHz**
    3. Click the **OK** button

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  2. From the **Save As** window, type the Filename: **Ex_6_2_Radiation_Boundary**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



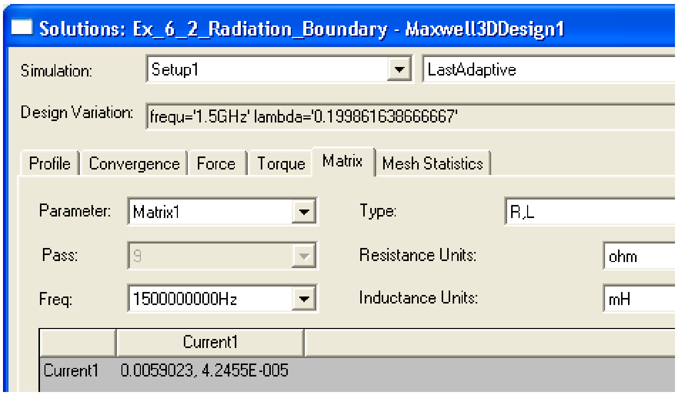
Note: To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Impedance Matrix**
       - Select the **Matrix** tab
       - Set Parameter to **Matrix1**
       - Set Type to **R,L**
    2. Press **Close**



Create Parameters

- **Note:** We will create a Poynting vector expression in the calculator that is accessible as a field plot and for power calculations. The expression will be:

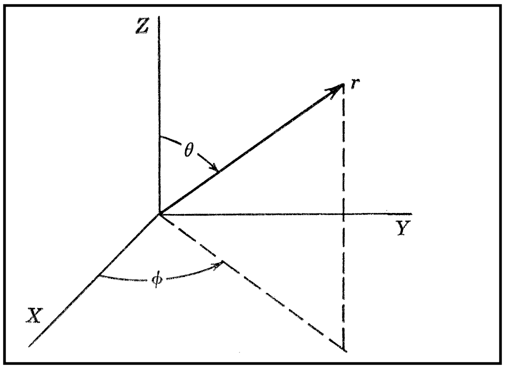
$$\vec{P} = \frac{1}{2}\mathrm{Re}\left\{\vec{E} \times \vec{H}^{*}\right\}$$

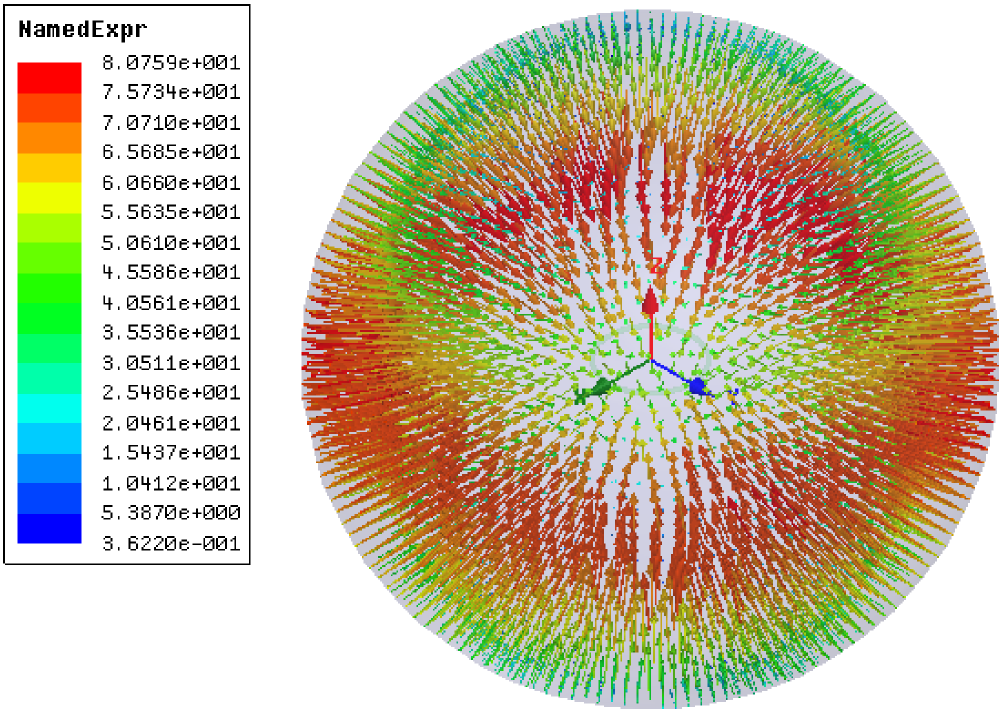
- **Create parameter Poynting**
  - Goto ***Maxwell 3D > Fileds > Calculator***
    1. Select **Input > Quantity > E**
    2. Select **Input > Quantity > H**
    3. Select **General > Complex > Conj**
    4. Select **Vector > Cross**
    5. Select **General > Complex > Real**
    6. Select **Input > Number**
       - Type: **Scalar**
       - Value: **0.5**
       - Press **OK**
    6. Select **General > ***
    7. Select **Add** and set the name of expression as **Poynting**
    8. Press **Done**

Example (Eddy Current) - Radiation Boundary

Create Vector Plot

- **To Create Vector Plot**
  - Select outer surface of the object **Region**
  - Select the menu item ***Maxwell 3D > Fields > Fields > Named Expressions***
    1. Select the parameter **Poynting**
    2. Press **OK**
  - In Create Field Plot window
    1. Press **Done**
- A plot similar to the following should appear, with strong fields near theta=90°.





Calculate the Radiation Resistance

- Note:
  - The radiation resistance of an antenna is determined by the following formula:

$$R_r = \frac{P_{av}}{I_{rms}^2}$$

  - Where the radiated Power is defined as the integral of the Poynting vector over the surface S (outer surface of the region).

$$P_{av} = \frac{1}{2} \oint\!\!\!\oint_S \mathrm{Re}\left(\vec{E} \times \vec{H}^*\right) dS$$

  - Since the rms current is 1 A (the peak current is defined as 1.414 A), the radiation resistance is simply the result of this integral.
- **To Calculate Radiated Power**
  - Goto ***Maxwell 3D > Fileds > Calculator***
    1. Select **Poynting** from Named Expressions list
    2. Select **Copy to Stack**
    3. Select **Input > Geometry**
       - Select **Surface**
       - Select **Outside** from the list
       - Press **OK**
    4. Select **Vector > Normal**
    5. Select **Scalar >** ∫ **(Integrate)**
    6. Press **Eval**
  - The resulting value should be around **-2.165 W**
  - Since RMS current comes out to be 1A, the radiation resistance should also be **-2.165 Ohms**

- Note:
  - According to the literature, the radiation resistance of a Magnetic Dipole with no thickness is:

$$R_{rad\_air} = 20\pi^2 \beta^4 a^4$$

    where β is the wave number and a is the radius of the loop
  - This equation results in 1.923 ohms for a radius of 0.01m and a wavelength of 0.2m.

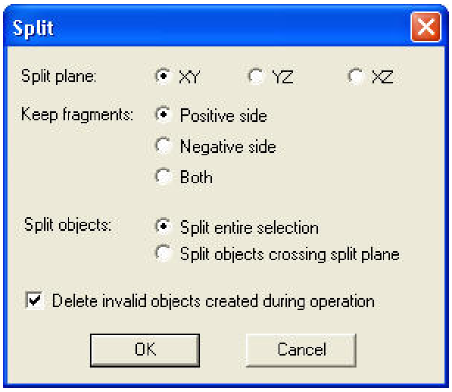
PART 2: Create a 1/16th Section Model

Create Symmetry Design

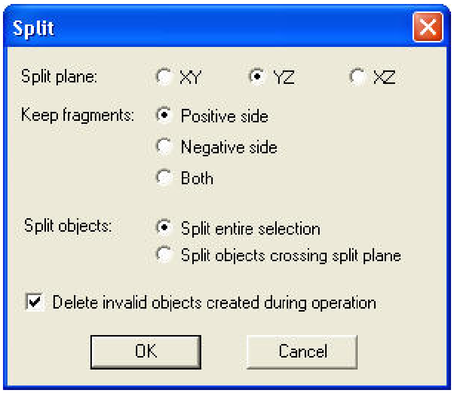
- **Copy Design**
  1. Select the design **Maxwell3D Design1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_6_2_Radiation_Boundary** in Project Manager window and select **Paste**
  3. Change the name of the design to **1_16_Model**

Split Model for 1/16th Section

- **Divide by XY Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XY**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

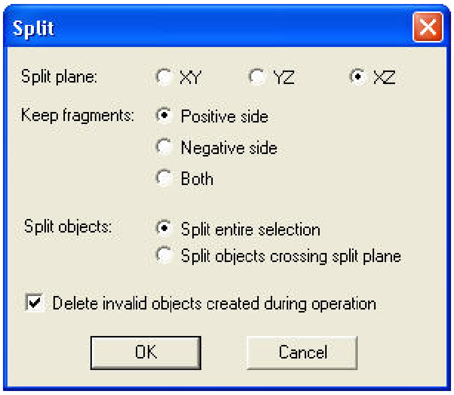


- **Divide by YZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

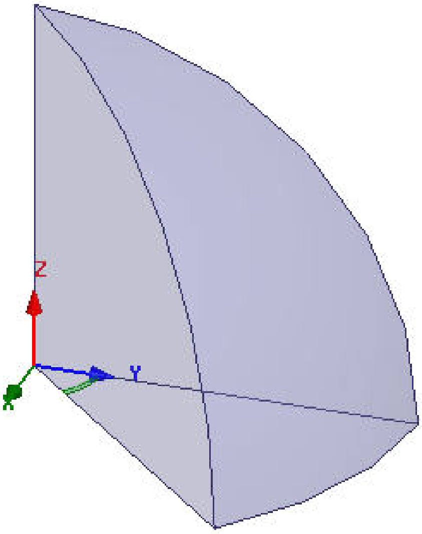


Example (Eddy Current) - Radiation Boundary

- **Create Relative Coordinate System**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Rotated***
    1. Using the coordinate entry field, enter the X axis position
       - X: **0.01,** Y: **0.01,** Z: **0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the XY Plane position in Absolute values
       - X: **-0.01,** Y: **0.01,** Z: **0**, Press the **Enter** key
- **Divide by XZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



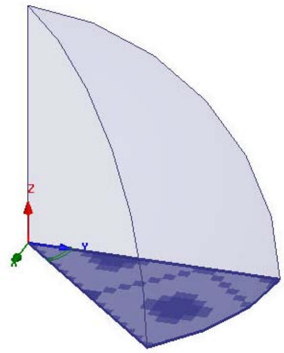
- **Change Work Coordinate System**
  - Goto ***Modeler > Coordinate System > Set Working CS***
  - In Select Coordinate System Window
    1. Select **Global**
    2. Press **Select**



Example (Eddy Current) – Radiation Boundary

Assign Boundaries

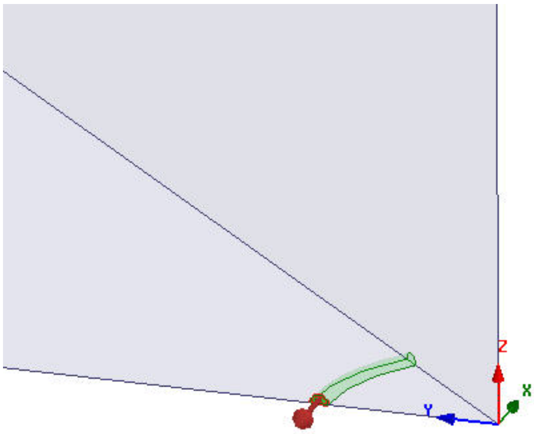
- **To Assign Symmetry Boundaries**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard to change selection to faces
  - Select the face of the Region with which touch with XY Plane
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Symmetry***
  - In Symmetry Boundary window,
    1. Symmetry: **Even (Flux Normal)**
    2. Press **OK**



  - **Note:** Only one symmetry boundary is required even though three surfaces define distinct symmetries. The two surfaces perpendicular to the ring have a symmetry that is accounted for by the default boundary condition of Zero Tangential H Field – i.e. this is a special case of symmetry, where the symmetric tangential fields are zero. The surface that lies in the plane of the ring has a Flux Normal symmetry, so we need to define this separately.

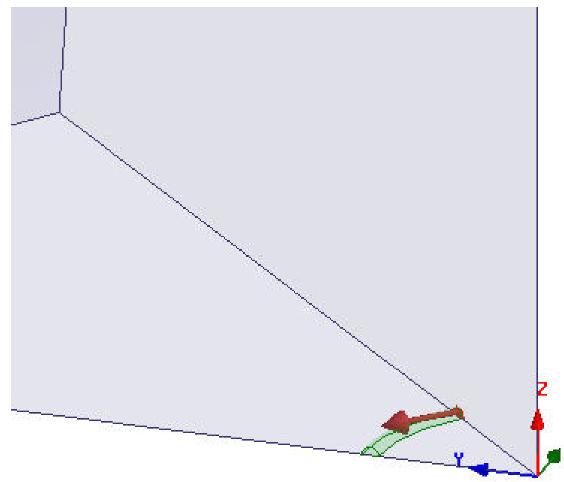
Create Excitations

- **Assign Excitation Current_Out**
  - Select the face of **Ring** which touches the YZ Plane
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **Current_Out**
    2. Value: **0.707 A**
    3. Type: **Solid**
    4. Direction: **Out of Model**
    5. Press **OK**



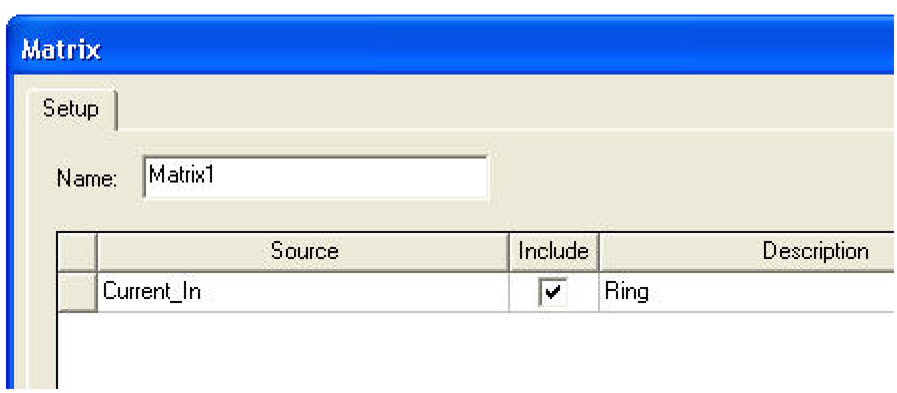
Example (Eddy Current) - Radiation Boundary

- **Assign Excitation Current_In**
  - Select the face of **Ring** as shown in image
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **Current_In**
    2. Value: **0.707 A**
    3. Type: **Solid**
    4. Direction: **Going in to Model**
    5. Press **OK**



Assign Matrix Parameters

- **To Assign Parameters**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window
    1. Current_In
       - Include: ☑ **Checked**
    2. Press **OK**



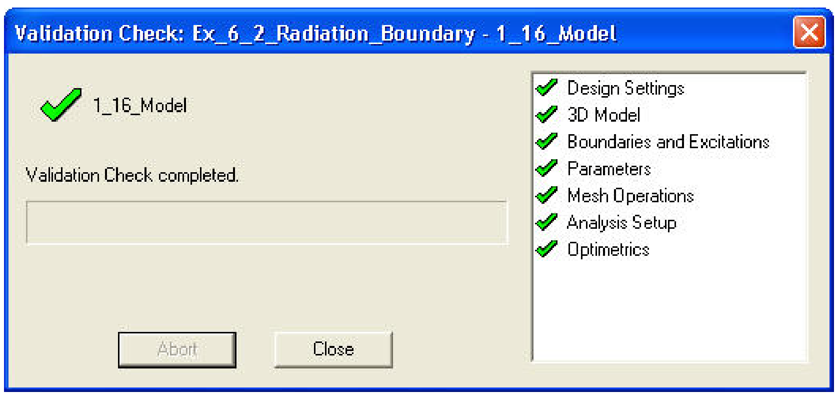
- **Note:** The excitations available to the Matrix are only those excitations that define the current entering the conduction path. The **Current_Out** excitation defines current <u>exiting</u> the conduction path, and so is not available to the matrix.

Save Project

- **To save the project:**
  1. In an Ansoft Maxwell window, select the menu item ***File > Save***

Model Validation

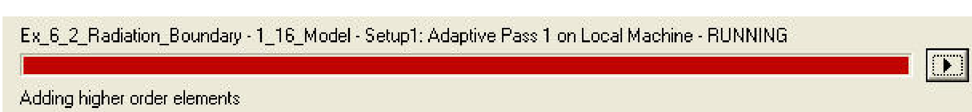
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

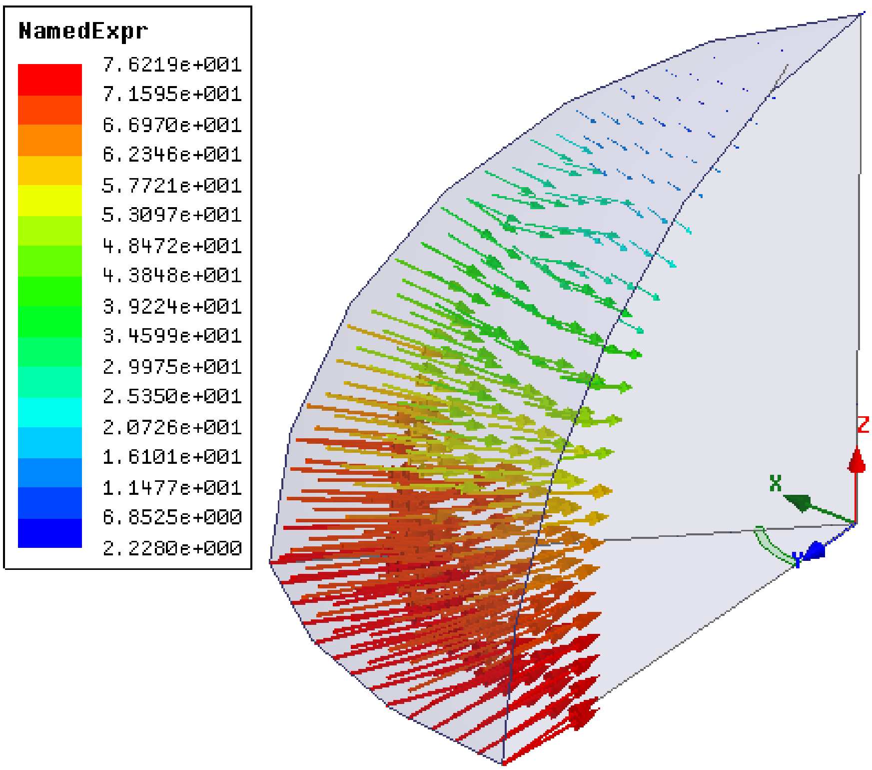
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***



Plot Power Flow

- **To View vector Plot**
  - In the Project Manager tree, expand Field Overlays and NamedExpr to find Poynting1
  - Double click on Poyntning1 to refresh the plot
  - The plot should look similar to the following



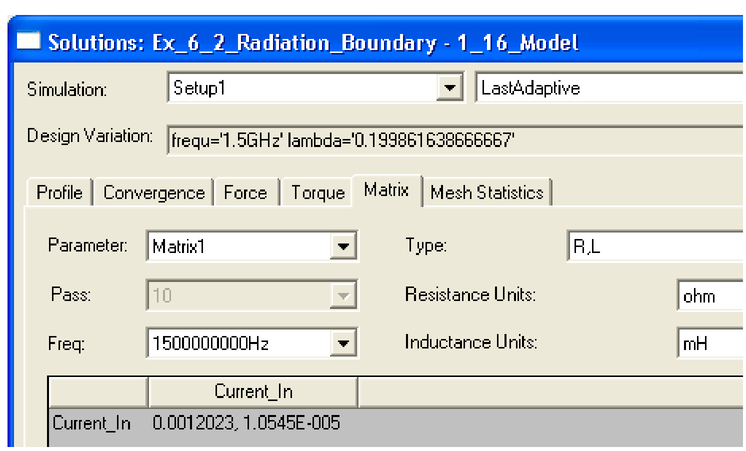
Example (Eddy Current) - Radiation Boundary

Calculate the Radiation Resistance

- **To Calculate Radiated Power**
  - Goto ***Maxwell 3D > Fileds > Calculator***
    1. Select **Poynting** from Named Expressions list
    2. Select **Copy to Stack**
    3. Select **Input > Geometry**
       - Select **Surface**
       - Select **Outside** from the list
       - Press **OK**
    4. Select **Vector > Normal**
    5. Select **Scalar >** $\int$ **(Integrate)**
    6. Press **Eval**
  - The resulting value should be around **-0.1336 W** which is around 1/16th of previous value

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Impedance Matrix**
       - Select the **Matrix** tab
       - Set Parameter to **Matrix1**
       - Set Type to **R,L**
    2. Press **Close**



Instantaneous Forces on Busbars in Maxwell 2D and 3D

This example analyzes the forces acting on a busbar model in Maxwell 2D and 3D. Specifically, it provides a method for determining the instantaneous force on objects having sinusoidal AC excitation in the Eddy Current Solver. Force vectors in AC problems are a combination of a time-averaged "DC" component and an alternating "AC" component. The alternating component fluctuates at a frequency twice the excitation frequency. Both of these components can be calculated using the formulas below so that the instantaneous force can be determined. Three different force methods are used in this example: Virtual, Lorentz, and the Maxwell Stress Tensor.

$$F_{DC} = \frac{1}{2}\int \mathrm{Re}\left| \bar{J} \times \bar{B}^{*} \right| dV$$

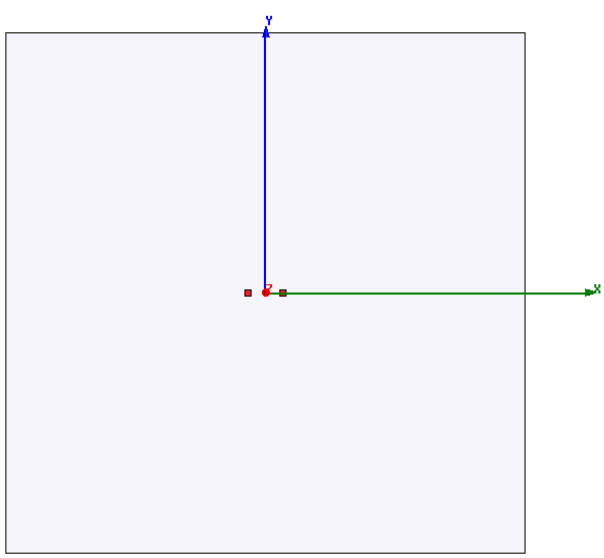
$$F_{AC} = \frac{1}{2}\int \left| \bar{J} \times \bar{B} \right| dV \qquad evaluated\ at\ phase\ (\omega t = degrees)$$

$$F_{INST} = F_{DC} + F_{AC}$$

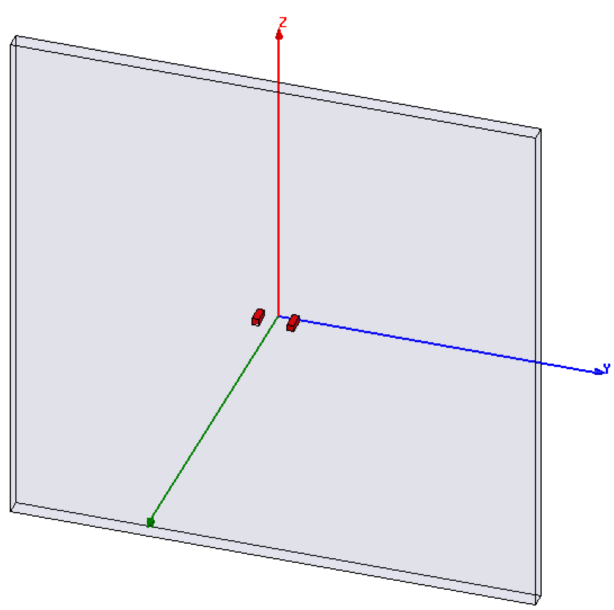
Description

This example will be solved in two parts using the 2D Eddy Current and 3D Eddy Current solvers. The model consists of two 4mm parallel copper busbars separated by a center-center spacing of 16mm. The excitation frequency is 100kHz.

2D Model



3D Model



PART 1 - The 2D Eddy Project

A 2D model of the busbars will be simulated first. Access the Maxwell Project Manager and create a new 2D project called 2dbars. Open the project and change to the Eddy Current solver with an XY drawing plane.

Setup the Design

1. Click on the menu item **Project > Insert Maxwell 2D Design**
2. Click on the menu item **Maxwell 2D > Solution Type ...**
   - Set Geometry Mode: **Cartesian, XY**
   - Select the radio button Magnetic: **Eddy Current**

- Draw the Solution Region
  - Click on Draw > Rectangle *(Enter the following points using the tab key).*
    - X: **-150**, Y: **-150**, Z: **0**
    - dX: **300**, dY: **300**, dZ: **0**
  - Change its properties:
    - Name: **Region**
    - Display Wireframe: **Checked**
  - Select **View > Fitall > Active View** to resize the drawing window.

Create the Model

Now the model can be created. This model also consists of a left and right busbar that have a 4mm square cross-section, however a length of 1 meter is assumed so that the results must be scaled to compare to 3D.

Create the Left Busbar

- Click on Draw > Rectangle
  - X: **-12**, Y: **-2**, Z: **0**
  - dX: **4**, dY: **4**, dZ: **0**
- Change its properties:
  - Name: **left**
  - Material: **Copper**
  - Color: **Red**

Create the Right Busbar

- Click on Draw > Rectangle
  - X: **8**, Y: **-2**, Z: **0**
  - dX: **4**, dY: **4**, dZ: **0**
- Change its properties:
  - Name: **right**
  - Material: **Copper**
  - Color: **Red**

Assign the Boundaries and Sources

The current is assumed to be 1A at 0 degrees in the left busbar and -1A at 60 degrees in the right busbar. A no-fringing vector potential boundary will be assigned to the outside of the 2D problem region which is also the default boundary for all 3D projects. This forces all flux to stay in the solution region.

1. The boundary must be set on the solution region.
   - Choose **Edit** > **Select** > **Edges** to change the selection mode from object to edge.
   - While holding down the **CTRL** key, choose the four outer edges of the region.
   - Click on **Maxwell 2D** > **Boundaries**> **Assign** > **Vector Potential**
     - Value: **0**
     - Phase: **0**
     - **OK**
   - When done, choose **Edit** > **Select** > **Object** to object selection mode.

1. Select **left** from the history tree
   - Click on **Maxwell 2D** > **Excitations** > **Assign** > **Current**
     - Name: **Current1**
     - Value: **1A**
     - Phase: **0**
     - Type: **Solid**
     - Reference Direction: **Positive**

2. Select **right** from the history tree.
   - Click on **Maxwell 2D** > **Excitations** > **Assign** > **Current**
     - Name: **Current2**
     - Value: **1A**
     - Phase: **60**
     - Type: **Solid**
     - Reference Direction: **Negative**

Turn on the Eddy Effects in the winding

In order to consider the skin effects in the busbars, you must manually turn on the eddy effect.

1. Choose **Maxwell 2D > Excitations > Set Eddy Effects ...**
2. Verify that the eddy effect is checked for both the **left** and **right** conductors.

Assign the Parameters

In order to automatically calculate force on an object, it must be selected in the Parameters panel. In 2D, only the virtual force can be automatically calculated. Later, the Lorentz force will be calculated manually in the Post Processor after solving the project.

1. Select the **left** busbar by clicking on it.
2. Click on **Maxwell 2D > Parameters > Assign > Force**
3. Click **OK** to enable the force calculation.

Add an Analysis Setup

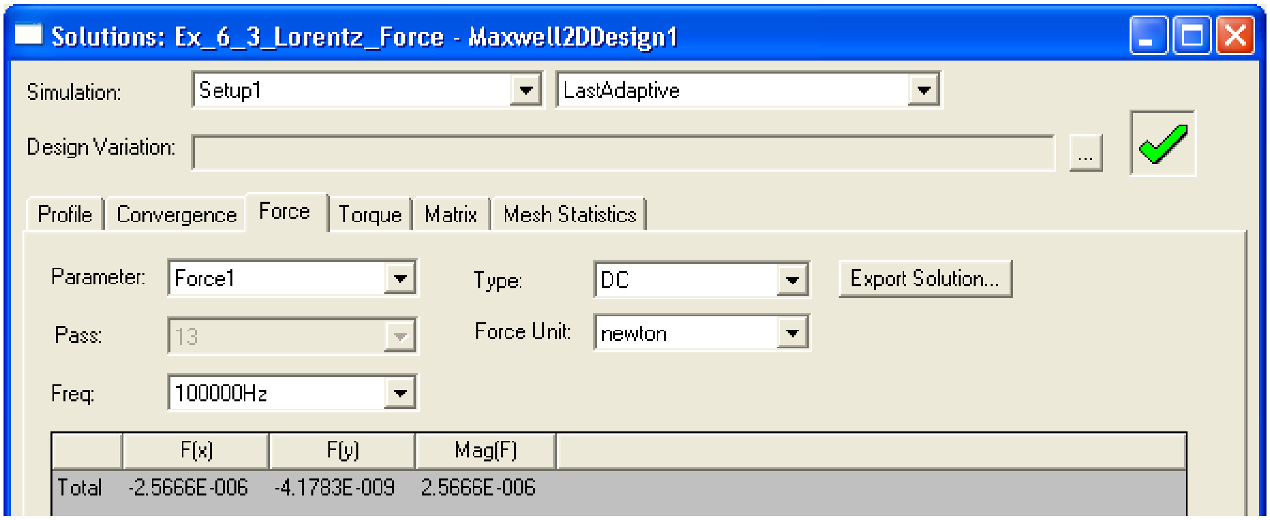
1. Click Right on the **Analysis** folder in the Model Tree and select **Add Solution Setup…**
2. On the General tab, re-set the Number of passes to **15.**
3. Percent Error to **0.01**
4. On the Solver tab, re-set the Adaptive Frequency to **100kHz.**

Solve the Problem

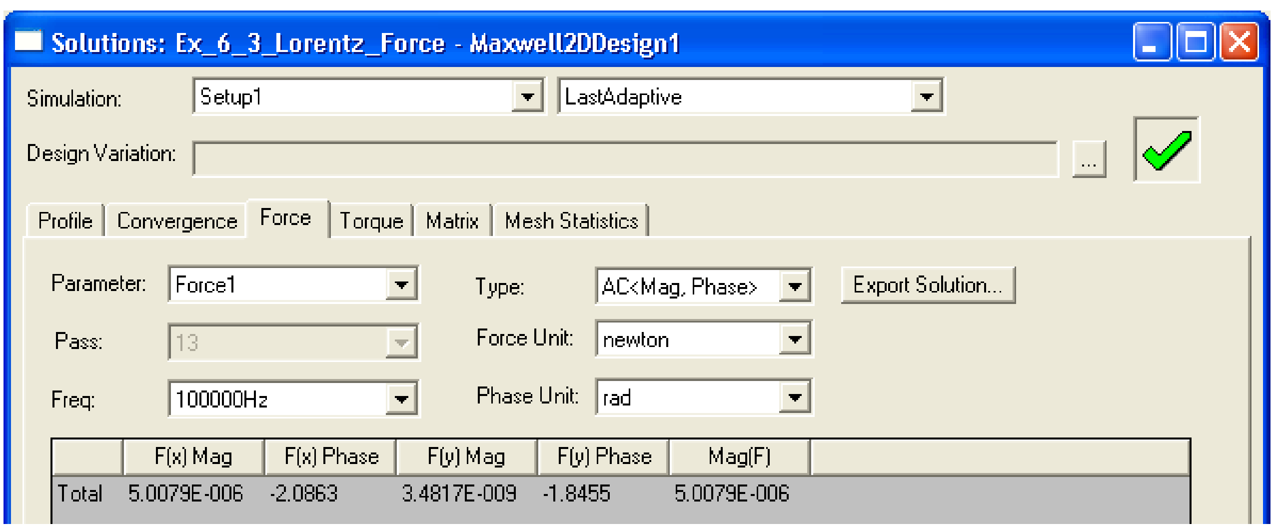
1. Save the project by clicking on menu item **File > Save As**
2. Select the menu item **Maxwell 2D > Validation Check** to verify problem setup
3. Click on **Maxwell 2D > Analyze All**.

View the Results

1. Select **Maxwell 2D > Results > Solution Data…** and click on the **Force** tab. The force results are reported for a 1 meter depth of the model. The DC forces are shown below.



2. Now select Type:**AC<Mag,Phase>** This shows the magnitude of the force **F(x)Mag** is approximately **5e-6 (N)** and the phase **F(x)Phase** is **-2.0 radians** or -120 degrees.



Create a Plot of Force vs. Time

The average, AC, and instantaneous components of the Lorentz force can be plotted vs. phase by creating named expressions in the calculator using the formulas at the beginning of the application note.

1. Determine the time-averaged component of Lorentz force:
   - Click on **Maxwell 2D > Fields > Calculator** and then perform the following:
   - **Quantity > J**
   - **Quantity > B > Complex > Conj > Cross**
   - **Scalar X > Complex > Real**
   - **Number > Scalar > 0.5 > OK**
   - **Multiply**
   - **Geometry > Volume > left > OK**
   - **Integrate**
   - **Add…** Name: **Force_DC**
   - Click **OK**

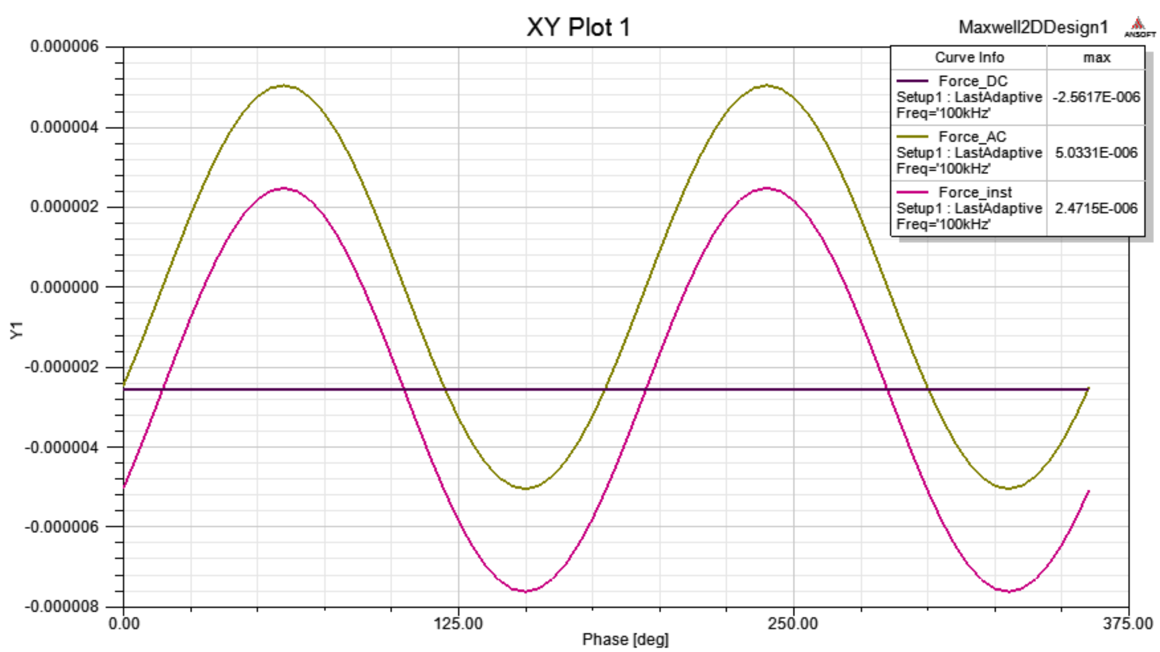
2. Determine the AC component of Lorentz force:
   - **Quantity > J**
   - **Quantity > B > Cross**
   - **Scalar X**
   - **Function > Phase > OK**
   - **Complex > AtPhase**
   - **Number > Scalar > 0.5 > OK**
   - **Multiply**
   - **Geometry > Volume > left > OK**
   - **Integrate**
   - **Add…** Name: **Force_AC**
   - Click **OK**

3. Determine the instantaneous (DC + AC) component of Lorentz force. In the **Named Expressions** panel:
   - In the Named Expressions window, select **Force_DC** and **Copy to stack**
   - Select **Force_AC** and **Copy to stack**
   - **Add**
   - **Add…** Name: **Force_inst**
   - Click **OK** and **Done** to close the calculator window.

4. Create a plot of Force vs. Phase. Now that the force quantities have been created, a plot of these named expressions can been created.

- Select **Maxwell 2D > Results > Create Fields Report** > **Rectangular Plot**
- Change the **Primary Sweep**: from the default **Freq** to **Phase.**
- Category: **Calculator Expressions**
- Quantity: **Force_DC, Force_AC, Force_inst** (hold down shift key to select all three at once)
- **New Report** > **Close**
- Right mouse click on the legend and select: **Trace Characteristics > Add…**
- Category: **Math**
- Function: **max**
- **Add** > **Done**
- Double left mouse click on the legend and change from the **Attribute** to the **General** tab.
- Check **Use Scientific Notation** and click on **OK**.

Note: The "max" values match the results from **Solution Data > Force**. I can also be observed that the forces fluctuate at 2 times the excitation frequency since there are two complete cycles over 360 degrees as shown below.



6. Finally, the instantaneous force on the left busbar can be calculated using an alternate method, the Maxwell Stress Tensor method. This method is different than both the Lorentz force and virtual force methods. The Maxwell Stress Tensor method is extremely sensitive to mesh. The force on an object can be determined by the following equation:

$$F_{MST} = \int \left(\overline{B} \cdot \overline{n}\right) \overline{H} - 0.5\,(\overline{B} \cdot \overline{H})\overline{n}\; dV \quad \textit{evaluated at phase}\ (\omega t = \textit{degrees})$$

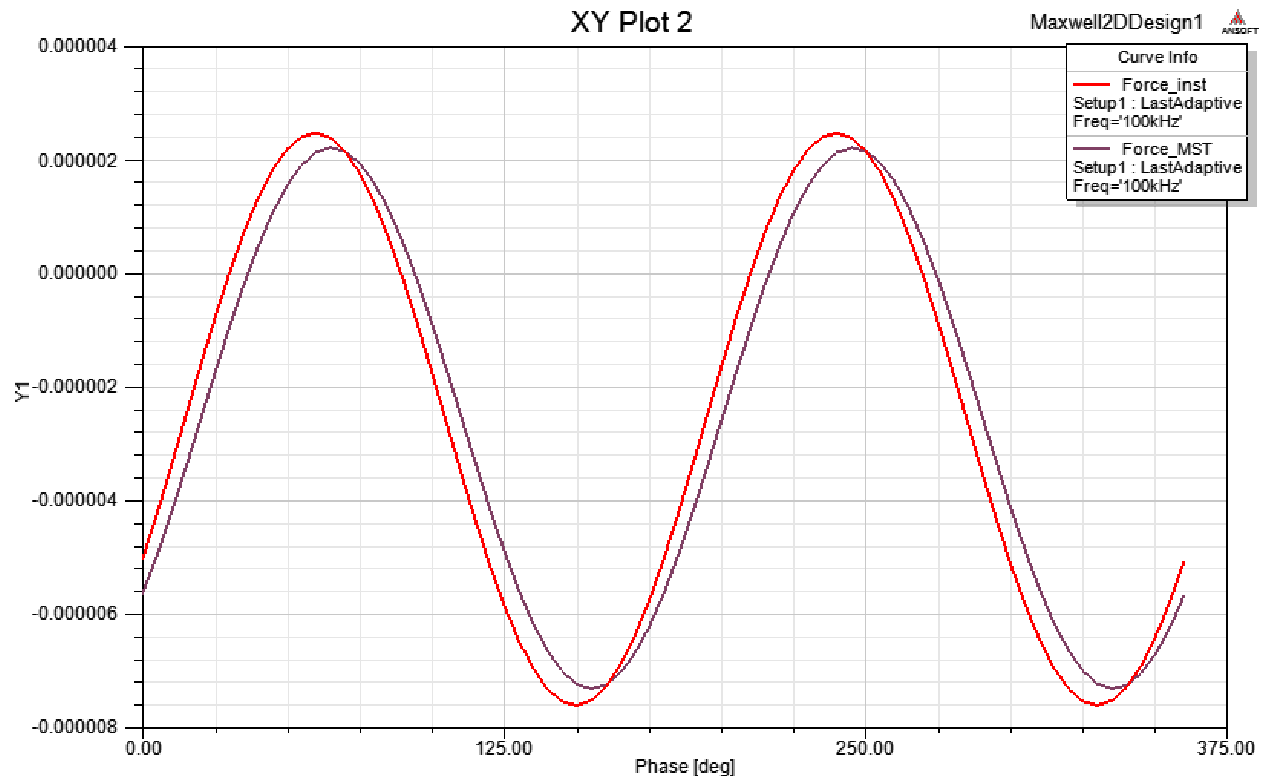
Determine the instantaneous component of force at time wt=0 using the Maxwell Stress Tensor method in the calculator:

Quantity > B	Loads the B vector
Function > Phase > OK	Loads the function Phase
Complex > At Phase	Evaluates the B vector at phase = wt
Geometry > Line > left > OK	This enters the edge of the left busbar
Unit Vector > Normal	To determine the unit normal vector for left busbar
Dot	To take B-dot-Unit Normal
Quantity > H	Loads the H vector
Function > Phase > OK	Loads the function Phase
Complex > At Phase	Evaluates the H vector at phase = wt
Multiply	This multiplies B and H
Quantity > B	Loads the B vector
Function > Phase > OK	Loads the function Phase
Complex > At Phase	Evaluates the B vector at phase = wt
Quantity > H	Loads the H vector
Function > Phase > OK	Loads the function Phase
Complex > At Phase	Evaluates the H vector at phase = wt
Dot	Computes B-dot-H
Number > Scalar > 0.5 > OK	
Multiply	Multiplies the quantity by 0.5
Geometry > Line > left > OK	Enters the edge of the left busbar
Unit Vector > Normal	To determine the unit normal vector for left busbar
Multiply	This multiplies the quantity times unit normal vector
Neg	This takes the negative
Add	
Scal? > ScalarX	To extract the x-component of the quantity
Geometry > Line > left > OK	Enters the edge of the left busbar
Integrate	To integrate the force density and obtain the force in newtons
Add…	Name: Force_MST

7. Create a plot of the Maxwell Stress Tensor Force vs. Phase.

- Select **Maxwell 2D > Results > Create Fields Report** > **Rectangular Plot**
- Change the **Primary Sweep**: from the default **Freq** to **Phase.**
- Category: **Calculator Expressions**
- Quantity: **Force_inst, Force_MST**

Note: The slight difference in these curves is due to mesh error in the stress tensor calculation.



This completes PART 1 of the exercise.

PART 2 - The 3D Eddy Project

Now the identical model will be simulated in Maxwell 3D.

Setup the Design

1. Click on the menu item **Project > Insert Maxwell 3D Design**
2. Click on the menu item **Maxwell 3D > Solution Type ...**
   - Select the radio button Magnetic: **Eddy Current**
3. Draw the Solution Region
   - Click on Draw > Box *(Enter the following points using the tab key).*
     - X: **0**, Y: **-150**, Z: **-150**
     - dX: **10**, dY: **300**, dZ: **300**
   - Change its properties:
     - Name: **Region**
     - Display Wireframe: **Checked**
   - Select **View > Fitall > Active View** to resize the drawing window.

Create the Model

Now the model can be created. This model also consists of a left and right busbar that have a 4mm square cross-section and a length of 10mm.

Create the Left Busbar

- Click on **Draw > Box**
  - X: **0** Y: **-12**, Z: **-2**
  - dX: **10**, dY: **4**, dZ: **4**
- Change its properties:
  - Name: **left**
  - Material: **Copper**
  - Color: **Red**

Create the Right Busbar

- Click on **Draw > Box**
  - X: **0** Y: **8**, Z: **-2**
  - dX: **10**, dY: **4**, dZ: **4**
- Change its properties:
  - Name: **Right**
  - Material: **Copper**
  - Color: **Red**

Assign the Boundaries and Sources

The current is assumed to be 1A at 0 degrees in the left busbar and -1A at 60 degrees in the right busbar. The default boundary in Maxwell 3D in no-fringing. So a boundary does not need to be explicitly assigned.

1. To assign the source current, the four (4) end faces of the conductors must be selected. Choose **Edit > Select > Faces** to change the selection mode from object to face.
2. Zoom in to the busbars using:**View > Zoom In**
3. Click on the front face of the **left** busbar.
    - Click on **Maxwell > Excitations > Assign > Current**
        - Name: **Current1**
        - Value: **1A**
        - Phase: **0**
        - Type: **Solid**
3. Select the other face of the left busbar. Select it and then:
    - Click on **Maxwell > Excitations > Assign > Current**
        - Name: **Current2**
        - Value: **1A**
        - Phase: **0**
        - Type: **Solid**
        - Click on **Swap Direction** to be sure that the red directional arrow is pointing out of the conductor
5. Click on the front face of the **right** busbar.
    - Click on **Maxwell > Excitations > Assign > Current**
        - Name: **Current3**
        - Value: **1A**
        - Phase: **60**
        - Type: **Solid**
6. Select the other face of the left busbar. Select it and then:
    - Click on **Maxwell > Excitations > Assign > Current**
        - Name: **Current4**
        - Value: **1A**
        - Phase: **60**
        - Type: **Solid**
        - Click on **Swap Direction** to be sure that the red directional arrow is pointing out of the conductor

Turn on the Eddy Effects in the winding

In order to consider the skin effects in the busbars, the eddy effect must be turned on.

1. Choose **Maxwell 3D > Excitations > Set Eddy Effects ...**
2. Verify that the eddy effect for **left** and **right** is checked.
3. Un-check the displacement current calculation.

Assign the Parameters

In order to automatically calculate force on an object, it must be selected in the Parameters panel. In Maxwell 3D, you can calculate both virtual and Lorentz force. Note however that Lorentz force is only valid on objects with a permeability = 1.

1. Select the **left** busbar by clicking on it in the history tree or on the screen.
2. Click on **Maxwell > Parameters > Assign > Force**
3. Name: **Force_Virtual**
4. Type: **Virtual**
5. Click **OK** to enable the virtual force calculation.
6. Click on **Maxwell > Parameters > Assign > Force**
7. Name: **Force_Lorentz**
8. Type: **Lorentz**
9. Click **OK** to enable the lorentz force calculation.

Add an Analysis Setup

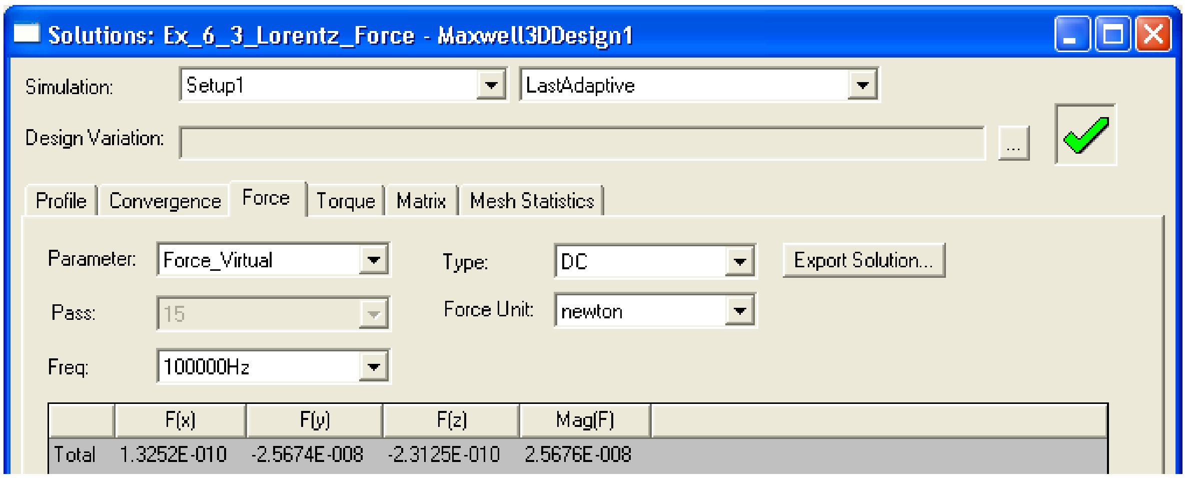
1. Click Right on the **Analysis** folder in the Model Tree and select **Add Solution Setup…**
2. On the General tab, re-set the Number of passes to **15.**
3. Percent Error to **0.01**
4. On the Solver tab, re-set the Adaptive Frequency to **100kHz.**
5. Click **OK** to save the setup.

Solve the Problem

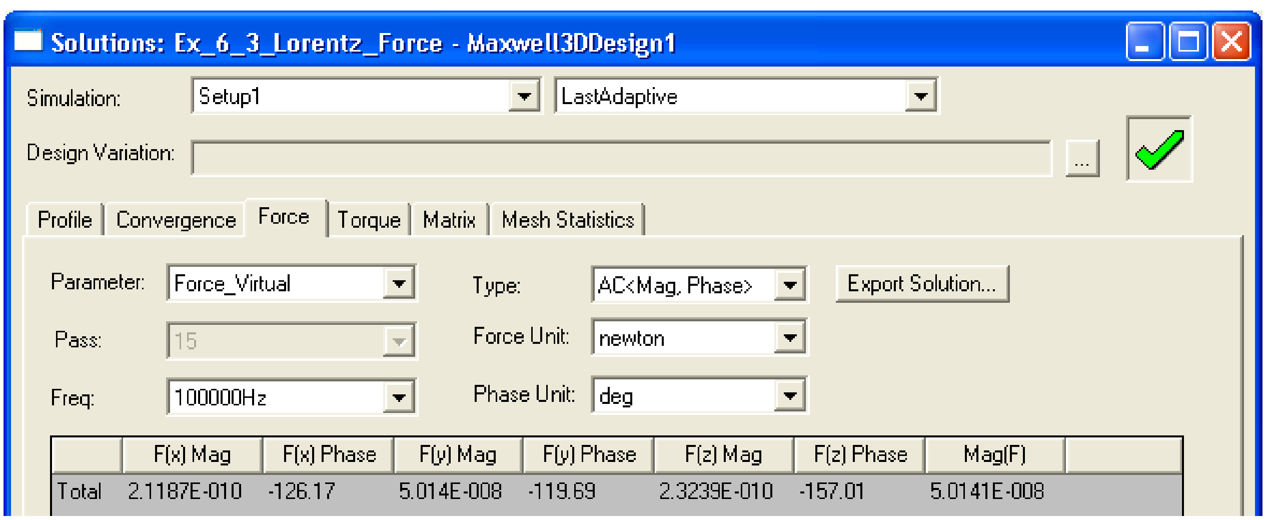
1. Save the project by clicking on menu item **File > Save**
2. Select the menu item **Maxwell 3D > Validation Check** to verify problem setup
3. Click on **Maxwell 3D > Analyze All**.

View the Results

3. Select **Maxwell 3D > Results > Solution Data…** and click on the **Force** tab. Notice that the 3D results are reported for a 10mm depth while the 2D results were for 1meter depth. The DC forces are shown below.



4. Now select Type:**AC<Mag,Phase>** This shows the magnitude of the force **F(y)Mag** is approximately **5e-8 (N)** and the phase **F(y)Phase** is **-2.0 radians** or -120 degrees. **F(y)** in 3D is same direction as **F(x)** in 2D and has a magnitude 1/100th of 2D since the modeled length is 10mm compared to 1m in 2D.

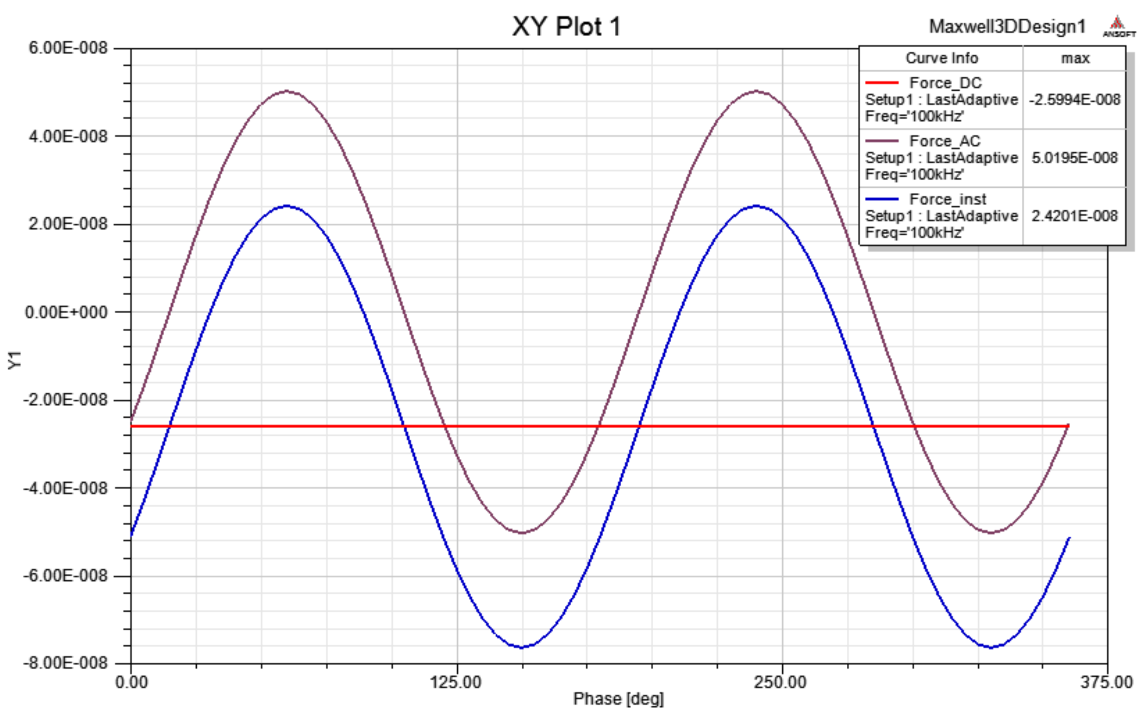


Create a Plot of Force vs. Time

The time-averaged, AC, and instantaneous components Lorentz force can be plotted vs. time by creating named expressions in the calculator using the formulas at the beginning of the application note.

1. Determine the time-averaged component of Lorentz force:
    - Click on **Maxwell 3D > Fields > Calculator** and then perform the following:
    - **Quantity > J**
    - **Quantity > B > Complex > Conj > Cross**
    - **Scalar Y > Complex > Real**
    - **Number > Scalar > 0.5 > OK**
    - **Multiply**
    - **Geometry > Volume > left > OK**
    - **Integrate**
    - **Add…** Name: **Force_DC**
    - **OK**
2. Determine the AC component of Lorentz force:
    - **Quantity > J**
    - **Quantity > B > Cross**
    - **Scalar Y**
    - **Function > Phase > OK**
    - **Complex > AtPhase**
    - **Number > Scalar > 0.5 > OK**
    - **Multiply**
    - **Geometry > Volume > left > OK**
    - **Integrate**
    - **Add…** Name: **Force_AC**
    - **OK**
3. Determine the instantaneous (DC + AC) component of Lorentz force. In the **Named Expressions** panel:
    - In the Named Expressions window, select **Force_DC** and **Copy to stack**
    - Select **Force_AC** and **Copy to stack**
    - **Add**
    - **Add…** Name: **Force_inst**
    - Click on **OK** and **Done** to close the calculator window.

4. Create a plot of Force vs. Phase. Now that the force quantities have been created, a plot of these named expressions can been created.
    - Select **Maxwell 3D > Results > Create Fields Report** > **Rectangular Plot**
    - Category: **Calculator Expressions**
    - Change the **Primary Sweep**: from the default **Freq** to **Phase.**
    - Quantity: **Force_DC, Force_AC, Force_inst** (hold down shift key to select all three at once)
    - **New Report > Close**
    - Right mouse click on the legend and select: **Trace Characteristics > Add…**
    - Category: **Math**
    - Function: **Max**
    - **Add > Done**
    - Double left mouse click on the legend and change from the **Attribute** to the **General** tab.
    - Check **Use Scientific Notation** and click on **OK**. Note that these values match the results on the **Solution Data > Force**. Also, since forces fluctuate at 2 times the excitation frequency, there are two complete cycles in 360 degrees shown below.



This completes PART 2 of the exercise.

Reference:

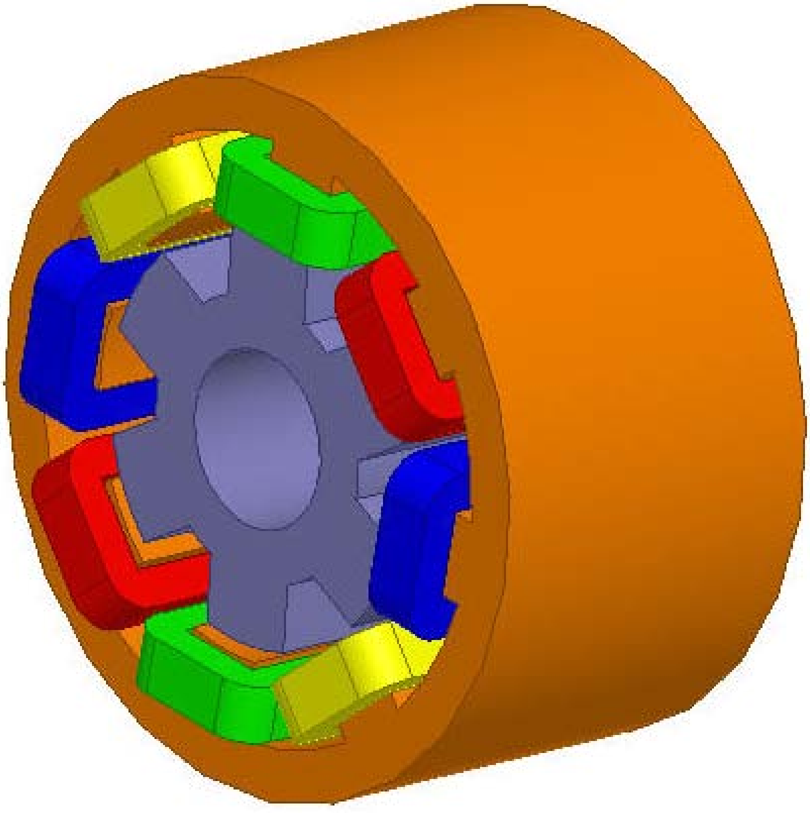
MSC Paper #118 "Post Processing of Vector Quantities, Lorentz Forces, and Moments in AC Analysis for Electromagnetic Devices" MSC European Users Conference, September 1993, by Peter Henninger, Research Laboratories of Siemens AG, Erlangen

Chapter 7.0 – Magnetic Transient

Example (Transient) – Stranded Conductors

Stranded Conductors

- This example is intended to show you how to create and analyze a transient problem on a Switched Reluctance Motor geometry using the Transient solver in the Ansoft Maxwell 3D Design Environment.
- Within the Maxwell 3D Design Environment, solid coils can be modeled as Stranded Conductors. There are many advantages to using Stranded Conductors when modeling coils that have multiple turns. The first obvious advantage is that a coil with multiple wires, say 2500, can be modeled as a single object as opposed to modeling each wire which would be impracticable. Defining a Stranded Conductor means that the current density will be uniform throughout the cross section of the conductor.
- The example that will be used to demonstrate how Stranded Conductors are implemented is a switched Reluctance Motor. This switched reluctance motor will have four phases and two coils per phase, thus we can show how independent coils can be grouped to create windings.
- Note: This tutorial shows how to setup a stranded conductor using Transient Solver and does not involve details regarding geometry creation. To see geometry creation details, please refer the example 5.3



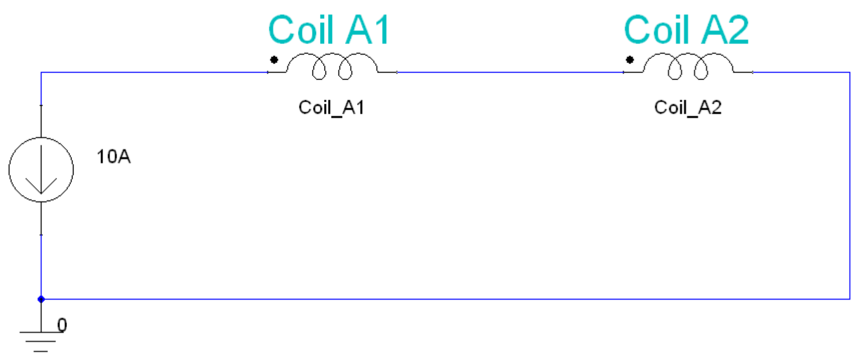
Example (Transient) – Stranded Conductors

Theory – Transient Solver

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

Theory – Transient Solver (Continued)

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



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

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Boolean Operations: **Split**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Transient**
  - Results
    - **Field Calculator**
  - Field Overlays:
    - **Magnitude B**

Example (Transient) - Stranded Conductors

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

Setting Tool Options

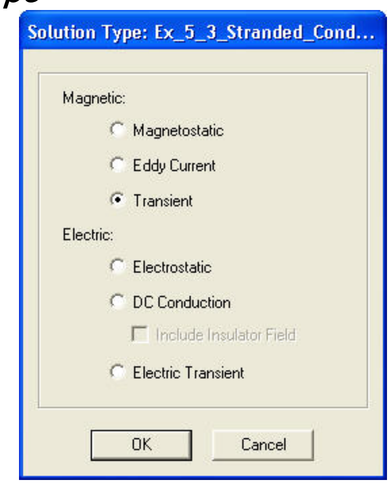
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item ***Tools > Options > Maxwell 3D Options***
     - Maxwell Options Window:
       1. Click the **General Options** tab
       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

Open Existing File

- **To Open a File**
  - Select the menu item ***File > Open***
  - Locate the file **Ex_5_3_Stranded_Conductors.mxwl** and **Open** it

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button

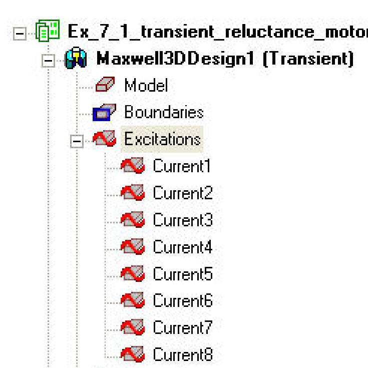


Save File

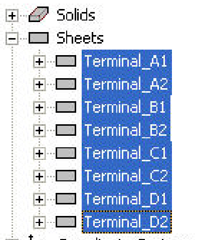
- **To Save File**
  - Select the menu item ***File > Save***
  - Save the file with a name **Ex_7_1_Transient_Reluctance_Motor**

Delete Excitations

- **Delete Specified Excitations**
  - As we have opened the file from a Magnetostatic setup, the excitation are already existing in the file
  - Delete all excitations from Project Manager tree as new excitations will be specified according to Transient Solver

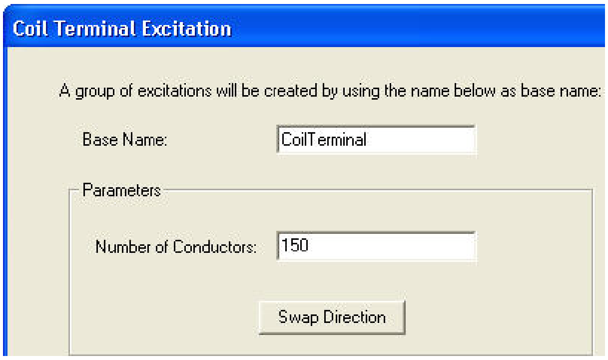


Example (Transient) - Stranded Conductors



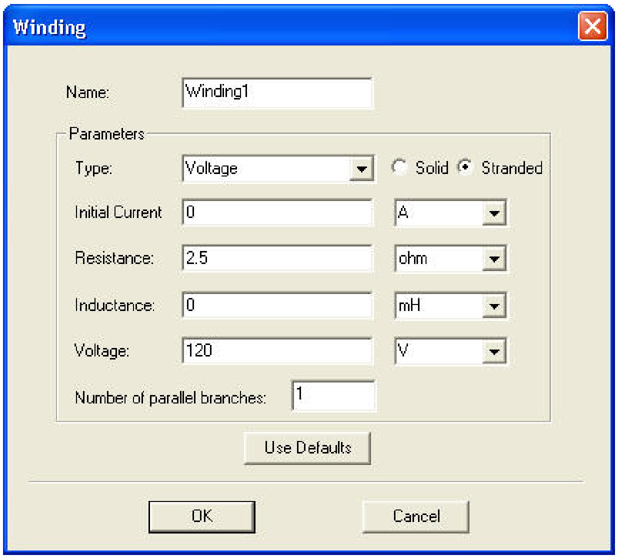
Specify Coil Terminals

- **To Specify Coil terminals**
  - Expand the history tree for Sheets
  - Press **Ctrl** and select the all sheet objects
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
  - In Coil Terminal Excitation window,
    1. Base Name: **CoilTerminal**
    2. Number of Conductors: **150**
    3. Press **OK**



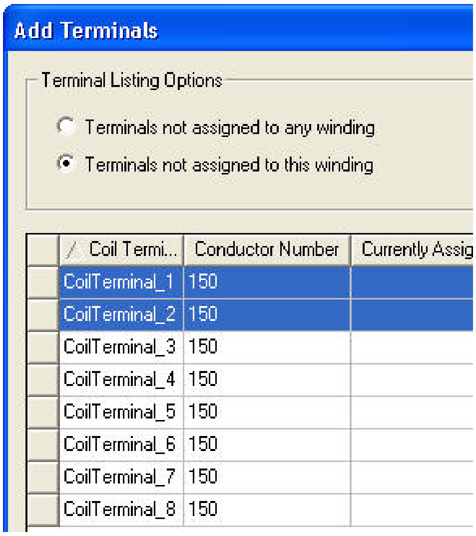
Specify Windings

- **To Add Winding**
  - Select the menu item ***Maxwell 3D > Excitations > Add Winding***
  - In Winding window,
    1. Name: **Winding1**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **2.3 ohm**
    6. Inductance: **0 mH**
    7. Voltage: **120 V**
    8. Press **OK**



Example (Transient) - Stranded Conductors

- **Add Terminals to Winding**
  - Expand the Project tree to display terminals
  - Right click on the **Winding1** from the Project tree and select **Add Terminals**
  - In Add Terminals window,
    1. Press **Ctrl** and select the terminals **CoilTerminal_1** and **CoilTerminal_2**
    2. Press **OK**
- Repeat the same steps to three more windings
  - **Winding2**
    - **CoilTerminal_3**
    - **CoilTerminal_4**
  - **Winding3**
    - **CoilTerminal_5**
    - **CoilTerminal_6**
  - **Winding4**
    - **CoilTerminal_7**
    - **CoilTerminal_8**



Assign Mesh Operations

- **Note:** The transient solver does not use automatic adaptive meshing.
- **Assign Mesh Operations for Coils**
  - Press **Ctrl** and select all the object corresponding to coils from history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict Length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **16000** (2000/tets per coil)
    4. Click the **OK** button

Example (Transient) - Stranded Conductors

- **Assign Mesh Operations for Stator and Rotor**
  - Press **Ctrl** and select the objects **Stator** and **Rotor** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict Length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **4000** (2000/tets per object)
    4. Click the **OK** button

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **General** tab:
       - Stop time: **0.02s**
       - Time step: **0.002s**
    2. Click the **OK** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button

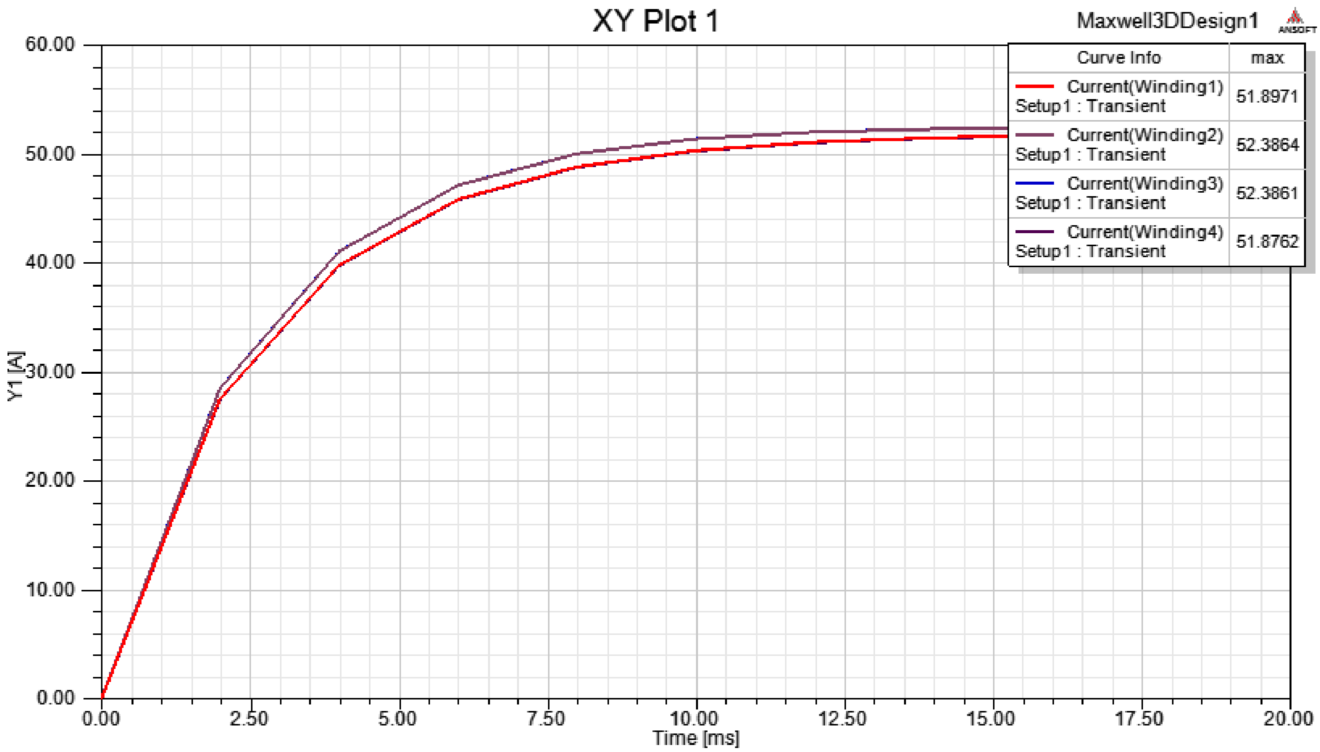
  Note: To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

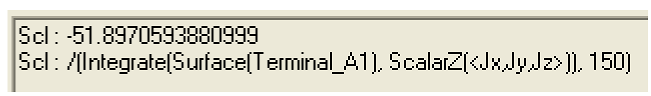
Create Quick Report

- **To Create a Report**
  - Select the menu item ***Maxwell 3D > Results > Create Transient Reports > Rectangular Plot***
  - In Report Window,
    1. Category: **Winding**
    2. Quantity: Press **Ctrl** and select **Current(Winding1), Current(Winding2), Current(Winding3), Current(Winding4)**
    3. Select the button **New Report**
    4. Press **Close**
  - Right Click on the plot and select ***Trace Characteristics > Add***
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **Max**
    - Select **Add** and **Done**



Calculate Current

- **To Calculate Current**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - In Fields Calculator window,
    1. Select **Input > Quantity > J**
    2. Select **Vector**: **Scal?** > **Scalar Z**
    3. Select **Input > Geometry**
       - Set the radio button to **Surface**
       - From the list select **Terminal_A1**
       - Press **OK**
    4. Select **Scalar >** ∫ (Integrate)
    5. Select **Input > Number**
       - Type: **Scalar**
       - Value: **150** (Number of Conductors)
       - Press **OK**
    6. Select **General > /**
    7. Select **Output > Eval**
    8. Press **Done** to close the calculator



  - Note that the value reported in calculator is same as the value shown in plot in the last step

Example (Transient) - Stranded Conductors

Create Quarter Symmetry Geometry

- So far we have been working with full geometry. Often it is useful to use symmetry in order to reduce the problem size and thus decrease the solution time. In this section, we'll show how to create the symmetric model and its impact on stranded conductors.

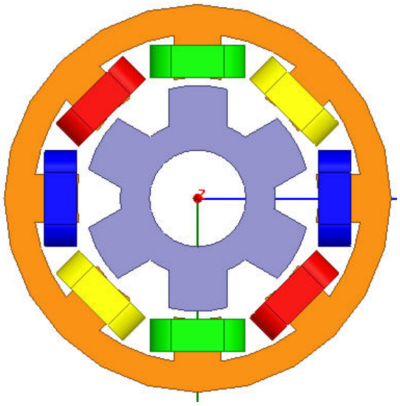
Create Symmetry Design

- **Copy Design**
  - Select the design **Maxwell3DDesign1** in Project Manager window, right click and select **Copy**
  - Select project **Ex_7_1_transient_reluctance_motor** in Project Manager window and select **Paste**

Rotate the Geometry to Create Quarter Symmetry

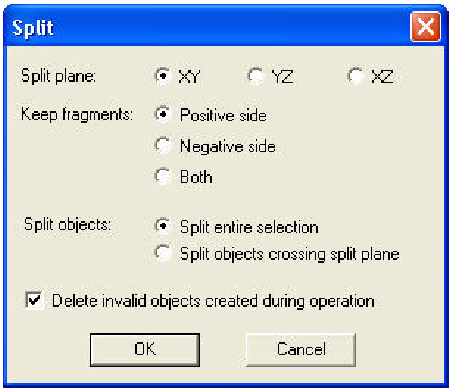
- Before splitting the model to create a ¼ model, all of the objects need to be rotated.
- **To Rotate Model**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate Window,
    1. Axis: **Z**
    2. Angle: **22.5 deg**
    3. Press **OK**

- **To Rotate the object Rotor**
  - Select the object **Rotor** from the history tree
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate Window,
    1. Axis: **Z**
    2. Angle: **7.5 deg**
    3. Press **OK**

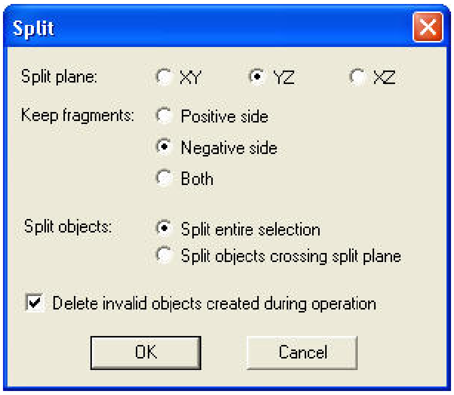


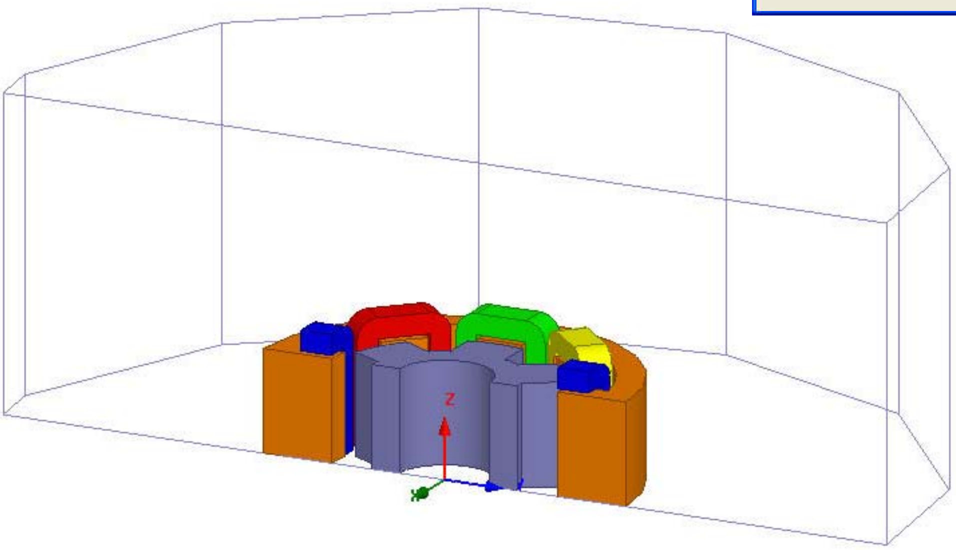
Split Model

- **Divide by XY Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XY**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



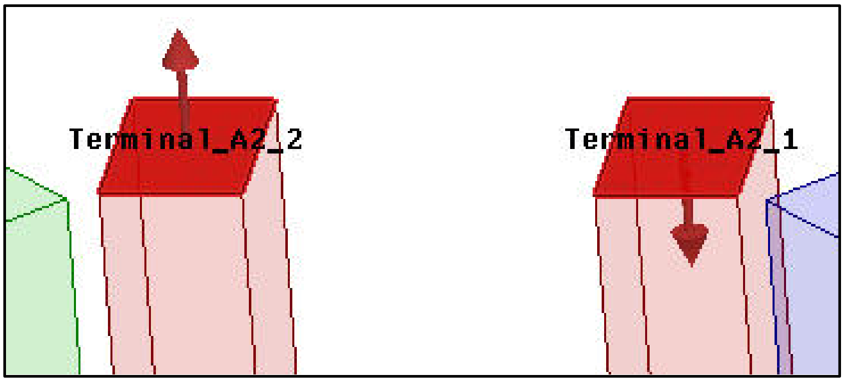
- **Divide by YZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Negative side**
    3. Split objects: **Split entire selection**
    4. Press **OK**





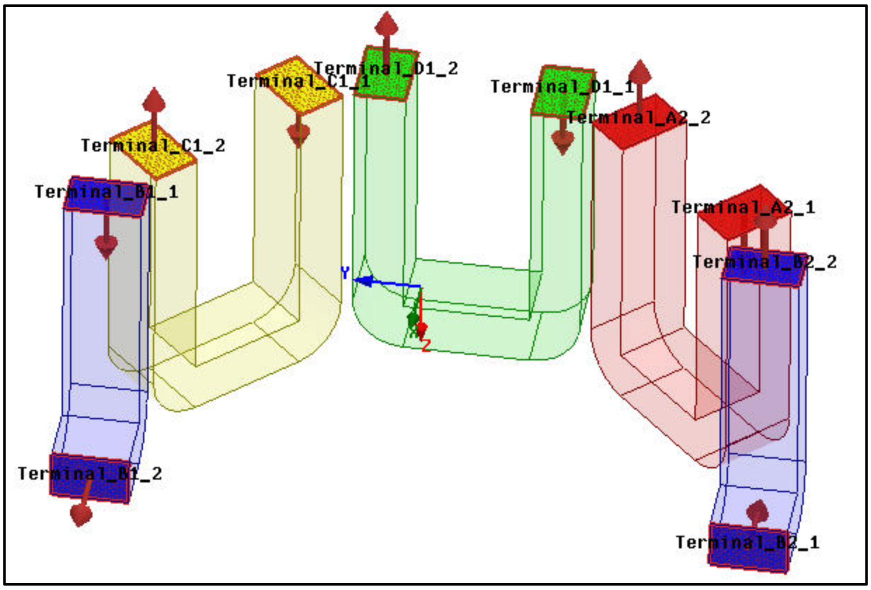
Redefining the Terminals

- When the entire coil is not being modeled and the coil cuts the surface of the solution boundary (called Region in this example), terminals need to be defined that are coincident with the Coil and the Region.
- **To Redefine Terminals**
  - Press **Ctrl** and select the objects **Region, Stator** and **Rotor**
  - Select the menu item ***View > Visibility > Hide Selection > Active View***
  - Select the menu item ***Edit > Select > Faces***
  - Select the face of **Coil_A2** that coincides with region as shown in below image
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
  - In Coil Terminal Excitation window,
    1. Name: **Terminal_A2_1**
    2. Number of Conductors: **150**
    3. Press **OK**
  - Select the other face of the object **Coil_A2** that touches with region
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
  - In Coil Terminal Excitation window,
    1. Name: **Terminal_A2_2**
    2. Number of Conductors: **150**
    3. Press the button **Swap Direction** to invert current direction
    4. Press **OK**



Example (Transient) – Stranded Conductors

- Redefine coil terminals as specified in below image for other objects
- Ensure the direction of current is consistent in all coils.



- ## Add Terminals to Windings
  - **To Add Terminals**
    - Right click on **Winding1** which exists from previous settings and select **Add Terminals**
    - In Add Terminals window,
      1. Press **Ctrl** and select the terminals **Terminal_A2_1** and **Terminal_A2_2**
      2. Press **OK**
  - Repeat the same steps to three more windings
    - **Winding2**
      - **Terminal_B1_1, Terminal_B1_2, Terminal_B2_1, Terminal_B2_2**
    - **Winding3**
      - **Terminal_C1_1, Terminal_C1_2**
    - **Winding4**
      - **Terminal_D1_1, Terminal_D1_2**

Modify Mesh Operations

- **To Modify Mesh Operations**
  - Expand the Project Manager tree to view Mesh Operations
  - Double click on the mesh operation **Length1** that corresponds to Coils
  - In Element Length Based Refinement window,
    1. Change Maximum Number of Elements to **4000** (1/4th of the value used for Full Model
    2. Press **OK**
  - Double click on **Length2** that corresponds to Stator and Rotor
  - In Element Length Based Refinement window,
    1. Change Maximum Number of Elements to **1000** (1/4th of the value used for Full Model
    2. Press **OK**

Set Symmetry Multiplier

- **To Set Symmetry Multiplier**
  - Select the menu item ***Maxwell 3D > Model > Set Symmetry Multiplier***
  - Set Symmetry Multiplier value of **4** in the window
  - Press **OK**

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button

  Note: To view any errors or warning messages, use the Message Manager.

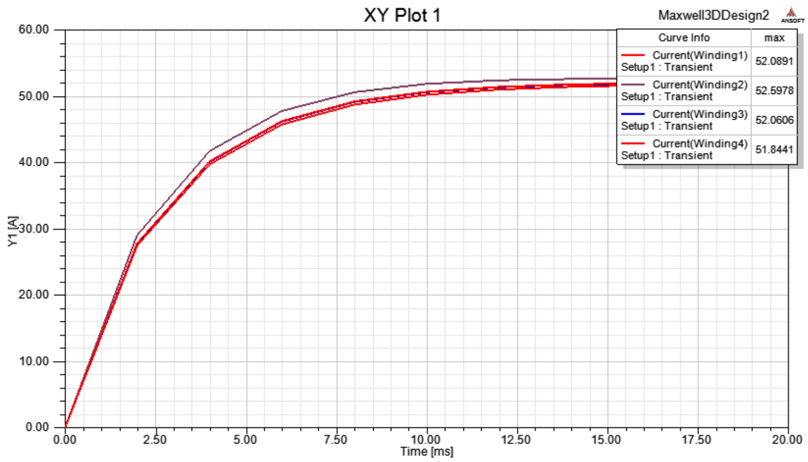
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Example (Transient) - Stranded Conductors

Results

- **To View Results**
  - Expand the Project Manager tree to view XY Plot 1 under Results



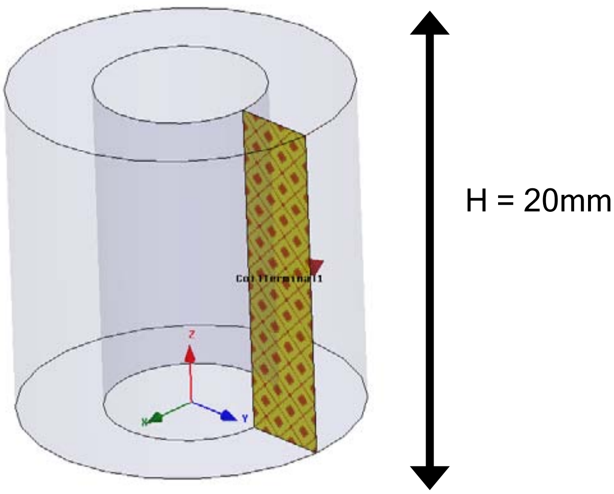
- **Create Sheet Object for Current Calculation**
  - Select any of the end faces of **Coil_A2**
  - Select the menu item ***Modeler > List > Create > Face List***
- **To Calculate Current**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - In Fields Calculator window,
    1. Select **Input > Quantity > J**
    2. Select **Vector**: **Scal?** > **Scalar Z**
    3. Select **Input > Geometry**
       - Set the radio button to **Surface**
       - From the list select **Facelist1**
       - Press **OK**
    4. Select **Scalar >** ∫ (Integrate)
    5. Select **Input > Number**
       - Type: **Scalar**
       - Value: **150** (Number of Conductors)
       - Press **OK**
    6. Select **General > /**
    7. Select **Output > Eval**
    8. Press **Done** to close the calculator

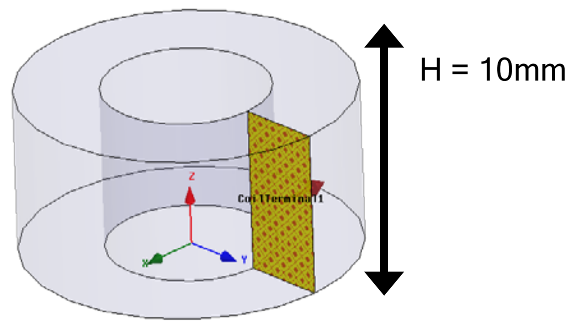
  - The value of current is around **52 A** which is save as previous

Example (Transient) – Stranded Conductors

Half Coils

- Note: If the cross section of the coil is cut in half, then the number of conductors is half. An example for a simple coil is shown below:



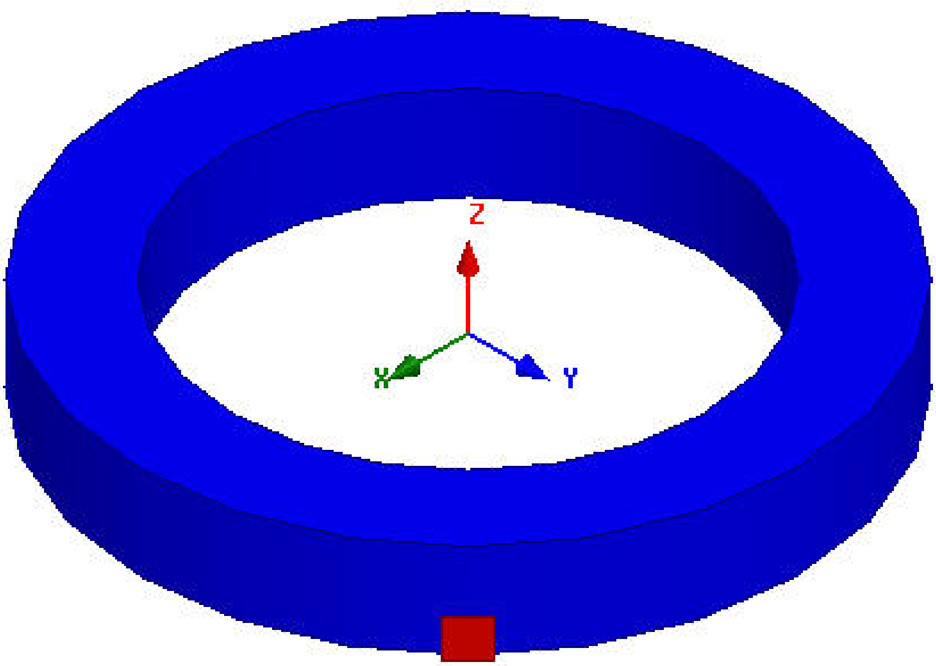


Modeling the Full Coil:
Conductor Number = 150
Symmetry Multiplier = 1

Cut the Full Coil in half and use an Odd Symmetry Boundary:
Conductor Number = 75

Rotational Cylindrical Motion using the Transient Solver

- Transient Solver in Maxwell allows to be combined with motion. This motion can be translational, rotational cylindrical or rotational non-cylindrical (relay-type motion). This example shows on a simple geometry how to set-up rotational cylindrical motion project in a Transient Solver. It teaches how to set-up a band for rotational cylindrical motion, create and post process time-dependent field variables and create an animation. It shows how to utilize the periodicity of the configuration to reduce the size of the problem.
- The geometry consists of a cylindrical 4-pole radially magnetized NdFeB magnet with sinusoidal distribution of magnetization. The magnet rotates with a constant speed of 360 degrees per second around its axis. There is a Hall Sensor at certain distance from the magnet. The goal is to model a time-dependent magnetic field created by rotating magnet and follow the magnitude of this field by employing the Hall sensor. The role of the sensor is to calculate the average flux density passing through the sensor.



Example (Transient) – Rotational Motion

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Regular Polyhedron, Rectangle**
    - Boolean Operations: **Split**
    - Sweep Operations: **Around Axis**
    - Model: **Band for rotational cylindrical motion**
  - Boundaries/Excitations
    - **Matching boundaries (Master/Slave)**
    - **Natural outer boundary**
  - Material properties
    - **Edit material (4-pole NdFeB radially magnetized magnet)**
  - Analysis
    - **Transient**
  - Results
    - **Fields Calculator**
    - **Rectangular Plot**
  - Field Overlays:
    - **Flux Density Vector Plots**
    - **Flux Density Vector Animation**

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

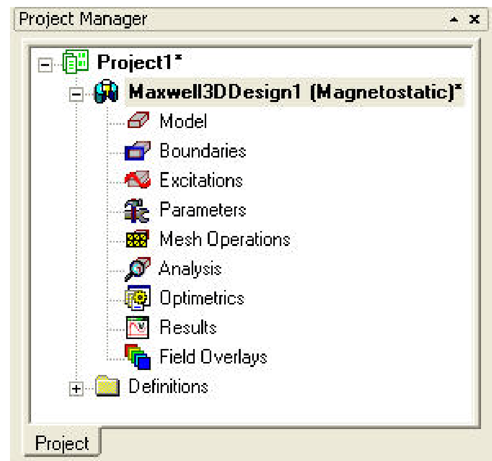
Setting Tool Options

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

Example (Transient) – Rotational Motion

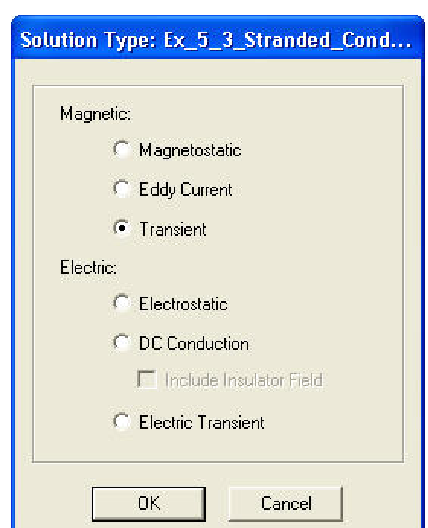
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

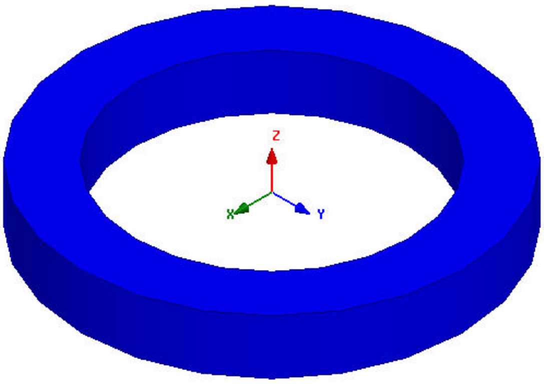
- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button



Create Magnet

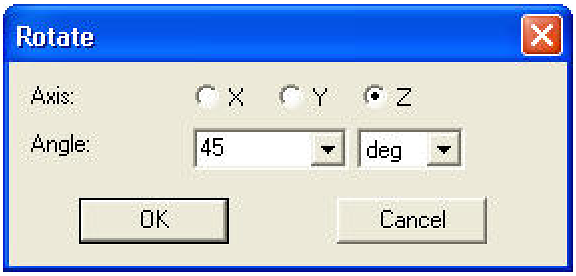
- Two different ways can be used to create the magnet
  - Using primitives from menu item ***Draw > Cylinder***
  - Create a rectangle and sweep it along an axis
- Here we will use second approach
- **To Create Rectangle**
  - Select the menu item ***Modeler > Grid Plane > XZ***
  - Select the menu item ***Draw > Rectangle***
    1. Using Coordinate entry field, enter the position of rectangle
       - **X = 5, Y = 0, Z = -1,** Press the **Enter** key
    2. Using Coordinate entry field, enter the opposite corner of rectangle
       - **X = 7, Y = 0, Z = 1,** Press the **Enter** key

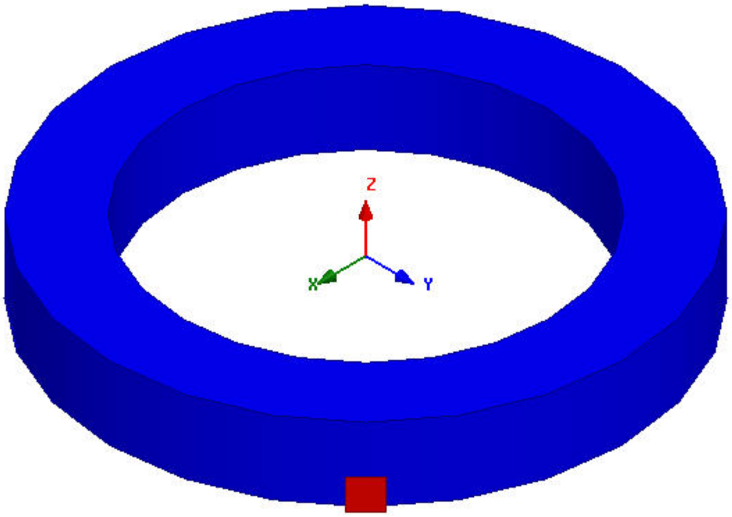
       (Note: Above values are in **Absolute** coordinates)
- **To Change Attributes**
  - Select the resulting sheet from the history tree and goto Properties window
    1. Change the name of the sheet to **Magnet**
    2. Change the color of the sheet to **Blue**
- **To Sweep Rectangle**
  - Select the sheet **Magnet** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Sweep Axis: **Z**
    2. Angle of Sweep: **360 deg**
    3. Press **OK**



Create Hall Sensor

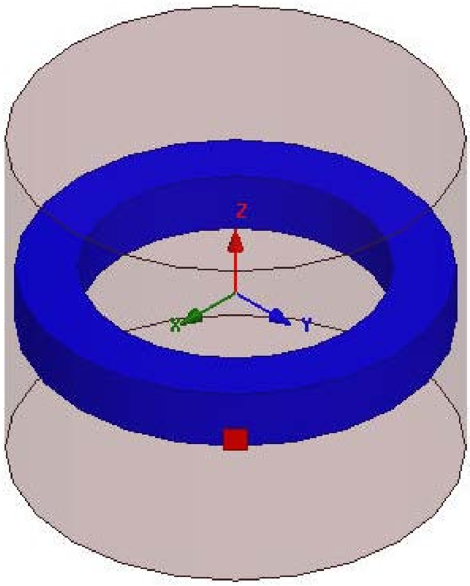
- **To Create Rectangle**
  - Select the menu item ***Modeler > Grid Plane > YZ***
  - Select the menu item ***Draw > Rectangle***
    1. Using Coordinate entry field, enter the position of rectangle
       - **X = 8, Y = -0.4, Z = -0.4,** Press the **Enter** key
    2. Using Coordinate entry field, enter the opposite corner of rectangle
       - **dX = 0, dY = 0.8, dZ = 0.8,** Press the **Enter** key
- **To Change Attributes**
  - Select the resulting sheet from the history tree and goto Properties window
    1. Change the name of the sheet to **Sensor**
    2. Change the color of the sheet to **Red**
- **To Rotate the Sheet**
  - Select the sheet **Sensor** from the history tree
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **45 deg**
    3. Press **OK**





Create Band

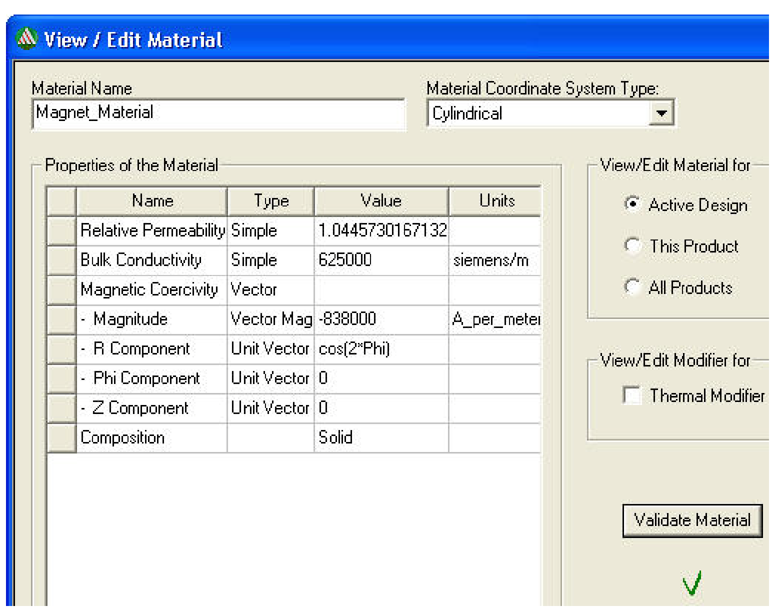
- In a Transient Solver, Band is an object that contains all rotating objects. It is a way to tell Maxwell which objects are steady and which are moving. In this example there is one moving (rotating) object: Magnet. For rotational cylindrical type of motion Band is usually a cylinder or polyhedron. It should be a solid cylinder, not a hollow cylinder. For translational type of motion Band cannot be a true surface object (cylinder). It must be a faceted object, such as Regular Polyhedron.
- **To Create a Band Object**
  - Select the menu item ***Modeler > Grid Plane > XY***
  - Select the menu item ***Draw > Cylinder***
    1. Using Coordinate entry field, enter the center of base
       - **X = 0, Y = 0, Z = -6,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - **dX = 7.3, dY = 0, dZ = 12,** Press the **Enter** key
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Band**
    2. Change the transparency of the object to **0.8**
    3. Change the color of the object to **Brown**



Assign Material: Magnet

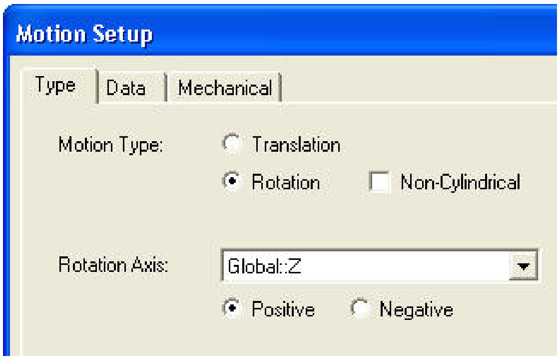
To Assign Magnet Material

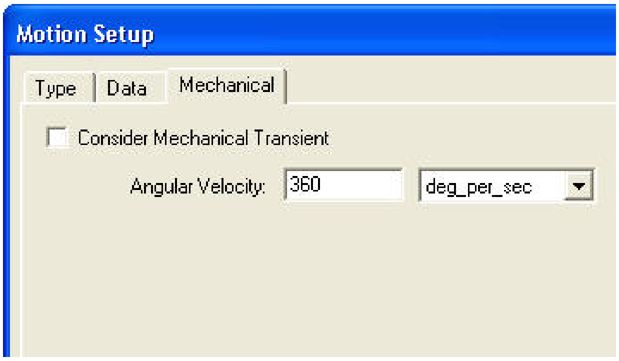
- Select the object **Magnet** from the history tree, right click and select **Assign Material**
- In Select Definition window,
  1. Type **NdFe30** in the **Search by Name** field
  2. Select option **Clone Material**
- In View/Edit Material window,
  1. Name: **Magnet_Material**
  2. Material Coordinate System Type: **Cylindrical**
- **Note:** The magnet is radially magnetized so it is advantageous to use the cylindrical coordinate system. The direction of magnetization is determined by Unit Vector. For the cylindrical coordinate system the components of Unit Vector are ***R***, ***Phi*** and ***Z***, standing for ***radial***, ***circumferential*** and ***z***-directions.
  3. Magnetic Coercivity:
     - Magnitude: **-838000**
     - R Component: **cos(2*Phi)**
- **Note:** Phi is a default symbol for an angle in cylindrical coordinate system. In order to get 4-pole magnet the argument in cos is multiplied by two.
  4. Select **Validate Material**
  5. Press **OK**



Specify Motion

- **To Specify Band**
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Model > Motion Setup > Assign Band***
  - In Motion Setup window,
    1. **Type** tab
       - Motion Type: **Rotational**
       - Rotation Axis: **Global:Z**
       - **Positive**
    2. **Mechanical** tab
       - Angular Velocity: **360 deg_per_sec**
    3. Press **OK**



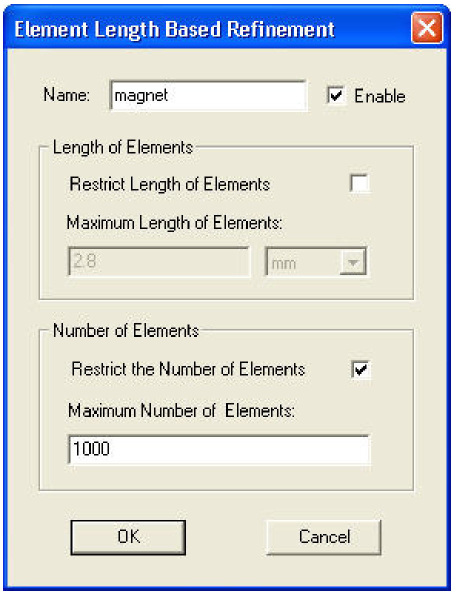


Create Region

- **To Create Region**
  - Select the menu item ***Draw > Cylinder***
    1. Using Coordinate entry field, enter the center of base
       - **X = 0, Y = 0, Z = -8,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - **dX = 16, dY = 0, dZ = 16,** Press the **Enter** key
- **To Change Attributes**
  - Select the resulting sheet from the history tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Change Display Wireframe to ☑ **Checked**

Assign Mesh Operations

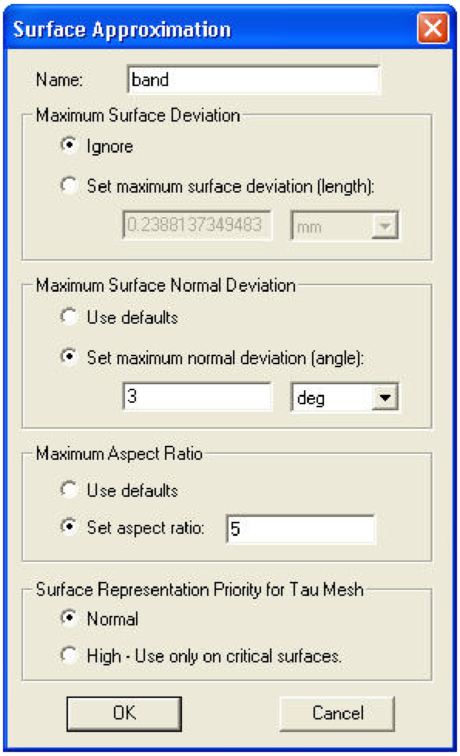
- Transient solver does not use Adaptive meshing as for other solvers. Hence mesh parameters need to be specified in order to get refined mesh.
- As we have defined Band region, Maxwell will automatically create a mesh operation in order to resolve cylindrical gap region between Band and the object inside it (Magnet). For rest, we need to manually specify mesh operations.
- **To Assign Mesh Operations for Magnet**
  - Select the object **Magnet** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **magnet**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**



- **To Assign Mesh Operations for Region**
  - Repeat the steps same as for Magnet and specify parameters as below
    1. Name: **region**
    2. Maximum Number of Elements: **8000**

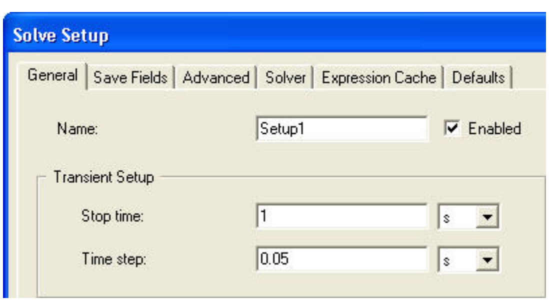
Example (Transient) – Rotational Motion

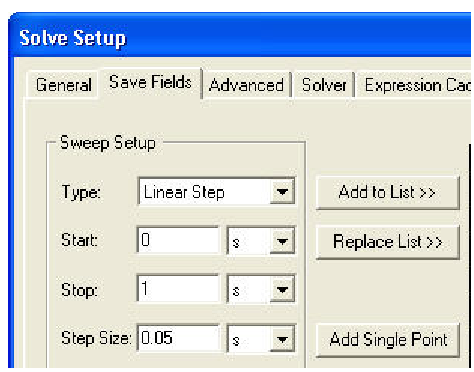
- **To Assign Mesh Operations for Band**
  - Since a cylinder is a true (curved) surface object, we will use Surface Approximation in order to create a good mesh for this rotational motion problem
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Surface Approximation***
  - In Surface Approximation window,
    1. Name: b**and**
    2. Maximum Surface Normal Deviation
       - Set maximum normal deviation (angle) : **3 deg**
    3. Maximum Aspect Ratio
       - Set aspect ratio: **5**
    4. Press **OK**



Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup Window,
    1. **General** tab
       - Stop time: **1 s**
       - Time step: **0.05 s**
  - **Note:** We will analyze one revolution. Since the speed is 360 degrees per second, to analyze one revolution, we have to solve 1 s in time. We will discretize the time domain into 20 time steps, meaning that the time step will be 0.05 s
    2. **Save Fields** tab
       - Type: **Linear Step**
       - Start: **0 s**
       - Stop: **1 s**
       - Step Size: **0.05 s**
       - Select the button **Add to List >>**
    3. Press **OK**





Save

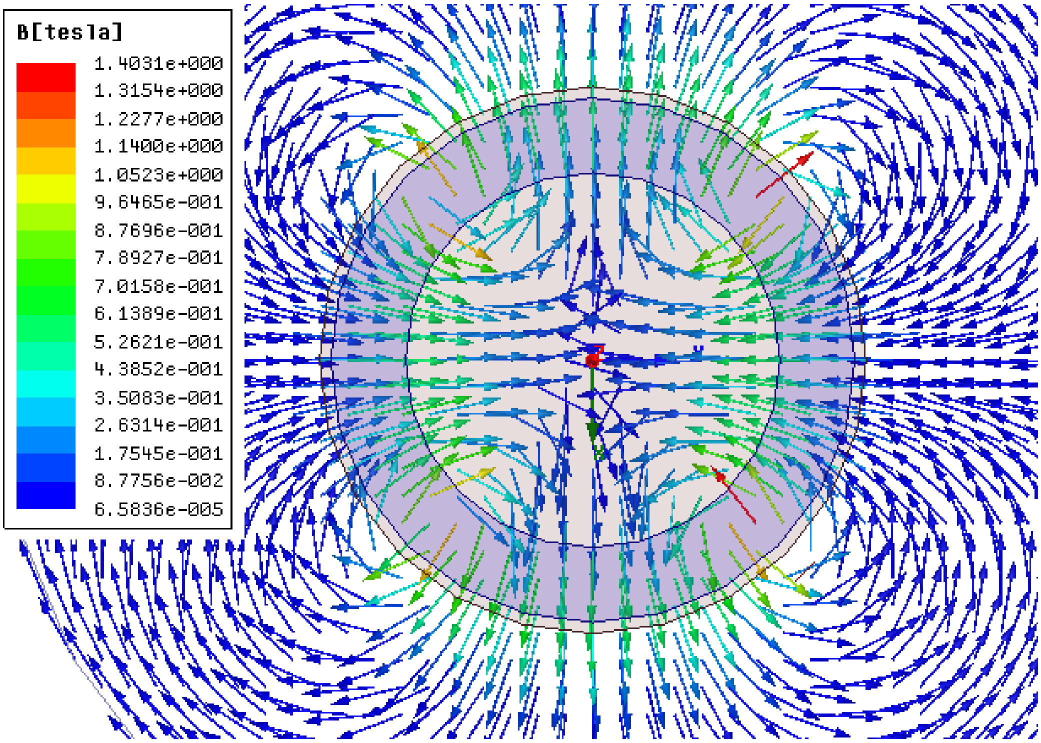
- **To Save File**
  - Select the menu item ***File > Save***
  - Save the file with the name **"Ex_7_2_Rotational_Motion.mxwl"**

Analyze

- **To Run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Plot Flux Density Vector on XY plane at time 0 s

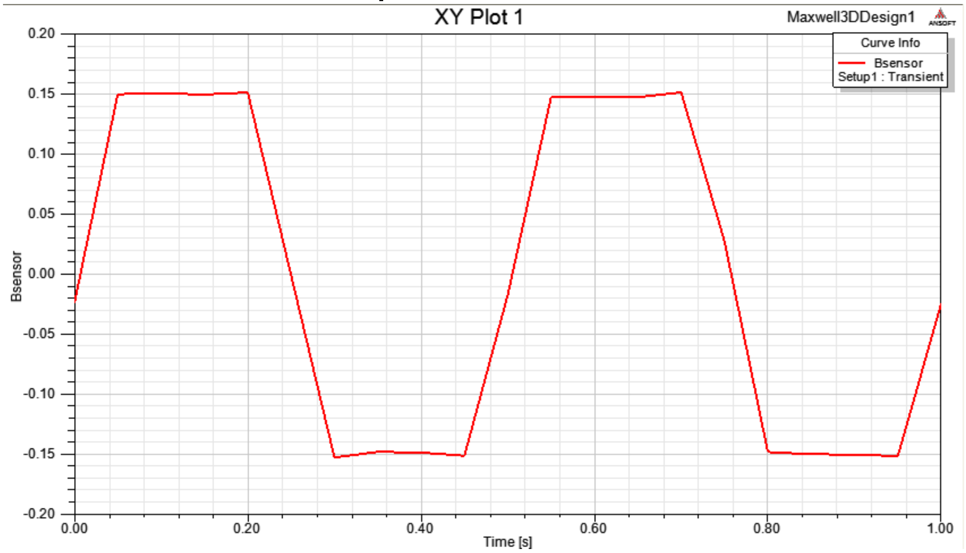
- **To Plot Flux Density at Time = 0 s**
  - Select the menu item *View > Set Solution Context*
  - In Set View Context window,
    1. Set Time to **0s**
    2. Press **OK**
  - Expand history tree for Planes and select **Global:XY**
  - Select the menu item *Maxwell 3D > Fields > Fields > B > B_Vector*
  - In Create Field Plot window,
    1. Press **Done**
- **To Modify Plot Attributes**
  - Double click on the legend to modify plot
  - In the window,
    1. **Marker/Arrow** tab
       - Size: **Set to appropriate value**
       - Map Size: ☐ **Unchecked**
    2. Press **Apply** and **Close**



Plot Hall Sensor Flux Density as a function of time

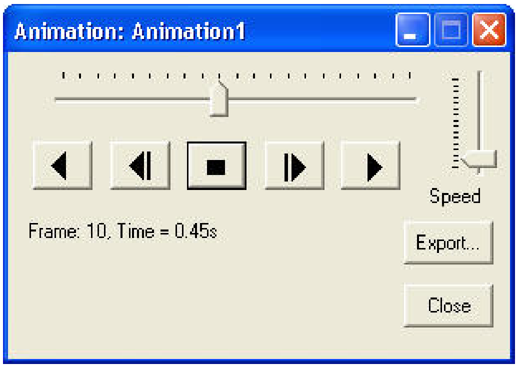
- **To define a function Bsensor**
  - Select the menu item *Maxwell 3D > Fields > Calculator*
  - In Calculator window,
    1. Select **Input > Quantity > B**
    2. Select **Input > Geometry**
       - Select the radio button **Surface**
       - Select **Sensor** from the list
       - Press **OK**
    3. Select **Vector > Normal**
    4. Press **Undo**
    5. Select **Scalar >** ∫ **Integrate**
    6. Select **Input > Number**
       - Type: **Scalar**
       - Value: **1**
       - Press **OK**
    7. Select **Input > Geometry**
       - Select the radio button **Surface**
       - Select **Sensor** from the list
       - Press **OK**
    8. Select **Scalar >** ∫ **Integrate**
    9. Select **General > /**
    10. Select **Add**
    11. Specify the name as **Bsensor**
    12. Press **OK**
    13. Press **Done** to close calculator

- **To Plot Bsensor Vs Time**
  - Select the menu item ***Maxwell 3D > Create Fields Reports > Rectangular Plot***
  - In Report window,
    1. Category: **Calculator Expressions**
    2. Quantity: **Bsensor**
    3. Select **New Report**



Create an Animation of Flux Density Vector Plot

- **To Animate Vector Plot**
  - Expand the project manager tree to view **Field Overlays > B_Vector1**
  - Right click on **B_Vector1** and select **Animate**
  - In Setup Animation window,
    1. Press **OK**
  - A window will appear to facilitate Play, Pause or Rewind animation



Utilize periodicity to reduce the size of the problem

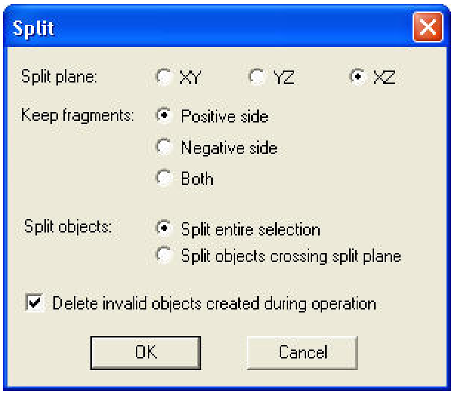
- It is always advisable to utilize possible symmetries present in the model in order to reduce the size of the problem and in turn to reduce the computation time. The geometry of the problem studied in this section is obviously symmetrical. It is a 4-pole geometry where one pole pitch can be identified as the smallest symmetry sector. The field pattern repeats itself every pole pitch, which in this case is 90 degrees (see plot on page 7-2.13). To be precise there is so called negative periodicity present in the field pattern because the field along the x-axis is exactly the same as the field along the y-axis but with the opposite direction. The negative periodicity boundary conditions can be set-up in Maxwell using Master and Slave boundaries.

Create Symmetry Design

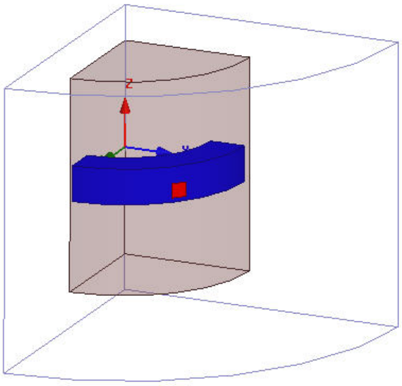
- **Copy Design**
  1. Select the design **Maxwell3DDesign1** in Project Manager window, right click and select **Copy**
  2. Select project **Ex_7_2_rotational_motion** in Project Manager window and select **Paste**

Split Model for 1/4th Section

- **Divide by YZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **XZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**

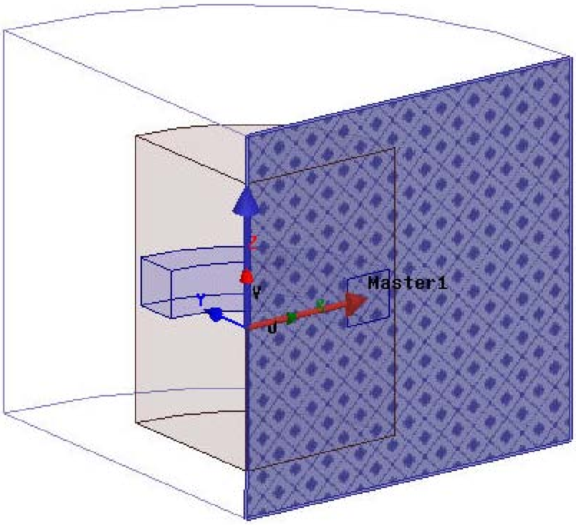


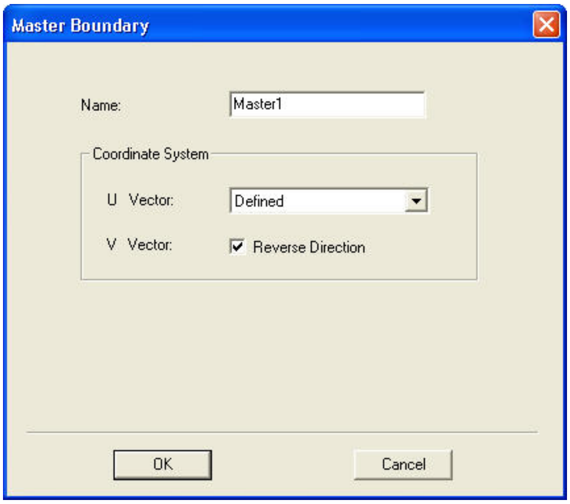
- **Divide by XZ Plane**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean >Split***
  - In Split window
    1. Split plane: **YZ**
    2. Keep fragments: **Positive side**
    3. Split objects: **Split entire selection**
    4. Press **OK**



Define Periodic

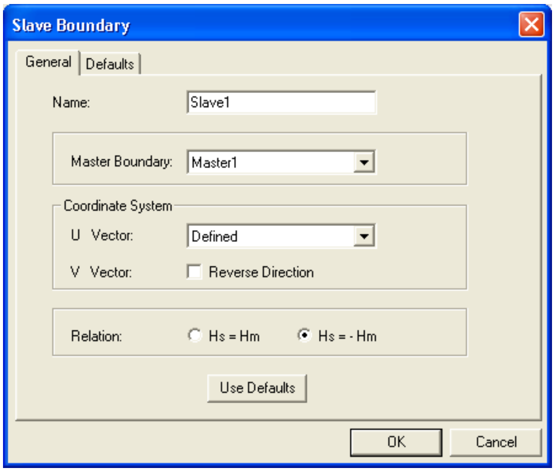
- **To Define Master Boundary**
  - Select the menu item ***Edit > Select > Faces*** or press **"F"** from the keyboard
  - Select the faces of the region lying on the plane XZ
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Master***
  - In Master Boundary window,
    1. U Vector: Select **New Vector**
       1. Using Coordinate entry field, enter the vertex
          - **X = 0, Y = 0, Z = 0,** Press the **Enter** key
       2. Using Coordinate entry field, enter the radius
          - **dX = 12, dY = 0, dZ = 0,** Press the **Enter** key
    2. V Vector: Check the box for **Reverse Direction**
    3. Press the **OK** button

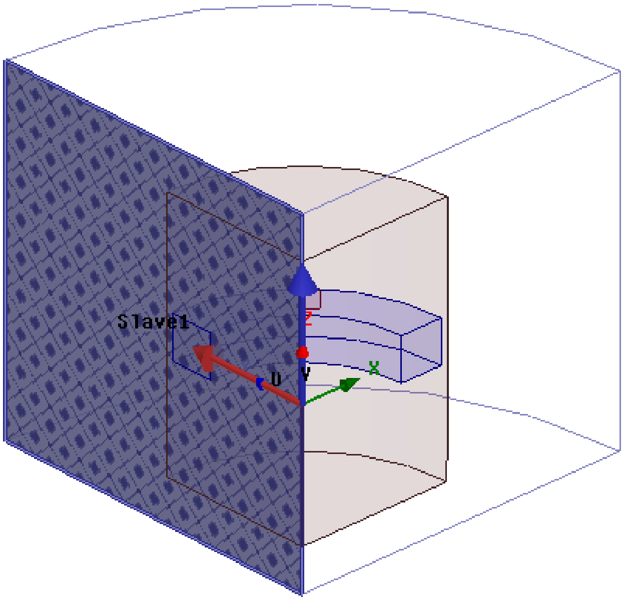




Example (Transient) – Rotational Motion

- **To Define Slave Boundary**
  - Select the face of the region that lies on YZ Plane
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Slave***
  - In Slave Boundary window,
    1. Master Boundary: Select **Master1**
    2. U Vector: Select **New Vector**
       1. Using Coordinate entry field, enter the vertex
          - **X = 0, Y = 0, Z = 0,** Press the **Enter** key
       2. Using Coordinate entry field, enter the radius
          - **dX = 0, dY = 12, dZ = 0,** Press the **Enter** key
    3. Relation: **Hs=-Hm** (negative periodicity)
    4. Press the **OK** button





Modify Mesh Operations

- Since we are using only 1/4th of the geometry, our mesh parameters should also be reduced to 1/4th to reduce number of elements
- **To Modify Mesh Parameters for magnet**
  - Expand the history tree to view mesh operations
  - Double click on the mesh operation **magnet** to modify its parameters
  - In Element Length based Refinement window,
    1. Maximum Number of Elements: Change to **250**
    2. Press **OK**
- Repeat the same process for the mesh operation **region** and change Maximum Number of Elements to **2000**
- Keep the mesh operation **band** unchanged

Set Symmetry Multiplier

- Since only one quarter of a geometry is considered, the energy, torque and force computed will be will be one quarter of the value computed with the full geometry. In order to change that, we have to set **Symmetry Multiplier** to **4**
- **To Set Symmetry Multiplier**
  - Select the menu item ***Maxwell 3D > Model > Set Symmetry Multiplier***
    1. Set Symmetry Multiplier to **4**
    2. Press **OK**

Analyze

- **To Run the Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

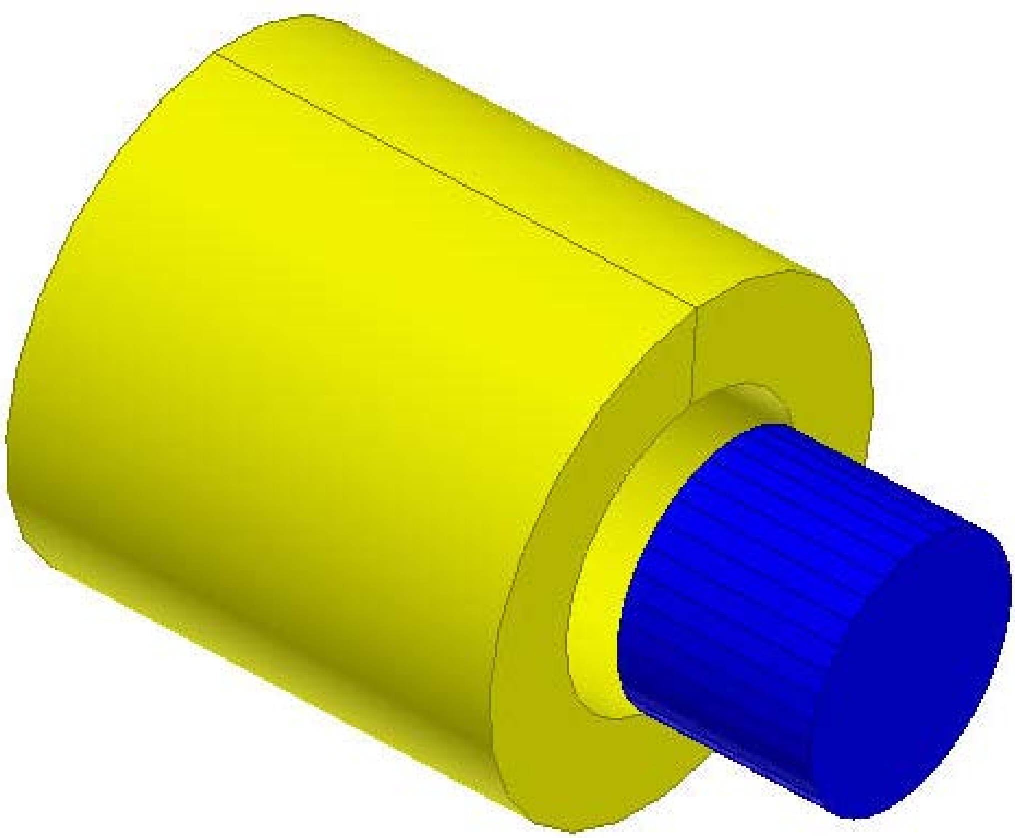
Results

- In the Project Manager Window expand Results and double-click on XY Plot 1 and view the results of Sensor Flux Density plot versus time. This plot should be very similar to the one already created for the full geometry

Example (Transient) – Translational Motion

Translational Motion using the Transient Solver

- Transient Solver in Maxwell allows to be combined with motion. This motion can be translational, rotational cylindrical or rotational non-cylindrical (relay-type motion). This example shows on a simple geometry how to set-up a translational motion project using the Transient Solver of Maxwell.
- The geometry consists of a cylindrical NdFeB magnet with uniform distribution of magnetization in the axial direction (z-direction). The magnet moves through a copper hollow cylindrical structure (coil) with a constant speed of 1 mm per second. As the magnet moves the eddy currents are induced and flow in the copper structure. The goal is to model a time-dependent magnetic field created by moving magnet and determine the eddy currents in the copper structure.



Example (Transient) – Translational Motion

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Regular Polyhedron, Rectangle**
    - Boolean Operations: **Split**
    - Sweep Operations: **Around Axis**
    - Model: **Band for translational motion**
  - Boundaries/Excitations
    - **Natural outer boundary**
  - Material properties
    - **Edit material (NdFeB axially magnetized magnet with uniform distribution of magnetization)**
  - Analysis
    - **Transient**
  - Results
    - **Fields Calculator**
    - **Rectangular Plot**
  - Field Overlays:
    - **Current Density Vector Plots**
    - **Current Density Vector Animation**

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

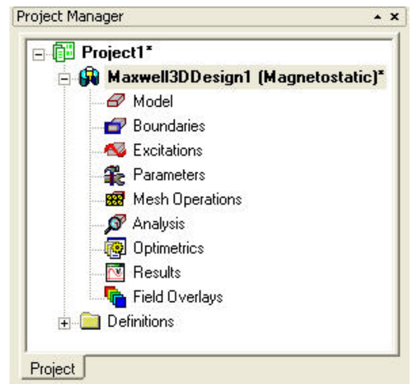
Setting Tool Options

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

Example (Transient) – Translational Motion

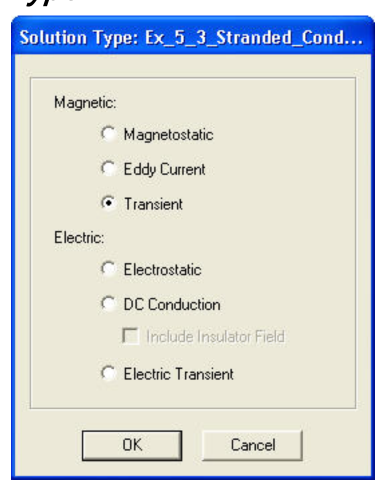
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item *File > New*.
  - Select the menu item *Project > Insert Maxwell 3D Design*, or click on the icon



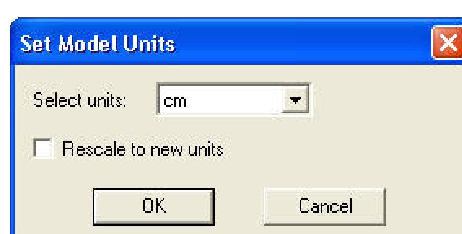
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 3D > Solution Type*
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button



Set Model Units

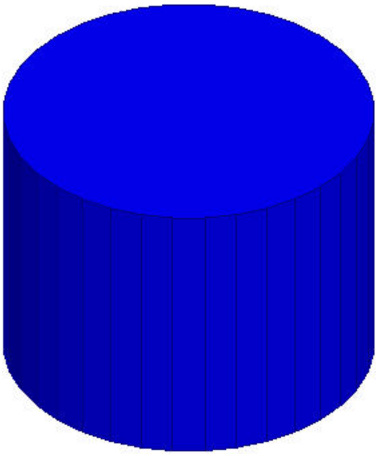
- **To Set the units:**
  - Select the menu item *Modeler > Units*
  - Set Model Units:
    1. Select Units: **cm**
    2. Click the **OK** button



Example (Transient) – Translational Motion

Create Magnet

- **To Create Magnet**
  - Select the menu item *Draw > Regular Polyhedron*
    1. Using Coordinate entry field, enter the center of base
       - **X = 0, Y = 0, Z = -1,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - **dX = 0, dY = 1.3, dZ = 2,** Press the **Enter** key
    3. Number of Segments: **36**
    4. Press **OK**
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Magnet**
    2. Change the color of the object to **Blue**



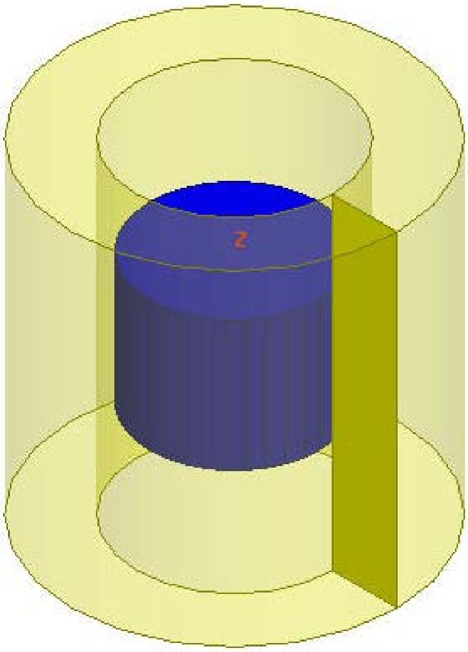
Create Coil

- **To Create Rectangle**
  - Select the menu item *Modeler > Grid Plane > YZ*
  - Select the menu item *Draw > Rectangle*
    1. Using Coordinate entry field, enter the position of rectangle
       - **X = 0, Y = 1.5, Z = -2.5,** Press the **Enter** key
    2. Using Coordinate entry field, enter the opposite corner of rectangle
       - **dX = 0, dY = 1, dZ = 5,** Press the **Enter** key
- **To Change Attributes**
  - Select the resulting sheet from the history tree and goto Properties window
    1. Change the name of the sheet to **Coil_Terminal**
    2. Change the color of the sheet to **Yellow**

Example (Transient) – Translational Motion

- **To Create a Copy**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Edit > Copy***
  - Select the menu item ***Edit > Paste***
  - A new sheet **Coil_Terminal1** is created
- **To Sweep Coil_Terminal1**
  - Select the sheet **Coil_Terminal1** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Sweep Axis: **Z**
    2. Angle of Sweep: **360 deg**
    3. Press **OK**

- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Coil**

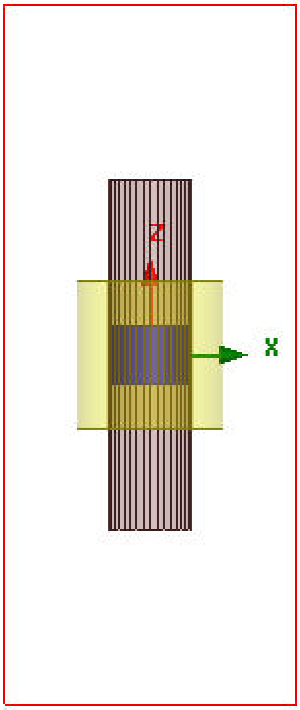


Create Band

- In Transient Solver, Band is an object that contains all moving objects. In other words it is a way how to tell Maxwell which objects are steady and which are moving. In this example there is one moving (translating) object: **Magnet**. For translational type of motion Band cannot be a true surface object (cylinder). It must be a faceted object, such as Regular Polyhedron:
- **To Create Band**
  - Select the menu item ***Modeler > Grid Plane > XY***
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using Coordinate entry field, enter the center of base
       - **X = 0, Y = 0, Z = -6,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - **dX = 0, dY = 1.4, dZ = 12,** Press the **Enter** key
    3. Number of Segments: **36**
    4. Press **OK**
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Band**
    2. Change the color of the object to **Brown**
    3. Change the transparency to **0.8**

Create Region

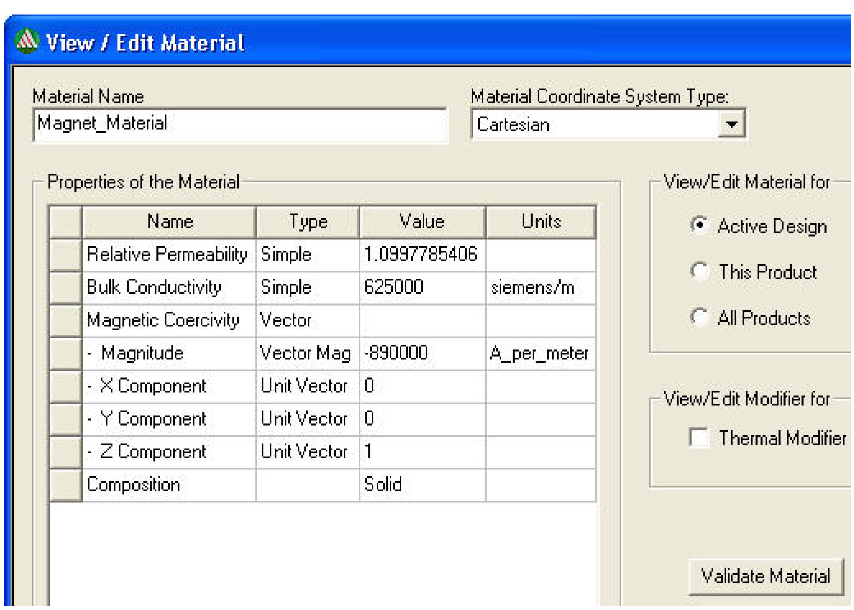
- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **50**
    4. Press **OK**



Assign Material: Magnet

To Assign Magnet Material

- Select the object **Magnet** from the history tree, right click and select **Assign Material**
- In Select Definition window,
  1. Type **NdFe35** in the **Search by Name** field
  2. Select option **Clone Material**
- In View/Edit Material window,
  1. Name: **Magnet_Material**
  2. Material Coordinate System Type: **Cartesian**
- **Note:** The direction of magnetization is determined by Unit Vector. For the Cartesian coordinate system the components of Unit Vector are ***X***, ***Y*** and ***Z***. The values of ***X***, ***Y*** and ***Z*** are 1, 0, 0 by default. For this example they have to be changed, because the magnet is magnetized in positive **z**-direction:
  3. Magnetic Coercivity:
     - Magnitude: **-890000**
     - X Component: **0**
     - Y Component: **0**
     - Z Component: **1**
  4. Select **Validate Material**
  5. Press **OK**

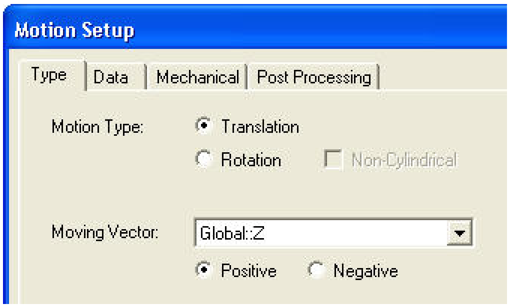


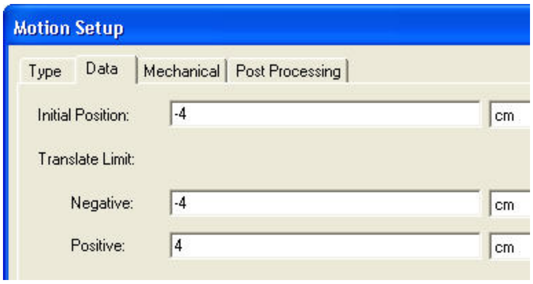
Example (Transient) – Translational Motion

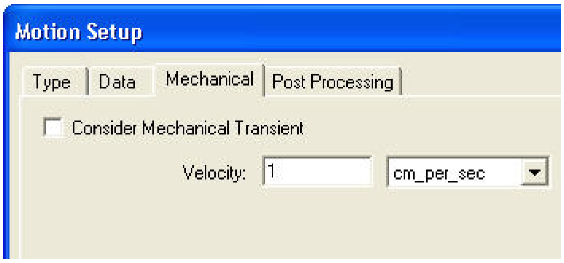
- **To Assign Material for Coil**
  - Select the object **Coil** from the history tree, right click and select **Assign Material**
  - In Select Definition window,
    1. Type **copper** in the **Search by Name** field
    2. Select **OK**

Assign Motion

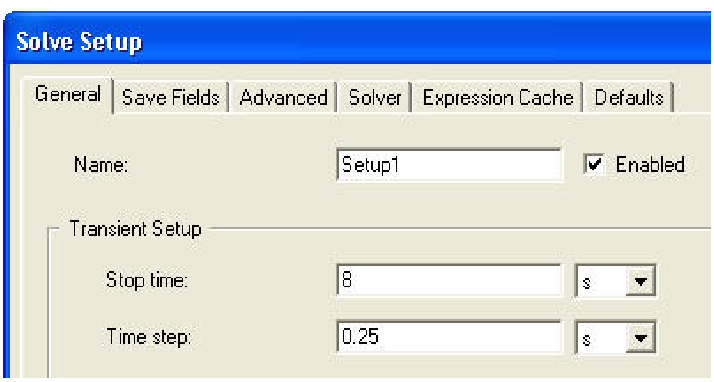
- **To Assign band Object**
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Model > Motion Setup > Assign Band***
  - In Motion Setup window,
    1. **Type** tab
       - Motion Type: **Translation**
       - Moving Vector: **Global:Z**
       - **Positive**
    2. **Data** tab
       - Initial Position: **-4 cm**
       - Translation Limit
         1. Negative: **-4 cm**
         2. Positive: **4 cm**
    3. **Mechanical** tab
       - Velocity: **1 cm_per_sec**
    4. Press **OK**

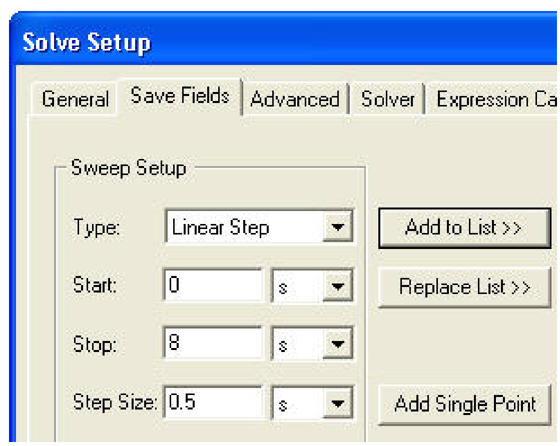






- ## Analysis Setup
  - **To Create Analysis Setup**
    - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
    - In Solve Setup Window,
      1. **General** tab
         - Stop time: **8 s**
         - Time step: **0.25 s**
      2. **Save Fields** tab
         - Type: **Linear Step**
         - Start: **0 s**
         - Stop: **8 s**
         - Step Size: **0.5 s**
         - Select the button **Add to List >>**
      3. Press **OK**

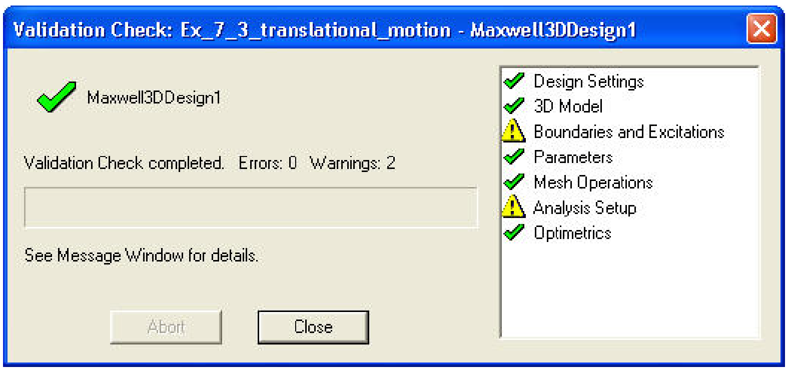




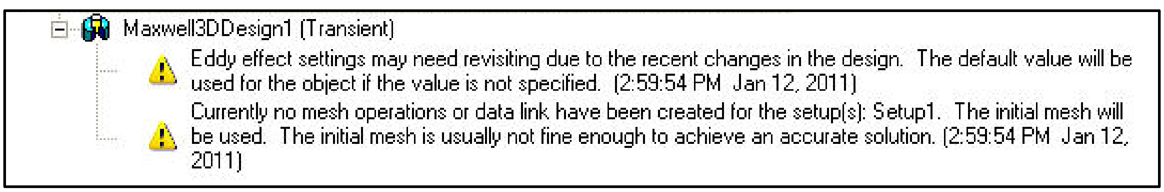
- ## Save
  - **To Save File**
    - Select the menu item ***File > Save***
    - Save the file with the name **“Ex_7_3_Translational_Motion.mxwl”**

Validity Check

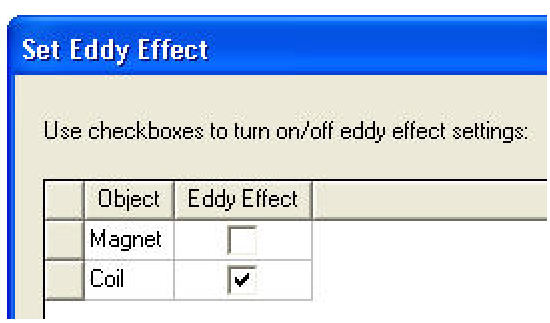
- **To Check Validation of the case**
  - Select the menu item ***Maxwell 3D > Validation Check***



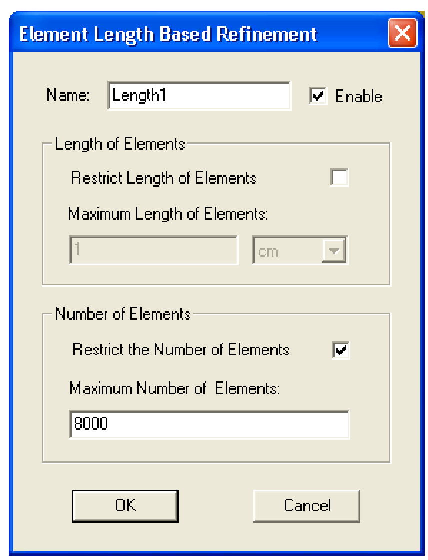
  - Two warning messages are shown in Validation Check
  - Details about warning can be sheen in Message Manager window



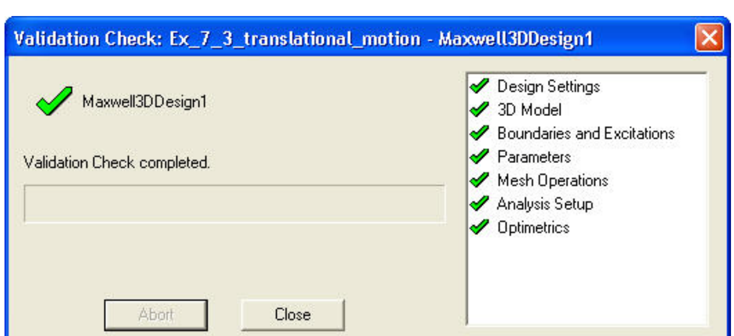
  - **Warning 1:** Eddy effect settings may need revisiting due to the recent changes in the design.
    1. In this example we are particularly interested to compute eddy currents induced in the solid copper coil. We have to make sure that the objects **Coil** is assigned for eddy current computation
    2. Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effects***
    3. In Set Eddy Effects window,
       - Coil
         - Eddy Effects: ☑ **Checked**
       - Press **OK**



- **Warning 2:** Currently no mesh operations or data link have been created for the setup(s): Setup1. The initial mesh will be used. The initial mesh is usually not fine enough to achieve an accurate solution
  1. For demonstration purposes in this particular example we are interested to see the induced eddy currents at the cross section of the coil. Let us therefore refine the cross section by assigning appropriate mesh operations
  2. Select the object **Coil** from the history tree
  3. Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  4. In Element Length Based Refinement window,
     - Name: **Length1**
     - Restrict Length of Elements: ☐ **Unchecked**
     - Restrict the Number of Elements: ☑ **Checked**
     - Maximum Number of Elements: **8000**
     - Press **OK**



- Select the menu item ***Maxwell 3D > Validation Check*** for verification

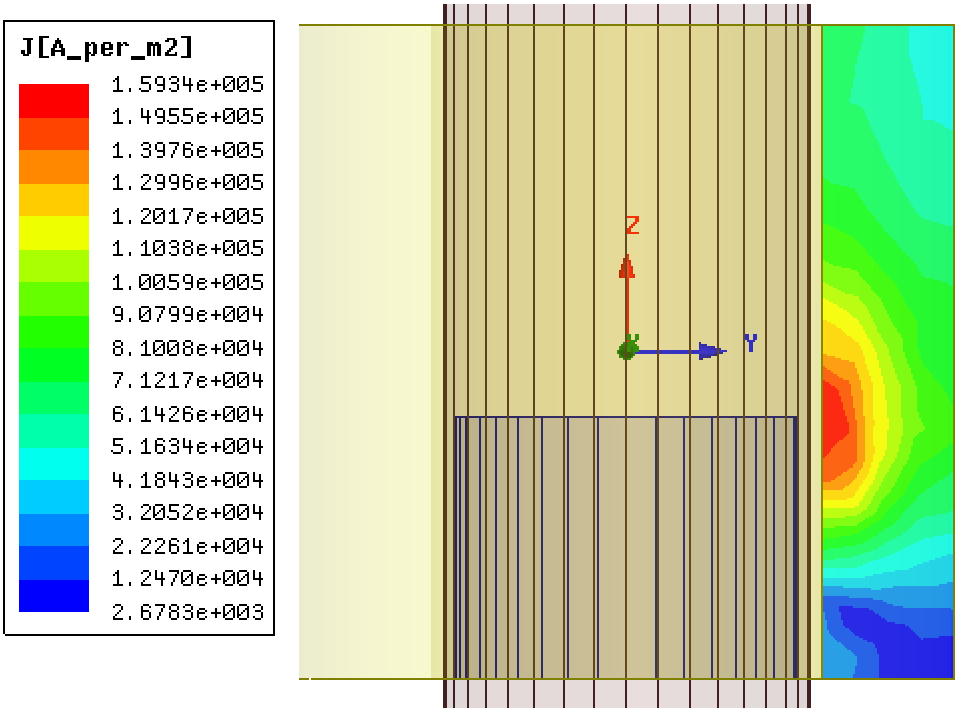


Analyze

- **To Run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Plot Current Density Magnitude on Coil Terminal at time 2.5 s

- **To Set Time= 2.5 s**
  - Select the menu item ***View > Set Solution Context***
  - In Set View Context window,
    1. Set Time to **2.5s**
    2. Press **OK**
- **To Plot Current Density on Coil_Terminal**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > J > Mag_J***
  - In Create Field Plot window,
    1. Press **Done**

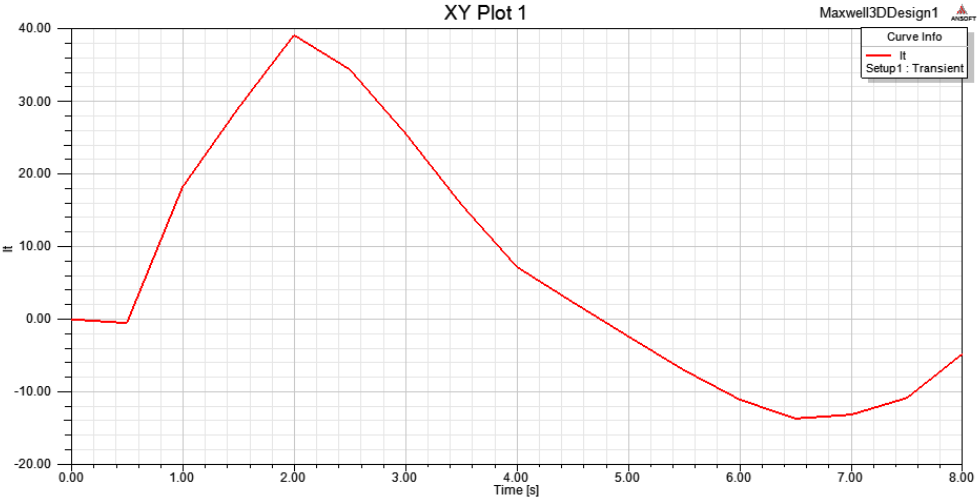


Example (Transient) – Translational Motion

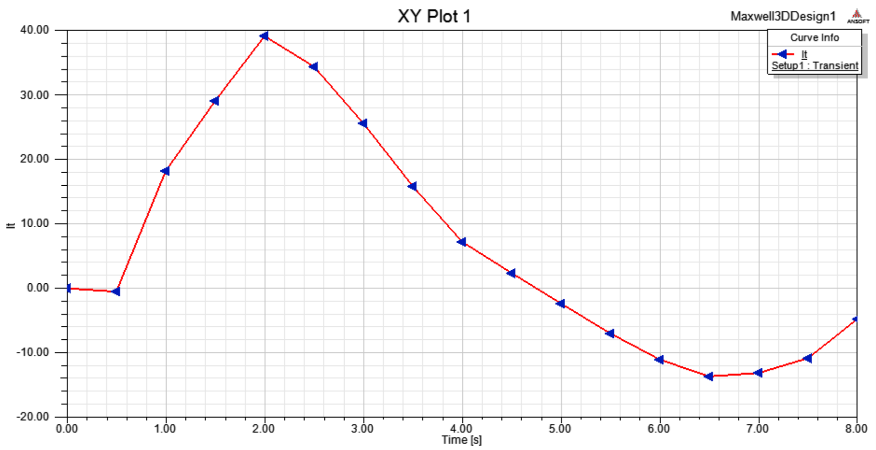
Plot Induced Current through the coil as a function of time

- **To Define Parameter for Current Through Terminal (It)**
  - Select the menu item *Maxwell 3D > Fields > Calculator*
  - In Fields Calculator window,
    1. Select **Input > Quantity > J**
    2. Select **Input > Geometry**
       - Select the radio button to **Surface**
       - Select the surface **Coil_Terminal** from the list
       - Press **OK**
    3. Select **Vector > Normal**
    4. Select **Scalar > Integrate**
    5. Select **Add**
    6. Specify the name as **It**
    7. Press **OK**
    8. Press **Done** to close calculator

- **To Plot It Vs Time**
  - Select the menu item *Maxwell 3D > Results > Create Fields Report > Rectangular Plot*
  - In Report window,
    - Category: **Calculator Expressions**
    - Quantity: **It**
    - Press **New Report**

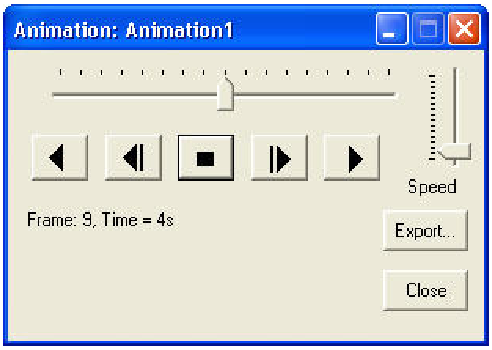


- **To Modify Plot Attributes**
  - Select the trace shown on the screen and double click on it to edit its properties
  - In Properties window,
    - **Attributes** tab
      - Show Symbol: ☑ **Checked**
      - Symbol Frequency: **1** (To show symbol for every data point)
    - Press **OK**



Create an Animation of Current Density Magnitude Plot

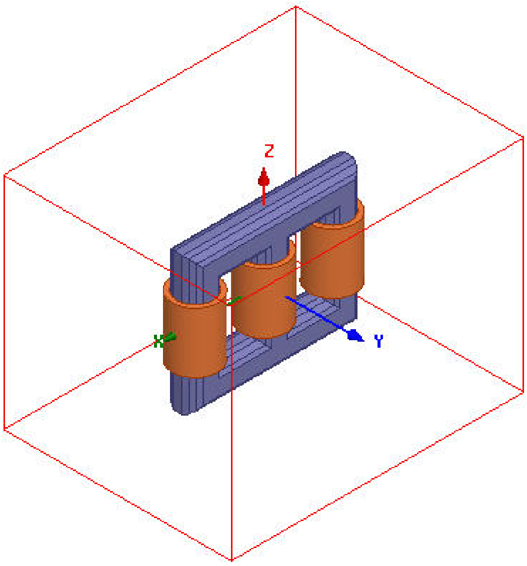
- **To Animate a Field Plot**
  - Expand the Project Manager tree to view **Field Overlays > J > Mag_J1**
  - Right click on the plot **Mag_J1** and select **Animate**
  - In Setup Animation window,
    - Press **Shift** key and select all time points for which plot needs to be animated and press **OK**
  - To start and stop the animation, to control the speed of the animation and to export the animation use the buttons on the Animation Panel:



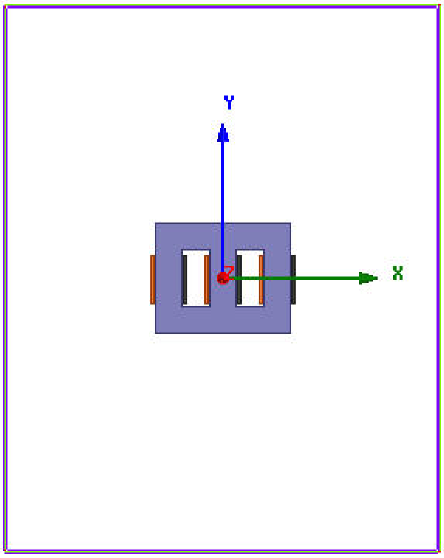
Transformer Core Loss Calculation in Maxwell 2D and 3D

- This example analyzes cores losses for a 3ph power transformer having a laminated steel core using Maxwell 2D and 3D. The transformer is rated 115-13.8kV, 60Hz and 30MVA. The tested power losses are 23,710W. It is important to realize that a finite element model cannot consider all of the physical and manufacturing core loss effects in a laminated core. These effects include: mechanical stress on laminations, edge burr losses, step gap fringing flux, circulating currents, variations in sheet loss values, … to name just a few. Because of this the simulated core losses can be significantly different than the tested core losses.
- This example will go through all steps to create the 2D and 3D models based on a customer supplied base model. For core losses, only a single magnetizing winding needs to be considered. Core material will be characterized for nonlinear BH and core loss characteristics. An exponentially increasing voltage source will be applied in order to eliminate inrush currents and the need for an unreasonably long simulation time (of days or weeks). Finally, the core loss will be averaged over time and the core flux density will be viewed in an animated plot.
- This example will be solved in two parts using the 2D Transient and 3D Transient solvers. The model consists of a magnetic core and low voltage winding on each core leg.

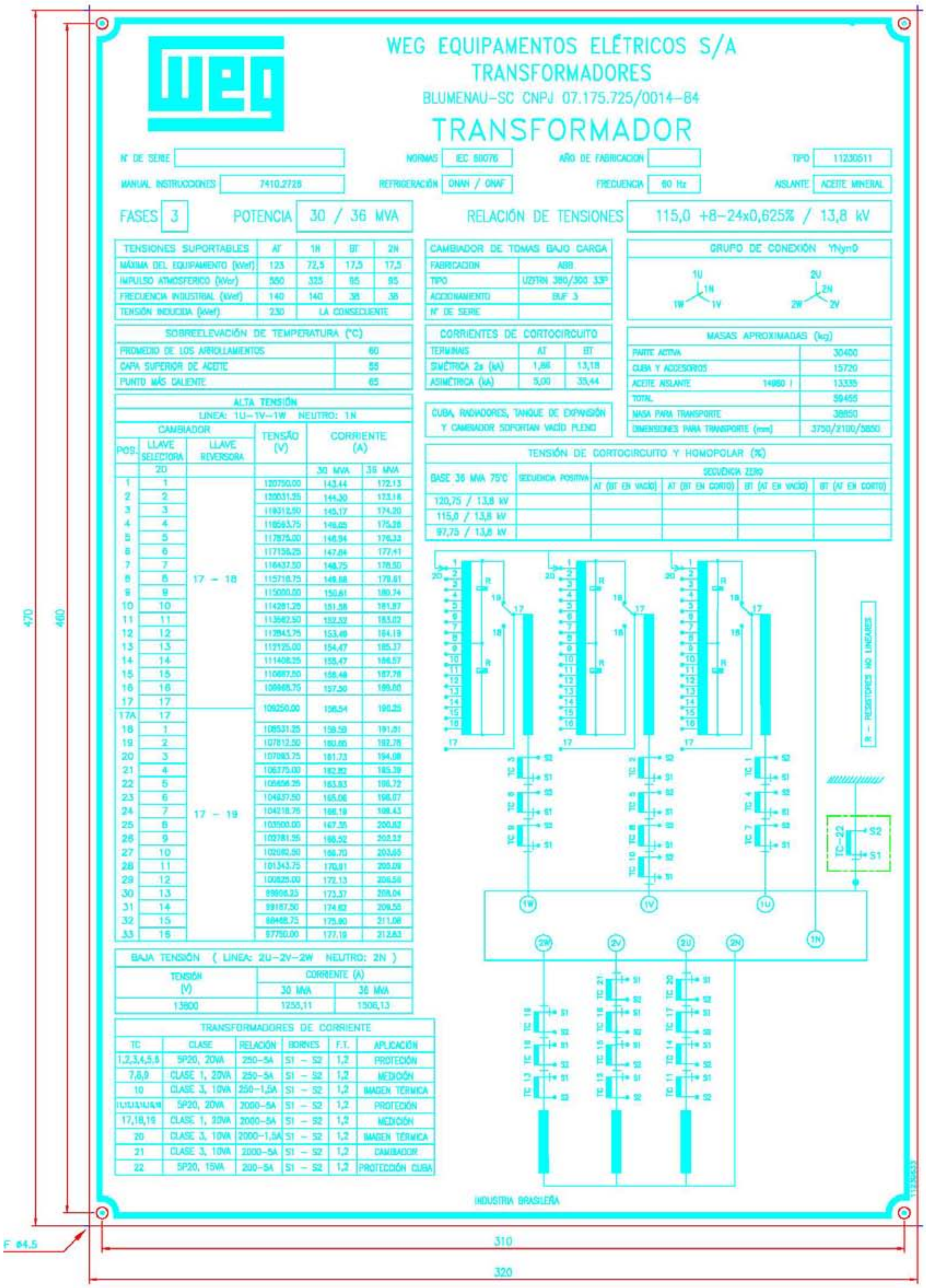
3D Model



2D Model



Example (2D/3D Transient) – Core Loss



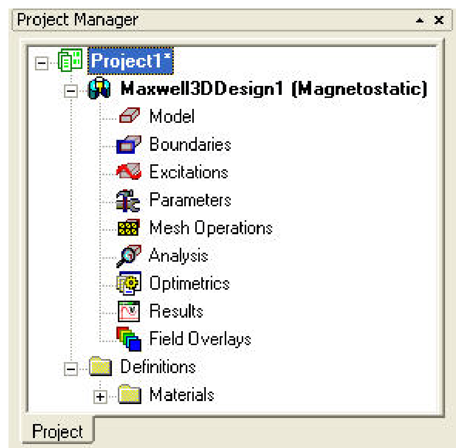
Example (2D/3D Transient) – Core Loss

Launch Maxwell

- **To access Maxwell**
  - Click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 15.**

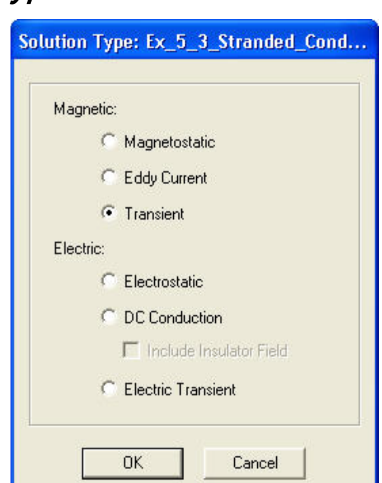
Opening a New Project

- **To Open a New Project**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To Set Solution Type**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button

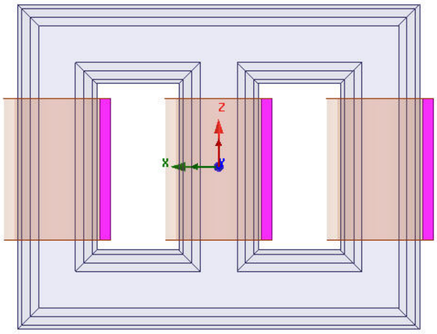


Prepare Geometry

- **To Import Geometry**
  - Select the menu item ***Modeler > Import***
  - Locate the parasolid file **“Ex_7_4_Core_Loss.x_t”** and **Open** it.
  - The geometry is of a transformer with core simplified in order to reduce the complexity. Users can bring the geometries directly and do simplification inside Maxwell.
- **Change Attributes**
  - Press **Ctrl** and select the objects **LV_A, LV_B** and **LV_C** and goto their properties window,
    1. Change the color of the objects to **Orange**
    2. Change the transparency of the objects to **0**
  - Select the object **Core** from the history tree and goto Properties window,
    1. Change the transparency of the object to **0**

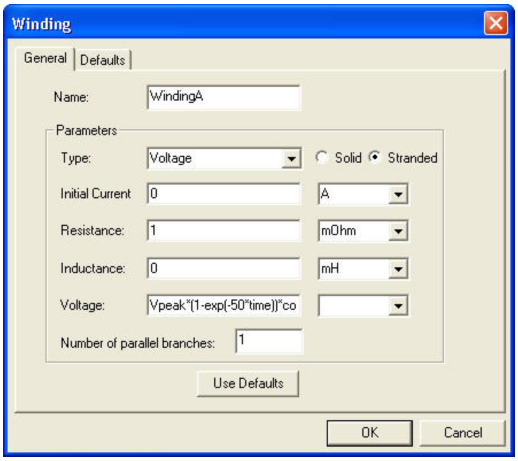
Specify Excitations

- **To Create Coil Terminals**
  - Press **Ctrl** and select the objects **LV_A, LV_B** and **LV_C**
  - Select the menu item ***Modeler > Surface Section***
  - In Section window,
    1. Section Plane: Select **XZ**
    2. Press **OK**
  - Rename the resulting sections to **SectionA, SectionB** and **SectionC** respectively
  - Select the sheets **SectionA, SectionB** and **SectionC** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - Delete the sheets **SectionA_Separate1, SectionB_Separate1** and **SectionC_Separate1**



Example (2D/3D Transient) – Core Loss

- **Assign Excitations**
  - Press **Ctrl** and select the sheets **SectionA, SectionB** and **SectionC** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
  - In Coil Terminal Excitation window,
    1. Base Name: **term_A**
    2. Number of Conductors: **76**
    3. Press **OK**
  - This will create three excitations corresponding to each section. Change their names as below:
    1. Rename the excitation corresponding to SectionA as **term_A**
    2. Rename the excitation corresponding to SectionB as **term_B**
    3. Rename the excitation corresponding to SectionC as **term_C**
- **Create Windings**
  - Select the menu item ***Maxwell 3D > Excitations > Add Winding***
  - In Winding window,
    1. Name: **WindingA**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **1 mOhm**
    6. Inductance: **0 mH** (Since this is calculated by solver)
    7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time)**
    8. Press **OK**
  - In Add Variable window,
    1. Unit Type: **Voltage**
    2. Unit: **V**
    3. Value: 13800√2 / √3 = **11268**
    4. Press **OK**

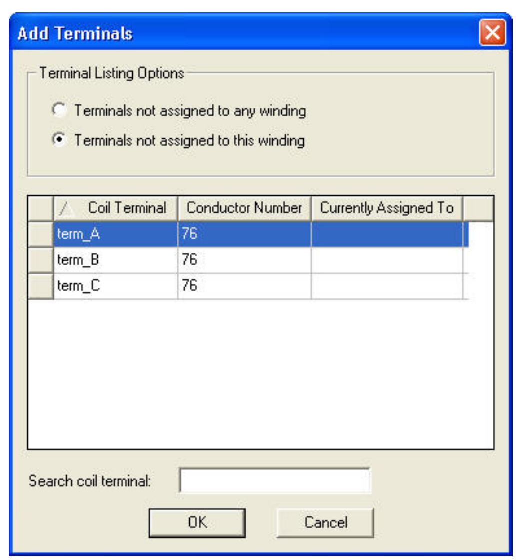


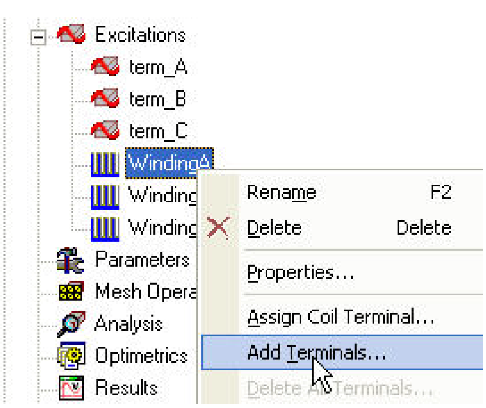
- **Note:** This is an exponentially increasing (in several cycles) sinusoidal 60Hz waveform with peak magnitude of 11,268V..

Example (2D/3D Transient) - Core Loss

- In Similar way add two more windings
- WindingB
  1. Name: **WindingB**
  2. Type: **Voltage**
  3. Stranded: ☑ **Checked**
  4. Initial Current: **0 A**
  5. Resistance: **1 mOhm**
  6. Inductance: **0 mH** (Since this is calculated by solver)
  7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time+(2/3*pi))**
- WindingC
  1. Name: **WindingC**
  2. Type: **Voltage**
  3. Stranded: ☑ **Checked**
  4. Initial Current: **0 A**
  5. Resistance: **1 mOhm**
  6. Inductance: **0 mH** (Since this is calculated by solver)
  7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time+(4/3*pi))**

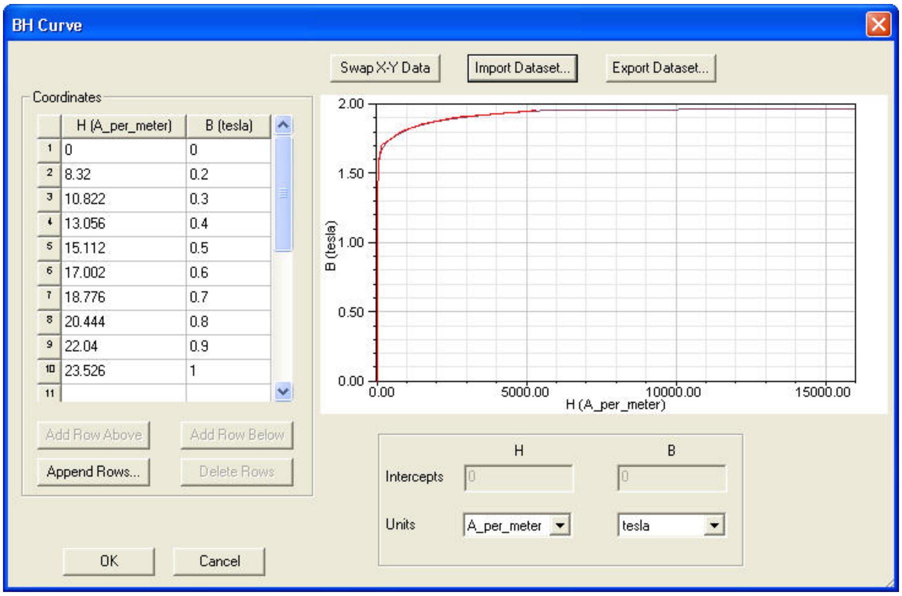
- **Add Terminals to Windings**
  - Expand the Project Manager tree to view Excitations
  - Right click on **WindingA** and select **Add Terminals**
  - In Add Terminals window,
    - Select **term_A**
    - Press **OK**
  - In Similar way add **term_B** to **WindingB**
  - Add **term_C** to **WindingC**





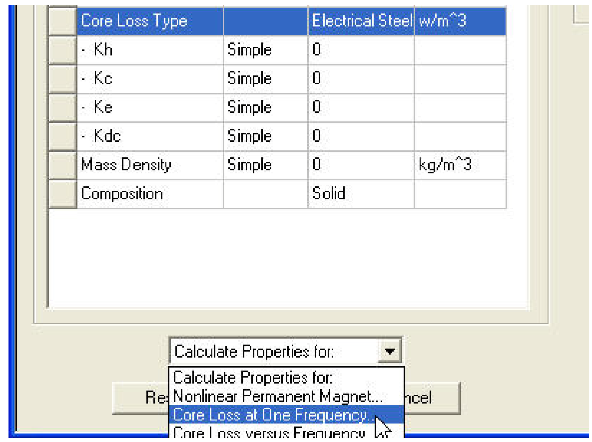
Assign Materials

- **To Assign Materials to Coils**
  - Press **Ctrl** and select the objects **LV_A, LV_B** and **LV_C ,** right click and select **Assign Material**
  - In Select Definition window,
    1. Type **copper** in Search by Name field
    2. Press **OK** to assign material
- **To Assign Material to Core**
  - Select the object **Core** from the history tree, right click and select **Assign Material**
  - In Select Definition window, select the button **Add Material**
  - In View/Edit Material window,
    - Material Name: **M125_027**
    - Relative Permeability:
      - Set the type to **Nonlinear**
      - Select the button **BH Curve** from value field
      - In BH Curve window,
        - Select the button **Import Dataset**
        - Set the File Type to ***.Tab**
        - Locate the file **Ex_7_4_core_loss_B_H.tab** and **Open** it
        - Press **OK** to close BH Curve window

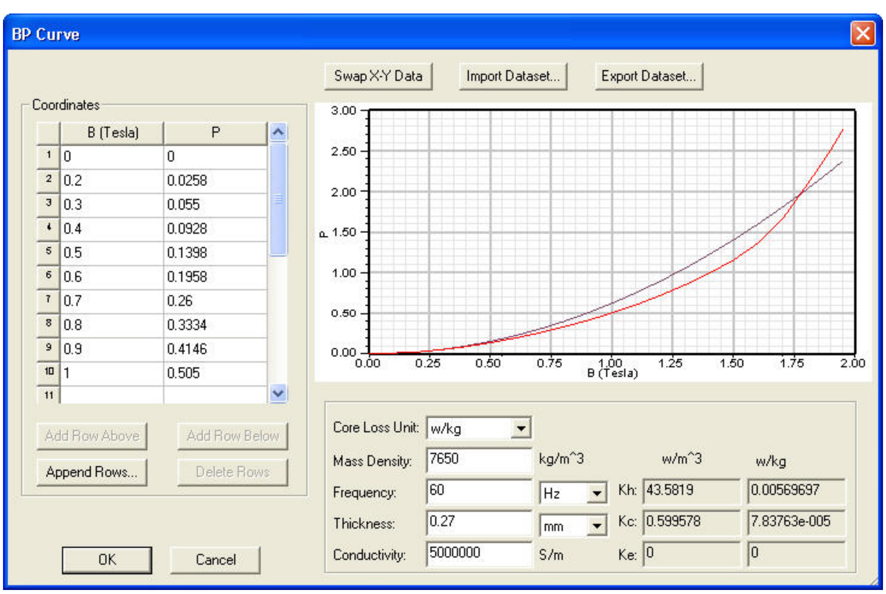


Example (2D/3D Transient) - Core Loss

- Return to View/Edit Material window,
  - Core Loss Type: Set to **Electrical Steel**
  - Set the tab at the bottom of window Calculate Properties for to **Core Loss at one Frequency**

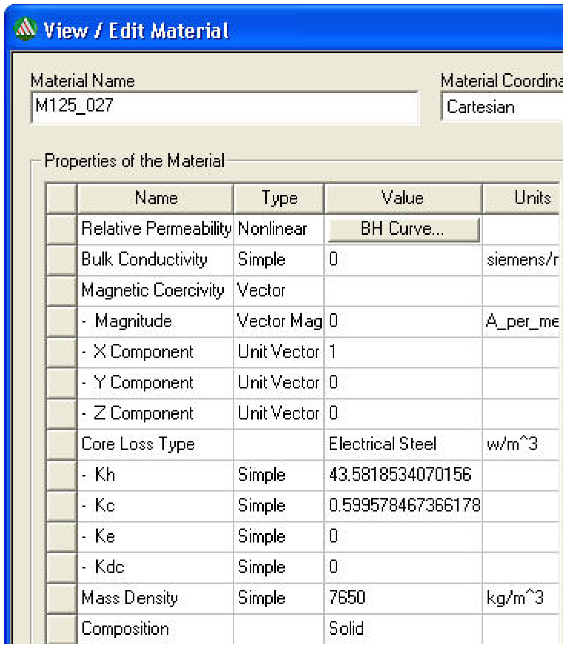


- In BP Curve window,
  - Select the button **Import Dataset**
  - Set the File Type to ***.Tab**
  - Locate the file **Ex_7_4_core_loss_B_loss.tab** and **Open** it
  - Core Loss Unit: **w/kg**
  - Mass Density: **7650 kg/m^3**
  - Frequency: **60 Hz**
  - Thickness: **0.27 mm**
  - Conductivity: **5000000 S/m**
  - Press **OK** to close BP Curve window



Example (2D/3D Transient) – Core Loss

- Note that the core loss coefficients are calculated automatically.



Name	Type	Value	Units
Relative Permeability	Nonlinear	BH Curve...	
Bulk Conductivity	Simple	0	siemens/r
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	0	A_per_me
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Core Loss Type		Electrical Steel	w/m^3
- Kh	Simple	43.5818534070156	
- Kc	Simple	0.599578467366178	
- Ke	Simple	0	
- Kdc	Simple	0	
Mass Density	Simple	7650	kg/m^3
Composition		Solid	

- Press **OK** to create the new material
- Press **OK** to close Select Definition window

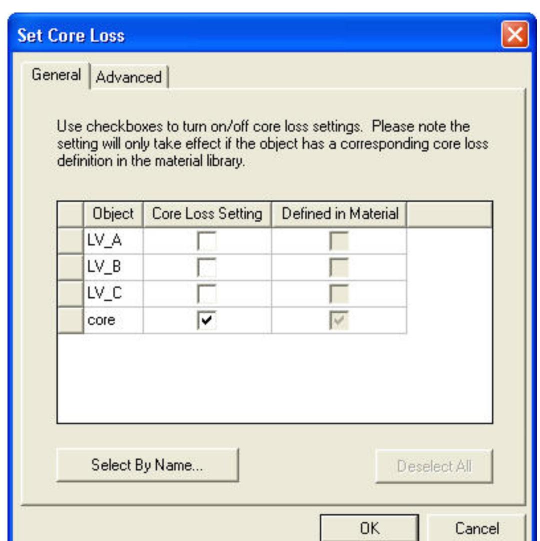
Assign Mesh Operations

- In the transient solvers, there is no automatic adaptive meshing. Therefore, the user must either link the mesh from an identical model solved using the magnetostatic and eddy current solvers, or alternatively a manual mesh must be created. In this example, a mesh is created manually using “inside selection” to create elements throughout the volume of the objects.
- **To Assign Mesh Operations for Core**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    - Name: **Length_Core**
    - Restrict Length of Elements: ☐ **Unchecked**
    - Restrict the Number of Elements: ☑ **Checked**
    - Maximum Number of Elements: **10000**
    - Press **OK**

- **To Assign Mesh Operations for Coils**
  - Press **Ctrl** and select the objects **LV_A, LV_B** and **LV_C** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    - Name: **Length_Coils**
    - Restrict Length of Elements: ☐ **Unchecked**
    - Restrict the Number of Elements: ☑ **Checked**
    - Maximum Number of Elements: **10000**
    - Press **OK**

Set Core Loss Calculations

- **To Set Core Loss calculations for Core**
  - Select the menu item ***Maxwell 3D > Excitations > Set Core Loss***
  - In Set Core Loss window,
    - Core:
      - Core Loss Settings: ☑ **Checked**
    - Press **OK**
  - **Note:** Once the core loss properties are defined in material definition, a tick mark will appear in the column Defined in Material indicating core loss coefficients are already specified

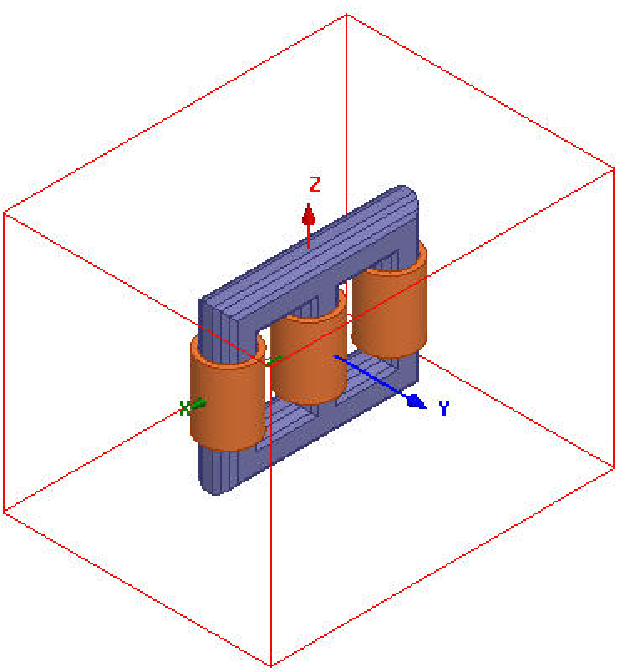


Set Eddy Effects

- Since winding is single object representing many strands and core is single object representing many laminations, eddy effect must be turned off them.
- **To Turn off Eddy Effects in Objects**
  - Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effects***
  - In Set Eddy Effects window,
    - Ensure Eddy Effects are **Unchecked** for all objects
    - Press **OK**

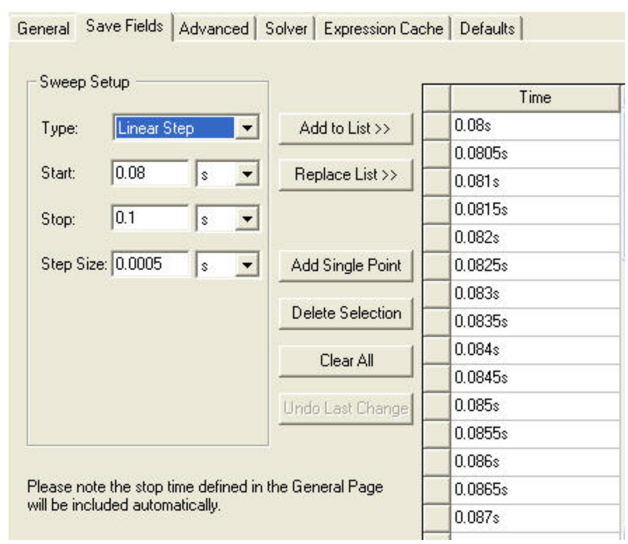
Create Simulation Region

- **To Create Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    - Padding Data: **Pad individual directions**
      - +/- X = **30**
      - +/- Y = **200**
      - +/- Z = **30**
  - **Note:** This small padding % is acceptable as fields are completely concentrated inside the magnetic core and there is little or no fringing



Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup Window,
    - **General** tab
      - Stop time: **0.1s**
      - Time step: **0.0005 s**
    - **Save Fields** tab
      - Type: **Linear Step**
      - Start: **0.08 s**
      - Stop: **0.1 s**
      - Step Size: **0.0005 s**
      - Select the button **Add to List >>**
    - **Solver** tab
      - Nonlinear Residuals: **1e-6** (To Provide accurate convergence for BH Curve
    - Press **OK**



Save

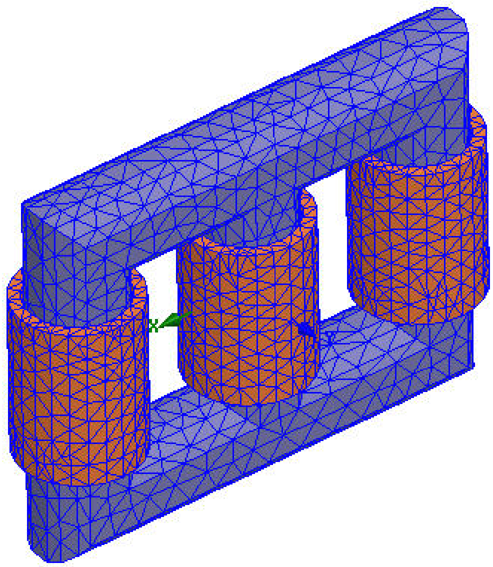
- **To Save File**
  - Select the menu item ***File > Save***
  - Save the file with the name **"Ex_7_4_Core_Loss.mxwl"**

Analyze

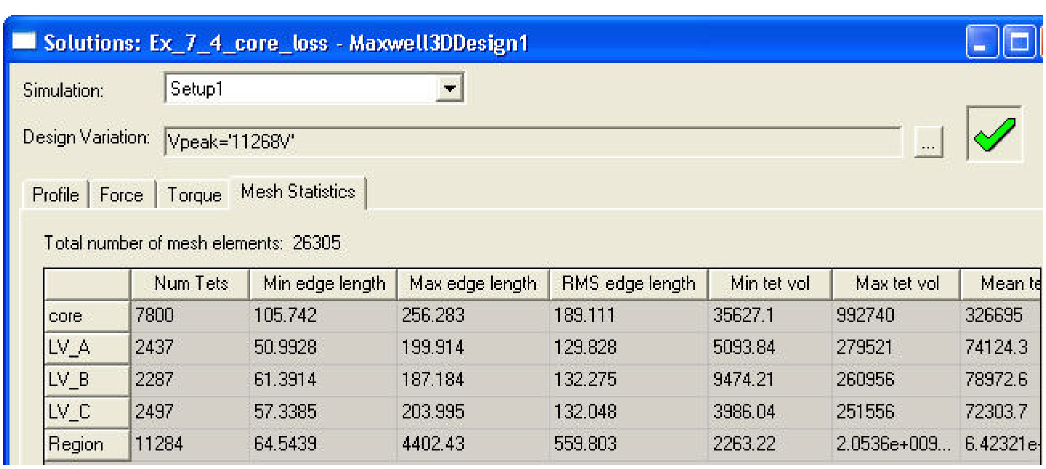
- **To Run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Mesh Information

- **To Plot Mesh on Core and Coils**
  - Select the object **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***
  - Select the menu item ***Edit > Select All Visible***
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    - Press **Done**



- **To View Mesh Information**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
  - In Solutions window,
    - Select the tab **Mesh Statistics** to view mesh information



Solutions: Ex_7_4_core_loss - Maxwell3DDesign1

Simulation: Setup1

Design Variation: Vpeak='11268V'

Profile | Force | Torque | Mesh Statistics

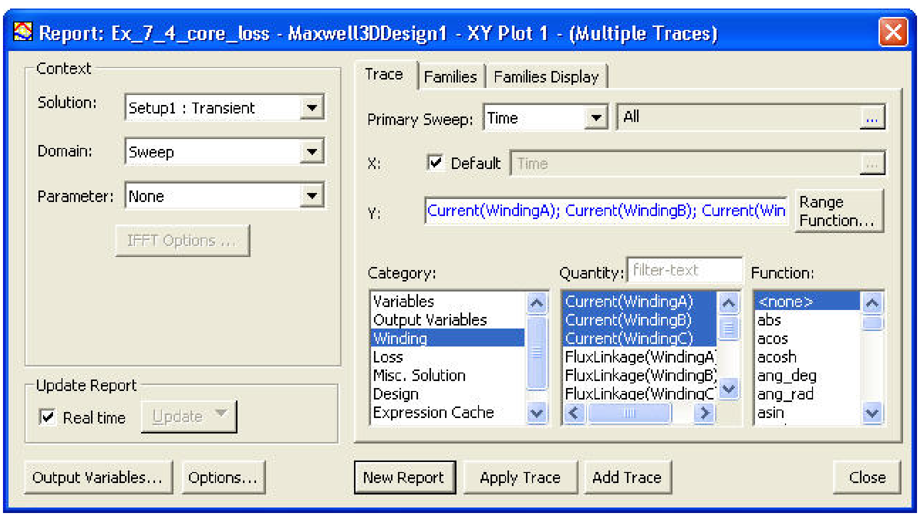
Total number of mesh elements: 26305

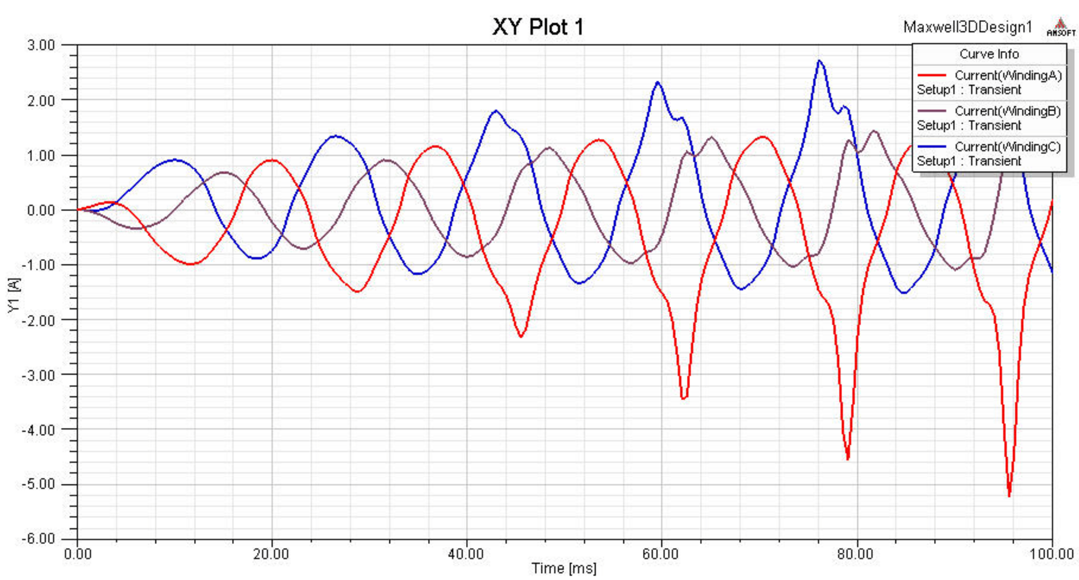
	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean te
core	7800	105.742	256.283	189.111	35627.1	992740	326695
LV_A	2437	50.9928	199.914	129.828	5093.84	279521	74124.3
LV_B	2287	61.3914	187.184	132.275	9474.21	260956	78972.6
LV_C	2497	57.3385	203.995	132.048	3986.04	251556	72303.7
Region	11284	64.5439	4402.43	559.803	2263.22	2.0536e+009...	6.42321e

Create Reports

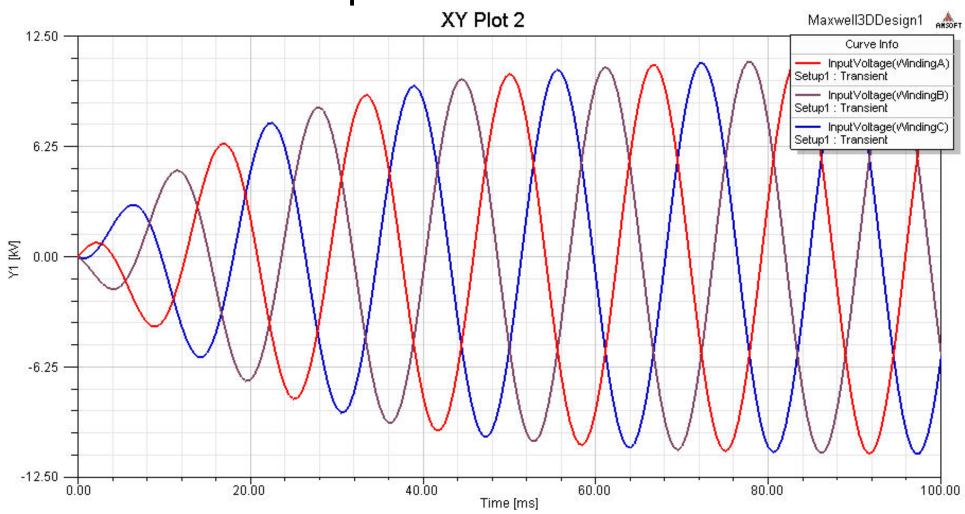
Plot Winding Currents Vs Time

- Select the menu item ***Maxwell 3D > Results > Create Transient Report > Rectangular Plot***
- In Report window,
  - Category: **Winding**
  - Quantity: Press **Ctrl** and select **Current(WindingA), Current(WindingB)** and **Current(WindingC)**
  - Select **New Report**
- **Note:** Do not close Report window as we will create more plots using same window

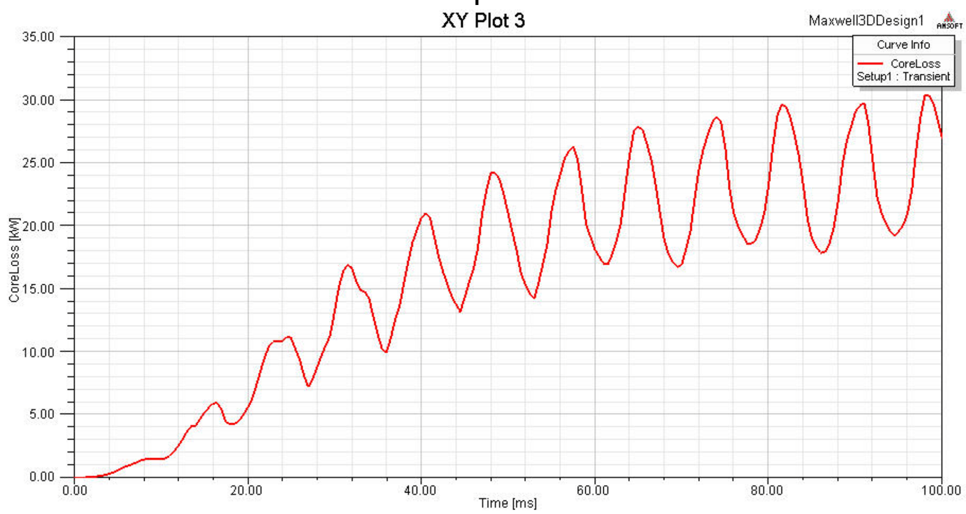




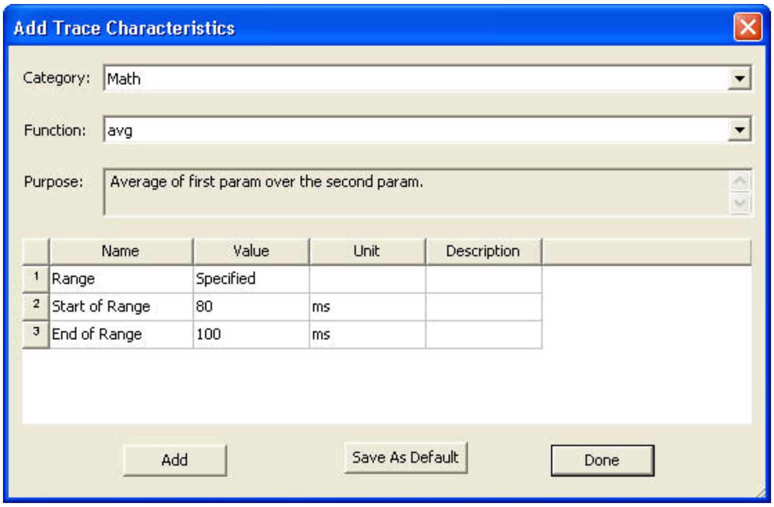
- **Plot Input Voltages Vs Time**
  - In Report window,
    - Quantity
      - Deselect the Current quantities already selected
      - Press **Ctrl** and select **InputVoltage(WindingA), InputVoltage(WindingB)** and **InputVoltage(WindingC)**
    - Select **New Report**

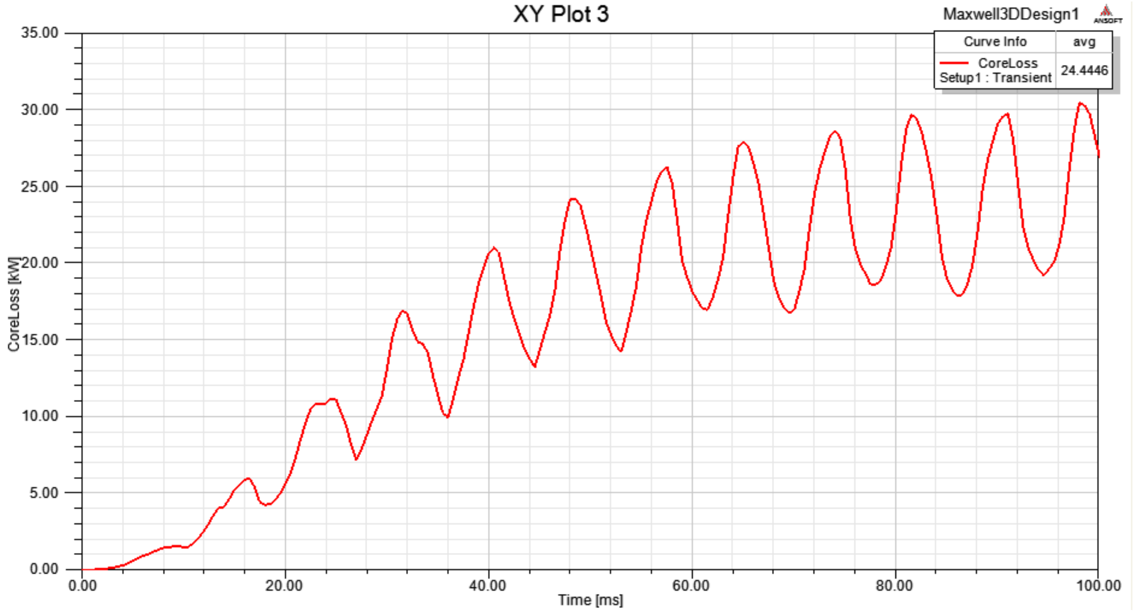


- **Plot Cores Loss vs Time**
  - In Report window,
    - Category: Change to **Loss**
    - Quantity: Select **CoreLoss**
    - Select **New Report**
    - Press **Close** to close report window



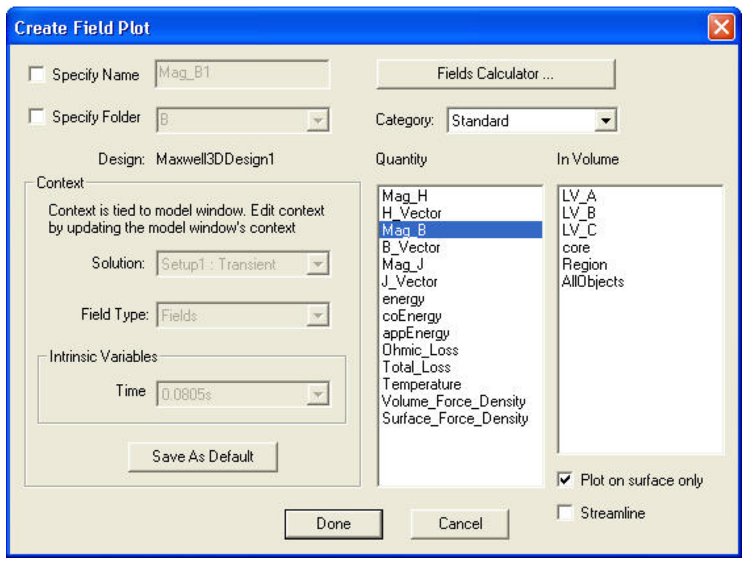
- **Calculate Avg Losses over a Time Range**
  - In XY Plot Corresponding to CoreLoss, right click on the Legend and select ***Trace Characteristics > Add***
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **Avg**
    - Change the Range from Full to **Specified**
    - Start of Range: **80 ms**
    - End of Range: **100 ms**
    - Select **Add** and **Done**

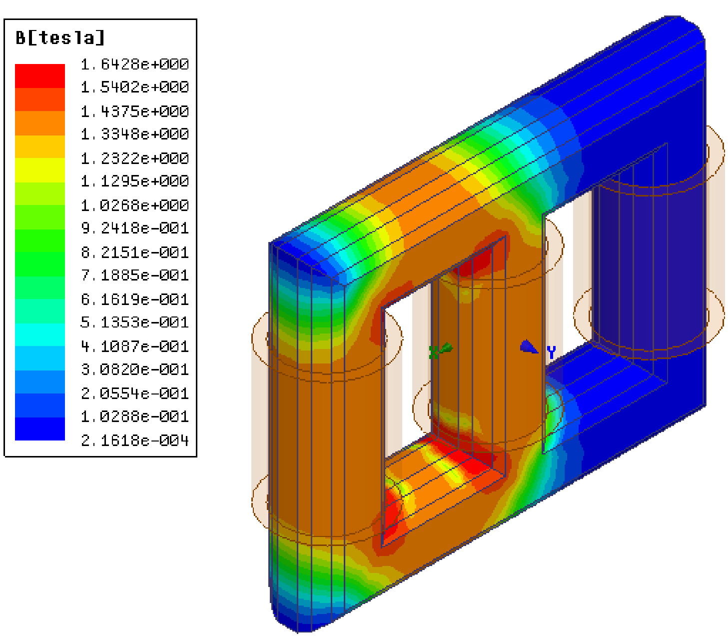




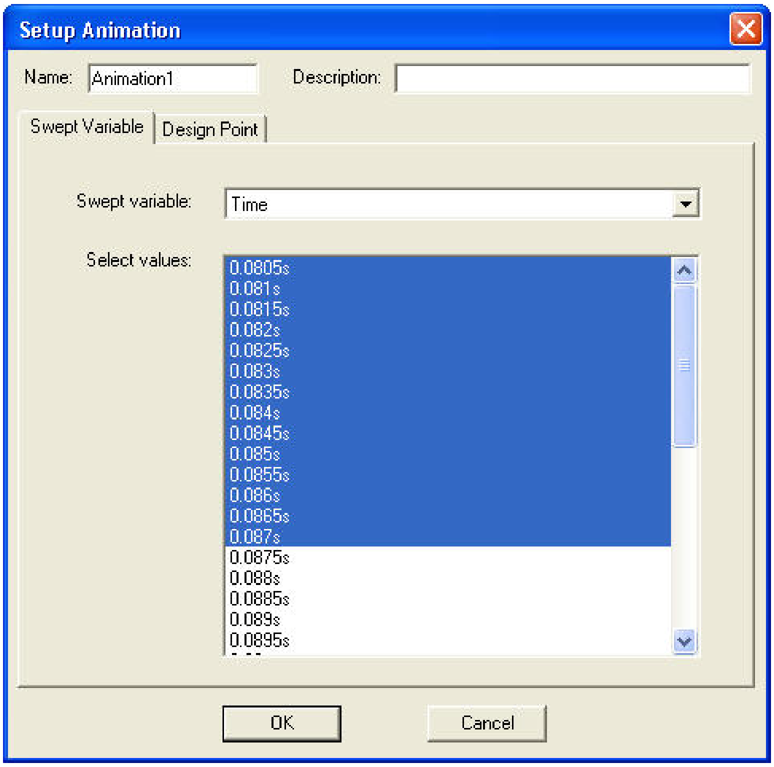
Create Flux Density Plot

- **To Plot Flux Density on Core**
  - Double click on **Maxwell3DDesign1** in Project Manager window to exit Plot view
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    - Plot on surface Only: ☑ **Checked**
    - Press **Done**

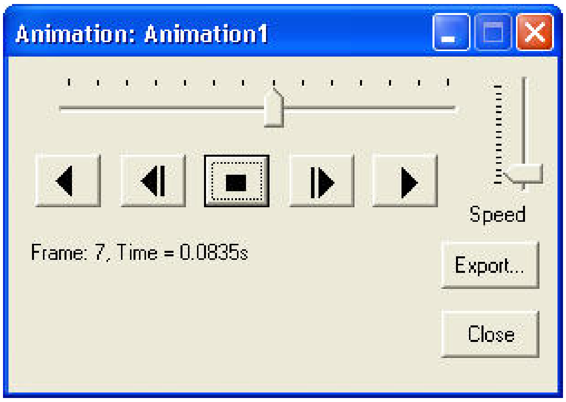




- **To Animate the Plot**
  - Select the menu item ***Maxwell 3D > Fields > Animate***
  - In Setup Animation window,
    - Sweep Variable: **Time**
    - Select values: Select the time range from **0.0805s** to **0.087s**
    - Press **OK**



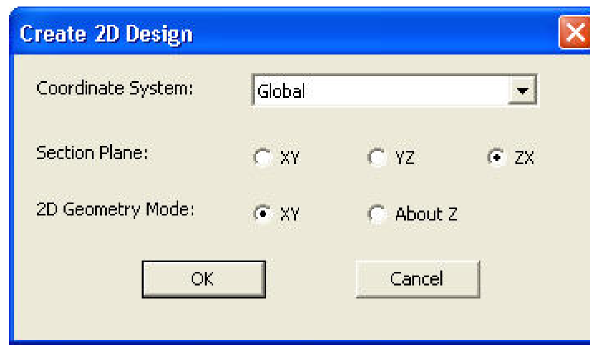
  - An Animation window will pop up which will enable to start, stop, pause the animation. Animation speed can also be varied using same window. The animation can be also exported in GIF or AVI format using Export button



Part 2: 2D Eddy Project

Create a 2D Design Automatically

- **To Create a 2D Design from 3D**
  - Select the menu item ***Maxwell 3D > Create 2D Design***
  - In Create 2D Design window,
    - Coordinate System: **Global**
    - Section Plane: **ZX**
    - 2D Geometry Mode: **XY**
    - Press **OK**

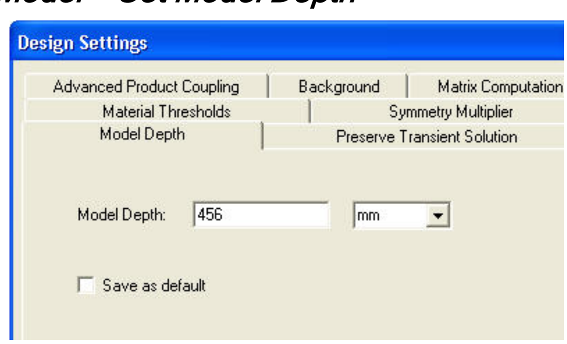


Set Solution Type

- **To Set Solution Type**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - In Solution Type window,
    - Verify that Geometry Mode is set to **Cartesian, XY**
    - Select the radio button to **Magnetic > Transient**
    - Press **OK**

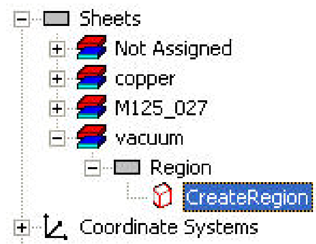
Set Model Depth

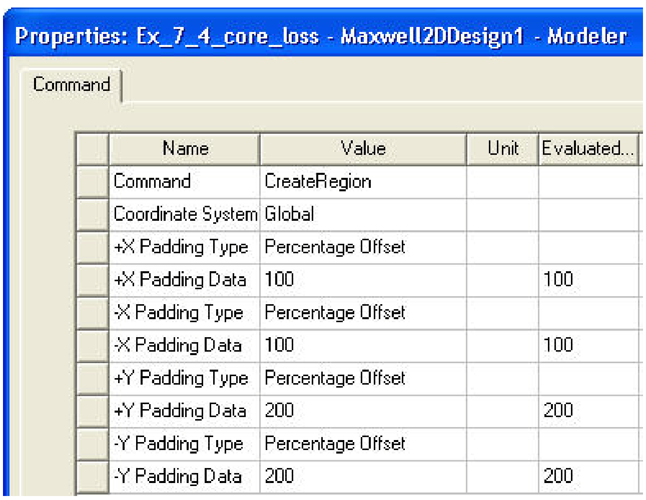
- Set the depth of the 2D XY model to give the same area as in Maxwell 3D = 264704 mm^2. Since the width of the core leg = 580mm, set the depth = 456mm.
- **To Set Model Depth**
  - Select the menu item ***Maxwell 2D > Model > Set Model Depth***
  - In Design Settings window,
    - Set Model Depth to **456 mm**
    - Press **OK**



Modify 2D Geometry

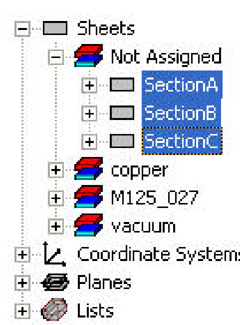
- **Modify Region**
  - Expand the history tree corresponding to the sheet **Region**
  - Double click on the command **CreateRegion** from the history tree
  - In Properties window,
    - Change +X Padding Data to **100**
    - Change -X Padding Data to **100**
    - Press **OK**





Name	Value	Unit	Evaluated...
Command	CreateRegion		
Coordinate System	Global		
+X Padding Type	Percentage Offset		
+X Padding Data	100		100
-X Padding Type	Percentage Offset		
-X Padding Data	100		100
+Y Padding Type	Percentage Offset		
+Y Padding Data	200		200
-Y Padding Type	Percentage Offset		
-Y Padding Data	200		200

- **Delete Unnecessary Sheets**
  - Press **Ctrl** and select the sheets **SectionA, SectionB** and **SectionC**
  - Select the menu item ***Edit > Delete***



- **Separate Coil Sections**
  - Press **Ctrl** and select the sheets **LV_A,LV_B** and **LV_C**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***

Specify Excitations

- **Assign Coil: Out**
  - Press **Ctrl** and select the sheets **LV_A, LV_B** and **LV_C** from history tree
  - Select the menu item ***Maxwell 2D > Excitations > Assign > Coil***
  - In Coil Excitation window,
    - Base Name: **out**
    - Number of Conductors: **76**
    - Polarity: **Positive**
  - Rename the excitations created to **A_out, B_out** and **C_out**
- **Assign Coil: In**
  - Press **Ctrl** and select the sheets **LV_A_Separate1, LV_B_Separate1** and **LV_C_Separate1** from history tree
  - Select the menu item ***Maxwell 2D > Excitations > Assign > Coil***
  - In Coil Excitation window,
    - Base Name: **in**
    - Number of Conductors: **76**
    - Polarity: **Negative**
  - Rename the excitations created to **A_in, B_in** and **C_in**
- **Add Windings**
  - Select the menu item ***Maxwell 2D > Excitations > Add Winding***
  - In Winding window,
    1. Name: **WindingA**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **1 mOhm**
    6. Inductance: **0 mH** (Since this is calculated by solver)
    7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time)**
    8. Press **OK**
  - In Add Variable window,
    1. Unit Type: **Voltage**
    2. Unit: **V**
    3. Value: $13800\sqrt{2} / \sqrt{3}$ = **11268**
    4. Press **OK**

- In Similar way add two more windings
- WindingB
    1. Name: **WindingB**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **1 mOhm**
    6. Inductance: **0 mH** (Since this is calculated by solver)
    7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time+(2/3*pi))**
- WindingC
    1. Name: **WindingC**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **1 mOhm**
    6. Inductance: **0 mH** (Since this is calculated by solver)
    7. Voltage: **Vpeak*(1-exp(-50*time))*cos(2*pi*60*time+(4/3*pi))**

Add Coils to the Winding

- Expand the Project Manager tree to view Excitations
- Right click on **WindingA** and select **Add Coils**
- In Add Terminals window,
    - Press **Ctrl** and select **A_in** and **A_out**
    - Press **OK**
- In Similar way add **B_in** and **B_out** to **WindingB**
- Add **C_in** and **C_out** to **WindingC**

Assign Boundary

- The current is assumed to be 1A at 0 degrees in the left busbar and -1A at 60 degrees in the right busbar. A no-fringing vector potential boundary will be assigned to the outside of the 2D problem region which is also the default boundary for all 3D projects. This forces all flux to stay in the solution region.
- **To Assign Boundary**
  - Select the menu item ***Edit > Select > Edges***
  - Select all external edges of the Region
  - Select the menu item ***Maxwell 2D > Boundaries > Assign > Vector Potential***
  - In Vector Potential Boundary window,
    - Value: Set to **0**
    - Press **OK**
  - Select the menu item ***Edit > Select > Objects*** to change selection filter

Assign Mesh Operations

- As in the 3D transient solver, there is no adaptive meshing in the 2D transient solver. A manual mesh is created manually using “inside selection” to create elements throughout the volume of the objects.
- **To Assign Mesh Operation**
  - Press **Ctrl** and select the core and all six sheets corresponding to coils
  - Select the menu item ***Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    - Restrict Length of Elements: ☑ **Checked**
    - Maximum Length of Elements: **100 mm**
    - Restrict the Number of Elements: ☐ **Unchecked**
    - Press **OK**

Set Eddy Effects

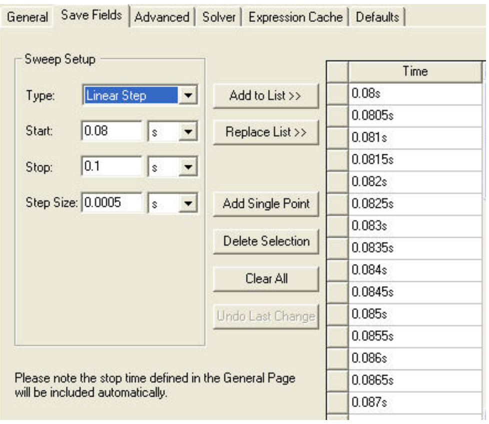
- **To Turn off Eddy Effects in Objects**
  - Select the menu item ***Maxwell 2D > Excitations > Set Eddy Effects***
  - In Set Eddy Effects window,
    - Ensure Eddy Effects are **Unchecked** for all objects
    - Press **OK**

Set Core Loss Calculations

- **To Set Core Loss calculations for Core**
  - Select the menu item ***Maxwell 2D > Excitations > Set Core Loss***
  - In Set Core Loss window,
    - Core:
      - Core Loss Settings: ☑ **Checked**
    - Press **OK**

Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 2D > Analysis Setup > Add Solution Setup***
  - In Solve Setup Window,
    - **General** tab
      - Stop time: **0.1s**
      - Time step: **0.0005 s**
    - **Save Fields** tab
      - Type: **Linear Step**
      - Start: **0.08 s**
      - Stop: **0.1 s**
      - Step Size: **0.0005 s**
      - Select the button **Add to List >>**
    - **Solver** tab
      - Nonlinear Residuals: **1e-6**
    - Press **OK**



Save

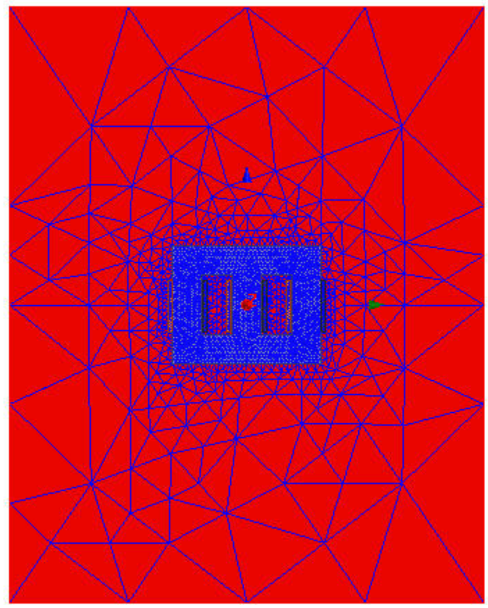
- **To Save File**
  - Select the menu item ***File > Save***

Analyze

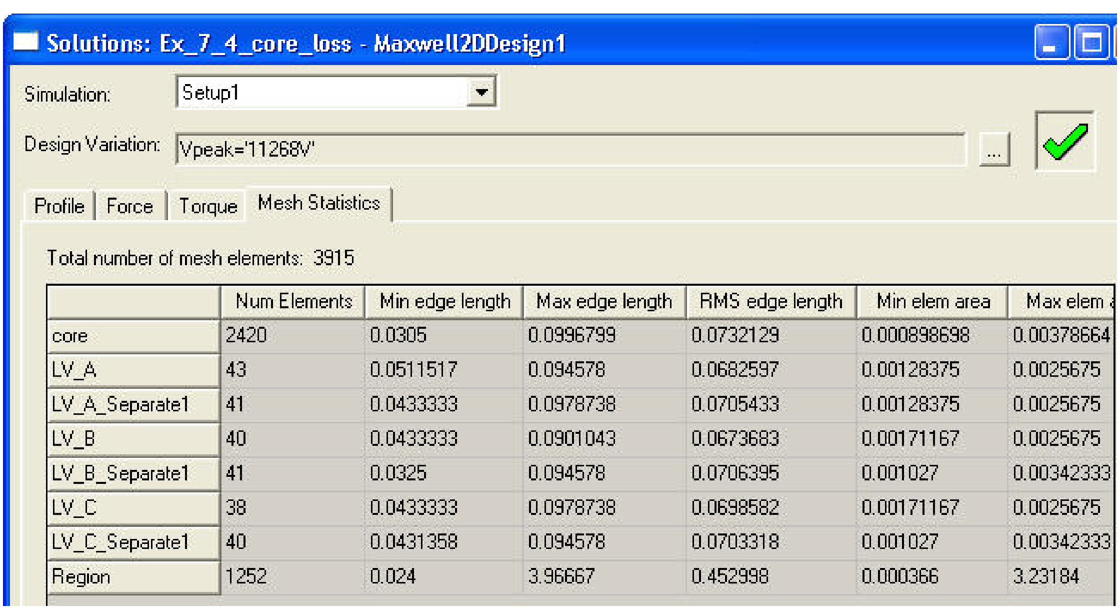
- **To Run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Mesh Information

- **To Plot Mesh on Core and Coils**
  - Select the menu item ***Edit > Select All Visible***
  - Select the menu item ***Maxwell 2D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    - Press **Done**



- **To View Mesh Information**
  - Select the menu item ***Maxwell 2D > Results > Solution Data***
  - In Solutions window,
    - Select the tab **Mesh Statistics** to view mesh information

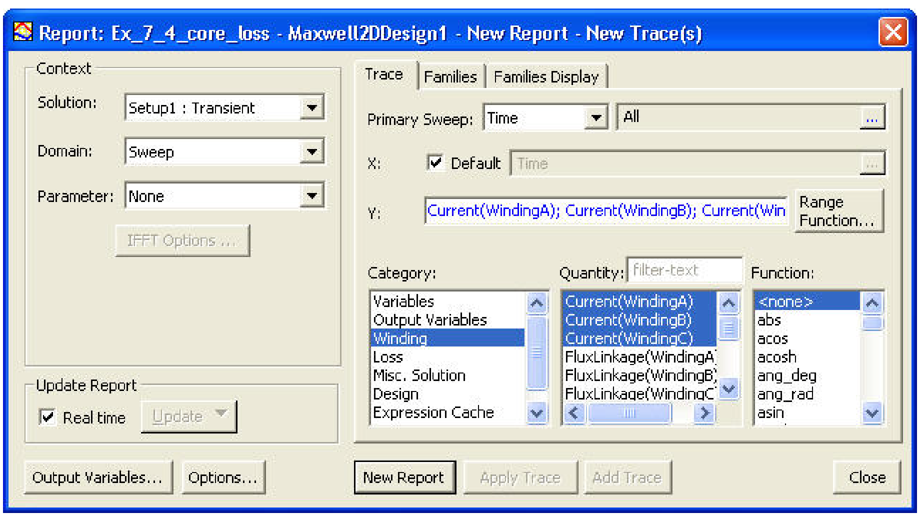


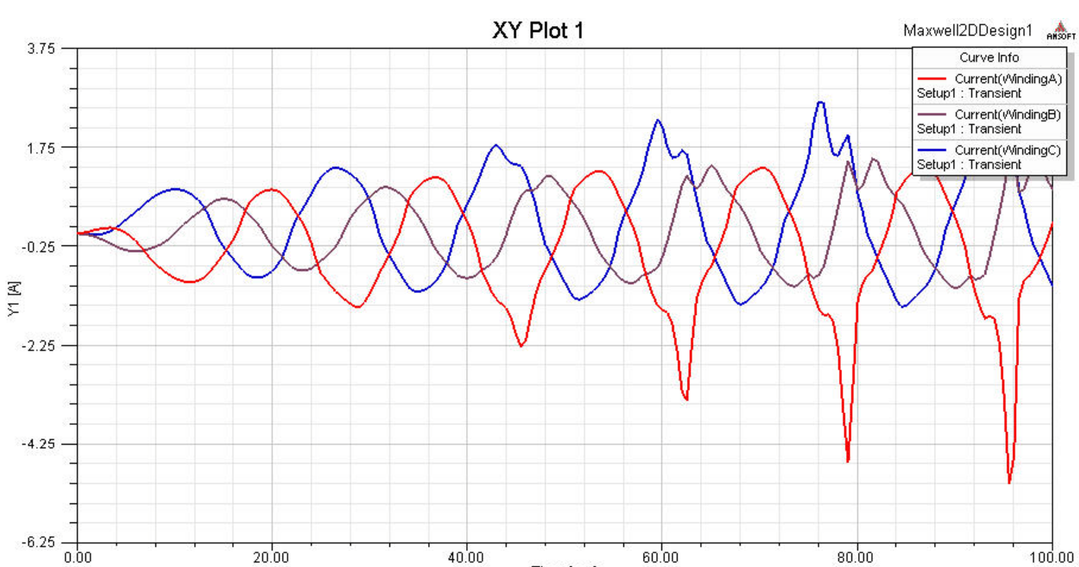
	Num Elements	Min edge length	Max edge length	RMS edge length	Min elem area	Max elem a
core	2420	0.0305	0.0996799	0.0732129	0.000898698	0.00378664
LV_A	43	0.0511517	0.094578	0.0682597	0.00128375	0.0025675
LV_A_Separate1	41	0.0433333	0.0978738	0.0705433	0.00128375	0.0025675
LV_B	40	0.0433333	0.0901043	0.0673683	0.00171167	0.0025675
LV_B_Separate1	41	0.0325	0.094578	0.0706395	0.001027	0.00342333
LV_C	38	0.0433333	0.0978738	0.0698582	0.00171167	0.0025675
LV_C_Separate1	40	0.0431358	0.094578	0.0703318	0.001027	0.00342333
Region	1252	0.024	3.96667	0.452998	0.000366	3.23184

Create Reports

Plot Winding Currents Vs Time

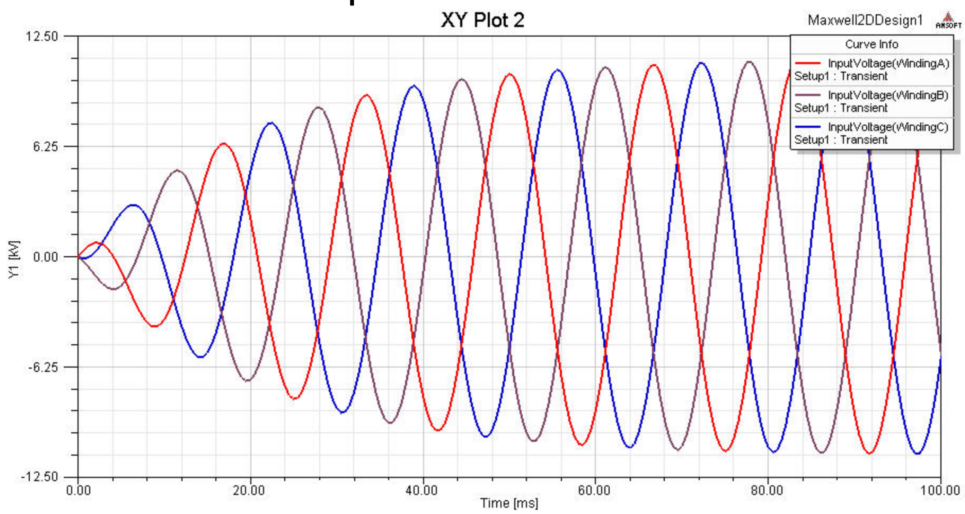
- Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
- In Report window,
  - Category: **Winding**
  - Quantity: Press **Ctrl** and select **Current(WindingA), Current(WindingB)** and **Current(WindingC)**
  - Select **New Report**
- **Note:** Do not close Report window as we will create more plots using same window



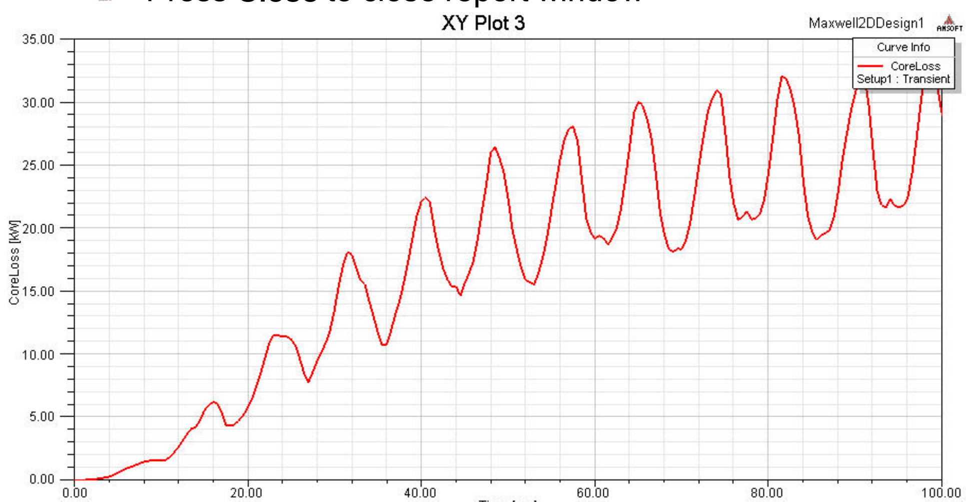


Example (2D/3D Transient) – Core Loss

- **Plot Input Voltages Vs Time**
  - In Report window,
    - Quantity
      - Deselect the Current quantities already selected
      - Press **Ctrl** and select **InputVoltage(WindingA), InputVoltage(WindingB)** and **InputVoltage(WindingC)**
    - Select **New Report**

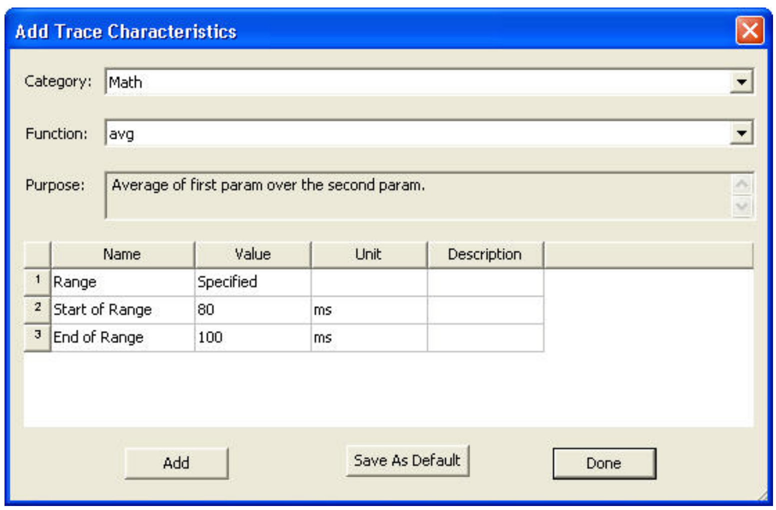


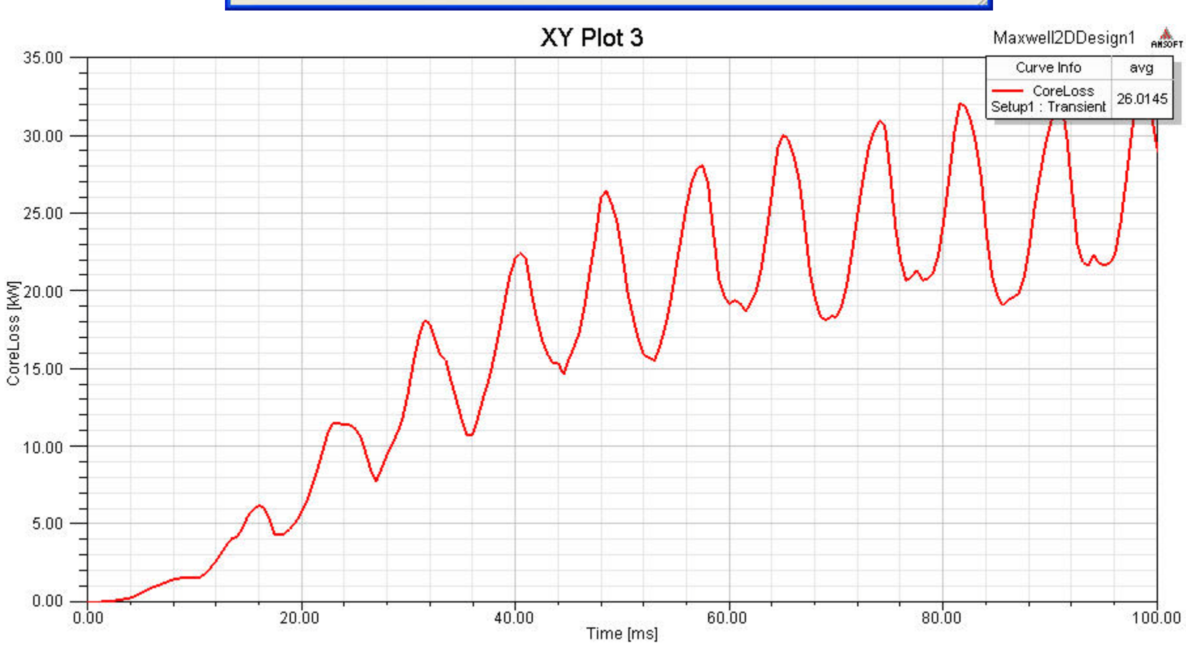
- **Plot Cores Loss vs Time**
  - In Report window,
    - Category: Change to **Loss**
    - Quantity: Select **CoreLoss**
    - Select **New Report**
    - Press **Close** to close report window



Example (2D/3D Transient) – Core Loss

- **Calculate Avg Losses over a Time Range**
  - In XY Plot Corresponding to CoreLoss, right click on the Legend and select ***Trace Characteristics > Add***
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **Avg**
    - Change the Range from Full to **Specified**
    - Start of Range: **80 ms**
    - End of Range: **100 ms**
    - Select **Add** and **Done**

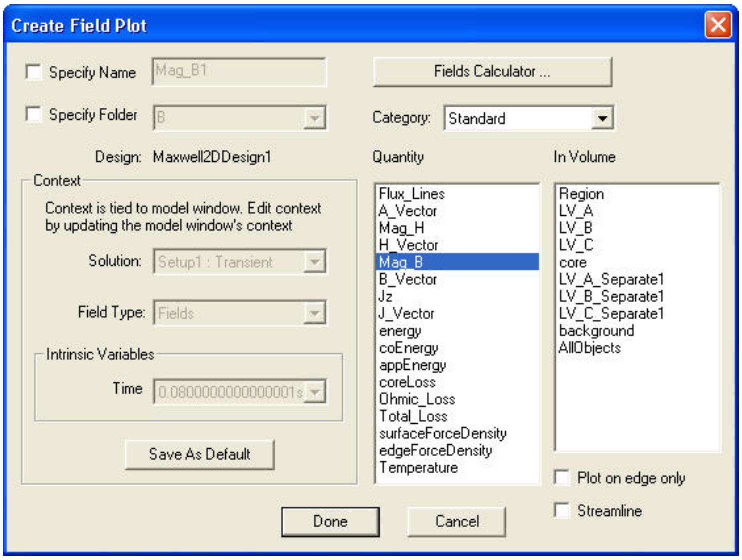


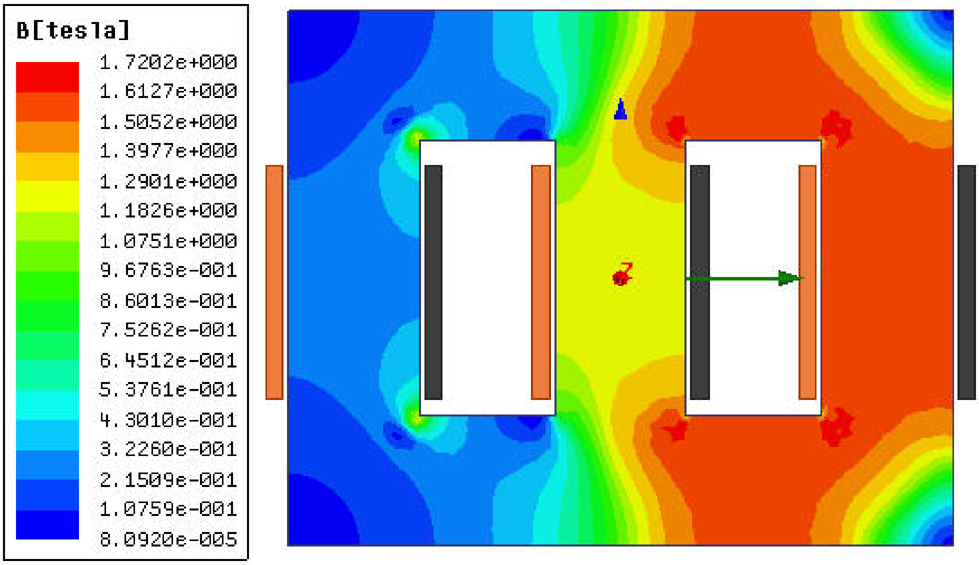


Create Flux Density Plot

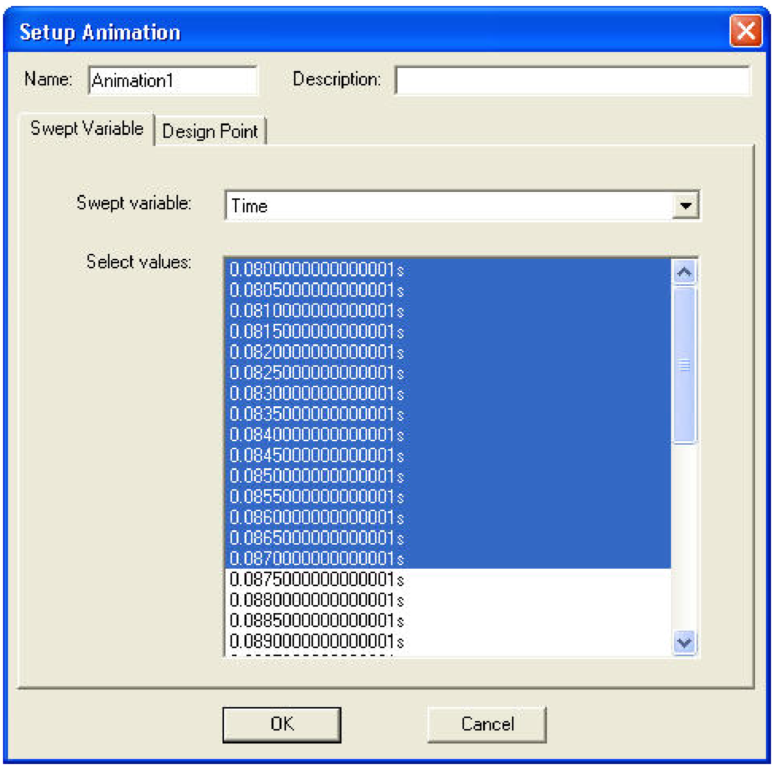
To Plot Flux Density on Core

- Double click on **Maxwell2DDesign1** in Project Manager window to exit Plot view
- Select the sheet **Core** from the history tree
- Select the menu item ***Maxwell 2D > Fields > Fields > B > Mag_B***
- In Create Field Plot window,
  - Press **Done**

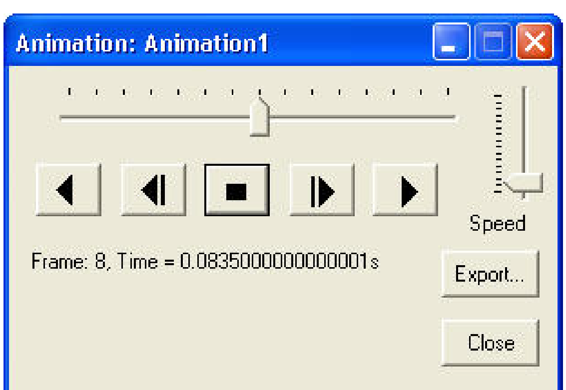




- **To Animate the Plot**
  - Select the menu item ***Maxwell 2D > Fields > Animate***
  - In Setup Animation window,
    - Sweep Variable: **Time**
    - Select values: Select the time range from **0.0805s** to **0.087s**
    - Press **OK**



  - An Animation window will pop up which will enable to start, stop, pause the animation. Animation speed can also be varied using same window. The animation can be also exported in GIF or AVI format using Export button

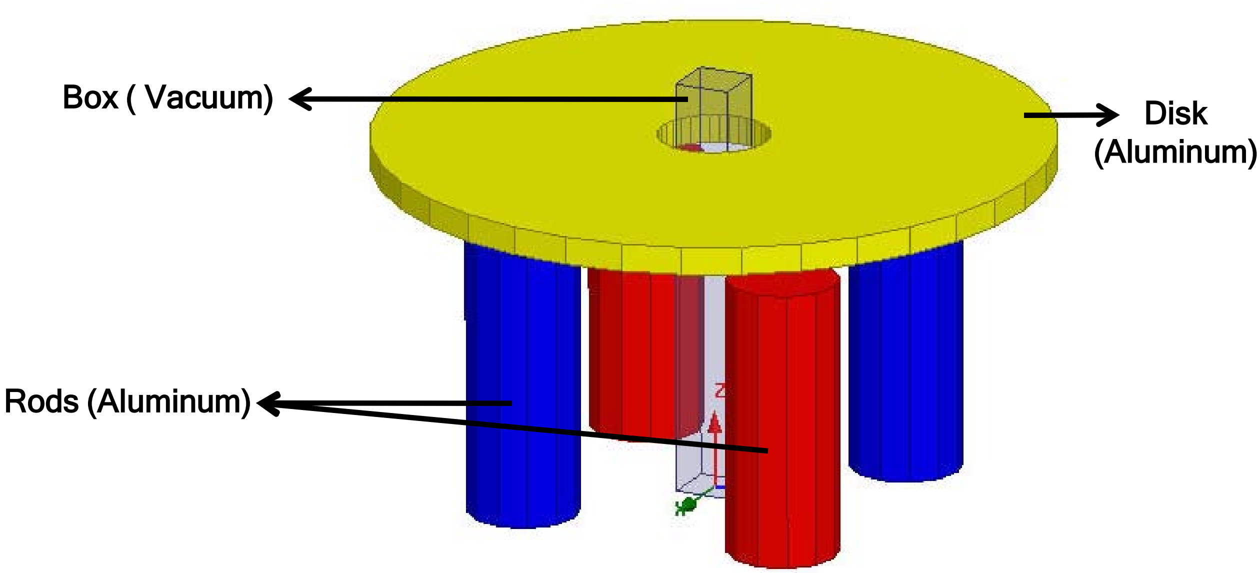


Chapter 8.0 – Electrostatic

Example (Electrostatic) – Mass Spectrometer

Quadrupole Mass Spectrometer

- This example uses the Electrostatic solver in the ANSYS Maxwell 3D Design Environment.
- It models a Quadrupole Mass Spectrometer which is used to identify particles present in a substance. The particles are ionized and subjected to an electric field which deflects them in angles directly proportional to their mass. The objective of this simulation is to determine the voltage magnitude and electric field stress in the center of the device. These results will be exported to a file in a uniform grid format so that they can be used in a separate program to calculate particle trajectories.
- The model consists of four vertical rods, one round disk with a hole in the center, a box for mesh refinement, and a cylindrical region. Although the actual length of this type of spectrometer is approximately 200 mm, only the last 30 mm will be modeled and analyzed. This is where the deflection of the particles occurs due to the fields.
- The application note covers the drawing of the model as well as the assignment of material properties and sources. Only one boundary condition: the ground plane, needs to be specified. Otherwise, the default Neumann boundaries on the remaining outer edges are adequate. The excitation consists of simple voltage sources. A mesh operation is assigned to the box object in order to increase the number of tetrahedra in that area. After the final solution is completed, plots are generated for voltage contours and electric field intensities. Also, voltage values are exported to a file in a uniform grid pattern.



Example (Electrostatic) - Mass Spectrometer

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Regular Polyhedron, Box**
    - Sweep Operations: **Around Axis**
  - Boundaries/Excitations
    - **Voltage**
  - Analysis
    - **Electrostatic**
  - Results
    - **Voltage**
  - Field Overlays:
    - **Mag_E**
    - **Voltage**

Example (Electrostatic) - Mass Spectrometer

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

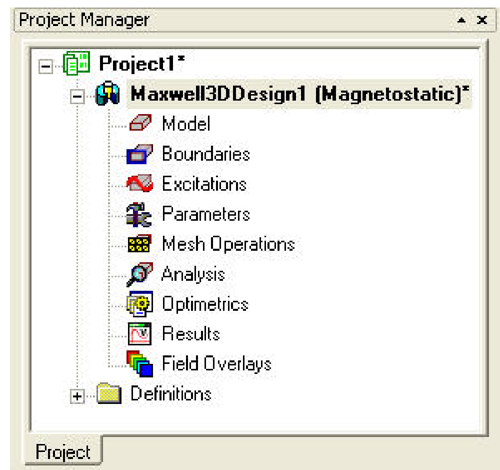
Setting Tool Options

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

Example (Electrostatic) - Mass Spectrometer

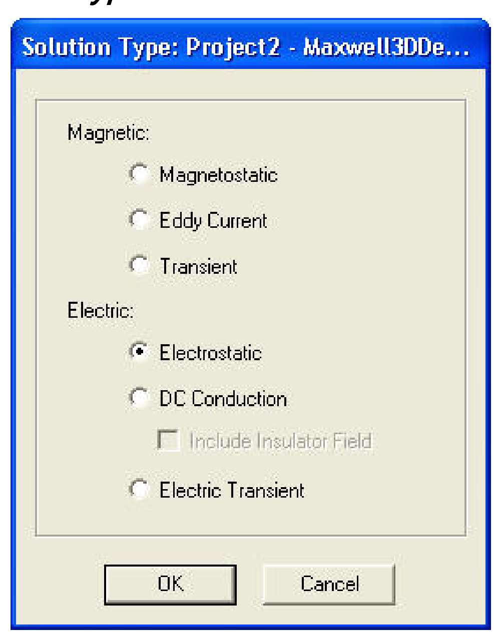
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



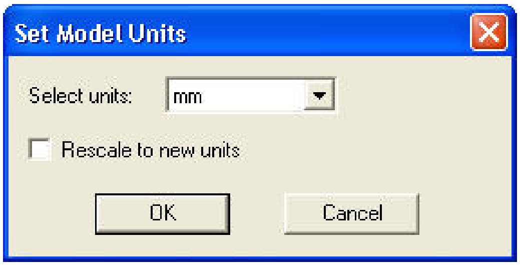
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Electric > Electrostatic**
    2. Click the **OK** button



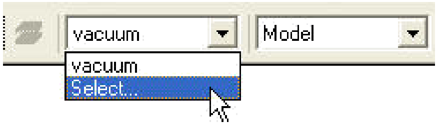
Set Model Units

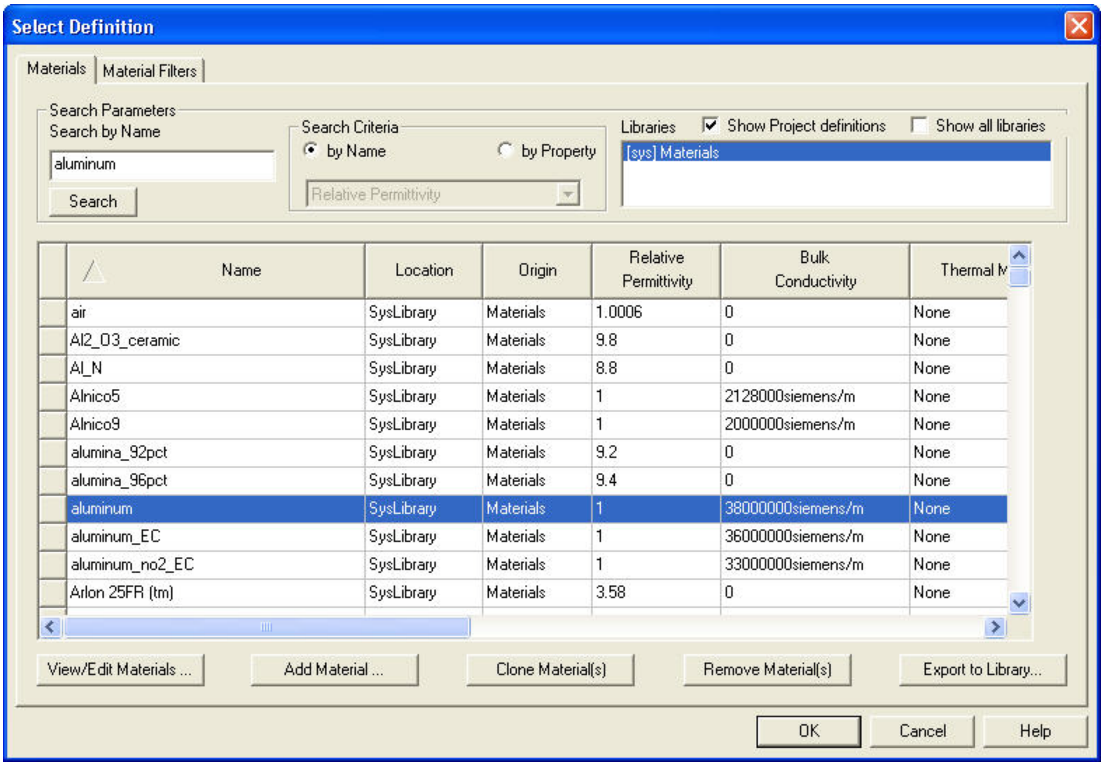
- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **mm**
    2. Click the **OK** button



Set Default Material

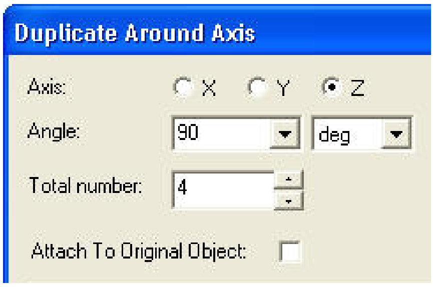
- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **aluminum** in the **Search by Name** field
    2. Click the **OK** button



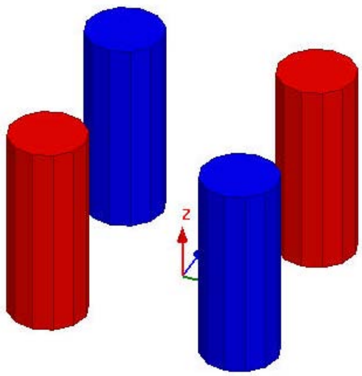


Create Rods

- **To Create Rod**
  - Select the menu item *Draw > Regular Polyhedron*
    1. Using Coordinate entry field, enter the center of base
       - X: **-10,** Y: **-10,** Z: **0,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - dX: **4,** dY: **0,** dZ: **20,** Press the **Enter** key
    3. Number of Segments: **12**
    4. Press **OK**
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Rod1**
    2. Change the color of the object to **Red**
- **Duplicate Rod1**
  - Select the object **Rod1** from the history tree
  - Select the menu item *Edit > Duplicate > Around Axis*
  - In Duplicate Around Axis window,
    - Axis: **Z**
    - Angle: **90 deg**
    - Total Number: **4**
    - Press **OK**

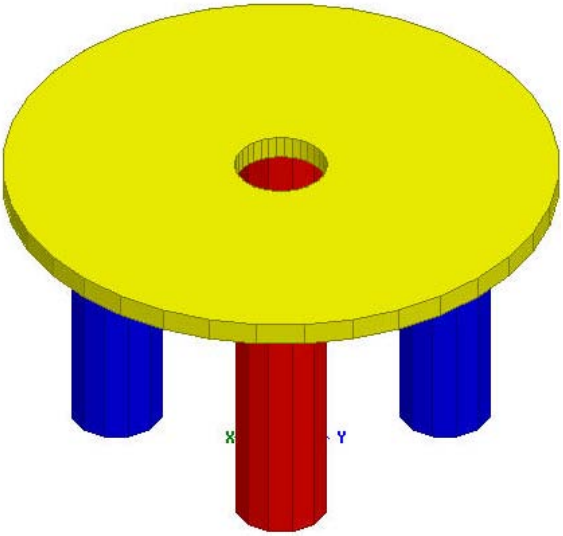


- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object Rod1_1 to **Rod2** and color to **Blue**
    2. Change the color of the object Rod1_2 to **Rod3** and color to **Red**
    3. Change the color of the object Rod1_3 to **Rod4** and color to **Blue**



Create Round Disk

- **Create Rectangle**
  - Select the menu item ***Modeler > Grid Plane > YZ***
  - Select the menu item ***Draw > Rectangle***
    1. Using Coordinate entry field, enter the position of rectangle
       - X: **0,** Y: **4,** Z: **24,** Press the **Enter** key
    2. Using Coordinate entry field, enter the opposite corner of rectangle
       - dX: **0,** dY: **20,** dZ: **2,** Press the **Enter** key
- **To Change Attributes**
  - Select the resulting sheet from the history tree and goto Properties window
    1. Change the name of the sheet to **Disk**
    2. Change the color of the sheet to **Yellow**
- **To Sweep the Sheet**
  - Select the sheet **Disk** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Sweep Axis: **Z**
    2. Angle of Sweep: **360 deg**
    3. Number of segments: **36**
    4. Press **OK**



Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Vacuum**

Create a Box

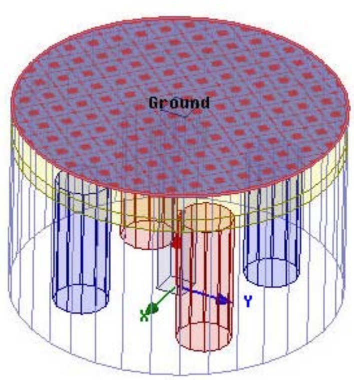
- You will need to create a box for the manual mesh refinement. This will be used to increase the number of tetrahedra in an area which is of primary interest but has low field strength.
- **To Create a Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X:**-2**, Y: **-2**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner of the box:
       - dX: **4**, dY: **4**, dZ: **30** , Press the **Enter** key
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Box**
    2. Change the transparency of the object to **0.8**

Create Solution Region

- **To Create Region**
  - Select the menu item ***Modeler > Grid Plane > XY***
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using Coordinate entry field, enter the center of base
       - X: **0,** Y: **0,** Z: **0,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - dX: **0,** dY: **24,** dZ: **30,** Press the **Enter** key
    3. Number of Segments: **36**
    4. Press **OK**
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Region**
    2. Change Display Wireframe: ☑ **Checked**

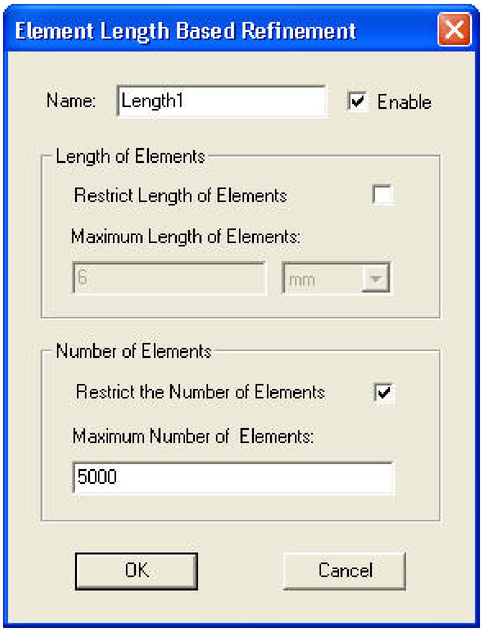
Specify Excitations

- **Specify Positive Excitation**
  - Press **Ctrl** and select the objects **Rod1** and **Rod3** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Positive**
    2. Value: **10 V**
    3. Press **OK**
- **Specify Negative Excitation**
  - Press **Ctrl** and select the objects **Rod2** and **Rod4** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Negative**
    2. Value: **-10 V**
    3. Press **OK**
- **Specify Voltage Source to Disk**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Source_Disk**
    2. Value: **5 V**
    3. Press **OK**
- **Specify Ground**
  - Select the menu item ***Edit > Select > Faces***
  - Select the top face of the **Region** as shown in image
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Ground**
    2. Value: **0 V**
    3. Press **OK**



Assign Mesh Operations

- The fields in the center of the assembly are of primary interest. In order to get a very accurate solution in this area of low field strength, it is necessary to manually refine the number of tetrahedra in this area.
- **To Assign Mesh Operations**
  - Select the menu item ***Edit > Select > Objects***
  - Select the object **Box** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length1**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**



Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

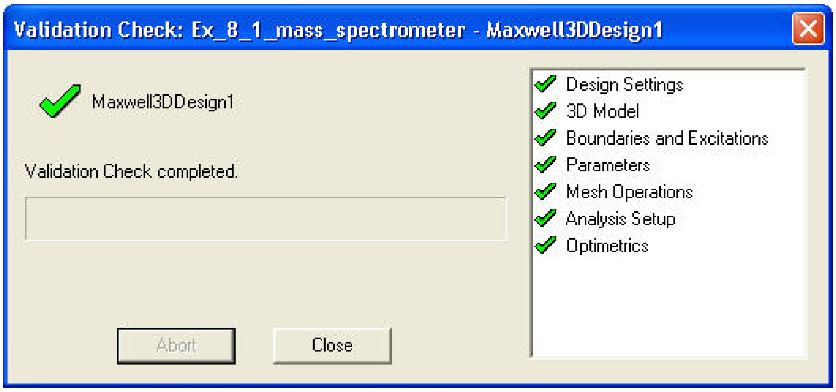
Example (Electrostatic) - Mass Spectrometer

Save Project

- **To save the project:**
  - Select the menu item *File > Save As*.
  - Save the file with the name: **Ex_8_1_Mass_Spectrometer**

Model Validation

- **To validate the model:**
  - Select the menu item *Maxwell 3D > Validation Check*
  - Click the **Close** button



- **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

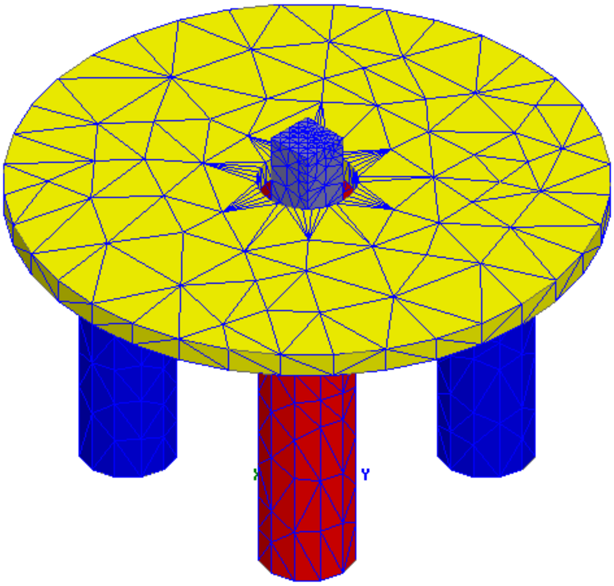
- **To start the solution process:**
  - Select the menu item *Maxwell 3D > Analyze All*

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To view the Mesh information:
      1. Click the **Mesh Statistics** Tab

Plot Mesh

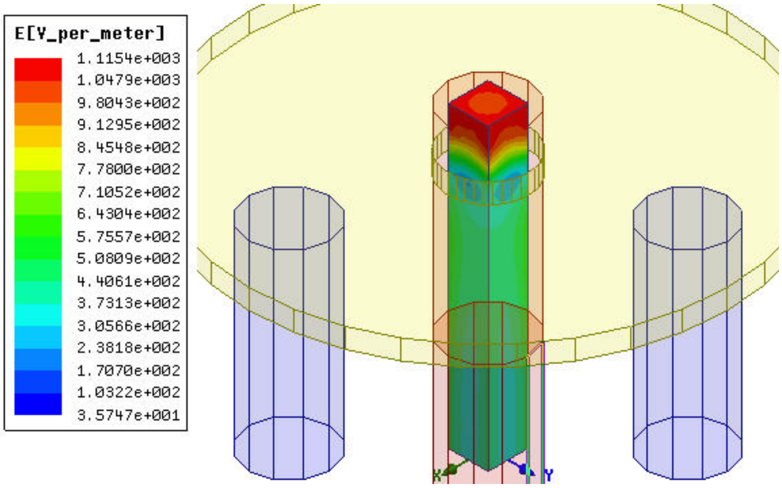
- **To Hide Region**
  - Select the object **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***
- **To Plot Mesh**
  - Select the menu item ***Edit > Select All Visible***
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    1. Press **Done**



Example (Electrostatic) - Mass Spectrometer

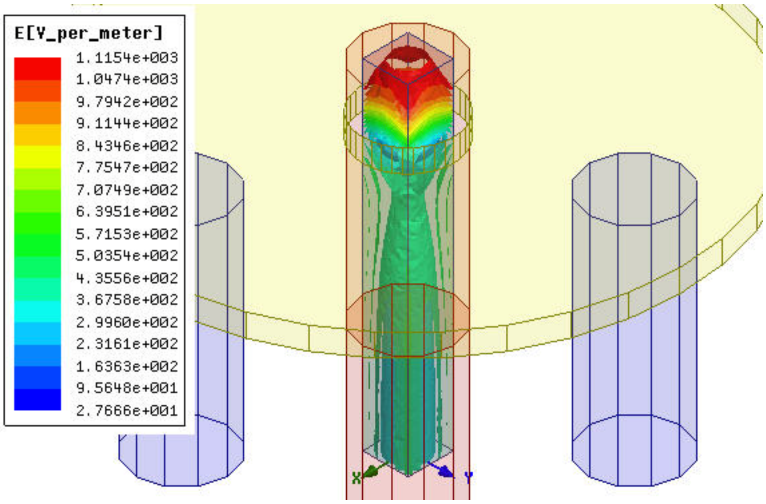
Plot the Mag_E field on surface of the box

- **To Plot Mag_E on the surface of the Box**
  - Select the object **Box** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > E > Mag_E***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



Plot the isovalues of Mag_E field in the volume of the box

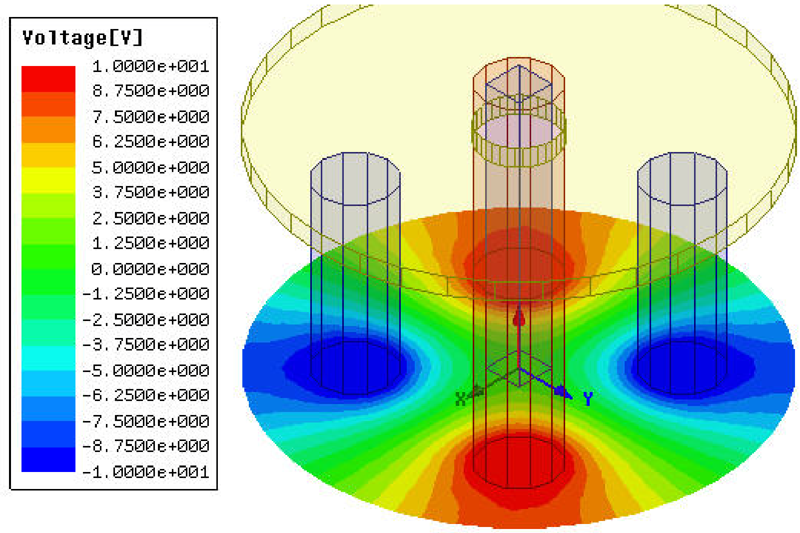
- **To Plot Mag_E in the volume of the Box**
  - Select the object **Box** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > E > Mag_E***
  - In Create Field Plot window,
    1. Plot on surface only: ☐ **Unchecked**
    2. Press **Done**



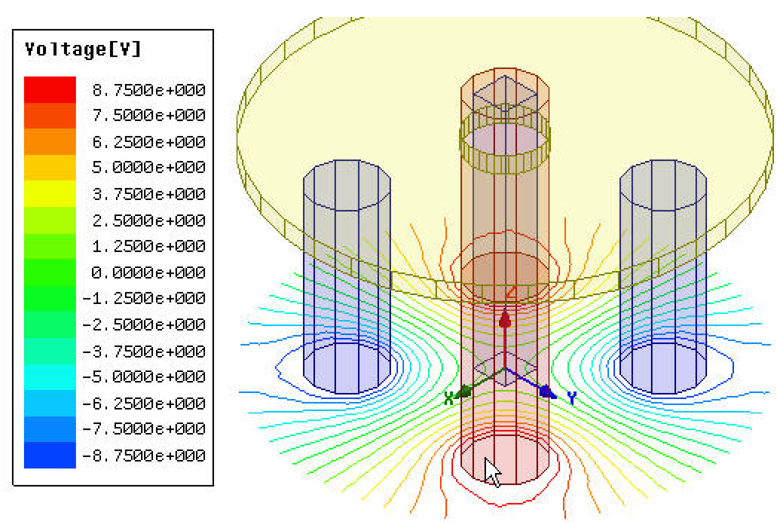
Example (Electrostatic) - Mass Spectrometer

Plot Voltage on Plane XY

- **To Plot Voltage on a Plane**
  - Expand the history tree for planes and select the plane **Global:XY**
  - Select the menu item ***Maxwell 3D > Fields > Fields > Voltage***
  - In Create Field Plot window,
    1. Press **Done**



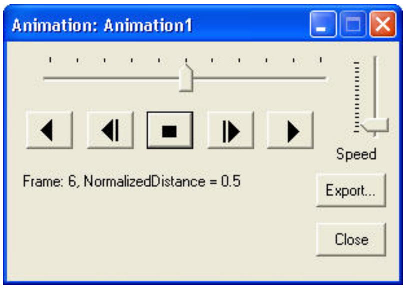
- **To Modify plot Attributes**
  - Double click on the legend to modify attributes of the plot
  - In the window, on **Plots** tab
    - Plot: **Voltage1**
    - IsoValType: **Line**
    - Press **Close**



Example (Electrostatic) - Mass Spectrometer

Animate the plot for Voltage

- **To Animate a Field Plot**
  - Expand the Project Manager tree to view **Field Overlays > Voltage > Voltage1**
  - Right click on the plot **Voltage1** and select **Animate**
  - In Setup Animation window,
    - Set Sweep variable to **Normalized Distance** and press **OK**
  - To start and stop the animation, to control the speed of the animation and to export the animation use the buttons on the Animation Panel:



Export the Voltage values to a file

- The voltage values are exported to an external file in order to be used in a particle trajectory program.
- **To Export Voltage values to a Text File**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - In Calculator window,
    1. Select **Input > Quantity > Voltage**
    2. Select **Output > Export**
    3. In Export Solution window,
       - Output file name: **Voltages.rtf**
       - Calculate grid points: ☑ **Checked**
       - X: **-1mm, 1mm, 1mm**
       - Y: **-1mm, 1mm, 1mm**
       - Z: **20mm, 22mm, 1mm**
    4. Press **OK**

References

1. International Journal of Mass Spectrometry and Ion Physics, Kevin Hunter and Bruce McIntosh, vol. 87 pp. 157-164, 1989
2. International Journal of Mass Spectrometry

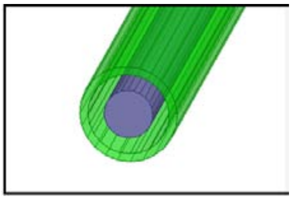
Chapter 9.0 – Basic Exercises

Introduction on the Electrostatic Solver

- This note introduces the "Electrostatic Solver" based on some simple examples. This solver is meant to solve the static electric field without current flowing in conductors (conductors are in electrostatic equilibrium). **The conductors are considered perfect such that there is no electric field inside conductors**.

Example 1 : Capacitance of a Cylindrical Capacitor

- Suppose we have a long coaxial line. We want to know what is the electric field distribution based on the potential (or the charges) that are applied on each conductor. We also want to determine the capacitance.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon
  - Rename design as **Coaxial Line**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose ***Electric > Electrostatic***
    2. Click the **OK** button

Set Default Material

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

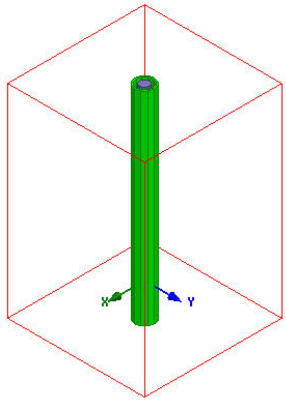
Create Geometry

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.6**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**25**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Inner**
- **Create Second Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **1**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**25**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Gap**
- **Create Third Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **1.2**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**25**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Outer**

- **Subtract Objects**
  - Press **Ctrl** and select the objects **Outer** and **Gap** from the history tree
  - Select the menu item, ***Modeler > Boolean > Subtract***
    1. Blank Parts: **Outer**
    2. Tool Parts: **Gap**
    3. Click the **OK** button

Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Region*** or click on the icon
  - In Region window,
    1. Pad individual directions: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - +X and -X = **300**
       - +Y and -Y = **300**
       - +Z and -Z = **0**
    3. Press **OK**



Assign Excitations

- Based on the assumptions that the conductors are in electrostatic equilibrium, we assign voltage potential on the object itself (and not on the surface of the object like in the other solvers).
- **Assign Excitation for Inner**
  - Select the object **Inner** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **-1000 V**
    2. Press **OK**
- **Assign Excitation for Outer**
  - Select the object **Outer** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **1000 V**
    2. Press **OK**

Assign Parameters

- In addition to the field, we are interested by the Capacitance value as well as the force applied to the inner armature.
- **Assign Matrix Parameter to Calculate Capacitance**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window
    1. Voltage1 and Voltage2
       - Include: ☑ **Checked**
    2. Press **OK**
- **Assign Force Parameter for Inner**
  - Select the Object **Inner** from the tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Force***
  - In Force Setup window,
    1. Type : **Virtual**
    2. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percent Error: **5**
    2. **Convergence** Tab
       - Refinement Per Pass: **50%**
    3. Click the **OK** button

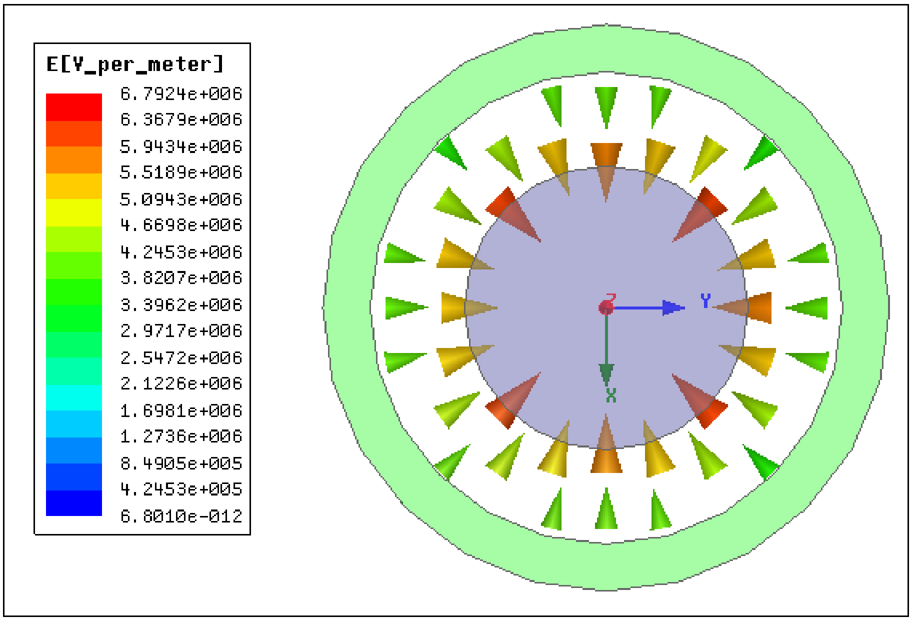
Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

  (You will need to save your project before analyzing. Note that, the project is always saved before the simulation starts).

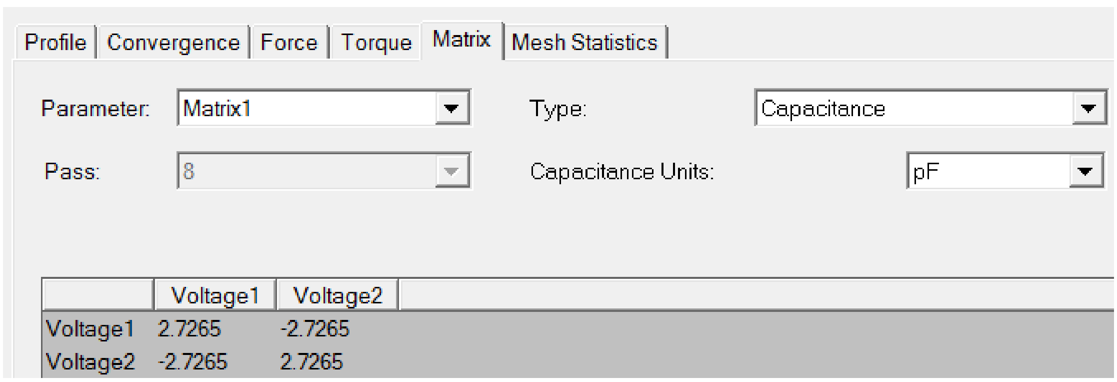
Create Vector Plot

- **To Create Vector Plot**
  - Select the plane **Global:XY** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > E > E_Vector***
  - In Create Field Plot window
    1. Press **Done**
  - To adjust spacing and size of arrows, double click on the legend and then go to **Marker/Arrow** and **Plots** tabs

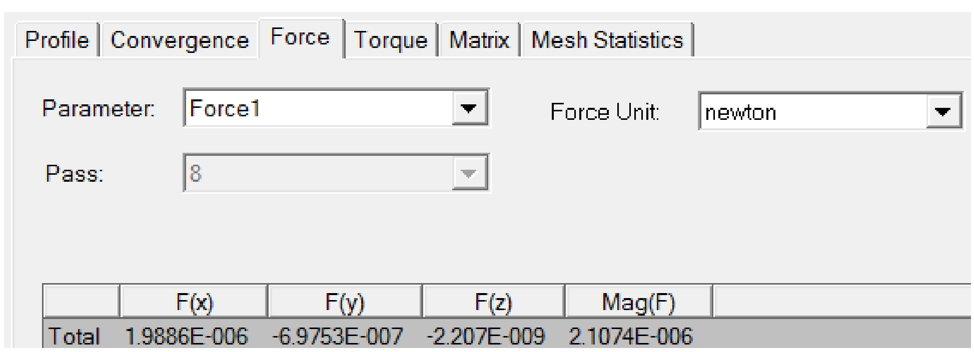


Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Capacitance**
       - Select the **Matrix** tab



       - In our problem, we only have two conductors, therefore the capacitance values are symmetrical
    2. To View **Force**
       - Select the **Force** tab



  - The analytical value of the capacitance per meter for an infinite long coaxial wire is given by the following formula:

    $C = 2\pi\varepsilon_0 / \ln(b/a)$ (a and b being the inside and outside diameters)
  - The analytical value would is therefore **1.089e-10 F/m** (a =0.6mm, b=1mm)
  - In our project, then length of the conductor is 25 mm, therefore the total capacitance is **2.72 pF**. We obtain a good agreement with the obtained result.

Example 2: Capacitance of a planar capacitor

- In this example we illustrate how to simulate a simple planar capacitor made of two parallel plates

Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***
  - Rename design as **Plate**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Electrostatic**
    2. Click the **OK** button

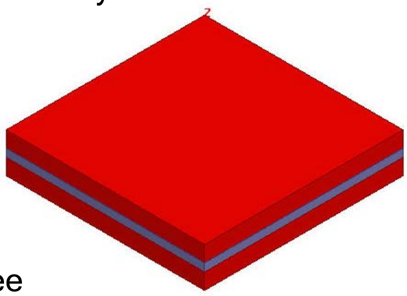
Set Default Material

- **To set the default material:**
  - Using the 3D Modeler Materials toolbar, choose **Select**
  - In Select Definition window,
    1. Type **pec** (Perfect Conductor) in the **Search by Name** field
    2. Click the **OK** button

Create Geometry

- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **25**, dY: **25**, dZ: **2**, Press the **Enter** key
  - Change the name of the Object to **DownPlate**
  - **Note:** It is not needed to draw the plates; it is possible in this case just to apply a voltage to the surface of the gap.

- **Create Gap**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **25**, dY: **25**, dZ: **1**, Press the **Enter** key
  - Change the name of the Object to **Gap** and material to **Vacuum**
- **Create Second Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **3**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **25**, dY: **25**, dZ: **2**, Press the **Enter** key
  - Change the name of the Object to **UpPlate**
  - Change the colors of the objects if needed



Assign Excitations

- **Assign Excitation for DownPlate**
  - Select the object **DownPlate** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **0 V**
    2. Press **OK**
- **Assign Excitation for UpPlate**
  - Select the object **UpPlate** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **1 V**
    2. Press **OK**

Assign Parameters

- **Assign Matrix Parameter to Calculate Capacitance**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window
    1. Voltage1 and Voltage2
       - Include: ☑ **Checked**

Analysis Setup

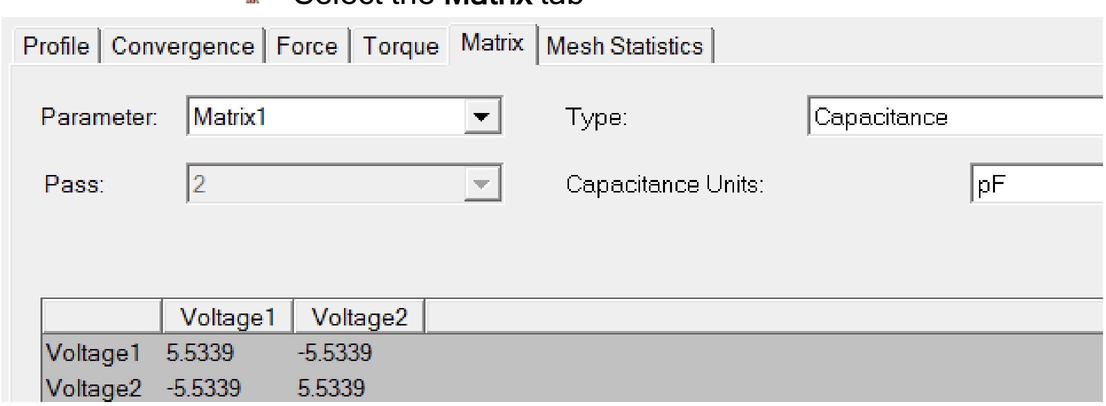
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **Convergence** Tab
       - Refinement Per Pass: **50%**
       - Maximum Number of Passes: **15**
    2. Click the **OK** button

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Capacitance**
       - Select the **Matrix** tab



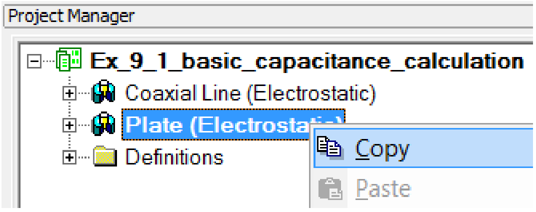
  - The analytical value of the capacitance for two parallel plates is given by: **C = A/ d *ε_0** (*A* is the area of the plate and *d* is the thickness of the dielectrics).
  - Using this formula, we obtain **5.53 pF.** This value matches the value obtain using the finite elements

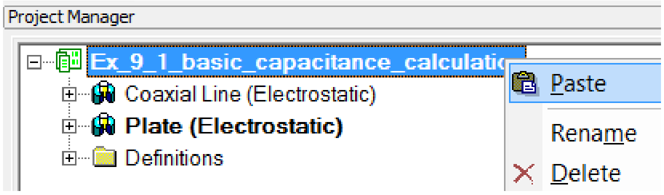
Example 3: Capacitance of planar capacitor with External Region

- Here we look at the consequence for the capacitance value when external surrounding is taken into consideration.

Copy Design

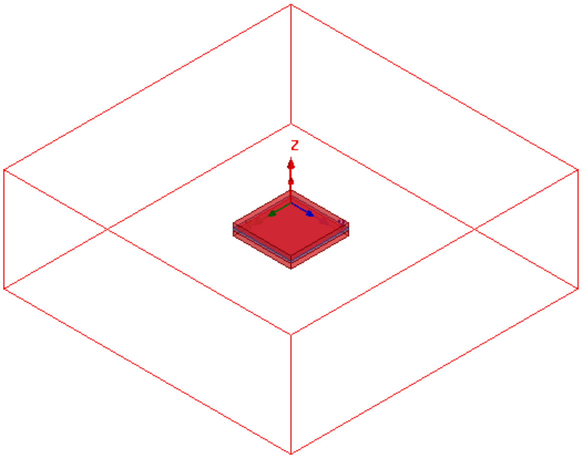
- **To Copy Design**
  1. Select the design **Plate** in Project Manager window, right click and select **Copy**
  2. Select project name in Project Manager window and select **Paste**





Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad individual directions: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - +X and -X = **200**
       - +Y and -Y = **200**
       - +Z and -Z = **400**
    3. Press **OK**

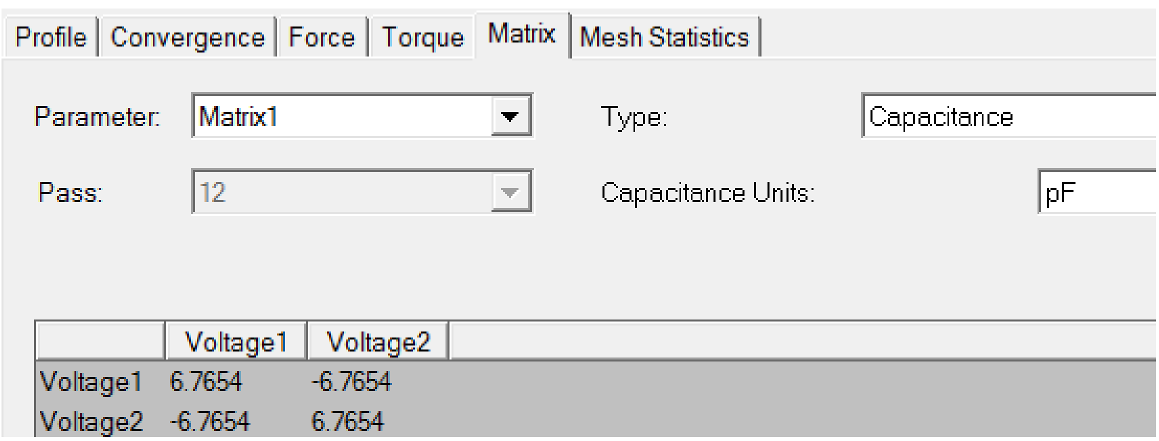


Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Capacitance**
       - Select the **Matrix** tab



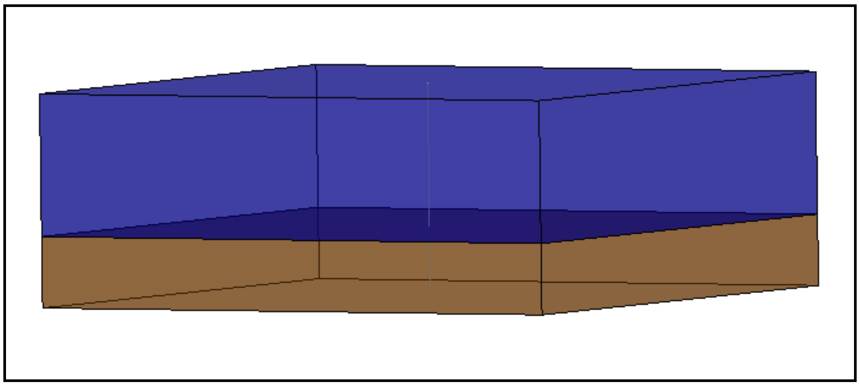
- Capacitance value reported is completely different from analytical value. But the analysis results match to more realistic conditions specially when dielectric size is relatively thick compared to electrodes.

Introduction on the DC Conduction Solver

- This note introduces the DC conduction solver. Only conductors are considered in the process.

Example 1: Parallel Plates with Non-uniform media

- In this example, we want to determine the DC resistance between two plates with a non-uniform media made of graphite and sea water. We do not need to draw the plates; we just need to draw the two media regions.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon
  - Change the name of the design to **Plates**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **DC Conduction**
       - Include Insulator Field : ☐ **Unchecked**
    2. Click the **OK** button

Create Geometry

- **Create Solid**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **4**, dY: **4**, dZ: **0.5**, Press the **Enter** key
  - Change the name of the Object to **Solid** and material to **graphite**
- **Create Liquid**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **4**, dY: **4**, dZ: **1**, Press the **Enter** key
  - Change the name of the Object to **Liquid** and material to **water_sea**

Assign Excitations

- We apply the voltage to the plates. We did not draw the plates, because we can apply a voltage to the top and bottom part of the regions.
- **Change Selection to Faces**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard
- **Assign Excitation for Top**
  - Select the top face of the Object **Liquid**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **10 V**
    2. Press **OK**
- **Assign Excitation for Top**
  - Select the bottom face of the Object **Solid**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **-10 V**
    2. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** to accept default settings

Analyze

- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

Compute the DC Resistance

- The DC resistance is not directly computed by Maxwell3D. However, we already know the applied voltage. We need the DC current that flows through the media. The current is obtained by taking the integral of Z component of **J** on a cross section of the region. To define any cross section, we define a local coordinate system, and use the planes defined by this local CS.
- **Create Local Coordinate System**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Offset***
    1. Using the coordinate entry fields, enter the origin position
       - X: **0**, Y: **0**, Z: **1**, Press the **Enter** key
- **Calculate Resistance**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - The field calculator is a tool that can compute any quantities using geometric entities and fields. The total current going through the media is the integral of the current J on the plane XY of the local CS.
    1. Select **Input > Quantity > J**
    2. Select **Vector > Scal? > ScalarZ**
    3. Select **Input > Geometry**
       - Select **Surface**
       - Select **RelativeCS1:XY** from the list
       - Press **OK**
    4. Select **Scalar >** **(Integrate)**
    5. Press **Eval**

- The total current appears, close to **-1.28A**. The value is very low; it makes sense because the conductivity of medium is very low. The negative sign is just a matter of sign convention due to the CS orientation.
- The DC resistance is given by R = Voltage / Current. The difference of potential between the two plate is 20 V. We obtain **R = 15.625 Ohm**

- **Analytical value of Resistance**
  - The analytical value of the resistance is given by the following formula

    $$R = \sigma_2 h_1 + \sigma_1 h_2 / (\sigma_1 \sigma_2 A)$$

    where σ_1, σ_2 are the conductivity of the two medium, h_1,h_2 the thickness of the two medium and A the surface of the plates.
  - If we take below values,
    - **σ_1 = 70000 Siemens/m**,
    - **h_1= 0.5e-3 m** (ferrite);
    - **σ_2 = 4 Siemens/m**,
    - **h_2 = 1e-3 m** (sea water) ;
    - **A = 16e-6 m^2**.
  - The value of resistance comes out to be:

    R = 15.625 Ohm
  - The values are similar to what we obtained using Maxwell

Example 2: DC current flow in conductors

- In this example, we illustrate the capability of the DC current solver to reconstruct the current paths flowing in different conductors

Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***
  - Rename design as **Conductors**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **DC Conduction**
    2. Click the **OK** button

Set Default Material

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

Create Geometry

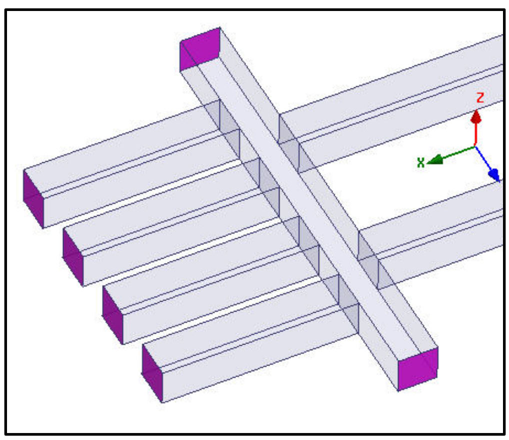
- **Create Conductor**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **1**, Y: **-0.6**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **1**, dY: **0.2**, dZ: **0.2**, Press the **Enter** key
  - Change the name of the Object to **Conductor**

- **Duplicate Conductor**
  - Select the object **Conductor** from the history tree
  - Select the menu item ***Edit > Duplicate > Along Line***
    1. Using coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using coordinate entry fields, enter the second point
       - dX: **0**, dY: **0.4**, dZ: **0**, Press the **Enter** key
    3. Total Number: **4**
    4. Press **OK**
- **Create Conductor_4**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.8**, Y: **-1**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.2**, dY: **2.2**, dZ: **0.2**, Press the **Enter** key
  - Change the name of the Object to **Conductor_4**
- **Create Conductor_5**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.8**, Y: **-0.4**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **-1.2**, dY: **0.2**, dZ: **0.2**, Press the **Enter** key
  - Change the name of the Object to **Conductor_5**
- **Duplicate Conductor_5**
  - Select the object **Conductor _5** from the history tree
  - Select the menu item ***Edit > Duplicate > Along Line***
    1. Using coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using coordinate entry fields, enter the second point
       - dX: **0**, dY: **0.8**, dZ: **0**, Press the **Enter** key
    3. Total Number: **2**
    4. Press **OK**

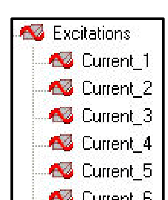
- **Create Conductor_Sink**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.4**, Y: **0.6**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **-0.4**, dY: **-1**, dZ: **0.2**, Press the **Enter** key
  - Change the name of the Object to **Conductor_Sink**

Assign Excitations

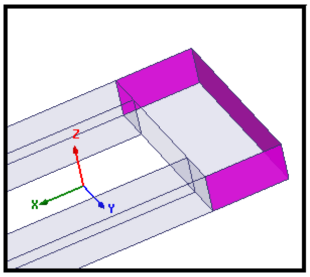
- **Change Selection to Faces**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard
- **Assign Excitation Current**
  - Select the top faces of the Objects **Conductor, Conductor_1, Conductor_2, Conductor_3** and **Conductor_4** as shown below



  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Set Value to **1 A**
    2. Press **OK**
  - Maxwell creates 6 current excitations of **1A**, each for every faces selected. The default current orientation is the current going inside the objects, so we do not need to swap any current direction



- **Assign Excitation for Top**
  - Select the faces of **Conductor_Sink** as shown in the image



  - Select the menu item ***Maxwell 3D > Excitations > Assign > Sink***
  - Press **OK**

Assign Mesh Operation

- **Change Selection to Objects**
  - Select the menu item ***Edit > Select > Objects*** or press **O** from the keyboard
- **To Assign Mesh Operation**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict Length of Elements: ☐ **Unchecked**
    2. Restrict the Number of Elements: ☑ **Checked**
    3. Maximum Number of Elements: **10000**
    4. Press **OK**

Analysis Setup

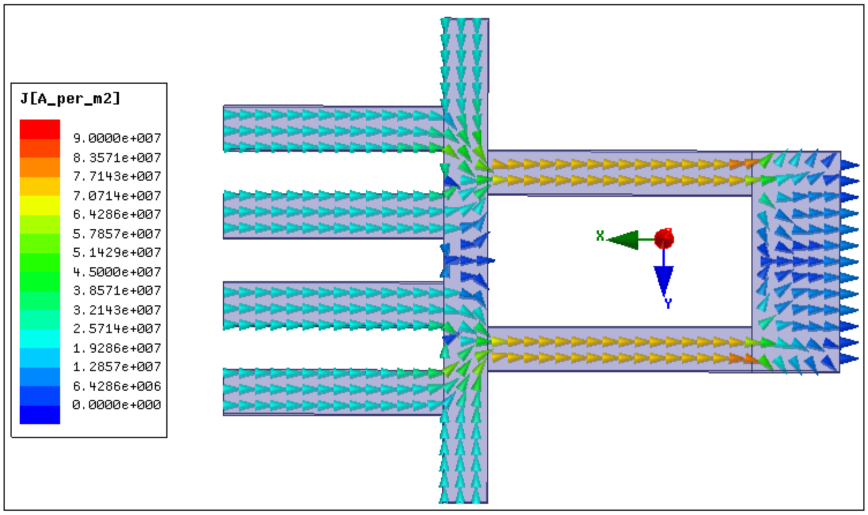
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** to accept default settings

Analyze

- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

Plot Current Density Vector

- **To Plot J Vectors**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Maxwell 3D > Fields > Fields > J > J_Vector***
  - In Create Field Plot window
    1. Press **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - User Limits: ☑ **Checked**
         1. Min: **0**
         2. Max: **9e+7**
    2. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **One space from right**
         2. Map Size: ☐ **Unchecked**
         3. Arrow tail: ☐ **Unchecked**
    3. **Plots** tab
       - Plot: **Vector_J1**
       - Vector Plot
         1. Spacing: **Set to Minimum**
    4. Press **Apply** and **Close**

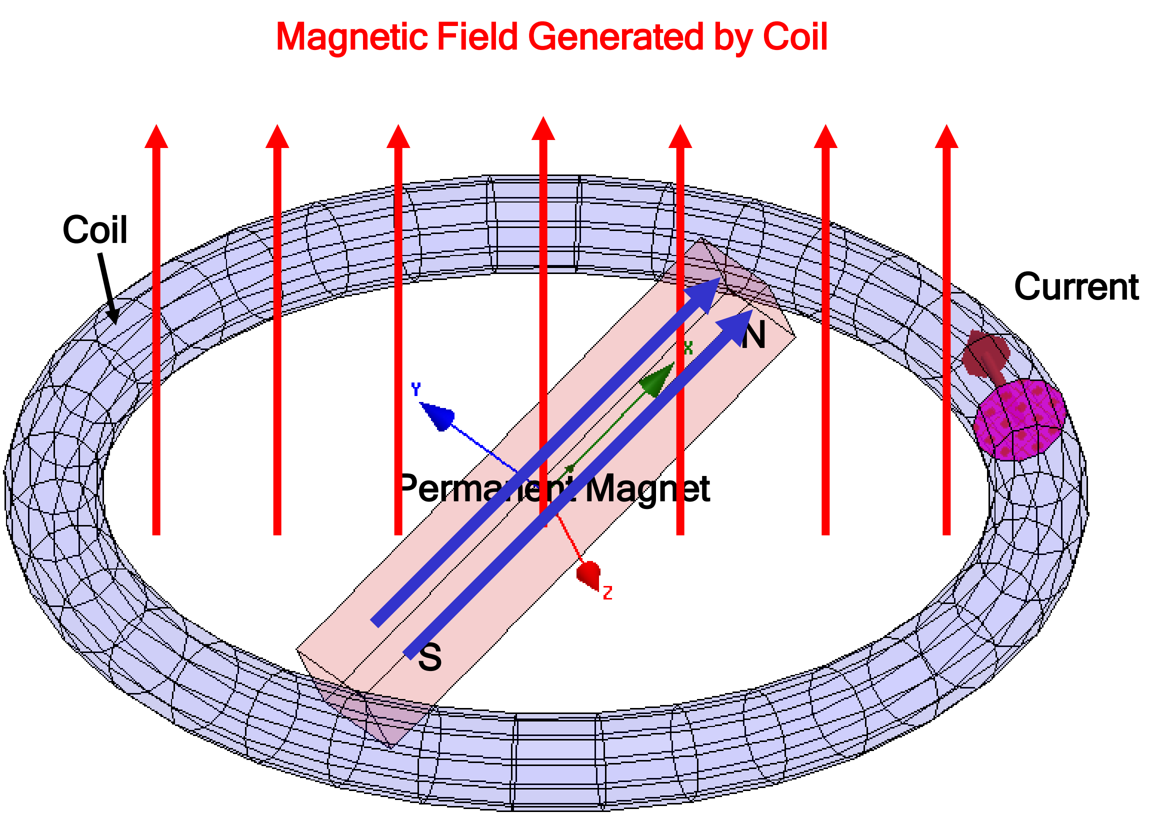


Torque Calculation in Magnetostatic Solver

- This exercise will discuss how to set up a torque calculation in the Magnetostatic Solver.

Problem Description

- As shown in the following image, the current in the coil generates a magnetic field pointing upward. The permanent magnet in the middle is magnetized along X-axis, hence there is a torque generated along Z-axis.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Set Solution Type

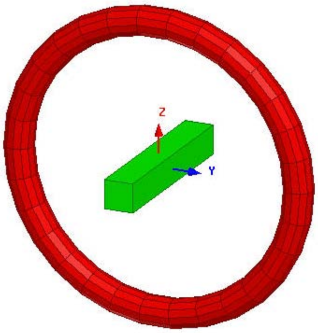
- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button

Create Coil

- **Create Profile for Sweep**
  - Select the menu item ***Draw > Regular Polygon***
    1. Using the coordinate entry fields, enter the center position
       - X: **0**, Y: **5**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.5**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Number of Segments: **12**
    4. Press **OK**
  - Change the name of resulting object to **Coil**
- **Sweep Profile**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window
    1. Sweep Axis: **X**
    2. Angle of Sweep: **360 deg**
    3. Number of Segments: **30**
    4. Press **OK**
  - Change material of resulting object to **Copper**
  - Change its Color and transparency if desired

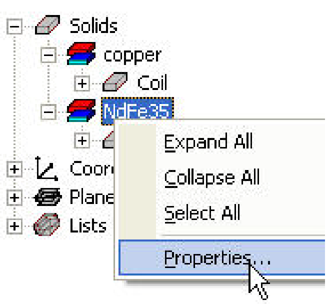
Create Magnet

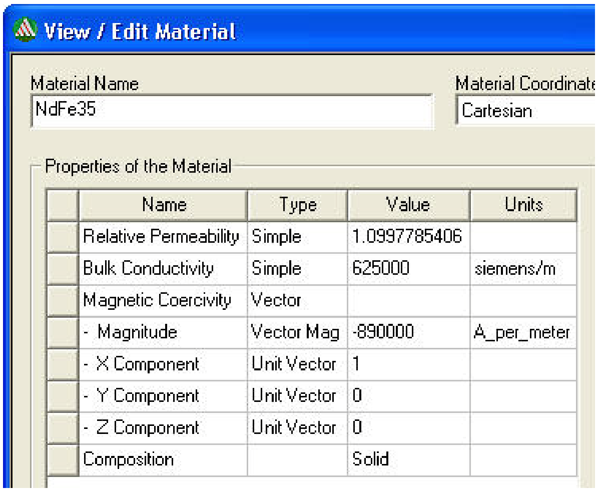
- **Create permanent Magnet**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-3**, Y: **-0.5**, Z: **-0.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **6**, dY: **1**, dZ: **1**, Press the **Enter** key
  - Change the name of the resulting object to **Magnet**
  - Change material of the object to **NdFe35**
  - Change its color and transparency if desired



Check Magnetization Direction

- As we have applied magnetic material **"NdFe35"** to the object **Magnet,** it is important to check the direction in which the material is magnetized.
- **To Check Properties of Material**
  - Right click on **NdFe35** from the history tree and select **Properties**
  - In Select Definition window, select **View/Edit Materials**
    - In View/Edit material window
      1. Ensure X Component is set to **1**
      2. Ensure Y and Z Component is set to **0**





Name	Type	Value	Units
Relative Permeability	Simple	1.0997785406	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-890000	A_per_meter
- X Component	Unit Vector	1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Composition		Solid	

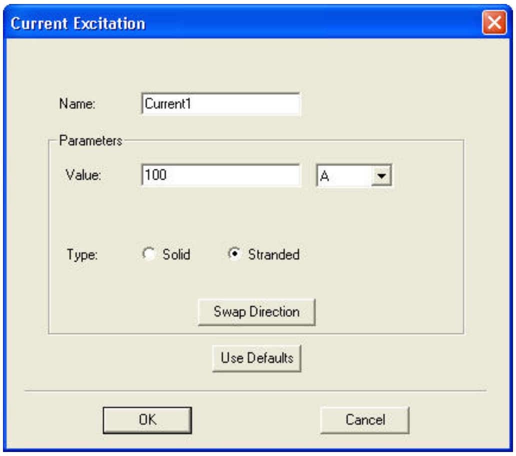
- In Maxwell All materials are magnetized in X Direction. Users can change the direction as needed

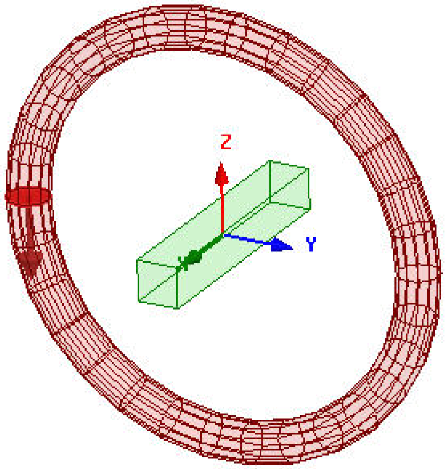
Create Coil Terminal

- **To Create Coil terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
    1. Section Plane: **XY**
    2. Press **OK**
  - Change the name of the resulting sheet to **Terminal**
- **Separate Sheets**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheet**
  - Select the sheet **Terminal_Separate1** from the history tree
  - Select the menu item ***Edit > Delete***

Assign Excitation

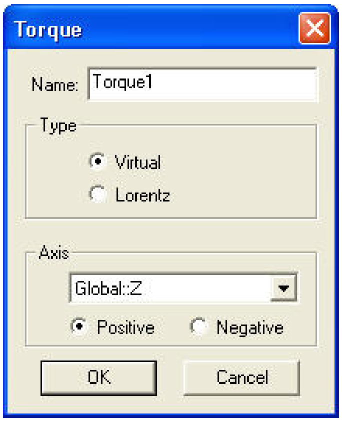
- **To Assign Excitation**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **100 A**
    3. Type: **Stranded**
    4. Press **OK**





Assign Torque Calculation

- **To Assign Torque Calculation**
  - Select the object **Magnet** from the history tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Torque***
  - In Torque window,
    1. Name: **Torque1**
    2. Type: **Virtual**
    3. Axis: **Global::Z**
    4. Positive: ☑ **Checked**
    5. Press **OK**



Rotate Coil

- **To Rotate the Coil**
  - Press **Ctrl** and select the object **Coil** and **Terminal** from the history tree
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **45 deg**
    3. Press **OK**

Create Region

- **To Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - Value: **100**
    3. Press **OK**

Analysis Setup

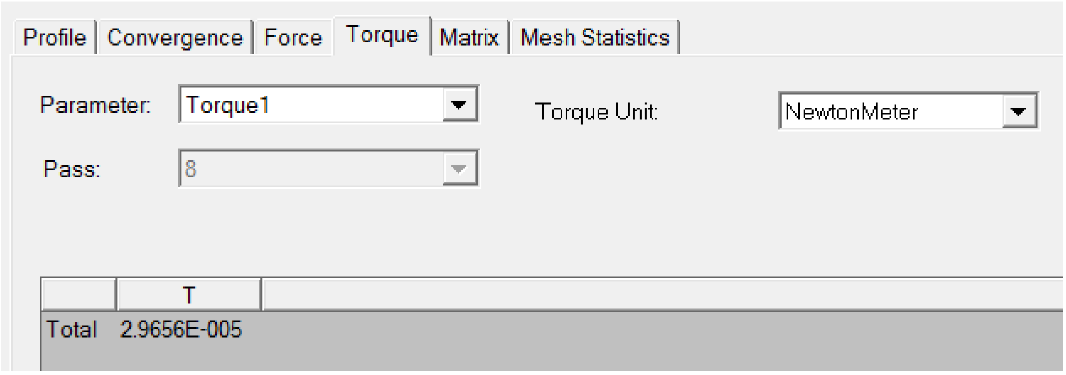
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** to accept default settings

Analyze

- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

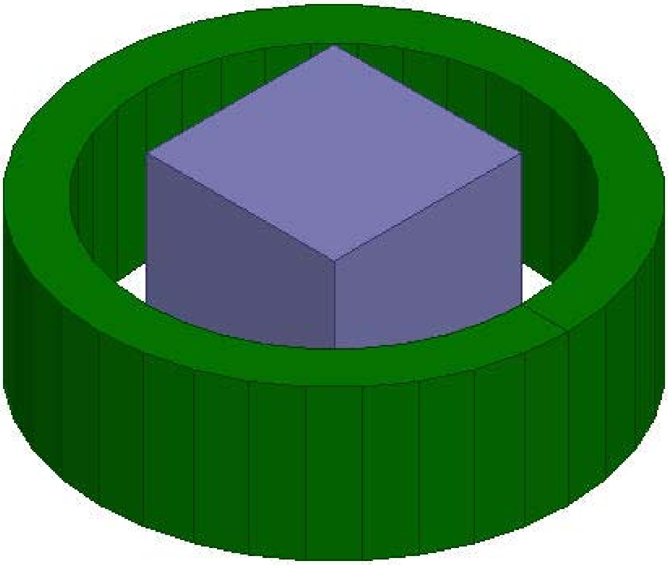
View Calculation results

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Torque** values
       - Select the **Torque** tab



Parametric study using a coil and Iron slug

- A Magnetostatic problem will be used to demonstrate the setup of a parametric solution using Optimetrics. The coil current and the dimensional length of an iron slug will be varied and the force on the slug will be observed.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button

Create Coil

Create Regular Polyhedron

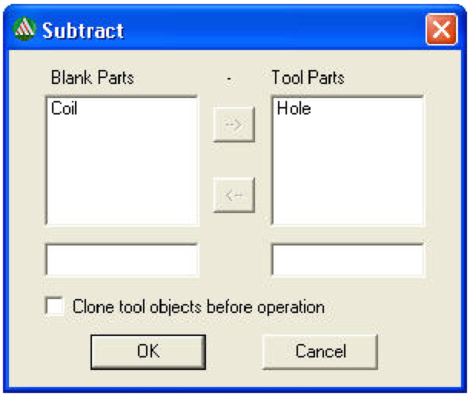
- Select the menu item ***Draw > Regular Polyhedron***
  1. Using the coordinate entry fields, enter the center of the base
     - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the radius
     - dX: **1.25**, dY: **0**, dZ: **0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the height
     - dX: **0**, dY: **0**, dZ:**0.8**, Press the **Enter** key
  4. Number of Segments: **36**
- Change the name of the Object to **Coil**
- Change the material of the object to **Copper**
- Change the color if desired

Create Second Polyhedron

- Select the menu item ***Draw > Regular Polyhedron***
  1. Using the coordinate entry fields, enter the center of the base
     - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the radius
     - dX: **1**, dY: **0**, dZ: **0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the height
     - dX: **0**, dY: **0**, dZ:**0.8**, Press the **Enter** key
  4. Number of Segments: **36**
- Change the name of the Object to **Hole**

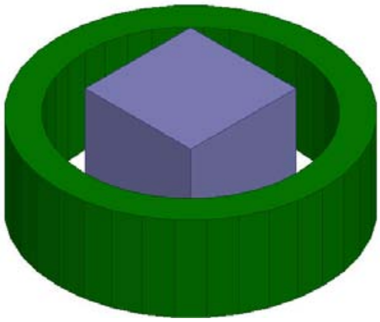
Subtract Objects

- Press **Ctrl** and select the objects **Coil** and **Hole** from the history tree
- Select the menu item, ***Modeler > Boolean > Subtract***
  1. Blank Parts: **Coil**
  2. Tool Parts: **Hole**
  3. Click the **OK** button



Create Slug

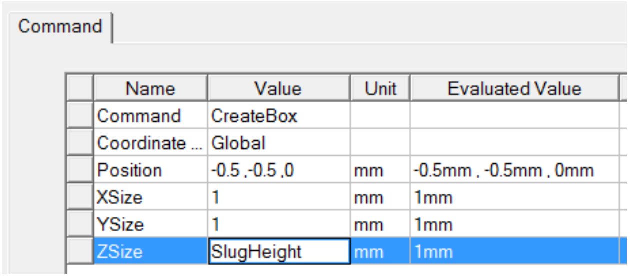
- **To Create Iron Slug**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.5**, Y: **-0.5**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **1**, dY: **1**, dZ: **1**, Press the **Enter** key
  - Change the name of the resulting object to **Slug**
  - Change material of the object to **Iron**
  - Change its color and transparency if desired



  - **Note:** The material properties for iron has a linear permeability. This means that no non-linear BH curve is being used in this example.

Add Parameter SlugHeight

- **To Add parameter for Slug Height**
  - Expand the history tree for the object **Slug** and double click on **CreateBox** command
  - In Properties window,
    1. For ZSize specify value as **SlugHeight**, press Tab to accept
    2. In Add Variable window,
       - Unit Type: **Length**
       - Unit: **mm**
       - Value: **1**
       - Press **OK**



Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate ...	Global		
Position	-0.5 ,-0.5 ,0	mm	-0.5mm , -0.5mm , 0mm
XSize	1	mm	1mm
YSize	1	mm	1mm
ZSize	SlugHeight	mm	1mm

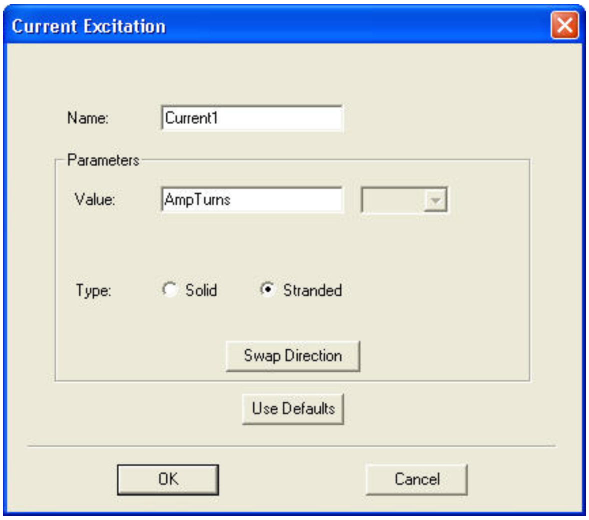
  - **Note:** By defining a variable name (SlugHeight) it becomes a design variable. The design variables are accessible by selecting menu item ***Maxwell 3D > Design Properties***

Create Coil Terminal

- **To Create Coil terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
    1. Section Plane: **YZ**
    2. Press **OK**
  - Change the name of the resulting sheet to **Terminal**
- **Separate Sheets**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheet**
  - Select the sheet **Terminal_Separate1** from the history tree
  - Select the menu item ***Edit > Delete***

Assign Excitation

- **To Assign Excitation**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **AmpTurns**
    3. Type: **Stranded**
    4. Press **OK**
  - In Add Variable window,
    1. Unit Type: **Current**
    2. Unit: **A**
    3. Value: **100**
    4. Press **OK**

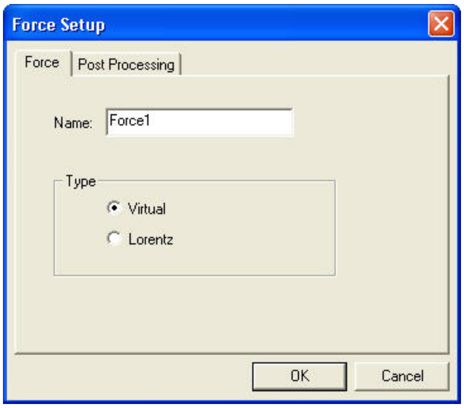


Create Region

- **To Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Padding all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - Value: **200**
    3. Press **OK**

Assign Force Calculation

- **To Assign Force Calculation**
  - Select the object **Slug** from the history tree
  - Select the menu item *Maxwell 3D > Parameters > Assign > Force*
  - In Torque window,
    1. Name: **Force1**
    2. Type: **Virtual**
    3. Press **OK**



Analysis Setup

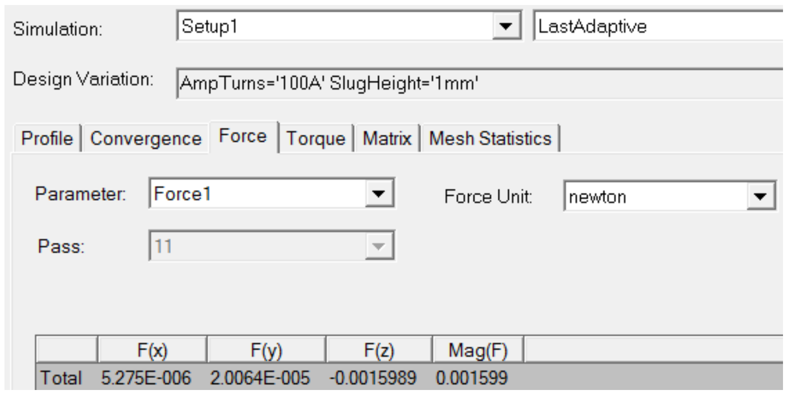
- **To create an analysis setup:**
  - Select the menu item *Maxwell 3D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** tab
       - Maximum Number of Passes: **15**
    2. Click the **OK** button

Analyze

- **To run Nominal Solution**
  - Select the menu item *Maxwell 3D > Analyze All*

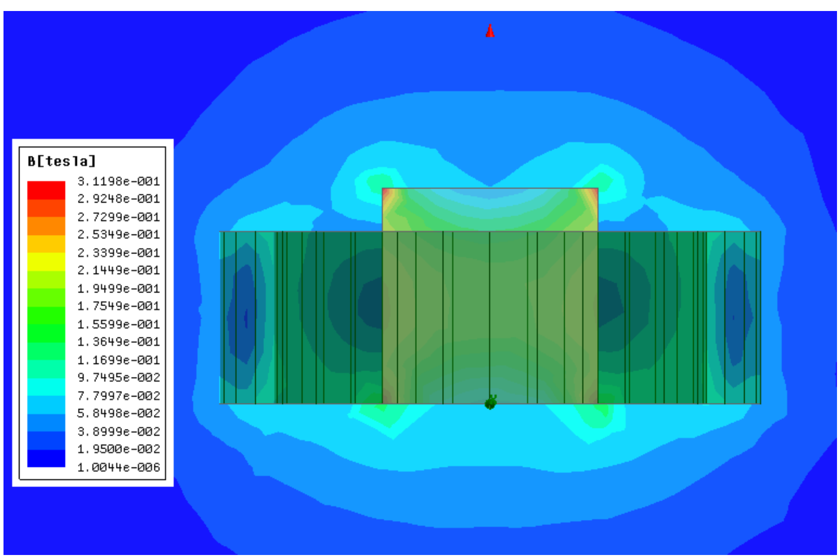
View Calculation results

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    1. To view **Force** values
       - Select the **Force** tab



Plot B Field

- **To Plot Mag_B**
  - Expand the history tree for Planes and select the plane **Global:YZ**
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    1. Press **Done**

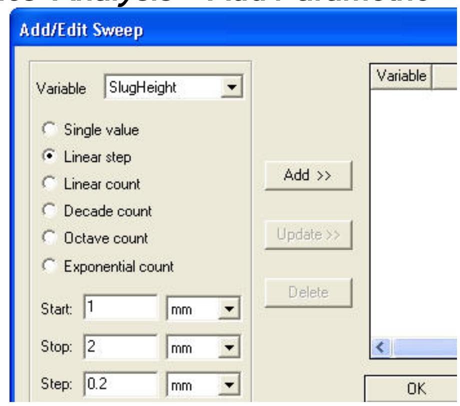


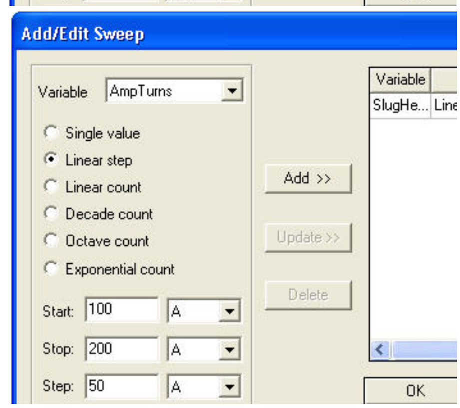
  - The results show about 0.2 Tesla field in the Center of the Slug which is well within the linear region of the BH curve for most steels

Create a Parametric solution

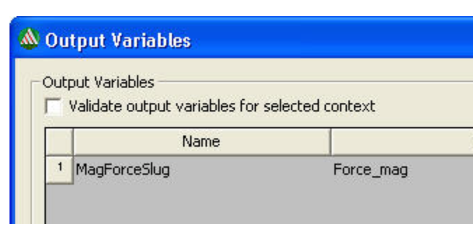
To create Parametric Sweep

- Select the menu item *Maxwell 3D > Optimetrics Analysis > Add Parametric*
- In Setup Sweep Analysis window,
  1. Select **Add**
  2. In Add/Edit Sweep window,
     - Variable: **SlugHeight**
     - Linear Step: ☑ **Checked**
     - Start: **1mm**
     - Stop: **2mm**
     - Step: **0.2mm**
     - Select **Add**
     - Change Variable to **AmpTurns**
     - Linear Step: ☑ **Checked**
     - Start: **100 A**
     - Stop: **200 A**
     - Step: **50 A**
     - Select **Add**
     - Press **OK**

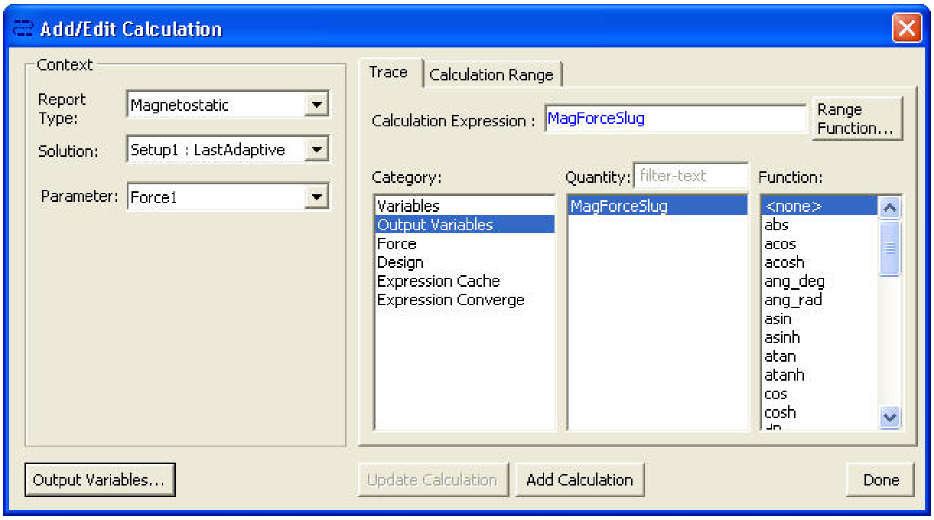




  3. Change tab to **Table** inspect the combination of solutions that have been created. There should be 18 solutions since we defined 6 variations of SlugHeight and 3 variations of AmpTurns.
  4. Select **Calculations** tab to set the output parameters
  5. Select **Setup Calculations**
  6. In Add/Edit Calculations window, Select **Output Variables**
  7. In Output Variables window,
     - Parameter: **Force1**
     - Category: **Force**
     - Quantity: **Force_mag**
     - Select **Insert Into Expression**
     - Name: **MagForceSlug**
     - Select **Add**
     - Select **Done** to close window

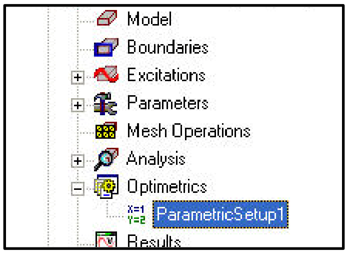


8. In Add/Edit Variable window,
   - Parameter: **Force1**
   - Category: **Output Variables**
   - Quantity: **MagForceSlug**
   - Select **Add Calculation**
   - Select **Done** to close window
9. Select **Options** tab on Setup Sweep Analysis window
   - Save Fields and Mesh: ☑ **Checked**
   - Copy geometrically equivalent meshes: ☑ **Checked**
10. Click **OK** to close



Solve the Parametric problem

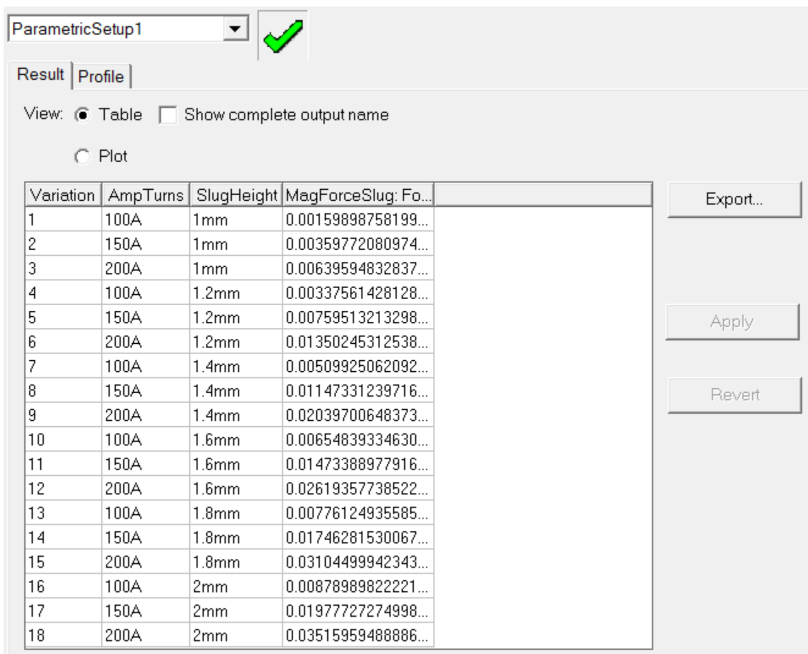
- **To Solve Parametric Setup**
  - In Project manager window, expand the tree for **Optimetrics**
  - Right click on the tab **Parametric Setup1** and select **Analyze**



  - **Note:** the solving criteria is taken from the nominal problem, **Setup1** . Each parametric solution will re-mesh if the geometry has changed or the energy error criteria is not met as defined in **Setup1**.

View Analysis Results

- **To view the Results of Parametric Sweep**
  - In Project manager window, expand the tree for **Optimetrics**
  - Right click on the tab **Parametric Setup1** and select **View Analysis Results**
  - In Post Analysis Display window,
    1. View: Select **Table** to view results in tabular form

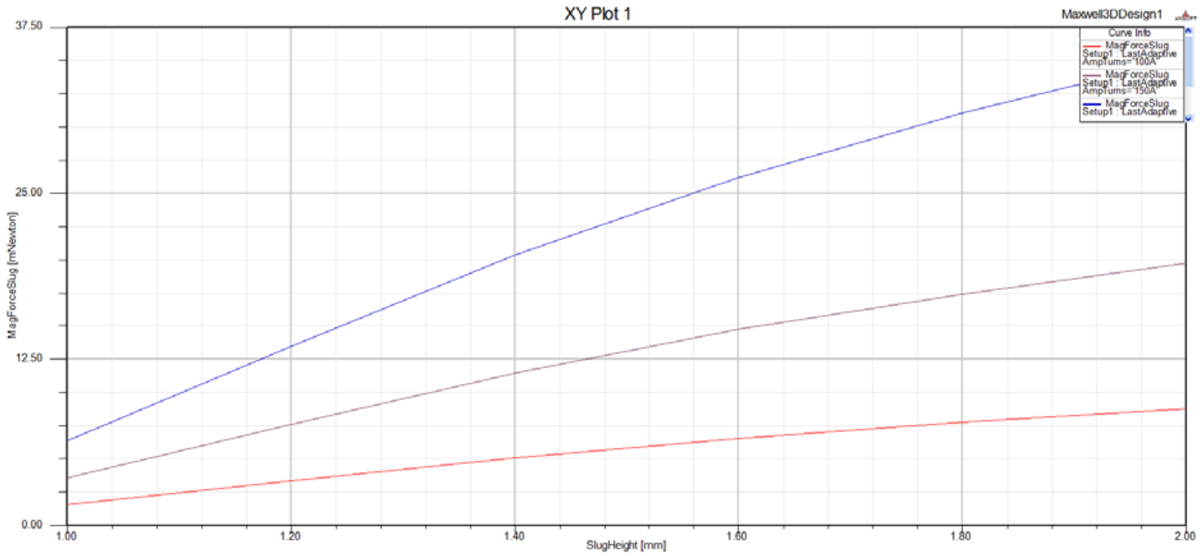


Variation	AmpTurns	SlugHeight	MagForceSlug: Fo...
1	100A	1mm	0.00159898758199...
2	150A	1mm	0.00359772080974...
3	200A	1mm	0.00639594832837...
4	100A	1.2mm	0.00337561428128...
5	150A	1.2mm	0.00759513213298...
6	200A	1.2mm	0.01350245312538...
7	100A	1.4mm	0.00509925062092...
8	150A	1.4mm	0.01147331239716...
9	200A	1.4mm	0.02039700648373...
10	100A	1.6mm	0.00654839334630...
11	150A	1.6mm	0.01473388977916...
12	200A	1.6mm	0.02619357738522...
13	100A	1.8mm	0.00776124935585...
14	150A	1.8mm	0.01746281530067...
15	200A	1.8mm	0.03104499942343...
16	100A	2mm	0.00878989822221...
17	150A	2mm	0.01977727274998...
18	200A	2mm	0.03515959488886...

Graph the Force vs. AmpTurns vs. SlugHeight

- **To Create a Plot**
  - Select the menu item ***Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot***
  - In report window,
    1. Parameter: **Force1**
    2. Parametric Sweep: **SlugHeight**
    3. X : **Default**
    4. Y :
       - Category: **Output Variables**
       - Qantity: **MagForceSlug**
    5. Select **New Report**

- The Plot should appear as shown in below
- Right click on the plot and select export to export the data in text file



Create a Table of SlugHeight, Force, and AmpTurns

- **To Create a Table**
  - Select the menu item ***Maxwell 3D > Results > Create Magnetostatic Report > Data Table***
  - In report window,
    1. Parameter: **Force1**
    2. Parametric Sweep: **SlugHeight**
    3. X : **Default**
    4. Y :
       - Category: **Output Variables**
       - Qantity: **MagForceSlug**
    5. Select **New Report**

Data Table 1 — Maxwell3DDesign1

	SlugHeight [mm]	MagForceSlug [mNewton] Setup1 : LastAdaptive AmpTurns='100A'	MagForceSlug [mNewton] Setup1 : LastAdaptive AmpTurns='150A'	MagForceSlug [... Setup1 : LastAd... AmpTurns='200A'
1	1.000000	1.598988	3.597721	6.395948
2	1.200000	3.375614	7.595132	13.502453
3	1.400000	5.099251	11.473312	20.397006
4	1.600000	6.548393	14.733890	26.193577
5	1.800000	7.761249	17.462815	31.044999
6	2.000000	8.789898	19.777273	35.159595

Inductor using Transient Source

- This exercise will discuss how to use transient sources as the excitation for an inductor coil.

Create Design

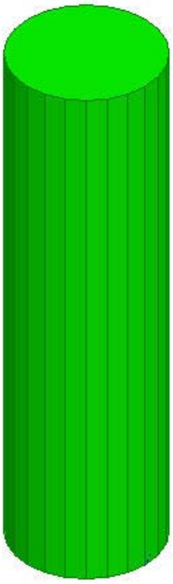
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon
  - Change the name of the Design to **BE_Trans**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button

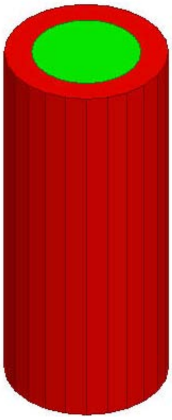
Create Core

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **2**, dY: **2**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**20**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Core**
  - Change the material of the object to **ferrite**



Create Coil

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **3**, dY: **3**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**20**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Coil**
  - Change the material of the object to **copper**
- **Subtract Objects**
  - Press **Ctrl** and select the objects **Coil** and **Core** from the history tree
  - Select the menu item, ***Modeler > Boolean > Subtract***
    1. Blank Parts: **Coil**
    2. Tool Parts: **Core**
    3. Clone tool objects before operation: ☑ **Checked**
    4. Click the **OK** button



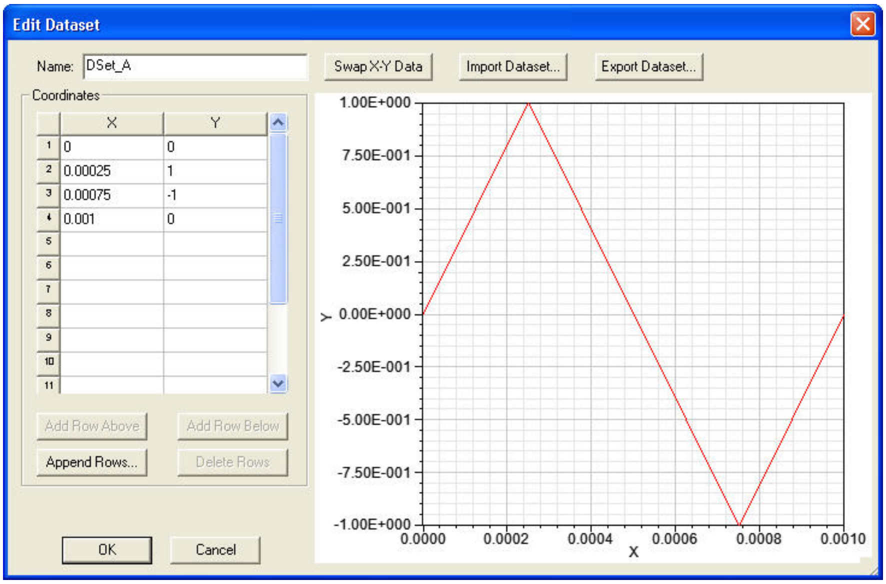
Create Coil Terminal

- **To Create Coil terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
    1. Section Plane: **YZ**
    2. Press **OK**
  - Change the name of the resulting sheet to **Terminal**
- **Separate Sheets**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheet**
  - Select the sheet **Terminal_Separate1** from the history tree
  - Select the menu item ***Edit > Delete***

Assign Excitation

Specify Dataset

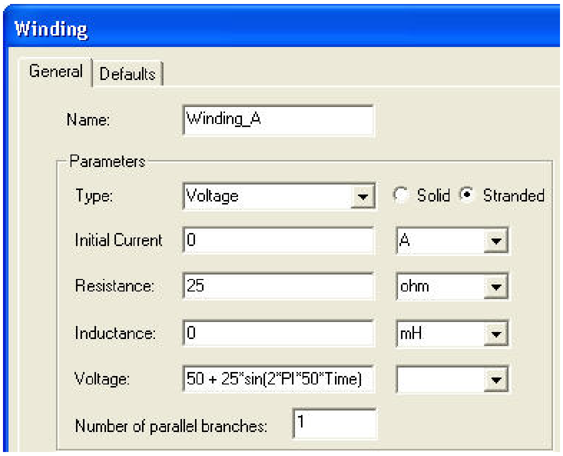
- The excitation for this problem will be a voltage source with a 1KHz triangular wave superimposed on a 50 Hz sine wave that has a 50 volt DC offset.Triangular wave will be specified through a design dataset
- Select the menu item ***Maxwell 3D > Design Datasets***
- In Datasets window, select **Add**
- In Add Dataset window,
  - Name: **DSet_A**
  - Coordinates:
    1. X1 = 0 Y1 = 0
    2. X2 = 250e-6 Y2 = 1
    3. X3 = 750e-6 Y3 = -1
    4. X4 = 1e-3 Y4 = 0
  - Select **OK** and **Done**



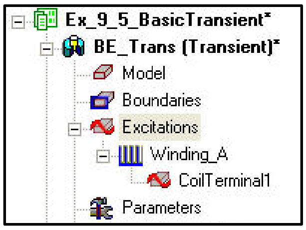
To Assign Coil Terminal

- Select the sheet **Terminal** from the history tree
- Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
- In Current Excitation window,
  1. Name: **CoilTerminal1**
  2. Number of Conductors: **150**
  3. Press **OK**

- **To Add Winding**
  - Select the menu item ***Maxwell 3D > Excitations > Add Winding***
  - In Winding window,
    1. Name: **Winding_A**
    2. Type: **Voltage**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Resistance: **25 ohm**
    6. Inductance: **0 H**
    7. Voltage: **50 + 25*sin(2*PI*50*Time) + 5*pwl_periodic(DSet_A, Time)**
    8. Press **OK**



  - **Note:** The expression specified for Voltage has three different components
    1. The first term is a 50 V DC offset
    2. The second term is a 25 Vp-p, 50 Hz sine wave
    3. The third term is a 5 Vp-p, 1 KHz triangular wave
- **Add Terminal to Winding**
  - In Project manager window, expand the tree for **Excitations**
  - Right click on the tab **CoilTerminal1** and select **Add to Winding**
  - In Add to Winding window,
    1. Select **Winding_A**
    2. Press **OK**



  - **Note:** An insulation boundary is not needed between the core and the coil because the ferrite core has a conductivity = 0.01S/m which below the default conductor/insulation threshold of 1S/m.

Create Region

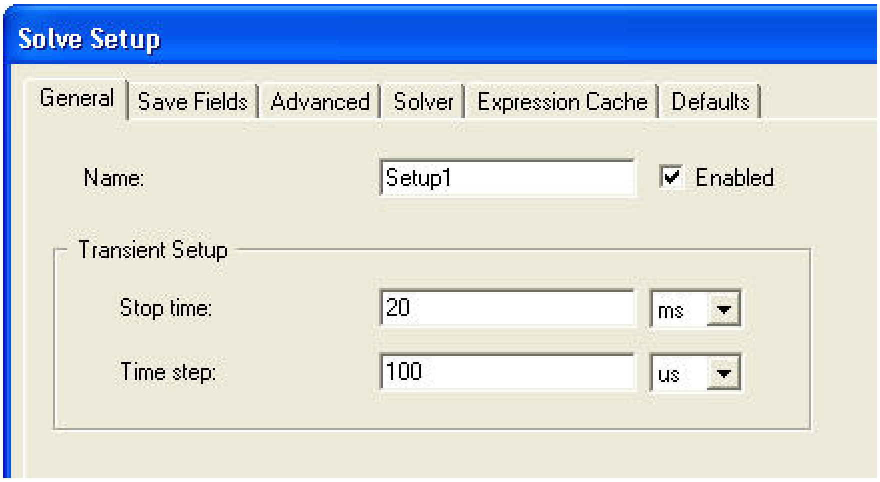
- **To Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - Value: **500**
    3. Press **OK**

Apply Mesh Operations

- The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created.
- **To Apply Mesh Operations for Core**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length1**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**
- **To Apply Mesh Operations for Coil**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length2**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** tab
       - Stop Time: **20 ms**
       - Time Step: **100 us**
    2. Click the **OK** button



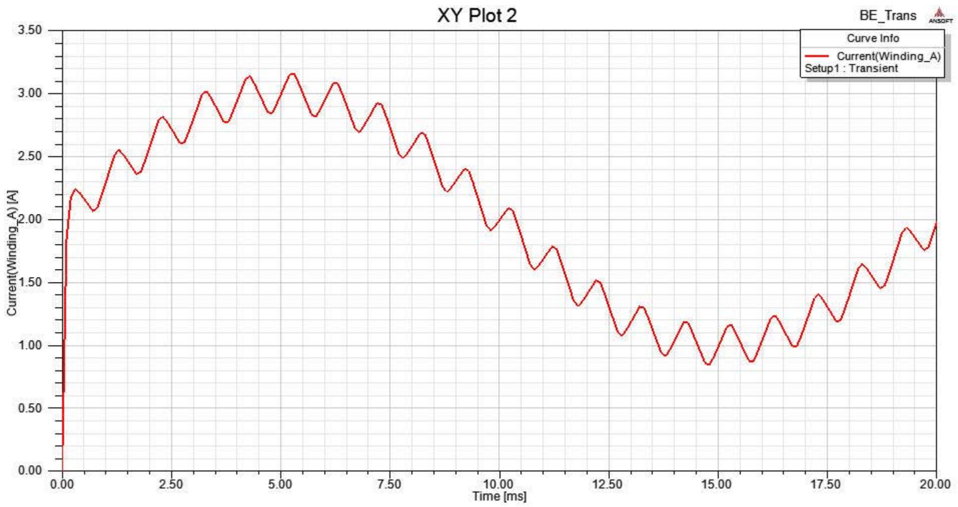
Analyze

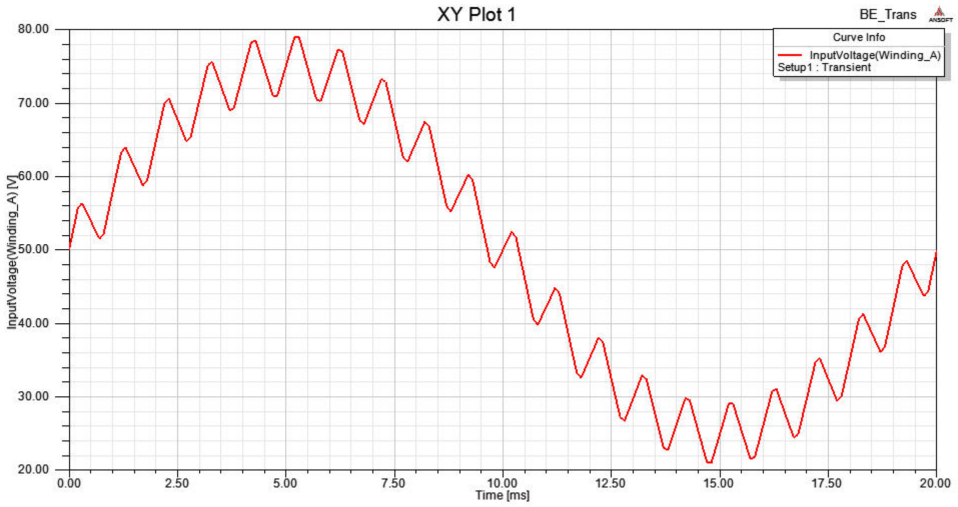
- **To run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Plot the Voltage and Current

To Create Plots

- Select the menu item *Maxwell 3D > Results > Create Transient Report > Rectangular Plot*
- In Reports window,
  1. Category: **Winding**
  2. Quantity: **Current (Winding_A)**
  3. Select **New Report**
  4. Change Quantity to **InputVoltage (Winding_A)**
  5. Select **New Report**





Inductor using Transient Source with External Circuits

- This exercise will discuss how to link a transient source generated by an external circuit to an inductor coil.

Create Design

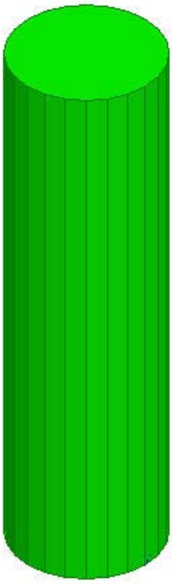
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon
  - Change the name of the Design to **BE_Trans_Ckt**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Transient**
    2. Click the **OK** button

Create Core

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **2**, dY: **2**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**20**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Core**
  - Change the material of the object to **ferrite**



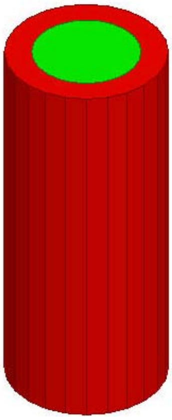
Create Coil

Create Regular Polyhedron

- Select the menu item *Draw > Regular Polyhedron*
  1. Using the coordinate entry fields, enter the center of the base
     - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the radius
     - dX: **3**, dY: **3**, dZ: **0**, Press the **Enter** key
  3. Using the coordinate entry fields, enter the height
     - dX: **0**, dY: **0**, dZ:**20**, Press the **Enter** key
  4. Number of Segments: **24**
- Change the name of the Object to **Coil**
- Change the material of the object to **copper**

Subtract Objects

- Press **Ctrl** and select the objects **Coil** and **Core** from the history tree
- Select the menu item, *Modeler > Boolean > Subtract*
  1. Blank Parts: **Coil**
  2. Tool Parts: **Core**
  3. Clone tool objects before operation: ☑ **Checked**
  4. Click the **OK** button



Create Coil Terminal

To Create Coil terminal

- Select the object **Coil** from the history tree
- Select the menu item *Modeler > Surface > Section*
  1. Section Plane: **YZ**
  2. Press **OK**
- Change the name of the resulting sheet to **Terminal**

Separate Sheets

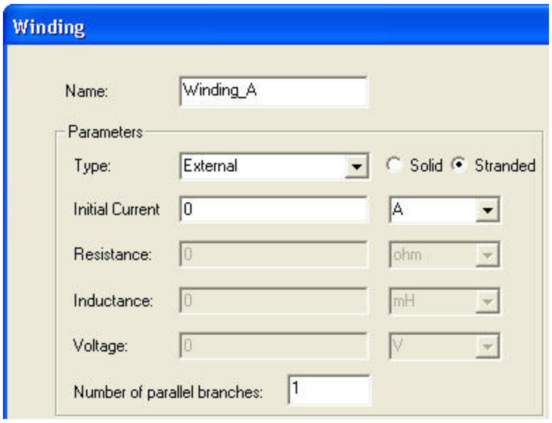
- Select the sheet **Terminal** from the history tree
- Select the menu item *Modeler > Boolean > Separate Bodies*

Delete Extra Sheet

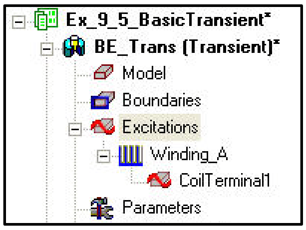
- Select the sheet **Terminal_Separate1** from the history tree
- Select the menu item *Edit > Delete*

Assign Excitation

- **To Assign Coil Terminal**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Coil Terminal***
  - In Current Excitation window,
    1. Name: **CoilTerminal1**
    2. Number of Conductors: **150**
    3. Press **OK**
- **To Add Winding**
  - Select the menu item ***Maxwell 3D > Excitations > Add Winding***
  - In Winding window,
    1. Name: **Winding_A**
    2. Type: **External**
    3. Stranded: ☑ **Checked**
    4. Initial Current: **0 A**
    5. Press **OK**



- **Add Terminal to Winding**
  - In Project manager window, expand the tree for **Excitations**
  - Right click on the tab **CoilTerminal1** and select **Add to Winding**
  - In Add to Winding window,
    1. Select **Winding_A**
    2. Press **OK**



  - **Note:** An insulation boundary is not needed between the core and the coil because the ferrite core has a conductivity = 0.01S/m which below the default conductor/insulation threshold of 1S/m.

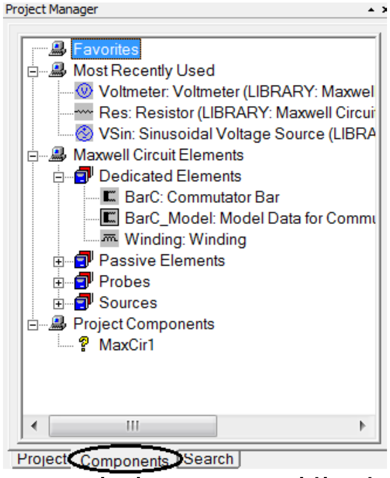
Create External Circuit

To Launch Maxwell Circuit Editor

- To access Maxwell Circuit Editor, click the Microsoft **Start** button, select **Programs** > **Ansoft > Maxwell 15> Maxwell Circuit Editor**

To Add Components

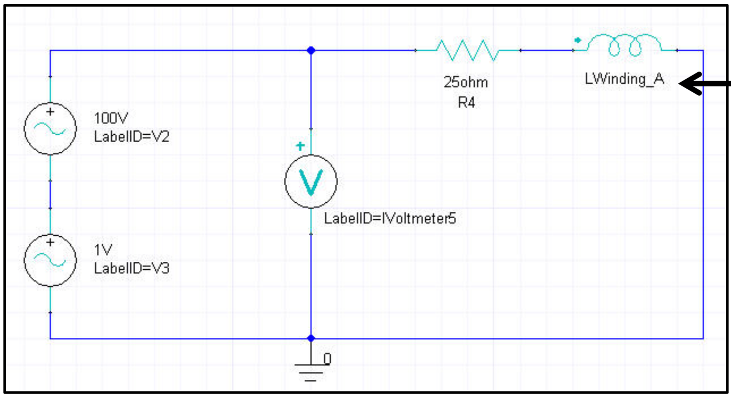
- Change the tab in Project Manager window to **Components**



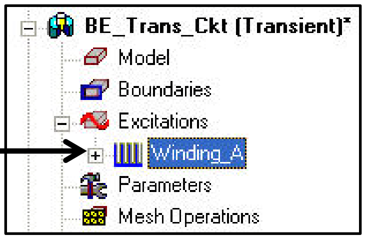
- In Project Manager window, expand the tree for ***Maxwell Circuit Elements > Dedicated Elements .*** Select the element **Winding** from the tree, drag and drop it on the worksheet
- Select the winding added in worksheet and go to the properties window,
  1. Change the name of the winding to **Winding_A**

     Note: This name has to be **exactly** the same as used in Maxwell in **Maxwell 3D > Excitations > Add Winding**
- In Project Manager window, select ***Sources > VSin*** from the tree, drag and drop it on the worksheet. Select the source added in worksheet and go to the properties window,
  1. Change the value of **Va** to **100 V**
  2. Change the value of **VFreq** to **50 Hz**
- Add another source **VSin** and change its properties as below
  1. Change the value of **Va** to **10 V**
  2. Change the value of **VFreq** to **1000 Hz**
- In Project Manager window, select ***Passive Elements > Res*** , drag and drop it on the worksheet. Select the resistor added in worksheet and go to the properties window,
  1. Change the value of **R** to **25 ohm**

- In Project Manager window, select ***Probes > Voltmeter***, drag and drop it on the worksheet
- Select the menu item ***Draw > Wire*** to draw wires
- Select the menu item ***Draw > Ground*** to add ground
- Build the circuit as shown in below image



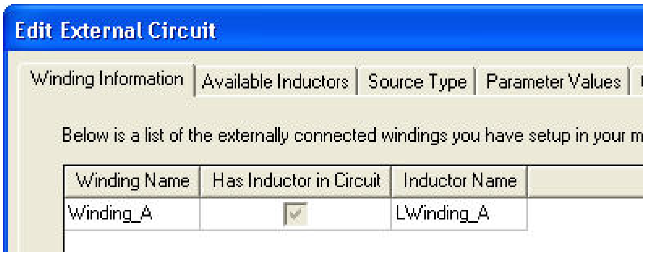
Note: Same name used: Winding_A



- **To Export Circuit**
  - Select the menu item ***Maxwell Circuit > Export Netlist***
    1. Save the file with the name **BE_Circuit.sph**
- **To Save File**
  - Select the menu item ***File > Save***
    1. Save the file with the name **BE_Circuit.amcp**

Link the circuit file to the Maxwell project

- **To Import circuit in Maxwell**
  - Return to Maxwell window
  - In Maxwell, Select the menu item ***Maxwell 3D > Excitations > External Circuit > Edit External Circuit***
  - In Edit External Circuit window,
    1. Select **Import Circuit**
    2. Browse to the file **BE_Circuit.sph** and **Open** it
    3. Press **OK**



Create Region

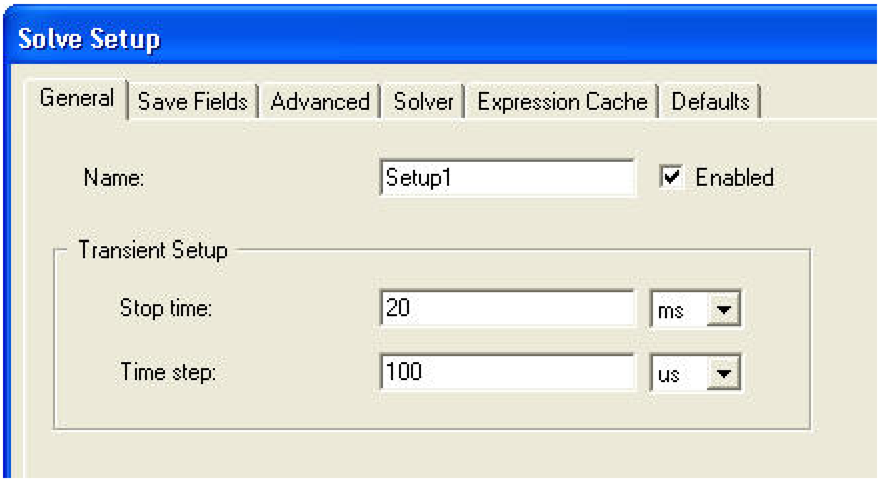
- **To Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - Value: **500**
    3. Press **OK**

Apply Mesh Operations

- The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created.
- **To Apply Mesh Operations for Core**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length1**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**
- **To Apply Mesh Operations for Coil**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length2**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** tab
       - Stop Time: **20 ms**
       - Time Step: **100 us**
    2. Click the **OK** button



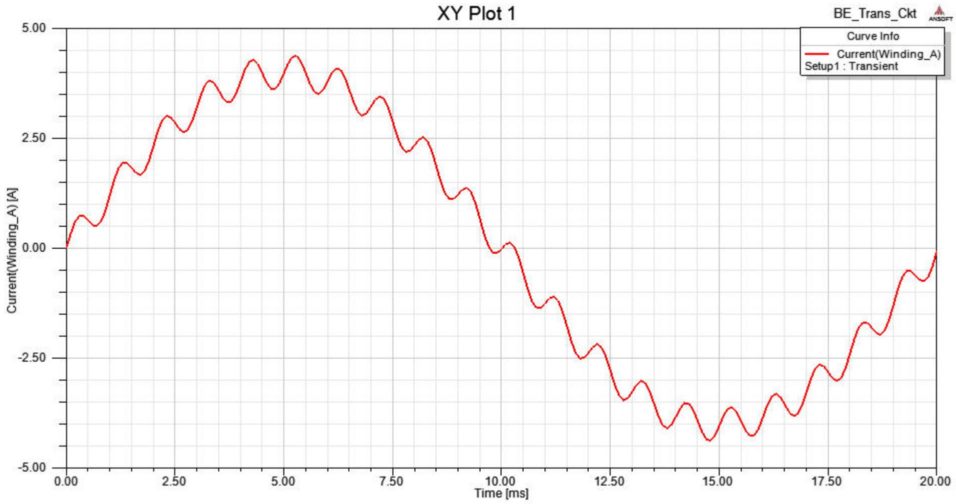
Analyze

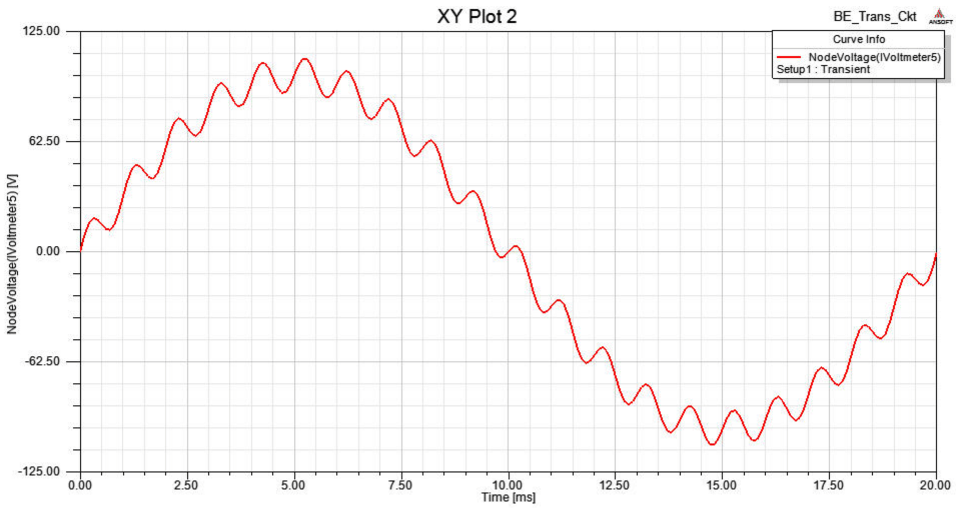
- **To run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Plot the Voltage and Current

To Create Plots

- Select the menu item *Maxwell 3D > Results > Create Transient Report > Rectangular Plot*
- In Reports window,
  1. Category: **Winding**
  2. Quantity: **Current (Winding_A)**
  3. Select **New Report**
  4. Change Category to **NodeVoltage**
  5. Quantity: **NodeVoltage**
  6. Select **New Report**





Post Processing in Maxwell 3D

- This exercise will discuss how to use the Maxwell 3D Post Processor. Field plots and calculator operations will demonstrated on an Eddy Current project. The following tasks will be performed:
  - Plot the mesh on the core and coil
  - Plot of MagB on a plane
  - Plot of B_vector on a plane
  - Plot of MagH along a line
  - Create a table of MagH along a line
  - Calculate average MagB in an object
  - Verify DivB = 0
  - Calculate flux flowing through the top surface of the core
  - Calculate loss in the conductor
  - Calculate net and total current flowing in the conductor
  - Export field results to a file.

Create Design

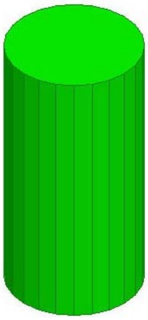
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon
  - Change the name of the Design to **Post_Exercise**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Eddy Current**
    2. Click the **OK** button

Create Core

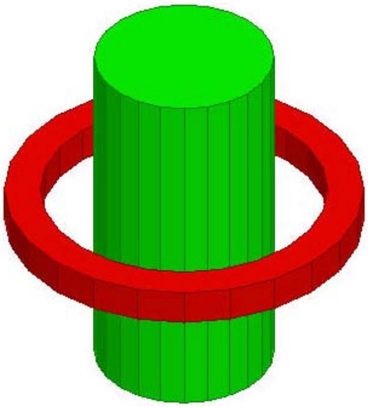
- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z:**-10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0**, dY: **5**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**20**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Core** and material to **iron**



Create Coil

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **10**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**2**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Coil** and material to **copper**
- **Create Hole**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **8**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**2**, Press the **Enter** key
    4. Number of Segments: **24**
  - Change the name of the Object to **Hole**

- **Subtract Objects**
  - Press **Ctrl** and select the objects **Coil** and **Hole** from the history tree
  - Select the menu item, ***Modeler > Boolean > Subtract***
    1. Blank Parts: **Coil**
    2. Tool Parts: **Hole**
    3. Click the **OK** button



Create Coil Terminal

- **To Create Coil terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
    1. Section Plane: **YZ**
    2. Press **OK**
  - Change the name of the resulting sheet to **Terminal**
- **Separate Sheets**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheet**
  - Select the sheet **Terminal_Separate1** from the history tree
  - Select the menu item ***Edit > Delete***

Create a line for plotting of the fields

- **To Create a Line**
  - Select the menu item ***Draw > Line***
    1. Using the coordinate entry fields, enter the first vertex
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second vertex
       - X: **0**, Y: **0**, Z: **20**, Press the **Enter** key
    3. Press **Enter** to exit line creation

Assign Excitation

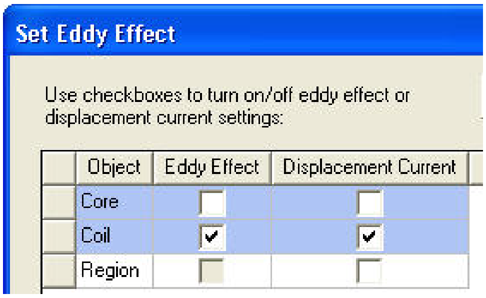
- **To Assign Excitation**
  - Select the sheet **Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **100 A**
    3. Phase: **0**
    4. Type: **Solid** (eddy effects can only be determined in solid sources)
    5. Direction: **Point out of terminal**
    6. Press **OK**

Create Region

- **To Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - Value: **100**
    3. Press **OK**

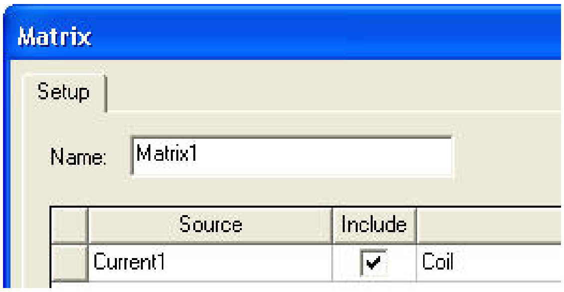
Set Eddy Effect

- **To Set Eddy Calculations**
  - Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effect***
  - In Set Eddy Effect window,
    1. Core
       - Eddy Effects: ☐ **Unchecked**
    2. Press **OK**



Setup the Impedance Calculation

- **To Setup Impedance Calculations**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Current1:
       - Include: ☑ **Checked**
    2. Press **OK**

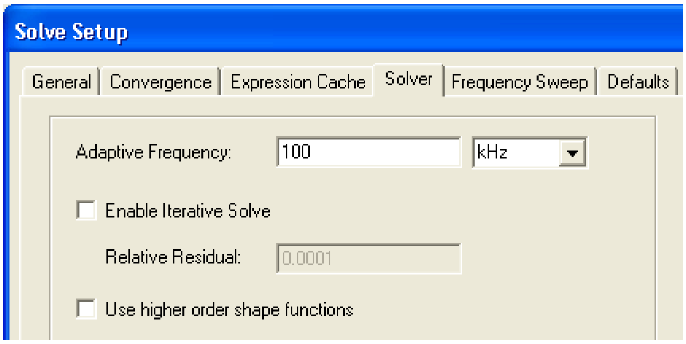


Apply Mesh Operations

- In order to create a fine initial mesh, several mesh operation will be performed.
- **To Apply Mesh Operations for Core**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length1**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**
- **To Apply Mesh Operations for Coil**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Length2**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**

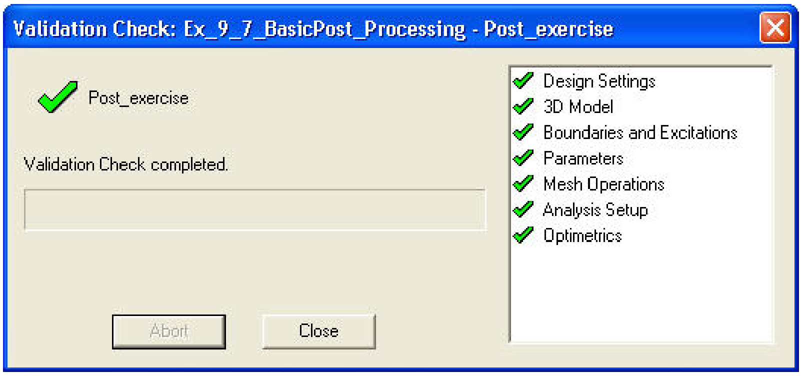
Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **Solver** tab
       - Adaptive Frequency: **100 kHz**
    2. Click the **OK** button



Validation Check

- **To Check the Validation of Case**
  - Select the menu item ***Maxwell 3D > Validation Check***

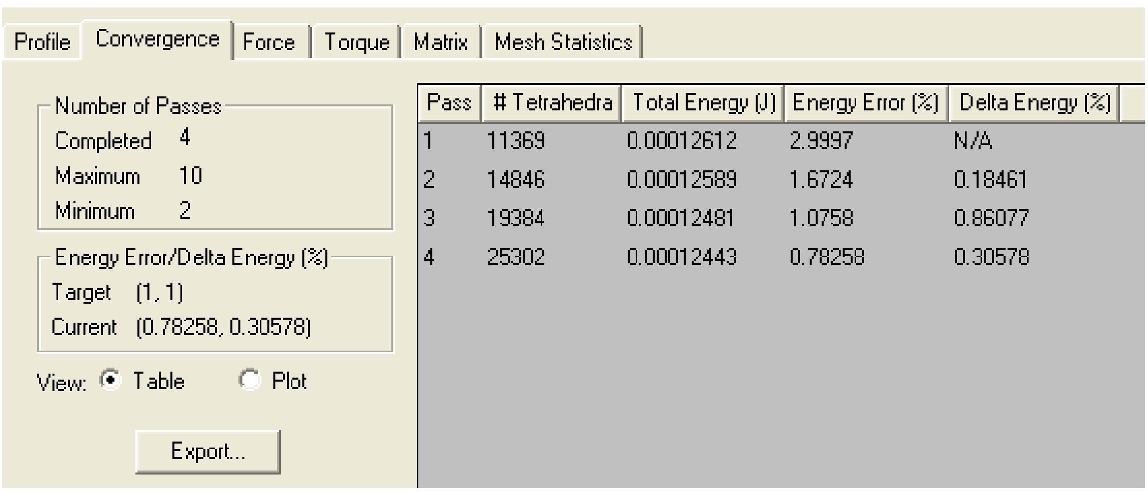


Analyze

- **To run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

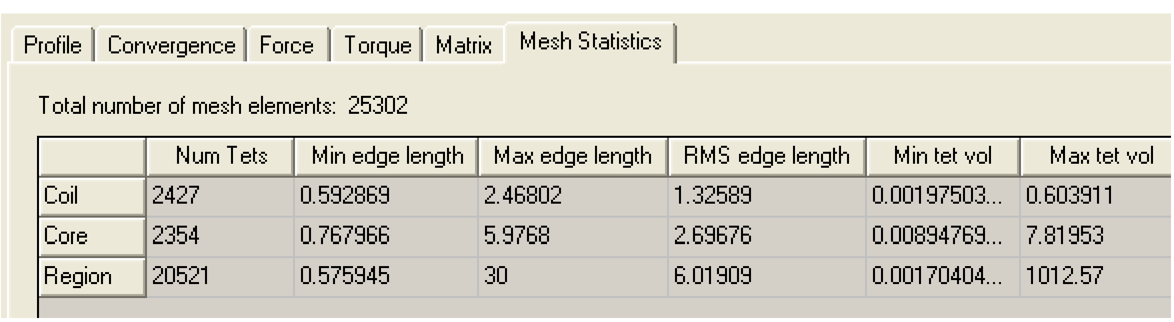
Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.



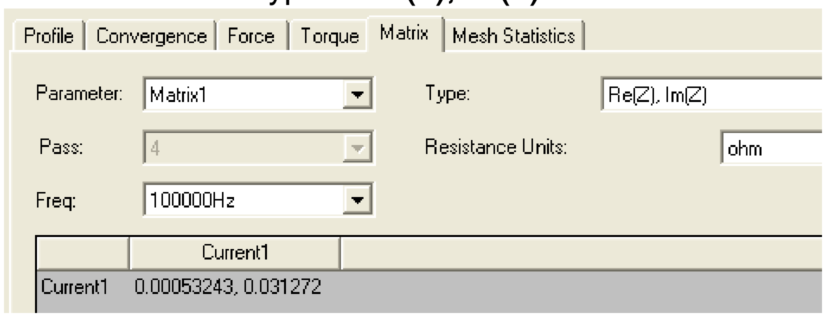
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	11369	0.00012612	2.9997	N/A
2	14846	0.00012589	1.6724	0.18461
3	19384	0.00012481	1.0758	0.86077
4	25302	0.00012443	0.78258	0.30578

- To View Mesh information
  1. Click **Mesh Statistics** Tab

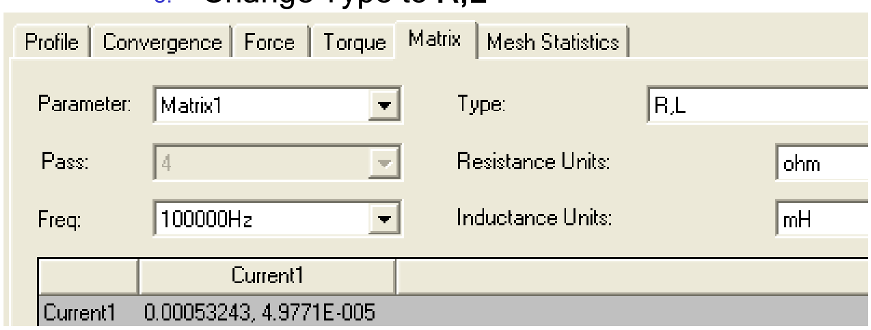


	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol
Coil	2427	0.592869	2.46802	1.32589	0.00197503...	0.603911
Core	2354	0.767966	5.9768	2.69676	0.00894769...	7.81953
Region	20521	0.575945	30	6.01909	0.00170404...	1012.57

- To View Impedance values
    1. Click **Matrix** tab
    2. Set Type to **Re(Z), Im(Z)**



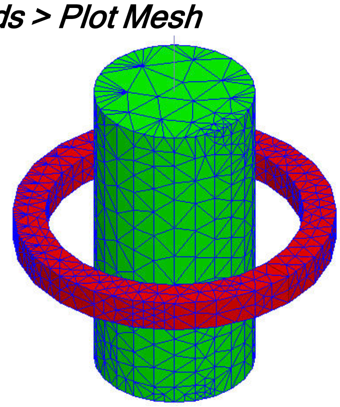
- **Note:** the imaginary term of the matrix includes both the inductive and capacitive reactance are reported in Ohms.
    3. Change Type to **R,L**



- **Note:** Imaginary term of the matrix includes only the inductance and is reported in Ohms and Henries

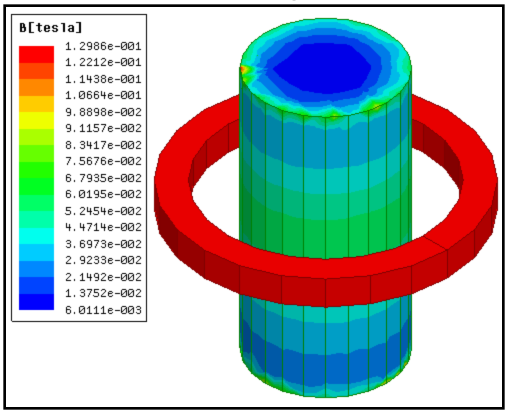
Plot Mesh

- **To Plot Mesh on Core and Coil**
    - Press **Ctrl** and select the objects **Core** and **Coil**
    - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
    - In Create Mesh Plot window,
        1. Press **Done**



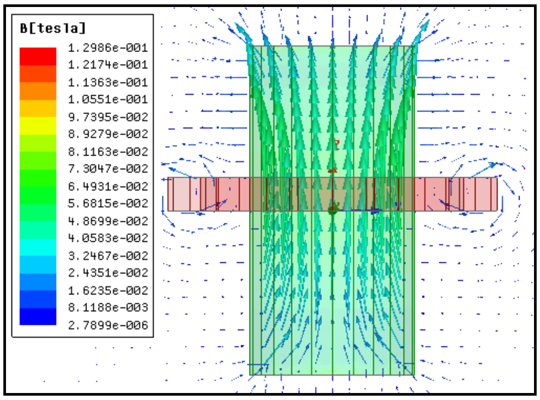
Plot Mag_B on Core

- **To Plot Mag_B**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



Plot Vector_B on YZ Plane

- **To Plot Vector_B on a Plane**
  - Expand the tree for Planes in History tree and select the pane **Global:YZ**
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Vector_B***
  - In Create Field Plot window
    1. Press **Done**
- **To Modify Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Marker/Arrow** tab
       - Arrow Options
         1. Size: **Set to appropriate value**
    2. **Plots** tab
       - Vector Plot
         1. Spacing: **Minimum**
         2. Min: **1**
         3. Max: **3**
    3. Press **Apply** and **Close**



Plot Mag_H along a Line

To Plot Mag_H

- Select the menu item *Maxwell 3D > Results > Create Field Reports > Rectangular Plot*
- In Report window.
  1. Geometry: **Polyline1**
  2. X : **Default**
  3. Category: **Calculator Expressions**
  4. Quantity: **Mag_H**
  5. Select **New Report**

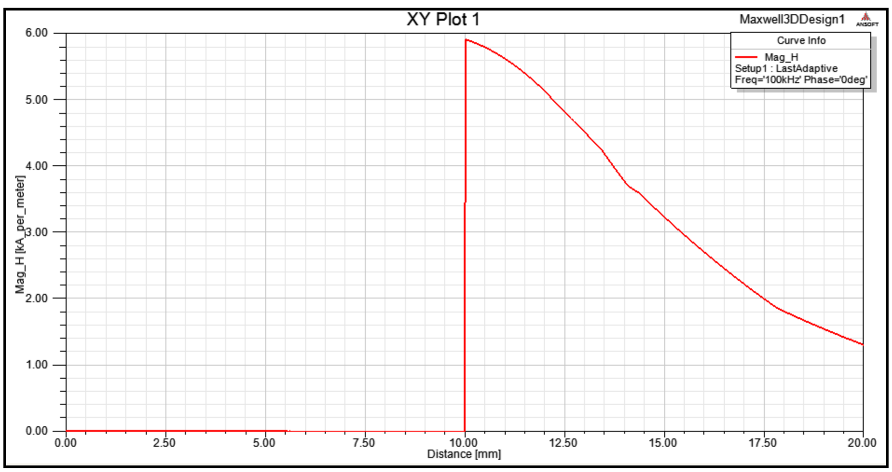


Table of Mag_H along a Line

- Select the menu item *Maxwell 3D > Results > Create Field Reports > Data Table*
  - In Report window.
    1. Geometry: **Polyline1**
    2. X : **Default**
    3. Category: **Calculator Expressions**
    4. Quantity: **Mag_H**
    5. Select **New Report**

	Distance [mm]	Mag_H [kA_per_meter] Setup1 : LastAdaptive Freq='100kHz' Phase='0deg'
1	0.000000	0.009419
2	0.020000	0.009427
3	0.040000	0.009436
4	0.060000	0.009444
5	0.080000	0.009452
6	0.100000	0.009460
7	0.120000	0.009468

Calculate Average Mag_B in an Object

- **To Calculate Mag_B Over a Volume**
  - Select the menu item *Maxwell 3D > Fields > Calculator*
    1. Select **Input > Quantity > B**
    2. Select **General > Complex > Real**
    3. Select **Vector > Mag**
    4. Select **Input > Geometry**
       - Volume: ☑ **Checked**
       - Select **Core** from the list
       - Press **OK**
    5. Select **Scalar >** **Integrate**
    6. Select **Input > Number**
       - Scalar: ☑ **Checked**
       - Value: **1**
       - Press **OK**
    7. Select **Input > Geometry**
       - Volume: ☑ **Checked**
       - Select **Core** from the list
       - Press **OK**
    8. Select **Scalar >** **Integrate**
    9. Select **General > /** (Divide)
    10. Select **Output > Eval**
  - The average value of flux density calculated is around **0.036 Tesla**

```
Scl : 0.0369052449051871
Scl : /(Integrate(Volume(Core), Mag(Real(<Bx,By,Bz>))), Integrate(Volume(Core), 1))
```

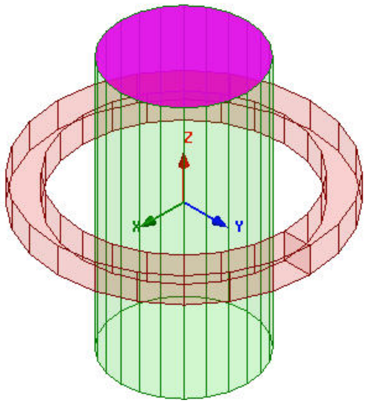
Calculate DivB = 0

- **To Calculate DivB**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **General > Complex > Real**
    3. Select **Vector > Divg**
    4. Select **Input > Geometry**
       - Volume: ☑ **Checked**
       - Select **Core** from the list
       - Press **OK**
    5. Select **Scalar >** $\int$ **Integrate**
    6. Select **Output > Eval**
  - The divergence of B is approximately zero

```
Scl : 1.81962671153383E-007
Scl : Integrate(Volume(Core), Divg(Real(<Bx,By,Bz>)))
```

Calculate flux flowing out of the top surface of the core

- **To Calculate flux**
  - Select the menu ***Edit > Select > Faces*** or press **F** from keyboard
  - Select the menu item ***Modeler > List > Create > Face List***
  - Select the top face of the **Core** as shown in below image
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity > B**
    2. Select **General > Complex > Real**
    3. Select **Input > Geometry**
       - Surface: ☑ **Checked**
       - Select **Facelist1** from the list
       - Press **OK**
    4. Select **Vector > Normal**
    5. Select **Scalar >** $\int$ **Integrate**
    6. Select **Output > Eval**



  - The net flux flowing out of the core is approximately (1.1 e-6 Webers)

```
Scl : 1.06547709374332E-006
Scl : Integrate(Surface(Facelist1), Dot(Real(<Bx,By,Bz>), SurfaceNormal))
```

Calculate loss in a conductor

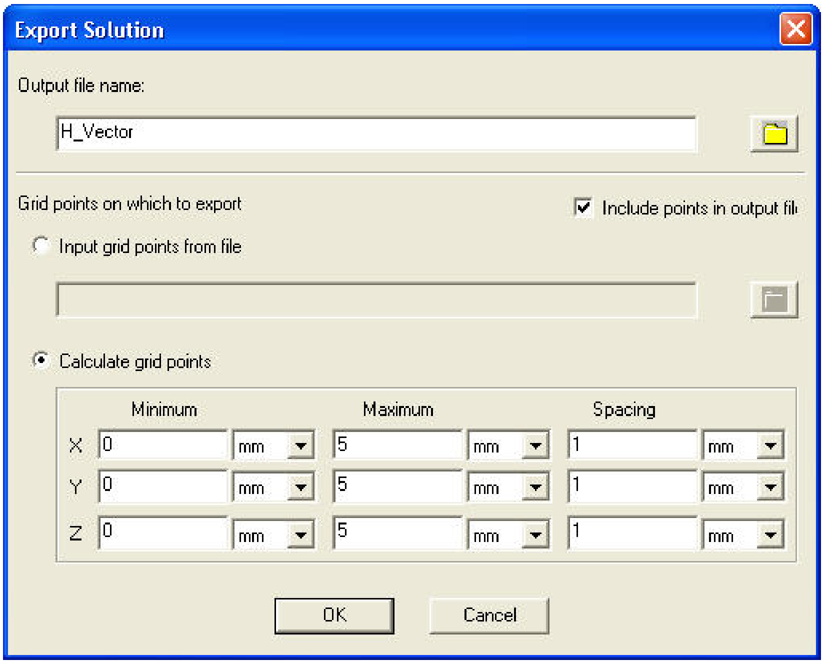
- **To calculate losses in Coil**
  1. Select **Input > Quantity > OhmicLoss**
  2. Select **Input > Geometry**
     - Volume: ☑ **Checked**
     - Select **Coil** from the list
     - Press **OK**
  3. Select **Scalar >** ∫ **Integrate**
  4. Select **Output > Eval**
  - The ohmic loss in the coil at 100 kHz is approximately 2.66 Watts.

Calculate the current flowing in a conductor

- **To Calculate Net Current in Coil**
  1. Select **Input > Quantity > J**
  2. Select **General > Complex > Real**
  3. Select **Input > Geometry**
     - Surface: ☑ **Checked**
     - Select **Terminal** from the list
     - Press **OK**
  4. Select **Vector > Normal**
  5. Select **Scalar >** ∫ **Integrate**
  6. Select **Output > Eval**
  - The net current in the coil is equal to the source amp-turns = +/- 100
- **Calculate Total Current including Eddy Component**
  1. Select **Input > Quantity > J**
  2. Select **General > Complex > CmplxMag**
  3. Select **Input > Geometry**
     - Surface: ☑ **Checked**
     - Select **Terminal** from the list
     - Press **OK**
  4. Select **Vector > Normal**
  5. Select **Scalar >** ∫ **Integrate**
  6. Select **Output > Eval**
  - Calculated value of current comes out to be around 120 A

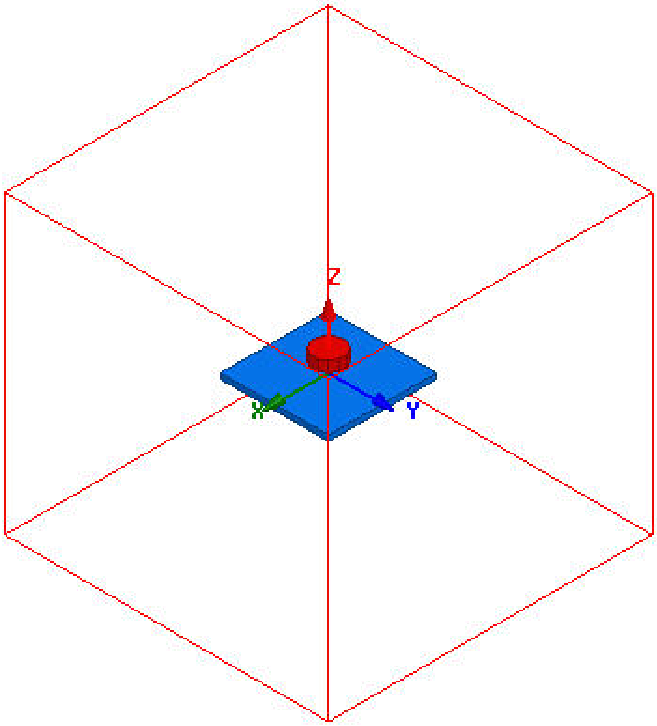
Export field results to a file

- In order to use the field results in another software program, you can export the fields on a uniform 3-dimensional grid.
- **To Export Results**
  1. Select **Input > Quantity > H**
  2. Select **Output > Export**
  3. In Export Solution window,
     - Output file name: **H_Vector**
     - Calculate grid points: ☑ **Checked**
     - Minimum: **(0, 0,0)**
     - Maximum: **(5, 5, 5)**
     - Spacing: **(1, 1, 1)**
- **Minimum:** The minimum coordinates of the grid, and unit of measure
- **Maximum:** The maximum coordinates of the grid, and unit of measure
- **Spacing:** The distance between grid points, and unit of measure



Puck Magnet Attractor

- This example describes how to create and optimize a puck magnet producing an optimal force on a steel plate using the 3D Magnetostatic solver and Optimetrics in the ANSYS Maxwell 3D Design Environment.
- The optimization obtains the desired force by varying the air gap between the plate and the puck using a local variable.



Create Design

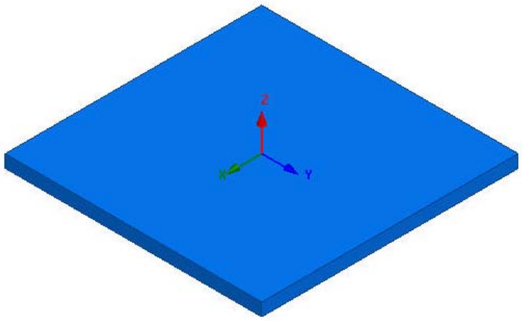
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the  icon
  - Change the name of the Design to **Puck_Attractor**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Magnetostatic**
    2. Click the **OK** button

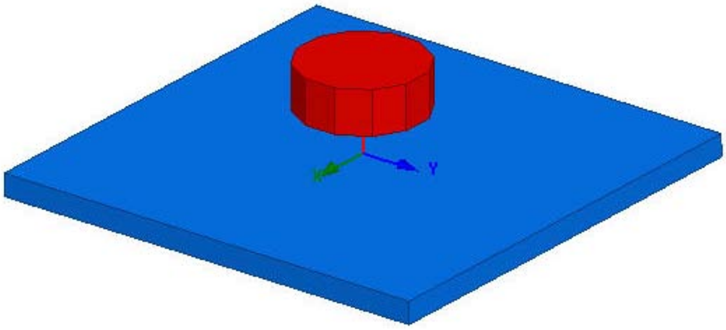
Create Steel Plate

- **To Create Steel Plate**
  - Select the menu item *Draw > Box*
    1. Using the coordinate entry fields, enter the position of box
       - X: **-10**, Y: **-10**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **20**, dY: **20**, dZ: **-1**, Press the **Enter** key
  - Change the name of the Object to **Plate**
  - Change the material of the Object to **steel_1008**



Create Puck

- **To Create Puck**
  - Select the menu item *Draw > Regular Polyhedron*
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **2**, dY: **2**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**2**, Press the **Enter** key
    4. Number of Segments: **12**
  - Change the name of the Object to **Puck**
  - Change the material of the Object to **NdFe30**

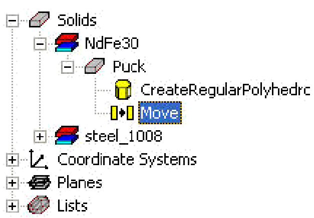


Create Parameter for Puck Motion

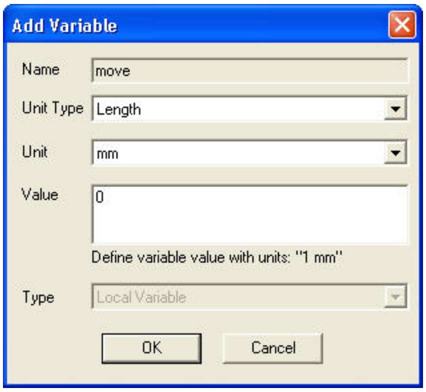
To Create Puck Motion

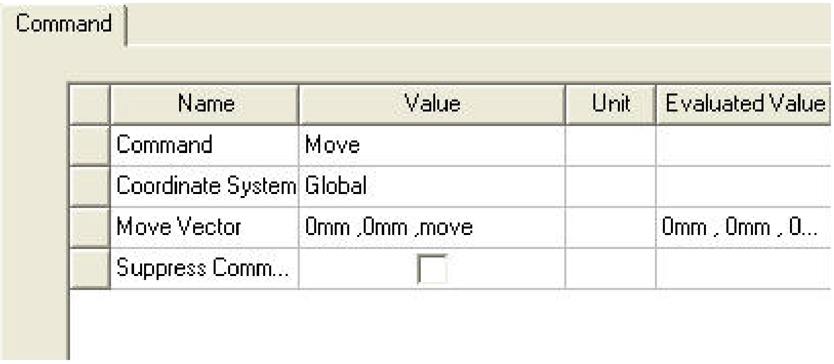
- Select the Object **Puck** from the history tree
- Select the menu item ***Edit > Arrange > Move***
  1. Using the coordinate entry fields, enter the reference point of move vector
     - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the target point of move vector
     - dX: **0**, dY: **0**, dZ: **0**, Press the **Enter** key

To Parameterize Puck Motion



- Expand the history tree for the Object **Puck**
- Double click on the command **Move** from the tree
- In Properties window,
  1. Move Vector: Specify as **0, 0, move**
  2. In Add variable window,
     - Unit Type: **Length**
     - Unit: **mm**
     - Value: **0**
     - Press **OK**
  3. Press **OK** to close Properties window



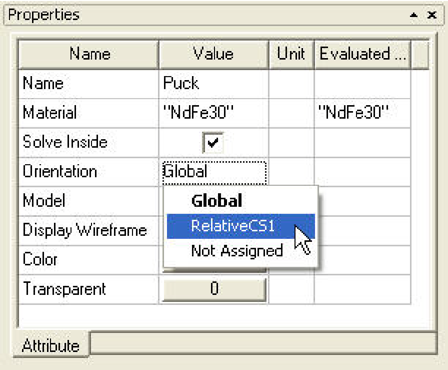


Command

Name	Value	Unit	Evaluated Value
Command	Move		
Coordinate System	Global		
Move Vector	0mm ,0mm ,move		0mm , 0mm , 0...
Suppress Comm...	☐		

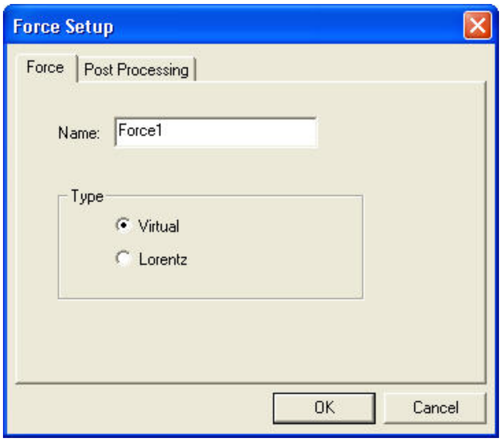
Create Relative Coordinate System

- **Note:** In Maxwell, all magnetic materials are magnetized in X direction. In this step, we will change the direction of magnetization for puck by creating relative coordinate system.
- **To Create Relative Coordinate System**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Rotated***
    1. Using the coordinate entry fields, enter the X axis
       - X: **0**, Y: **0**, Z: **1**, Press the **Enter** key
    2. Using the coordinate entry fields, enter a point on XY Plane
       - dX: **0**, dY: **1**, dZ: **0**, Press the **Enter** key
- **To Assign Coordinate System to Puck**
  - Select the object **Puck** from the history tree and goto the properties window
  - Change the Orientation of the object to **RelativeCS1**



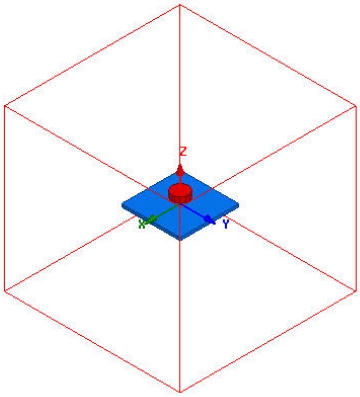
Assign Parameters Force

- **To Assign Force Parameter**
  - Select the object **Plate** from the history tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Force***
  - In Force Setup window,
    1. Name: **Force**
    2. Type: **Virtual**
    3. Press **OK**



Create Region

- **To Change Work Coordinate System**
  - Select the menu item ***Modeler > Coordinate System . Set Work CS***
    1. Select the coordinate System **Global** from the list
    2. Press **Select**
- **To Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding individual directions: ☑ **Checked**
    2. Value:
       - +/- X = **100**
       - +/- Y = **100**
       - +/- Z = **500**
    3. Press **OK**



Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** tab
       - Percentage Error: **2**
    2. **Convergence** tab
       - Refinement Per Pass: **50%**

Validation Check

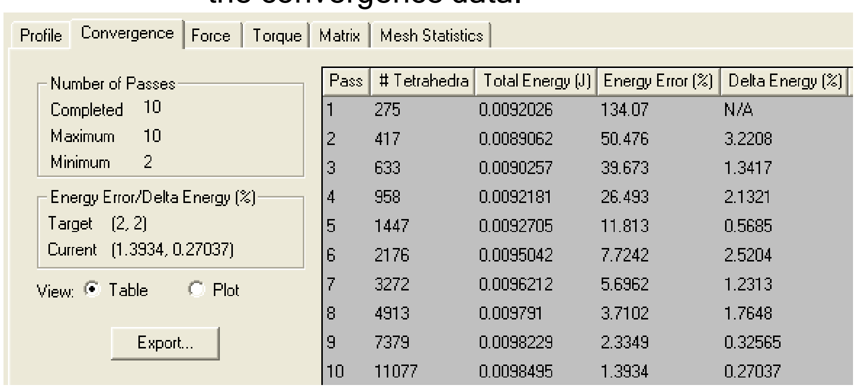
- **To Check the Validation of Case**
  - Select the menu item ***Maxwell 3D > Validation Check***
    - **Note:** To view any errors or warning messages, use the Message Manager.

Analyze

- **To run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.



Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

Number of Passes
Completed 10
Maximum 10
Minimum 2

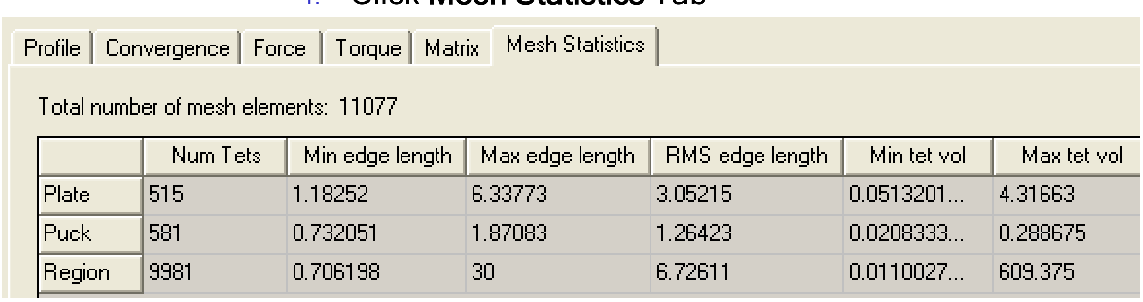
Energy Error/Delta Energy (%)
Target (2, 2)
Current (1.3934, 0.27037)

View: Table Plot

Export...

Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	275	0.0092026	134.07	N/A
2	417	0.0089062	50.476	3.2208
3	633	0.0090257	39.673	1.3417
4	958	0.0092181	26.493	2.1321
5	1447	0.0092705	11.813	0.5685
6	2176	0.0095042	7.7242	2.5204
7	3272	0.0096212	5.6962	1.2313
8	4913	0.009791	3.7102	1.7648
9	7379	0.0098229	2.3349	0.32565
10	11077	0.0098495	1.3934	0.27037

- To View Mesh information
  1. Click **Mesh Statistics** Tab

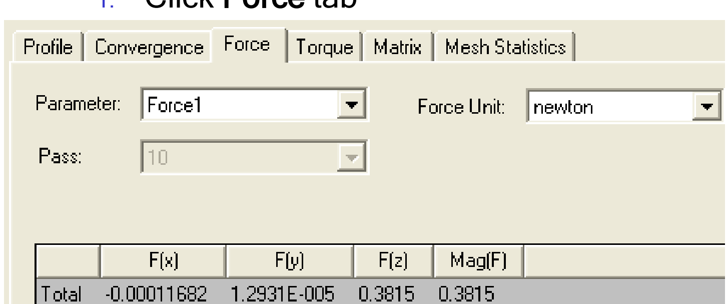


Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

Total number of mesh elements: 11077

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol
Plate	515	1.18252	6.33773	3.05215	0.0513201...	4.31663
Puck	581	0.732051	1.87083	1.26423	0.0208333...	0.288675
Region	9981	0.706198	30	6.72611	0.0110027...	609.375

- To View Force values
  1. Click **Force** tab



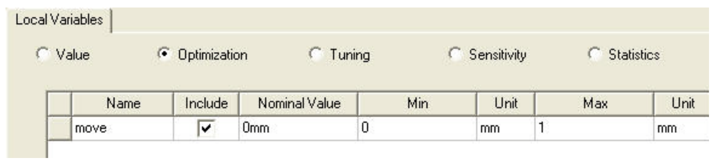
Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

Parameter: Force1
Force Unit: newton
Pass: 10

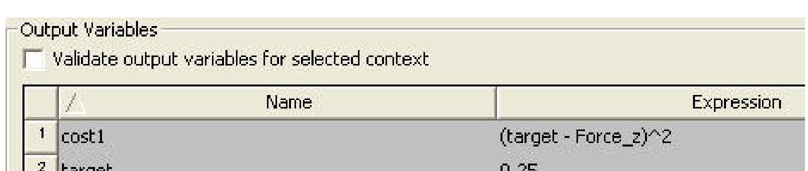
	F(x)	F(y)	F(z)	Mag(F)
Total	-0.00011682	1.2931E-005	0.3815	0.3815

Optimetrics Setup and Solution

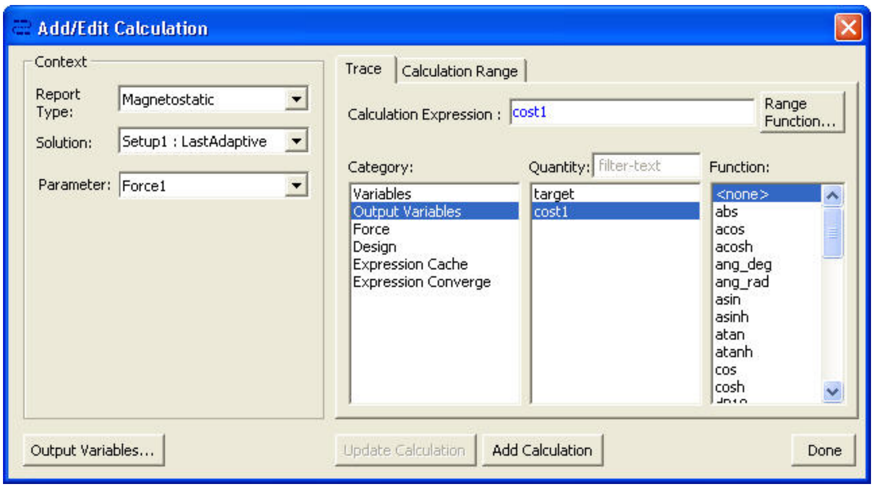
- It is possible to optimize position in order to obtain the specified force. For this optimization, the position will be varied to obtain a desired force of 0.25N.
- **Specify Parametric Variables**
  - Select the menu item ***Maxwell 3D > Design properties***
  - In Properties window,
    1. Optimization: ☑ **Checked**
    2. move:
       - Include: ☑ **Checked**
       - Min : **0 mm**
       - Max : **1 mm**
    3. Press **OK**



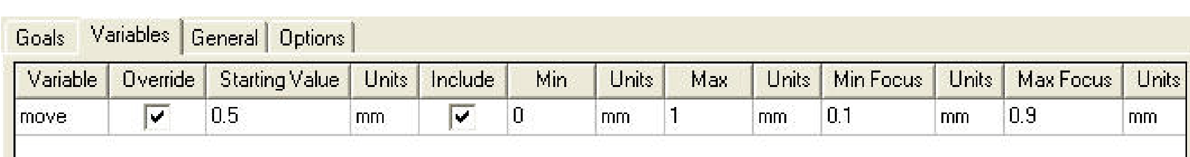
- **Setup an Optimization Analysis**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Optimization***
  - In Setup Optimization window,
    1. Optimizer: **Sequential Nonlinear Programming**
    2. Max. No. of Iterations: **10**
    3. Select **Setup Calculations**
    4. In Add/Edit calculations window, select **Output Variables**
    5. In Output variables window,
       - Name: **target**
       - Expression: **0.25**
       - Select **Add**
       - Without closing window, set name to **cost1**
       - Set Parameter to **Force1**
       - Expression: **(target – Force_z)^2**
       - Press **Done**



6. In Add/Edit Calculations window,
   - Parameter : **Force1**
   - Category: **Output Variables**
   - Quantity: **cost1**
   - Select **Add Calculation**
   - Press **Done**



7. Parameter **cost1** will be added to Setup Optimization window.
8. Set Condition for cost1 to **Minimize**
9. Change the tab to **Variables**
10. For the variable **move**
    - Starting value: **0.5mm**
    - Min: **0 mm**
    - Max: **1 mm**
    - Min Focus: **0.1 mm**
    - Max Focus: **0.9 mm**
    - Press **OK**



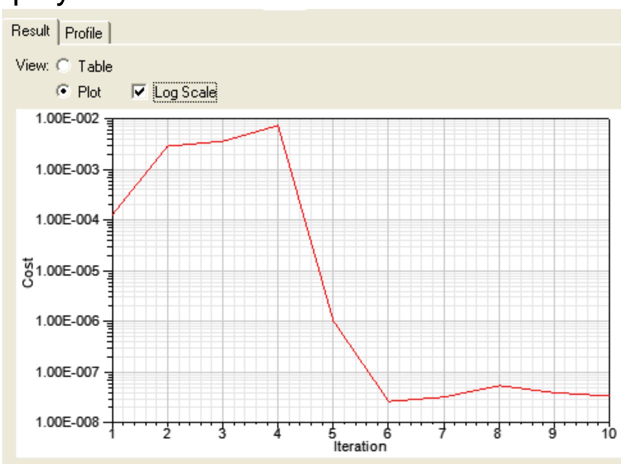
Variable	Override	Starting Value	Units	Include	Min	Units	Max	Units	Min Focus	Units	Max Focus	Units
move	✓	0.5	mm	✓	0	mm	1	mm	0.1	mm	0.9	mm

Solve Optimization Analysis

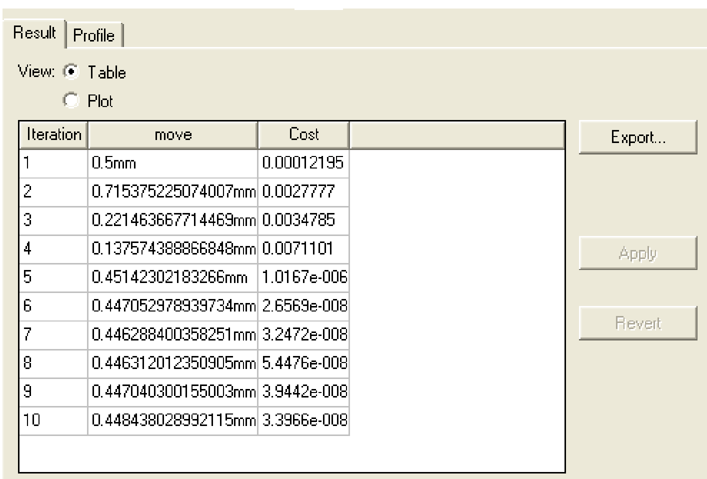
- **To Solve Optimization Analysis**
  - Select the menu item ***Maxwell 3D > Analyze All***

View Optimetrics Results

- **To View Results**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Optimetrics Results***
  - In Post Analysis Display window,
    1. Log Scale: ☑ **Checked**
  - Plot Display should look as below



  - Change View to **Table** to see actual values



Iteration	move	Cost
1	0.5mm	0.00012195
2	0.715375225074007mm	0.0027777
3	0.221463667714469mm	0.0034785
4	0.137574388866848mm	0.0071101
5	0.45142302183266mm	1.0167e-006
6	0.447052978939734mm	2.6569e-008
7	0.446288400358251mm	3.2472e-008
8	0.446312012350905mm	5.4476e-008
9	0.447040300155003mm	3.9442e-008
10	0.448438028992115mm	3.3966e-008

  - Press **Close**

Create Plot of Cost vs Move

- **To Create Report**
  - Select menu item *Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot*
  - In Report window,
    1. Parameter: **Force1**
    2. X : **Default**
    3. Category: **Output Variables**
    4. Quantity: **cost1**
    5. Function: **<none>**
    6. Select **New Report**
- **To Modify Plot Attributes**
  - Double click on Y axis of plot
  - In Properties window,
    1. Scaling tab
       - Axis Scaling: **Log**
       - Pres **OK**

- From the plot, it can be seen that minimum value of cost is obtained at around **445 um**

Introduction on Meshing Operations

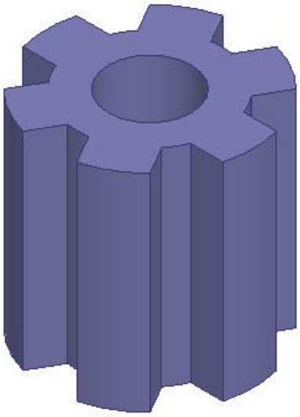
- This note introduces the different mesh operations that Maxwell offers. Mesh operation are used either to cut the number of adaptive passes or to optimize the mesh details on complicated objects. We will illustrate the meshing operations using an SRM core.

Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Create Core

- **To Create Core**
  - Select the menu item ***Draw> User Define Primitive > SysLib > RMxprt > SRMcore***
  - In User Defined Primitive Operation window,
    1. Press **OK** to accept default settings
  - Change the name of the Object to **Core**



Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup window,
    1. Press **OK** to accept default settings

Save File

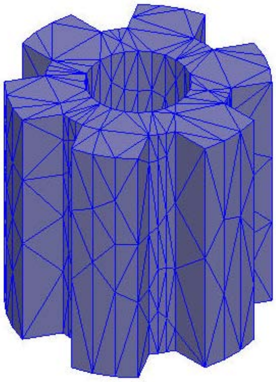
- **To Save File**
  - Select the menu item ***File > Save As***
    1. Save the file with name **Ex_9_9_BasicMeshing.mxwl**

Generate Initial Mesh

- **To Create Initial Mesh**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Apply Mesh Operations***

Plot Mesh

- **To Plot Mesh on the Object**
  - Select the Object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - In Create Mesh plot window,
    1. Press **Done**



  - **Note :** This is initial mesh created by Maxwell. Apart from transient solver, for all other solvers this mesh will be refined for each pass in order to increase accuracy of solution. For Transient solutions, mesh refinement is not done. So it is required to apply appropriate mesh sizes which will be discussed in next few slides.

Mesh Statistics

- **To View Statistics of Mesh**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
  - In Solutions window,
    1. Select the tab **Mesh Statistics**

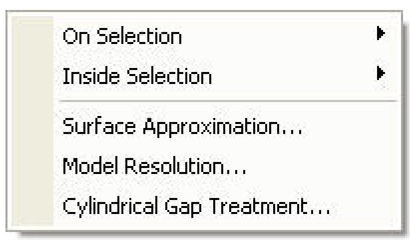
Profile | Convergence | Force | Torque | Matrix | Mesh Statistics

Total number of mesh elements: 1503

	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol...	Mean tet vol...	Std Devn (vol)
Core	1503	10.4389	100	25.1756	2.88159	2850.15	329.532	350.678

Mesh Operations

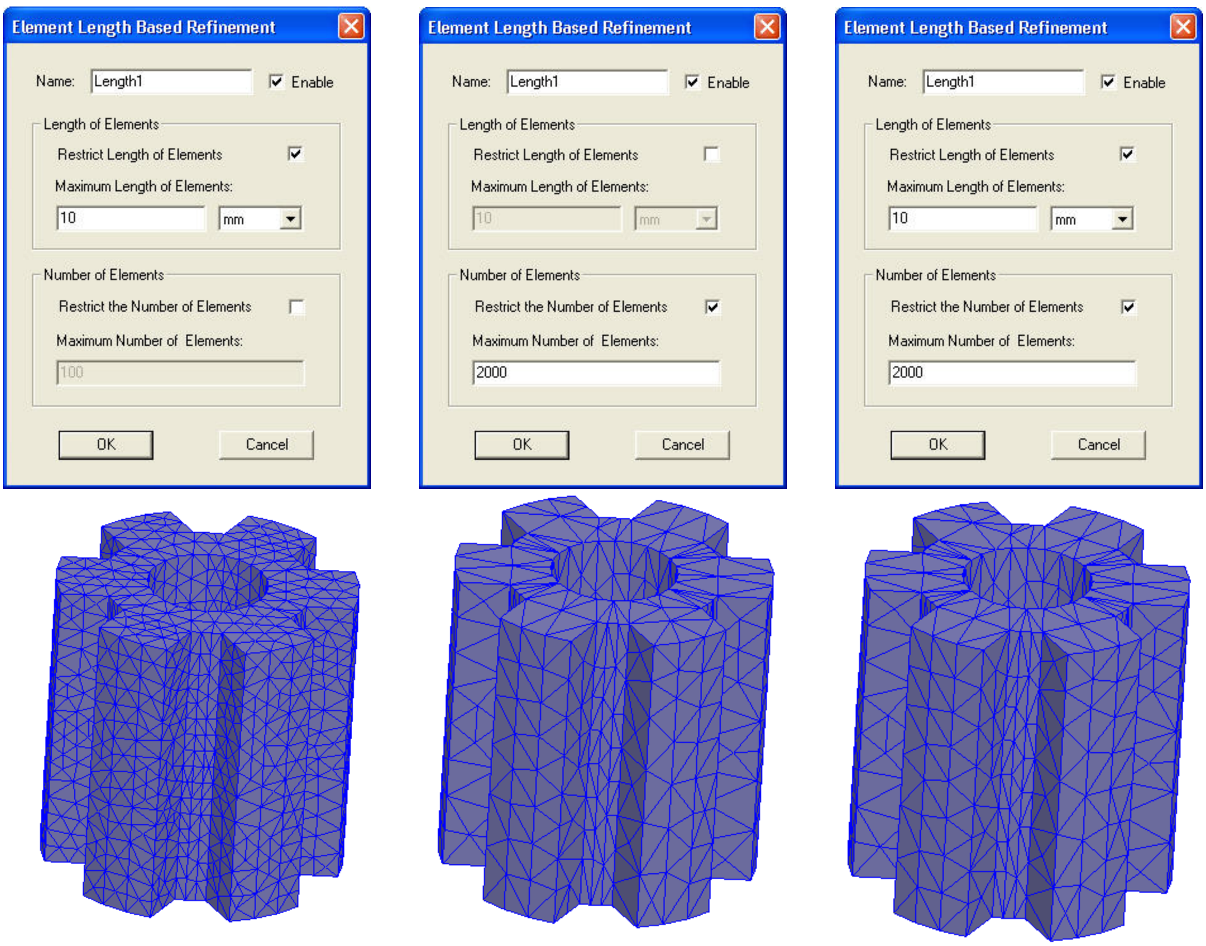
- Maxwell3D has number of meshing operations in order to construct a better initial mesh for pass one. These mesh operations enable you to add elements in a volume, on a surface. It is also possible to monitor the discretization of true surfaces.
- Note that when assigning a mesh operation, the mesh process will do its best to cope with the specified operations. Automatic meshing is a complex operation, so other constraints are to be taken into account. As a result, sometimes, the obtained mesh is not quite exactly what has been entered because internal constraints (seeding, facetting, ..) have imposed a different mesh topology. For instance, it will be difficult for the meshing engine to put 100,000 elements on the top face of a cylinder if the rest of the cylinder has a very coarse mesh.
- For selected objects, five different type of mesh operations can be applied



- First two operations are used to add elements either on the surface or in the volume of the objects
- The last three meshing operations will monitor the way the meshing will handle the geometry of the object.

Mesh operation: Add Elements in the Object

- We can add more elements in any object to obtain a better mesh density right away.
- **To Apply Mesh operations in an object**
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - Mesh operations can be assigned based on either Maximum Length of Elements or Maximum Number of elements in an object.

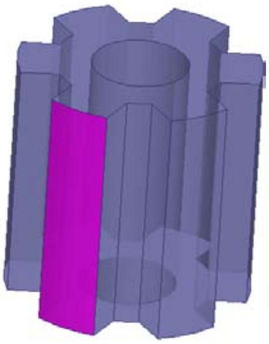


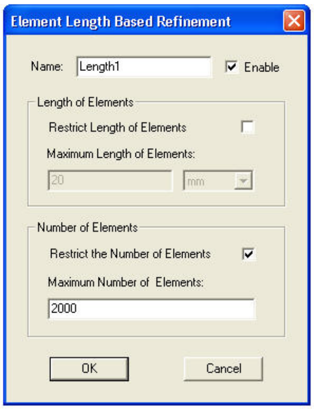
- When you enter both a number of elements and you restrict the maximum length of elements, Maxwell will stop when the first constraint is attained. In this particular case, entering only a maximum length of 10mm produces about 20,000 elements. Therefore, having both 10mm and 2,000 elements lead to have only 2,000 elements added.

Mesh operation: Add Elements on a Surface

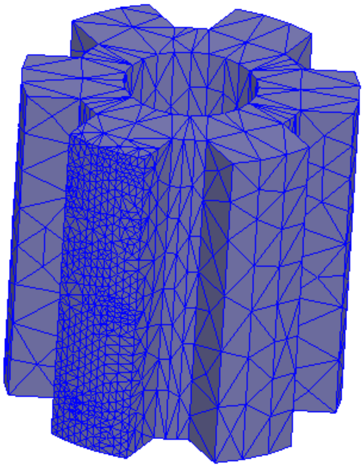
To Apply Mesh Operations on a Surface

- Delete already applied mesh operations
- Select the menu item ***Maxwell 3D > Analysis Setup > Revert to Initial Mesh***
- Select the menu item ***Edit > Select > Faces*** to change selection filter to faces
- Select the face of the object as shown in the image
- Select the menu item ***Maxwell 3D > Mesh Operations > Assign > On Selection > Length Based***
- Set Maximum Number of Elements to **2000**



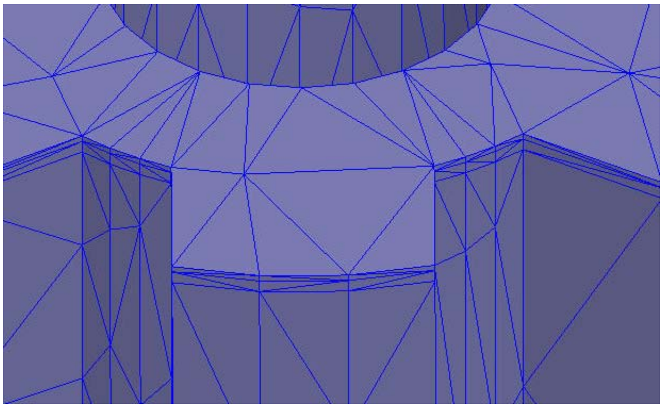


- Select the menu item ***Maxwell 3D > Analysis Setup > Apply Mesh Operations***
- The resulting mesh takes into account the operation by adding elements on the surface without affecting the other parts of the object.



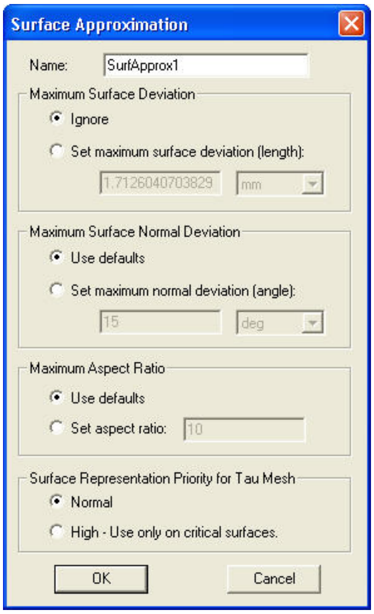
Mesh operation: Skin Depth Mesh

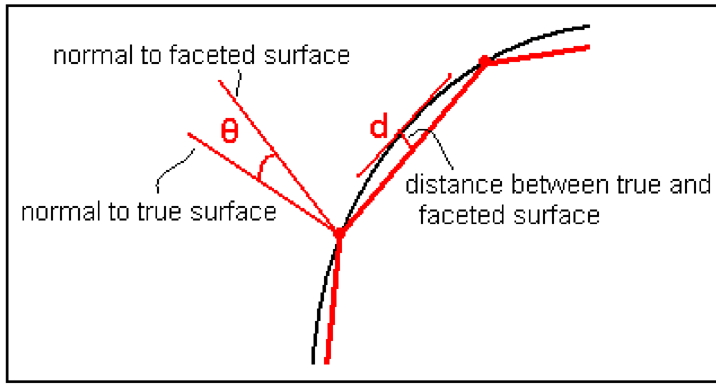
- This mesh operation has to be used with great caution. The goal is to impose to Maxwell to mesh in the skin depth with a given number of layers and a given mesh density. This can be used in the case where the skin depth is much smaller than the dimensions of the object and therefore there is a risk that the adaptive meshing won't capture the eddy current in the early adaptive passes.
- The major consequence of the mesh operation is that the adaptive meshing process does not have the full freedom when refining the mesh. Usually, it is preferred to use phantom objects or shells to take into account skin depth.



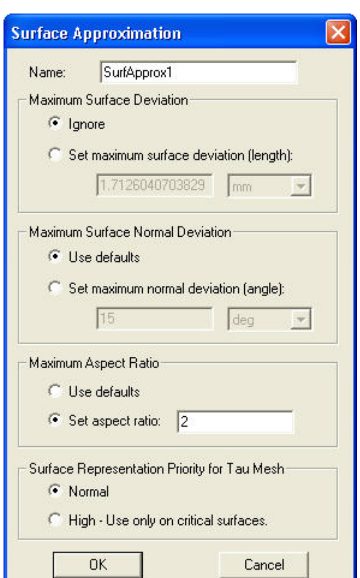
Mesh operation: Surface Approximation

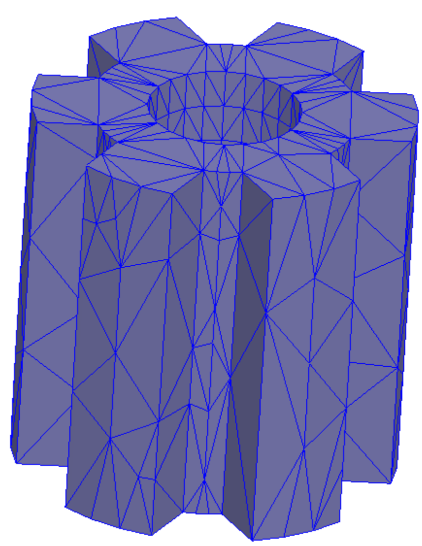
- This mesh operations control the way Maxwell handle true surfaces
- When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.
- The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees (15deg is the default).
- **To Apply Surface Approximation**
  - Delete already applied mesh operations
  - Select the menu item ***Maxwell 3D > Analysis Setup > Revert to Initial Mesh***
  - Select the Object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Surface Approximation***



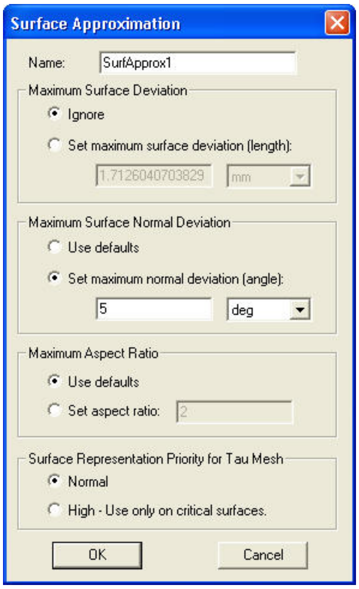


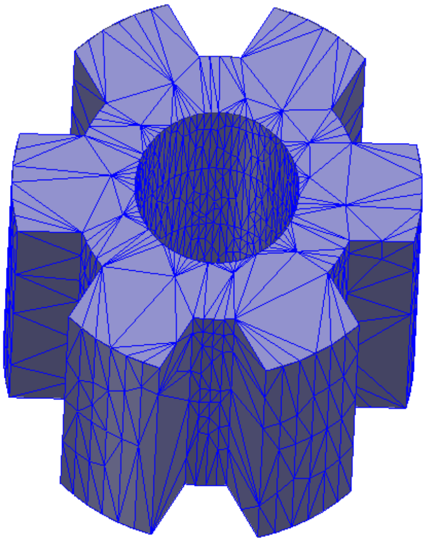
- Maximum surface deviation corresponds to the distance **d** above
- Normal Surface Normal Deviation corresponds to the angle **θ**
- In most of the cases initial mesh is created using Tau Mesher. Using Normal or High option in Surface Approximation window, we can set tolerance in capturing small features. Using High option will help in creating better mesh on small details with expense of mesh count.
- In Surface Approximation window,
  1. Set Maximum Aspect Ratio value to **2**
  2. Generate the mesh with these parameters
  3. Maxwell will make sure that the elements are not too flat



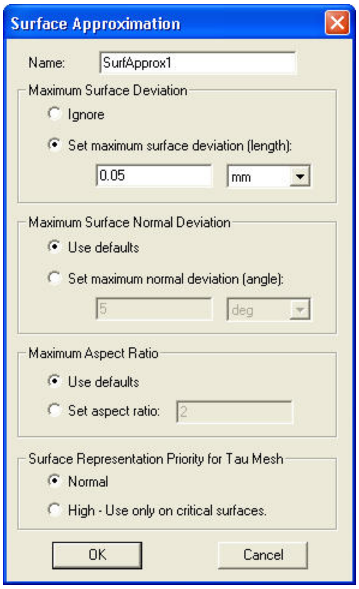


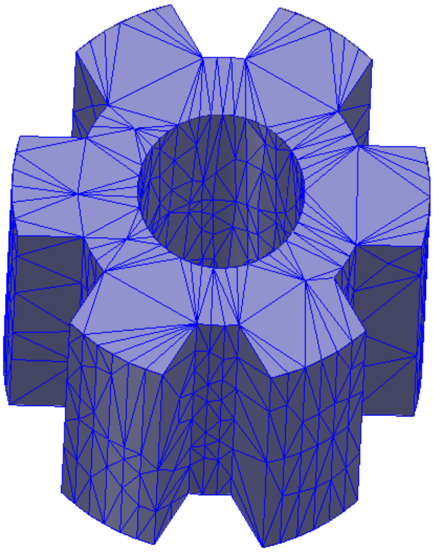
4. Edit mesh parameters and set the value of Maximum Normal Deviation angle to **5 degrees.** Set all other values to default and apply mesh operations
5. Putting 5 produces 3 times more elements to discretize a curved surface





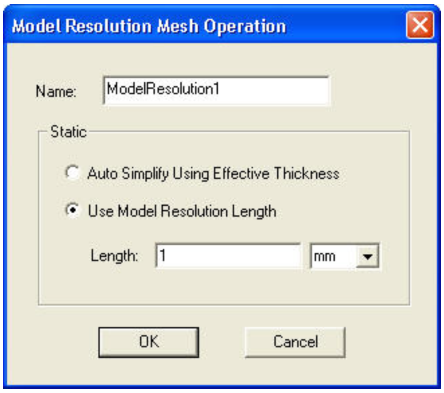
6. Set the value of maximum surface deviation length to 0.05 leaving all other values to default. This will ensure that the distance between curve lines and element edges won't be more than 0.05mm.
7. The mesh operation is used when dealing with objects that have 'small' true surfaces (fillet, some details) and 'large' true surfaces.



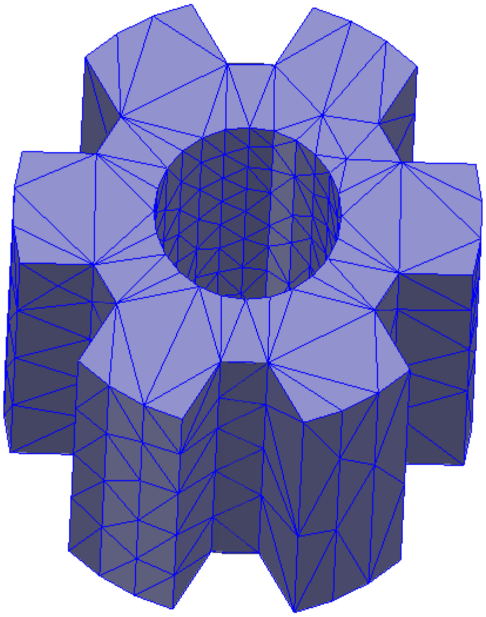


Mesh Operation: Model Resolution

- **To Set Model Resolution**
  - Delete the previous mesh operation
  - Select the object **Core** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Model Resolution***
  - In Model Resolution mesh Operation window,
    1. Set the tab to **Use Model Resolution Length**
    2. Set the Length value to **1mm**
    3. Press **OK**

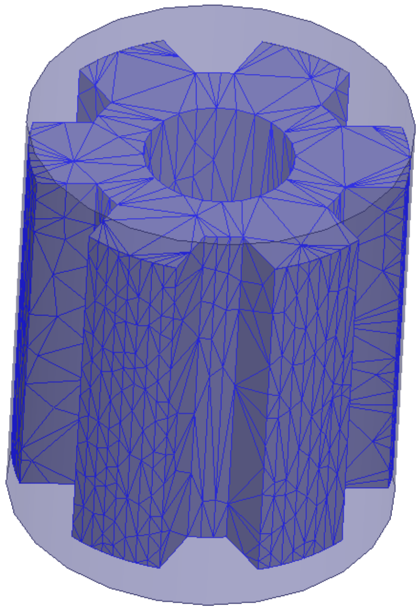


  - This operation is meant to disfeature an object. By putting 1mm, Maxwell will disregard any detail below 1mm. This can be useful when dealing with imported geometries where some details, inconstancies remain. This operation is seldom used. For this piece, the result is not spectacular at all.



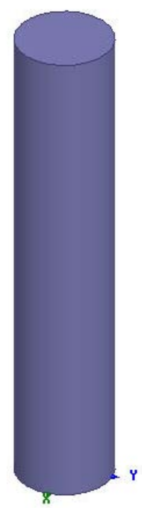
Mesh Operation: Cylindrical Gap Treatment

- These mesh operations are generally used to resolve narrow gaps in motor geometries . There are very small gaps between stator and rotor. The Band region created between them which needs to have fine mesh resolution between band region and stator or rotor. In such cases it is helpful to use this mesh operation on Band region.
- **Create Band Region**
  - Select the menu item ***Draw > Cylinder***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **-60**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **51**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**120**, Press the **Enter** key
  - Change the name of the object to **Band**
- **Assign Mesh operation**
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Cylindrical Gap Treatment***
  - Apply mesh operations
  - The refinement will be done in the region of small gap in order to capture the thin region.



Scripting the Creation of a Model Object

- This exercise will discuss how to record, modify and run a script for automating the generation of a cylinder. The following tasks will be performed:
  - Record a script in which a cylinder is created.
  - Modify the script to change the cylinder's radius and height.
  - Run the modified script.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the  icon

Save File

- **To Save File**
  - Select the menu item ***File > Save As***
    1. Save the file with name **Ex_9_10_BasicScripting.mxwl**

Start Recording the Script

- **To Record a Script**
  - Select the menu item ***Tools > Record Script to File***
  - Save the file with the name **Ex_9_10_BasicScripting.vbs**

Create Cylinder

- **To Create Cylinder**
  - Select the menu item ***Draw > Cylinder***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **1**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**10**, Press the **Enter** key

Stop Script Recording

- **To Stop Recording a Script**
  - Select the menu item ***Tools > Stop Script Recording***

Delete Cylinder

- **To Delete Cylinder**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Edit > Delete***

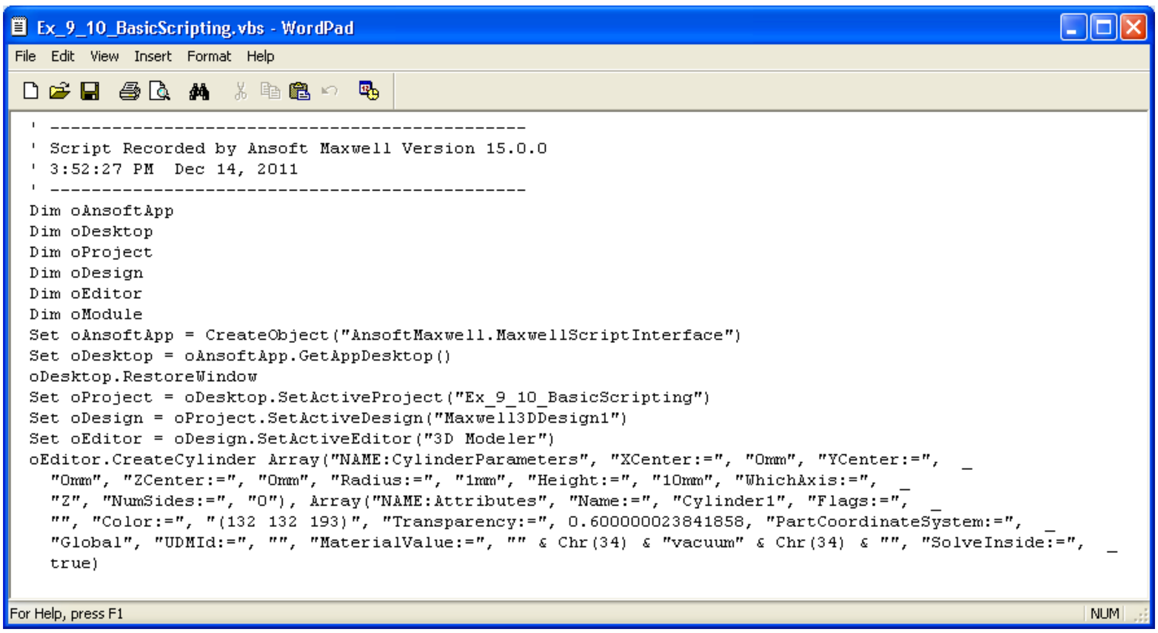
Run Script to Recreate Cylinder

- **To Run Script**
  - Select the menu item ***Tools > Run Script***
  - Locate the file **Ex_9_10_BasicScripting.vbs** and **Open** it
  - If successful, the original cylinder, **Cylinder1**, should be back.
  - We can now explore the contents of the script file.

Open Script for Editing

To View Script

- Open the script file using Textpad or Wordpad




```
' ----------------------------------------------
' Script Recorded by Ansoft Maxwell Version 15.0.0
' 3:52:27 PM  Dec 14, 2011
' ----------------------------------------------
Dim oAnsoftApp
Dim oDesktop
Dim oProject
Dim oDesign
Dim oEditor
Dim oModule
Set oAnsoftApp = CreateObject("AnsoftMaxwell.MaxwellScriptInterface")
Set oDesktop = oAnsoftApp.GetAppDesktop()
oDesktop.RestoreWindow
Set oProject = oDesktop.SetActiveProject("Ex_9_10_BasicScripting")
Set oDesign = oProject.SetActiveDesign("Maxwell3DDesign1")
Set oEditor = oDesign.SetActiveEditor("3D Modeler")
oEditor.CreateCylinder Array("NAME:CylinderParameters", "XCenter:=", "0mm", "YCenter:=",  _
  "0mm", "ZCenter:=", "0mm", "Radius:=", "1mm", "Height:=", "10mm", "WhichAxis:=",  _
  "Z", "NumSides:=", "0"), Array("NAME:Attributes", "Name:=", "Cylinder1", "Flags:=",  _
  "", "Color:=", "(132 132 193)", "Transparency:=", 0.600000023841858, "PartCoordinateSystem:=",  _
  "Global", "UDMId:=", "", "MaterialValue:=", "" & Chr(34) & "vacuum" & Chr(34) & "", "SolveInside:=",  _
  true)
```

Script File Contents

Definition of environment variables.

- **Dim** is the generic visual basic variable type.

```
' ----------------------------------------------
' Script Recorded by Ansoft Maxwell Version 15.0.0
' 3:52:27 PM  Dec 14, 2011
' ----------------------------------------------
Dim oAnsoftApp
Dim oDesktop
Dim oProject
Dim oDesign
Dim oEditor
Dim oModule
```

- **Reference defined environment variables**
  - These variables are defined with the term **Set**

```
Set oAnsoftApp = CreateObject("AnsoftMaxwell.MaxwellScriptInterface")
Set oDesktop = oAnsoftApp.GetAppDesktop()
oDesktop.RestoreWindow
Set oProject = oDesktop.SetActiveProject("Ex_9_10_BasicScripting")
Set oDesign = oProject.SetActiveDesign("Maxwell3DDesign1")
Set oEditor = oDesign.SetActiveEditor("3D Modeler")
```

- **Commands for Creating Cylinder**
  - All of the parameters needed to create the cylinder are defined in this line of code. Here we will modify the **Radius** and **Height** of the cylinder by changing the appropriate text.

```
oEditor.CreateCylinder Array("NAME:CylinderParameters", "XCenter:=",
"0mm", "YCenter:=", _ "0mm", "ZCenter:=", "0mm", "Radius:=", "1mm",
"Height:=", "10mm", "WhichAxis:=", _ "Z", "NumSides:=", "0"),
Array("NAME:Attributes", "Name:=", "Cylinder1", "Flags:=", _ "",
 "Color:=", "(132 132 193)", "Transparency:=", 0.600000023841858,
"PartCoordinateSystem:=", _ "Global", "UDMId:=", "", "MaterialValue:=", ""
& Chr(34) & "vacuum" & Chr(34) & "", "SolveInside:=", _ true)
```

Modify Script

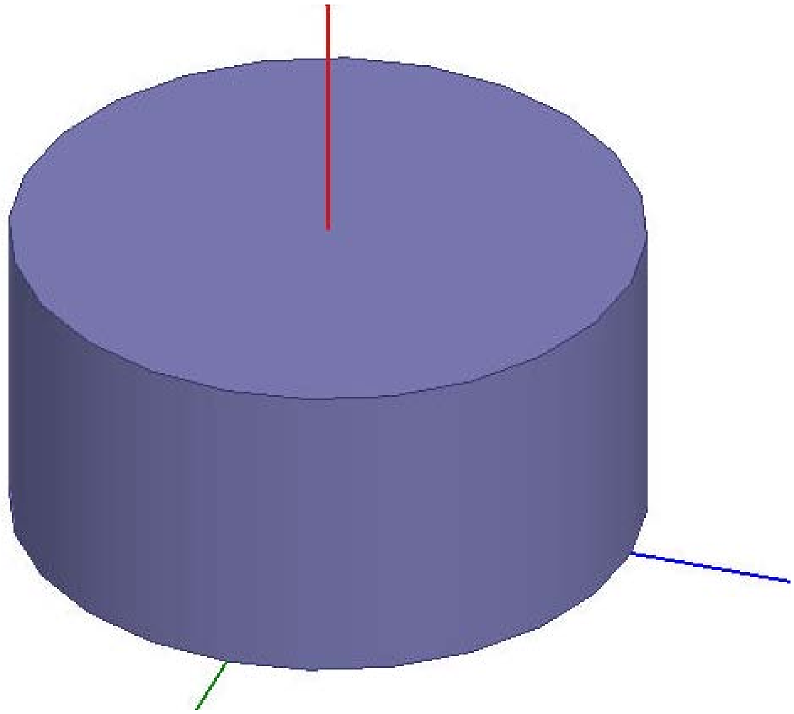
- **To Modify Cylinder parameters**
  - Locate the line containing the **Radius** and **Height** and change the numerical values to **5mm** and **5mm**, respectively.
  - **Original:** "**Radius:=", "1mm**", "**Height:=", "10mm**", "WhichAxis:=", _ "Z“
  - **Modified:** "**Radius:=", “5mm**", "**Height:=", “5mm**", "WhichAxis:=", _ "Z“
  - **Save** the modified script file

Delete Cylinder

- **To Delete Cylinder**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Edit > Delete***

Run Script to Recreate Cylinder

- **To Run Script**
  - Select the menu item ***Tools > Run Script***
  - Locate the file **Ex_9_10_BasicScripting.vbs** and **Open** it
  - If successful, the modified cylinder, **Cylinder1**, should appear.



Generalize the script to run in any Project and Design

- **To run the script in order to create your cylinder in a different project**
  - Change the following lines in the script.

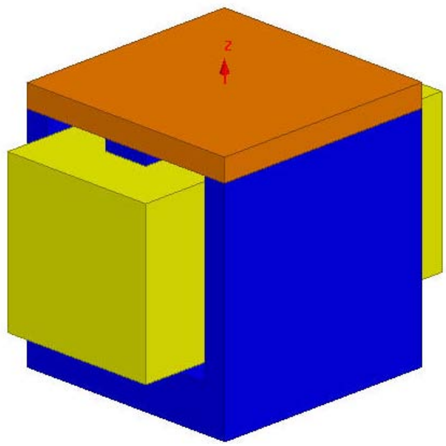
```
Set oProject = oDesktop.SetActiveProject("Ex_9_10_BasicScripting")
Set oDesign = oProject.SetActiveDesign("Maxwell3DDesign1")
```

  - Change the above two lines as below

```
Set oProject = oDesktop.GetActiveProject()
Set oDesign = oProject.GetActiveDesign()
```

Equivalent circuit extraction (ECE) of a linear actuator

- This exercise will discuss how to use Maxwell's magnetostatic solver combined with Optimetrics to create an equivalent circuit model to be used in Simplorer



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Set Solution Type

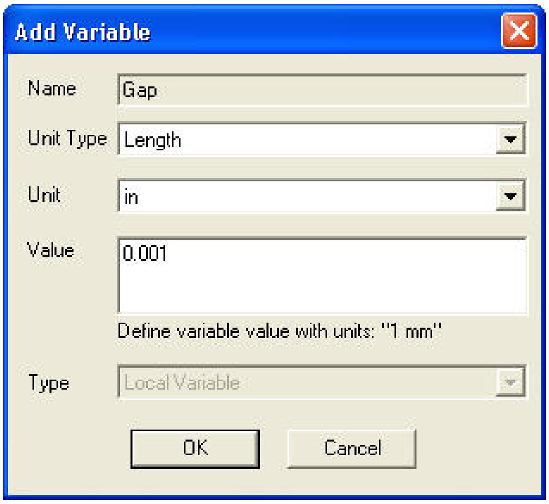
- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Magnetostatic**
    2. Click the **OK** button

Set Model Units

- **To Set Units**
  - Select the menu item ***Modeler > Units***
  - In Set Model Units window,
    1. Select units: **in (inches)**
    2. Press **OK**

Create Armature

- **To Create Armature**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.5**, Y: **0.5**, Z: **0.001**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **-1**, dY: **-1**, dZ: **0.1**, Press the **Enter** key
  - Change the name of the resulting object to **Armature** and color to **Orange**
  - Change the material of the object to **steel_1010**
- **Parameterize Armature Position**
  - Expand the history tree for the object **Armature**
  - Double click on the command **CreateBox**
  - In Properties window,
    1. For Position, type **0.5, 0.5, Gap**, Press the **Tab** key
    2. In Add Variable window,
       - Unit Type: **Length**
       - Unit: **in (inches)**
       - Value: **0.001**
       - Press **OK**



Create Yoke

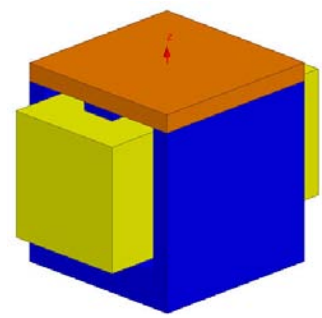
- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.5**, Y: **0.5**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **-1**, dY: **-1**, dZ: **-1**, Press the **Enter** key
  - Change the name of the resulting object to **Yoke** and color to **Blue**
  - Change the material of the object to **steel_1010**

- **Create another Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0.5**, Y: **0.4**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **-1**, dY: **-0.3**, dZ: **-0.8**, Press the **Enter** key
  - Change the name of the object to **Slot**
- **Duplicate Box**
  - Select the object **Slot** from the history tree
  - Select the menu item ***Edit > Duplicate > Mirror***
    1. Using the coordinate entry fields, enter the anchor point of mirror plane
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the target point
       - dX: **0**, dY: **-1**, dZ: **0**, Press the **Enter** key
- **Subtract Objects**
  - Press **Ctrl** and select the objects **Yoke, Slot** and **Slot_1**
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    1. Blank Parts: **Yoke**
    2. Tool Parts: **Slot, Slot_1**
    3. Press **OK**

Create Coil

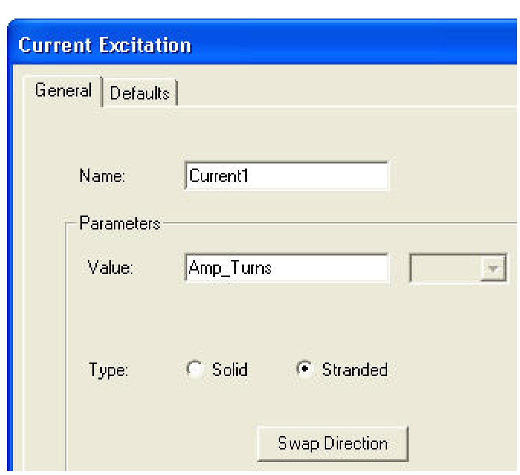
- **Create Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **-0.75**, Y: **-0.35**, Z: **-0.75**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **1.5**, dY: **0.7**, dZ: **0.7**, Press the **Enter** key
  - Change the name of the resulting object to **Coil** and color to **Yellow**
  - Change the material of the object to **copper**

- **Subtract Objects**
  - Press **Ctrl** and select the objects **Coil** and **Yoke**
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    1. Blank Parts: **Coil**
    2. Tool Parts: **Yoke**
    3. Clone tool objects before operation: ☑ **Checked**
    4. Press **OK**



Specify Excitations

- **To Create Coil Terminals**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Press **OK**
  - Change the name of the resulting sheet to **Coil_Terminal**
  - Select the sheet **Coil_Terminal** from history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - Select the sheet **Coil_Terminal_Separate1** and select the menu item ***Edit > Delete***
- **Assign Excitation**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **Amp_Turns**
    3. Type: **Stranded**
    4. Press **OK**
  - In Add Variable window,
    1. Value: **576**
    2. Press **OK**



Assign Boundary

- Since the Coil and the Yoke are conducting materials which touch, an insulating boundary must be applied to prevent the current from flowing from the Coil into the Yoke.
- **To Assign Insulating Boundary**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Insulating***
  - In Insulating Boundary window, press **OK**

Create Simulation Region

- **To Create Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding Data: **Pad all directions similarly**
    2. Padding Type: **Percentage Offset**
    3. Value: **100**
    4. Press **OK**

Assign Parameters

- **Assign parameter: Force**
  - Select the object **Armature** from the history tree
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Force***
  - In Force Setup window,
    1. Name: **Force1**
    2. Type: **Virtual**
    3. Press **OK**
- **Assign parameter: Inductance**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Current1
       - Include: ☑ **Checked**
    2. Press **OK**

Assign Mesh Operations

- **Assign Mesh Operations for Armature**
  - Select the object **Armature** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Armature_Inside**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**
- **Assign Mesh Operations for Yoke**
  - Select the object **Yoke** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Yoke_Inside**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**

Analysis Setup

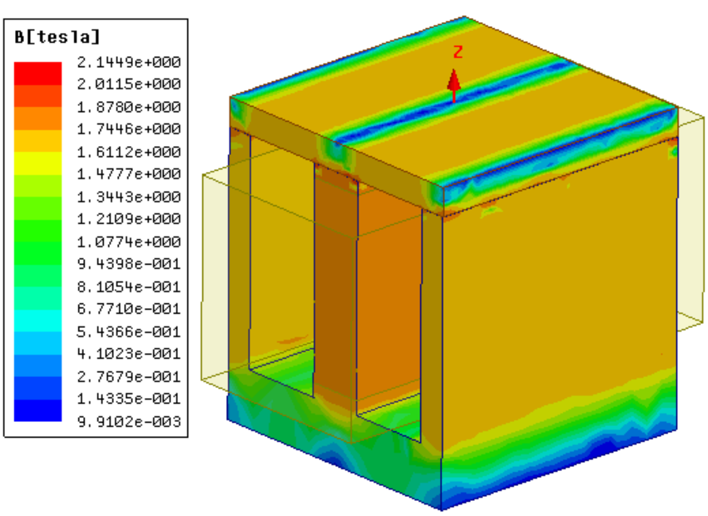
- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup window,
    1. **General** tab
       - Percentage Error: **3**
    2. **Convergence** tab
       - Refinement Per Pass: **15 %**
    3. **Solver** tab
       - Nonlinear Residuals: **0.005**
    4. Press **OK**

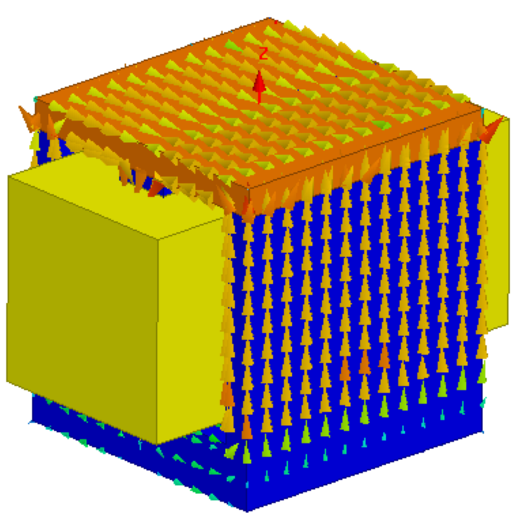
Analyze

- **To Start the Analysis**
  - Select the menu item *Maxwell 3D > Analyze All*

Results

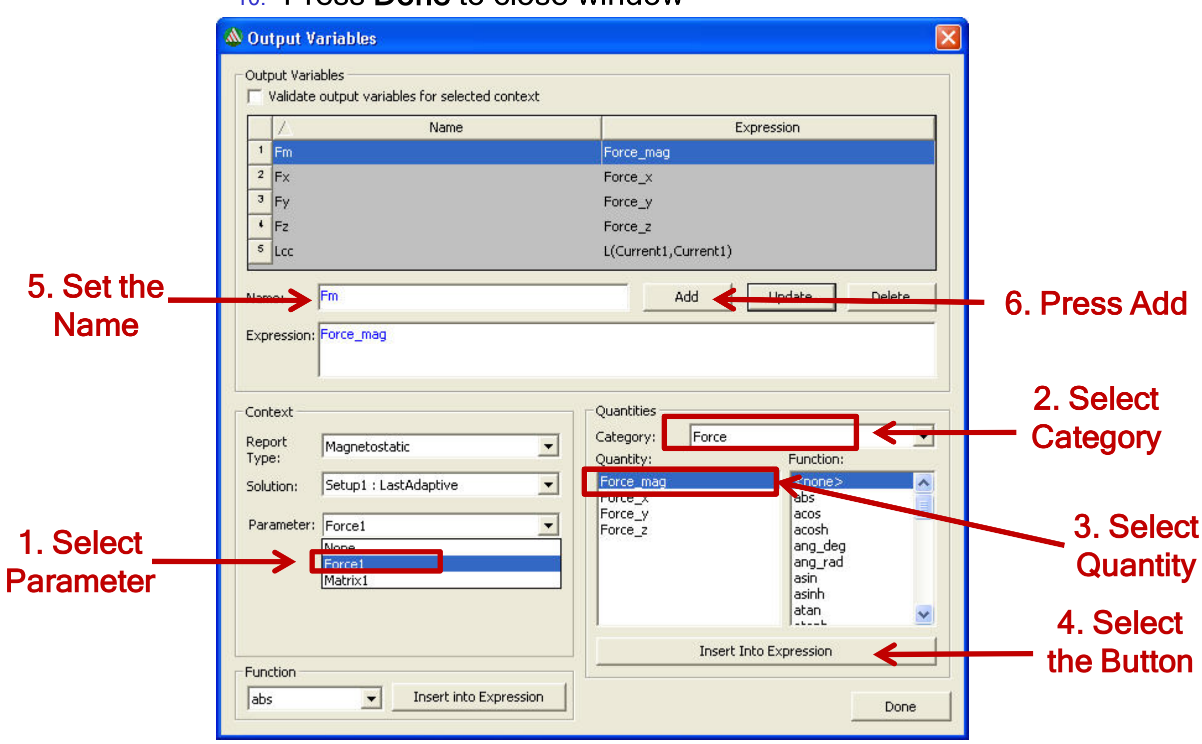
- **To View Solution Data**
  - Select the menu item *Maxwell 3D > Results > Solution Data*
  - To view force values
    1. Select the **Force** tab
    2. Reported force value is around **263 N**
  - To view inductance values
    1. Select the **Matrix** tab
    2. Reported value is around **415 nH**
- **To Create Field Plots**
  - Press **Ctrl** and select the objects **Armature** and **Yoke**
  - Select the menu item *Maxwell 3D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**
- Follow the same procedure to plot the B vector. Scale the size and length of the arrows by double clicking on the legend.





Setup the Parametric Analysis for ECE

- The ECE model will have two input and two output variables
  1. Input: **amp_turns** and **gap** (variables already defined)
  2. Output: Armature Force and Coil Inductance (need to create variables)
- **To Create Output Variables**
  - Select the menu item ***Maxwell 3D > Results > Output Variables***
  - In Output Variables window,
    1. Parameter: **Force1**
    2. Category: **Force**
    3. Qunatity: **Force_mag**
    4. Select the buttom **Insert Into Expression**
    5. Specify the name as **Fm** and select **Add**
    6. Similarly create variables **Fx, Fy** and **Fz** for Force_x, Force_y and Force_z respectively
    7. Change the Parameter to **Matrix**
    8. Change Category to **L** and Quantity to **L(Current1, Current1)** and select the button **Insert Into Expression**
    9. Specify the name as **Lcc** and select **Add**
    10. Press **Done** to close window



Setup Parametric Analysis

- **To Setup Parametric Sweep**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Parametric***
  - In Setup Sweep Analysis window, on **Sweep Definition** tab, select **Add**
  - In Add/Edit Sweep window,
    1. Variable: **Gap**
    2. Select **Linear Step**
    3. Start: **0.001 in**
    4. Stop: **0.006 in**
    5. Step: **0.001 in**
    6. Select **Add >>**
    7. Change Variable to **Amp_Turns**
    8. Select **Linear Count**
    9. Start: **1 A**
    10. Stop: **1000 A**
    11. Count: **4**
    12. Select **Add >>**
    13. Select **OK**
  - Select **Table** tab to view the entire parametric table
  - Select **Calculations** tab, select **Setup Calculations**
  - In Add/Edit Calculation window,
    1. Parameter: **Force1**
    2. Category: **Output Variables**
    3. Quantity: **Fm**
    4. Click on **Add Calculation**
    5. Repeat same steps for **Fx, Fy** and **Fz**
    6. Change Parameter to **Matrix1**
    7. Category: **Output Variables**
    8. Quantity: **Lcc**
    9. Click on **Add Calculation**
    10. Press **Done**
  - On **Options** tab
    1. Copy geometrically equivalent meshes: ☑ **Checked**
  - Press **OK** to close window

*	Amp_turns	Gap
1	1A	0.001in
2	334A	0.001in
3	667A	0.001in
4	1000A	0.001in
5	1A	0.002in
6	334A	0.002in
7	667A	0.002in
8	1000A	0.002in
9	1A	0.003in
10	334A	0.003in
11	667A	0.003in
12	1000A	0.003in
13	1A	0.004in
14	334A	0.004in
15	667A	0.004in
16	1000A	0.004in
17	1A	0.005in
18	334A	0.005in
19	667A	0.005in
20	1000A	0.005in
21	1A	0.006in
22	334A	0.006in
23	667A	0.006in
24	1000A	0.006in

Solve Parametric Sweep

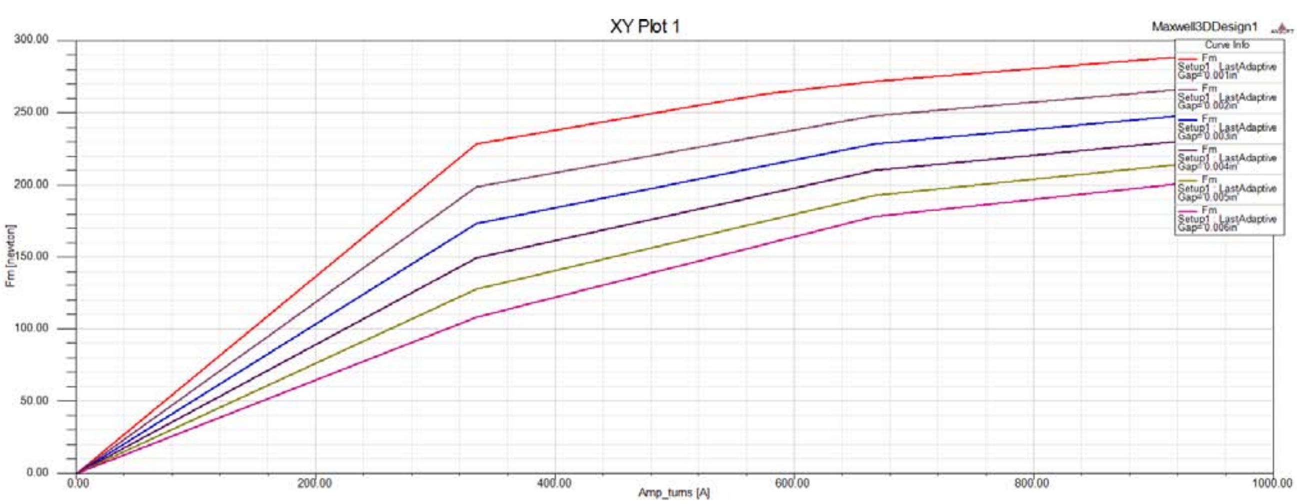
- **To Solve Parametric Analysis**
  - Select the **ParametricSetup1** tab under Optimetrics from project manager window
  - Right click and select **Analyze**

Solution Data for Parametric Sweep

- **To View Solution Data**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Optimetrics Results***

Create Plot of Force vs. Current for each gap

- **To Create Plot**
  - Select the menu item ***Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot***
  - In Report window,
    1. Parameter: **Force1**
    2. Primary Sweep: **Amp_Turns**
    3. Category: **Output Variables**
    4. Quantity: **Fm**
    5. Families: **Available variations**
    6. Select **New Report**

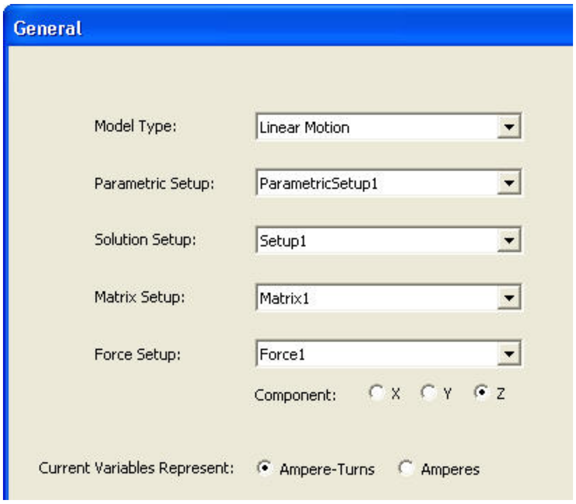


ECE from the Parametric Analysis

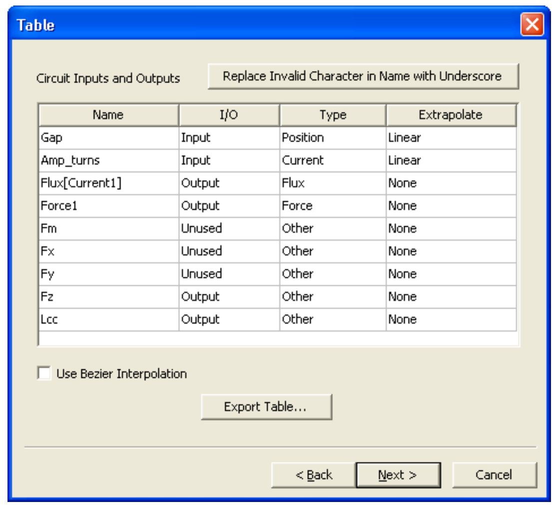
- In order to ultimately perform a system simulation in Simplorer an equivalent magnetic model must exported from Maxwell 3D. This model contains input pins for current and gap and outputs for force and flux linkage. The following steps show how to create this equivalent model. (Note: if you do not intend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

Extract Equivalent Circuit

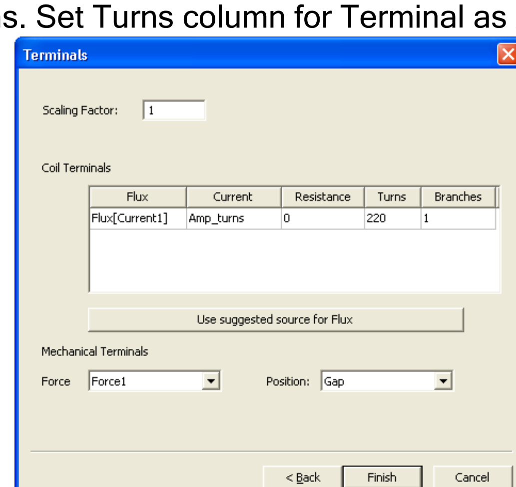
- **To Extract Equivalent Circuit**
  - Launch Simplorer and in Simplorer, select the menu item ***Simplorer Circuit > SubCircuit > Maxwell Component > Add Equivalent Circuit***
  - In Maxwell Equivalent Circuit Model window,
    1. Source Project File: Browse to the location of saved **Maxwell Project** file and select it
    2. Design Type: Select **3D**
    3. Select the Button **Extract Equivalent Circuit**
  - Maxwell will be launched and source file will open. In Maxwell window,
    1. Ensure Model Type is set to **Linear Motion**
    2. Under Parametric Setup, select the parametric setup for which circuit needs to extracted
    3. Select the component **Z** as only Z component of force will be transferred
    4. Select Current Variable as **Ampere-Turns** as we have specified ampere-turns value for current (as we specified stranded Conductor)
    5. Press **Next**



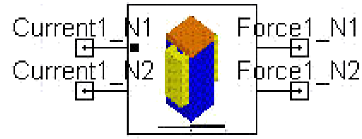
- In the next window
  1. As the parameters **Fm, Fx** and **Fy** will not be used in simplorer circuit, change the **I/O** column for these parameters to **Unused**
  2. Make sure that the data types are correct: Gap corresponds to **Position** and Amp_Turns corresponds to **Current**.
  3. Optionally if data needs to be exported in tabular format, select **Export Table** button
  4. Press **Next**



- Next window defines the scaling factor and the Terminals of the conservative nodes in Simplorer. The conservative nodes will have their Across and Through quantities solved by Simplorer, ensuring the physical meaning of the simulation. The Flux (and therefore the EMF) is the electrical Across quantity in this case. The current is the Through quantity. In the Mechanical domain, gap is the Across quantity whereas Force1 is the Through quantity.
- In Maxwell, we specified the conductor as stranded but we have not specified number of Turns. Set Turns column for Terminal as **220** and select **Finish**

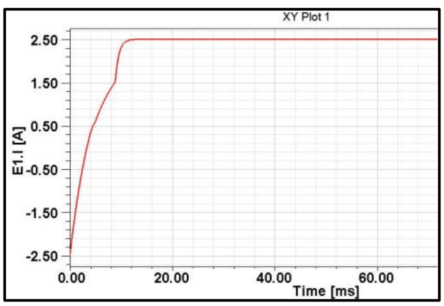


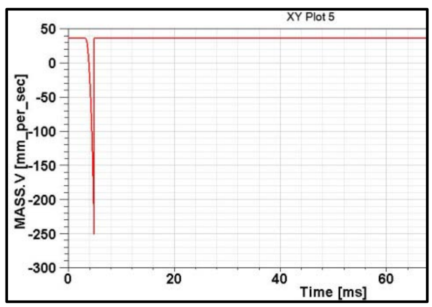
- Return to Simplorer window, and select **OK** in the Maxwell Equivalent Circuit Model window
- Place the resulting component on the Project Page

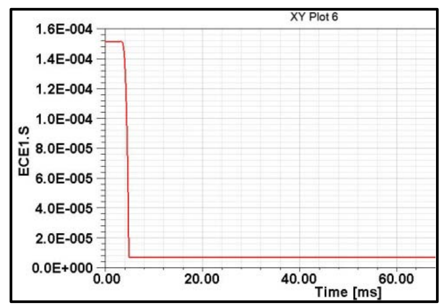


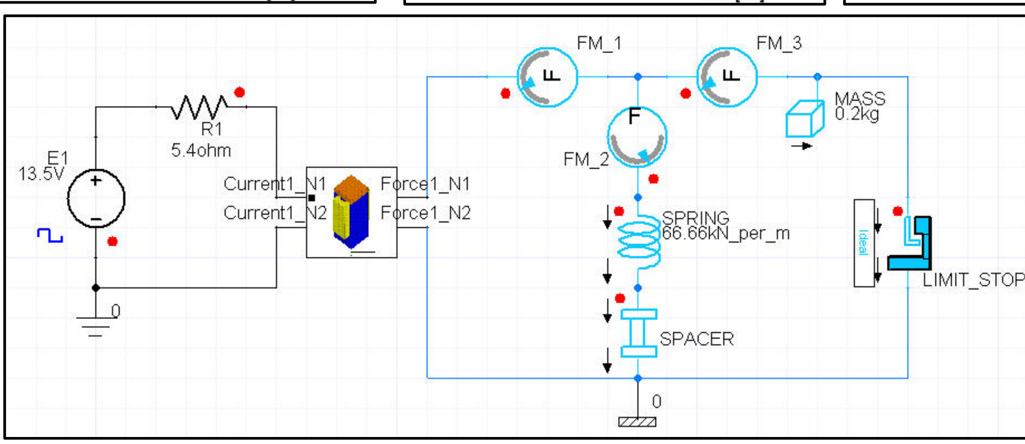
Simplorer Circuit

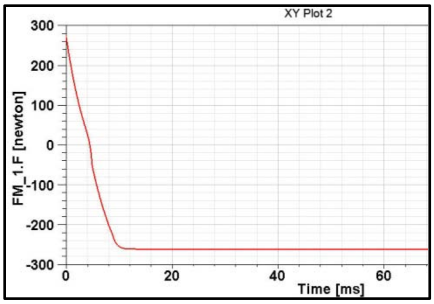
- **Build Simplorer Circuit**
  - Although beyond the scope of this example, build a circuit for a coupled electrical and mechanical simulation as shown below.
  - This coupled simulation takes into account flux linkage and back emf as well as mechanical mass and spring forces so that the closing time of the actuator can be determined.

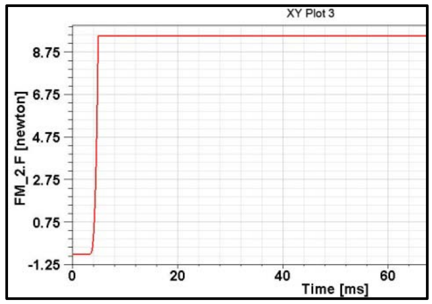


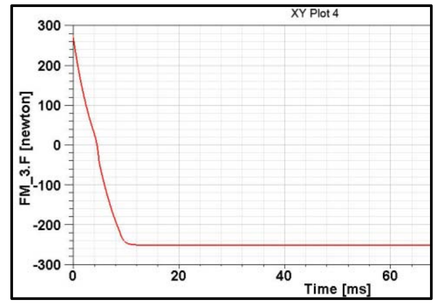












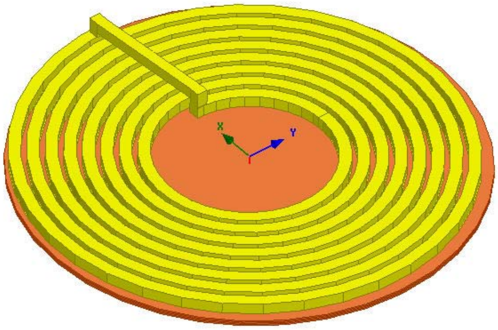
Basic Exercises – Maxwell Link with ANSYS Mechanical

Link between ANSYS Maxwell 3D and ANSYS Mechanical

- This exercise describes how to set up a Maxwell 3D Eddy Current project and then link the losses to ANSYS Mechanical for a thermal calculation

3D Geometry: Iron Disk above a Spiral Coil

- A sinusoidal 500 Hz current will be passed through a spiral coil which will induce eddy currents in an iron disk causing it to heat up.



Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Eddy Current**
    2. Click the **OK** button

Set Model Units

- **To Set Units**
  - Select the menu item ***Modeler > Units***
  - In Set Modeler Units window,
    1. Select units: **cm** (centimeters)
    2. Press the **OK** buttom

Create Coil

Draw Spiral

- Select the menu item ***Draw > User Defined Primitive > SysLib > SegmentedHelix > PolygonHelix***
- In User Defined Primitive Operation window,
  1. PolygonRadius: **1.5 cm**
  2. StartHelixRadius: **15 cm**
  3. RadiusChange: **3.1 cm**
  4. Pitch: **0 cm**
  5. Turns: **8**
  6. Leave other parameters as default and press **OK**
- Change the name of the object to **Coil** and color to **Yellow**
- Change the material of the object to **Copper**

Parameters | Info

Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate System	Global			
DLL Name	SegmentedHelix/Polyg...			
DLL Location	syslib			
DLL Version	1.0			
PolygonSegments	4		4	Number of cross-sectio...
PolygonRadius	1.5	cm	1.5cm	Outer radius of cross-se...
StartHelixRadius	15	cm	15cm	Start radius from polygo...
RadiusChange	3.1	cm	3.1cm	Radius change per turn
Pitch	0	cm	0cm	Helix pitch
Turns	8		8	Number of turns
SegmentsPerTurn	36		36	Number of segments p...
RightHanded	1		1	Helix direction, non-zer...

Draw Box

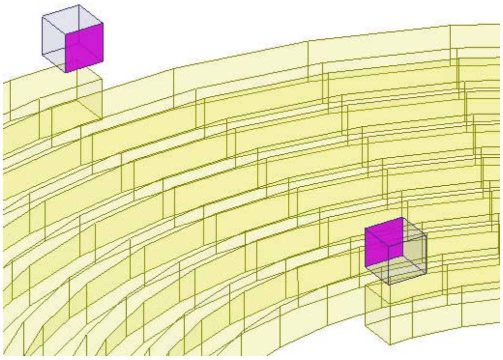
- Select the menu item ***Draw > Box***
  1. Using the coordinate entry fields, enter the box position
     - X: **14**, Y: **0**, Z: **-2**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the opposite corner
     - dX: **2**, dY: **2**, dZ: **-2**, Press the **Enter** key

Draw another Box

- Select the menu item ***Draw > Box***
  1. Using the coordinate entry fields, enter the box position
     - X: **40.5**, Y: **0**, Z: **-2**, Press the **Enter** key
  2. Using the coordinate entry fields, enter the opposite corner
     - dX: **-2**, dY: **-2**, dZ: **-2**, Press the **Enter** key

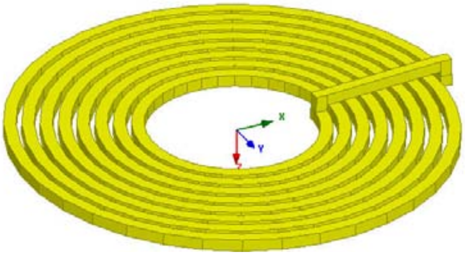
Basic Exercises – Maxwell Link with ANSYS Mechanical

- **Connect Surfaces**
  - Select the menu item ***Edit > Select > Faces***
  - Select the faces of the box as shown in below image



  - Select the menu item ***Modeler > Surface > Create Object from Face***
  - Select the resulting sheet objects from the history tree
  - Select the menu item ***Modeler > Surface > Connect***

- **Duplicate Boxes**
  - Select **Box1** and **Box2** from the history tree
  - Select the menu item ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **0**, dY: **0**, dZ: **1**, Press the **Enter** key
    3. Total Number: **2**
    4. Press **OK**
- **Unite Objects**
  - Select the menu item ***Edit > Select > Objects***
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Unite***



Create Disk

- **Create Regular Polyhedron**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z:**1.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **41**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**1**, Press the **Enter** key
    4. Number of Segments: **36**
  - Change the name of the Object to **Disk** and color to **Orange**
  - Change the material of the object to **cast_iron**

Specify Excitations

- **Create Coil Terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Press the **OK** button
  - Change the name of the resulting object to **Coil_Terminal**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - Delete all the resulting sheets apart from **Coil_Terminal**
- **Assign Excitation**
  - Select the object **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **I_Coil**
    2. Value: **125 A**
    3. Type: **Solid**
    4. Press **OK**

Resolve Skin Depth

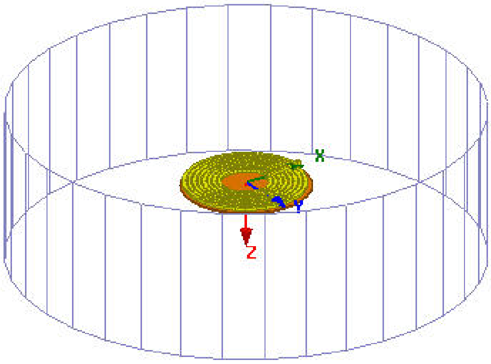
- **Compute Skin Depth**
  - Skin depth is a measure of how current density concentrates at the surface of a conductor carrying Alternating Current.
  - It is a function of the Permeability, Conductivity and frequency
  - Skin Depth in meters is defined as follows:

$$\delta = \sqrt{\frac{2}{\omega \sigma \mu_o \mu_r}}$$

  - Where:
    - ω is the angular frequency, which is equal to $2\pi f$. (f is the source frequency which in this case is 500Hz).
    - σ is the conductor's conductivity; for Cast Iron its 1.5e6 S/m
    - μ_r is the conductor's relative permeability; for Cast Iron its 60
    - μ_o is the permeability of free space, which is equal to $4\pi\times10^{-7}$ A/m.
  - For our model the skin depth is approximately 0.24 cm.
  - After three skin depths, the induced current will become almost negligible.
- **Create Surface layers to Assist with the Skin Depth Meshing**
  - Select the menu item ***Edit > Select > Faces***
  - Select the face on the disk that is closest to the coil
  - Select the menu item ***Modeler > Surface > Create Object from Face***
  - Select the resulting sheet objects from the history tree
  - Select the menu item ***Edit > Arrange > Move***
    1. Using the coordinate entry fields, enter the reference point of move vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the target point
       - dX: **0**, dY: **0**, dZ: **0.125**, Press the **Enter** key
  - Select the moved sheet **Disk_ObjectFromFace1**
  - Select the menu item ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **0**, dY: **0**, dZ:**0.125**, Press the **Enter** key
    3. Total Number: **2**
    4. Press **OK**

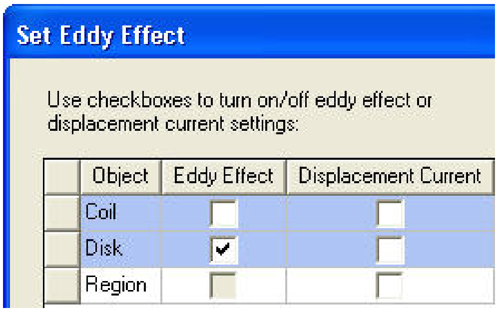
Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z:**-50**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **150**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**100**, Press the **Enter** key
    4. Number of Segments: **36**
  - Change the name of the object to **Region** and Change Display Wireframe: ☑ **Checked**



Set Eddy Effect

- **To Set Eddy Calculation for Disc**
  - Select the menu item ***Maxwell 3D > Excitations > Set Eddy Effects***
  - In Set Eddy Effects window,
    1. Coil
       - Eddy Effects: ☐ **Unchecked**
    2. Disk
       - Eddy Effects: ☑ **Checked**
       - Displacement Current: ☐ **Unchecked**
    3. Press **OK**



Analysis Setup

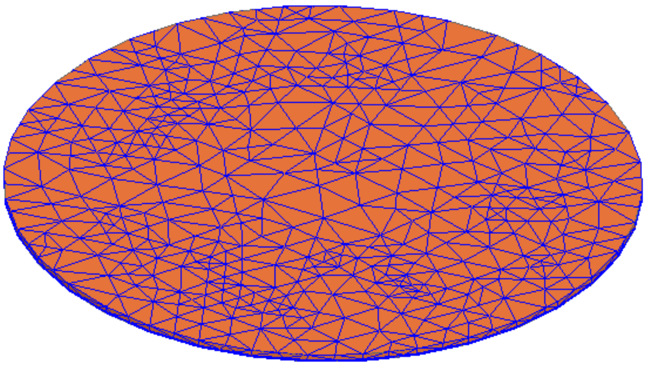
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **2**
    2. **Convergence** Tab
       - Refinement Per Pass: **15 %**
    3. **Solver** Tab
       - Adaptive Frequency: **500 Hz**
    4. Click the **OK** button

Analyze

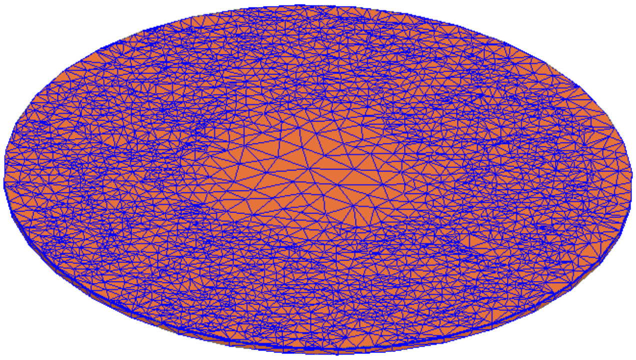
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Plot Mesh

- **To Plot Mesh on Disk**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    1. Press **Done**



Top View



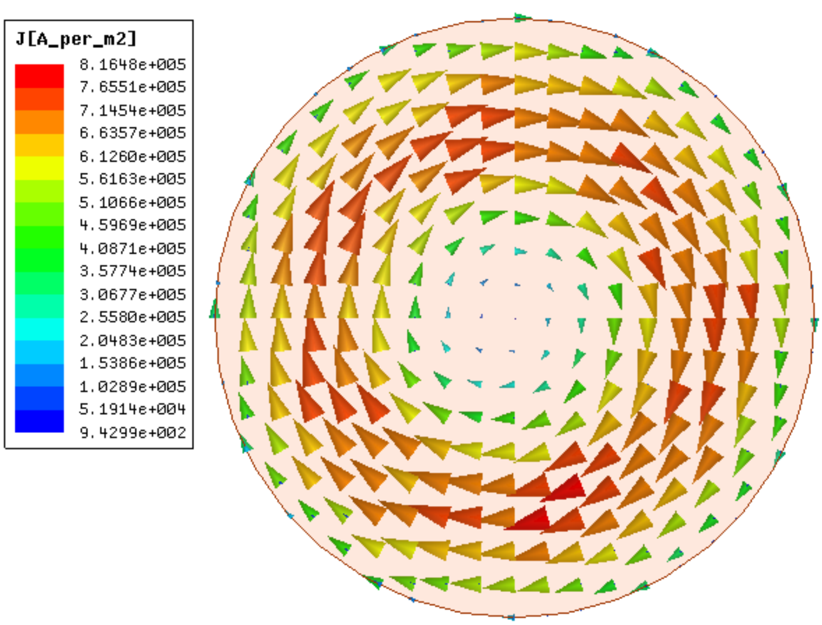
Bottom View: Notice the effect of the automatic adaptive meshing

Calculate Total Ohmic Loss

- **To Calculate Ohmic Losses in Disk**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
  - In Fields Calculator window,
    1. Select **Input > Quantity > OhmicLoss**
    2. Select **Input > Geometry**
       - Select **Volume**
       - Select **Disk** from the list
       - Press **OK**
    3. Select **Scalar >** ∫ **Integrate**
    4. Select **Output > Eval**
  - The evaluated value of losses in the **Disk** should be around **270.38 W**

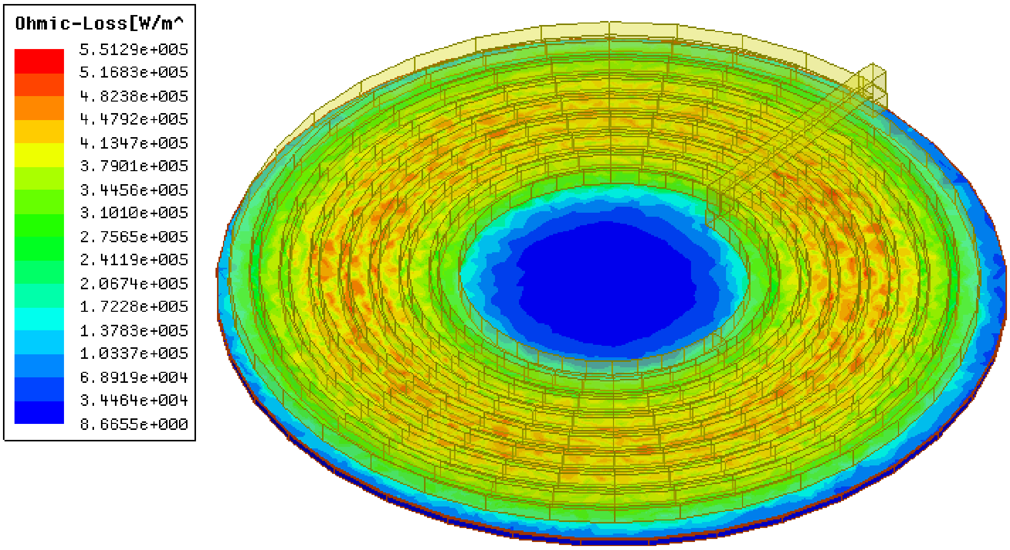
Plot Current Density Vectors

- **To Plot Current Density Vectors**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > J > Vector_J***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



Plot Ohmic Loss Distribution

- **To Plot Ohmic Losses on Disk**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > Other > Ohmic_Loss***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



This is the end of the Maxwell 3D Eddy Current Design

Save and Exit

- **Save the file**
  - Select the menu item ***File > Save As***
  - Save the file with the name **“Ex_9_12_BasicEddy_ANSYS_thermal”**
  - Select the menu item ***File > Exit***

Basic Exercises – Maxwell Link with ANSYS Mechanical

ANSYS Mechanical

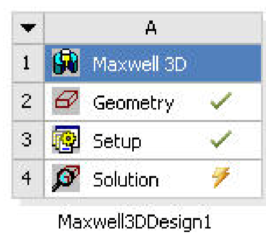
- In this section of tutorial, we will map the losses calculated by Maxwell to ANSYS Mechanical Steady State Thermal Solver. Solution in ANSYS Mechanical will give final temperature distribution of the objects.Mapping data from Maxwell to ANSYS Mechanical will be done through Workbench interface

Launch ANSYS Workbench

- **To Launch ANSYS Workbench**
  - Select the Microsoft **Start** button and Select **Programs > ANSYS 14.0 > Workbench**

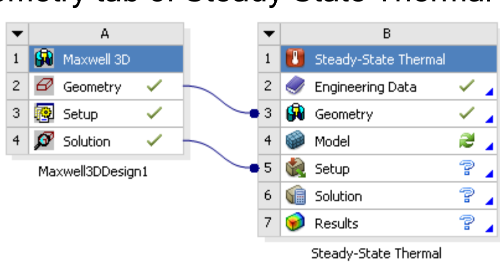
Import Maxwell File

- **To Import Maxwell Project**
  - In Workbench Project window, select the menu item ***File > Import***
  - Set the file type to **Maxwell Project File**
  - Browse to the file **Ex_9_12_BasicEddy_ANSYS_thermal.mxwl** and **Open** it
  - A Maxwell 3D Design is created as shown below



Create ANSYS Mechanical Design

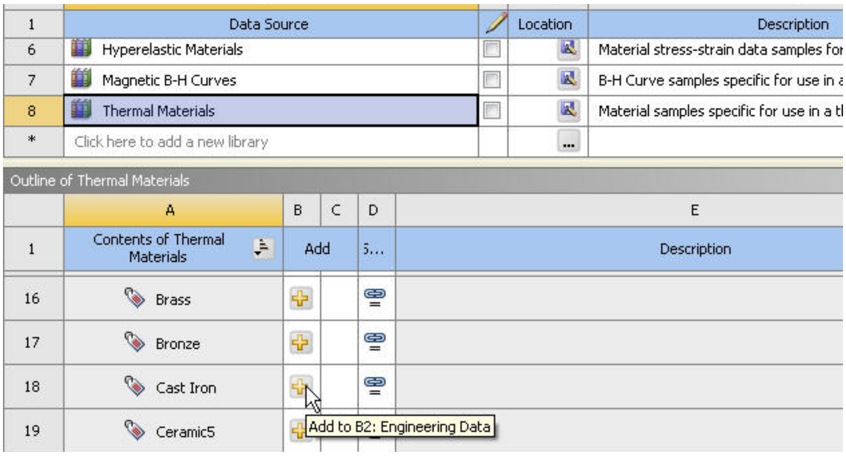
- **To Create Steady State Thermal Design**
  - Select a **Steady State Thermal** Analysis System from Analysis Systems list
  - Drag and drop it on the **Solution** tab of Maxwell 3D Design
  - Similarly drag and drop the **Geometry** tab of Maxwell on the **Geometry** tab of Steady State Thermal system. This will create a link between Maxwell Solution and Setup of ANSYS Mechanical
  - Right click on Solution tab of Maxwell 3D Design and Select **Update** to update the solution cell
  - Right click on the Geometry tab of Steady State Thermal system and select **Refresh**



Define Material Database

To Define Material Database

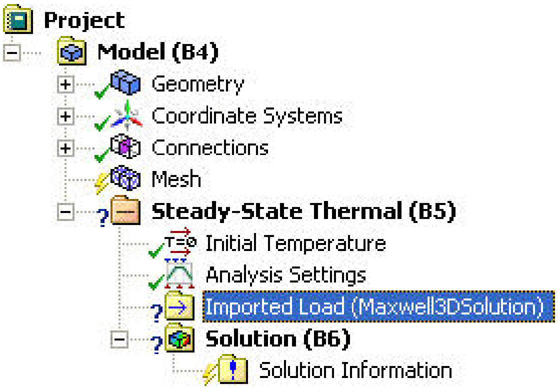
- Right click on the tab **Engineering Data** and select **Edit**
- In Engineering Data window,
  1. Select the icon **Engineering Data Sources** 
  2. Select the tab **Thermal Materials** from Data Sources
  3. Locate **Cast Iron** material from the list and select **Add**



Launch ANSYS Mechanical

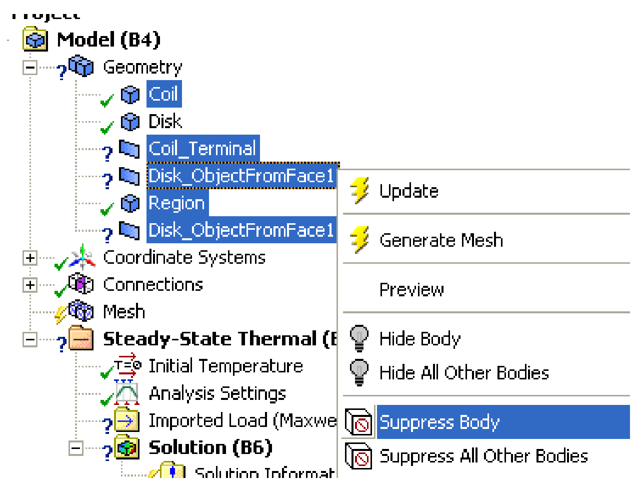
To Launch ANSYS Mechanical

- Right click on **Model** tab of Steady State Thermal analysis system and select **Edit**
- A tab corresponding to Maxwell data is automatically added in the tree to enable data mapping

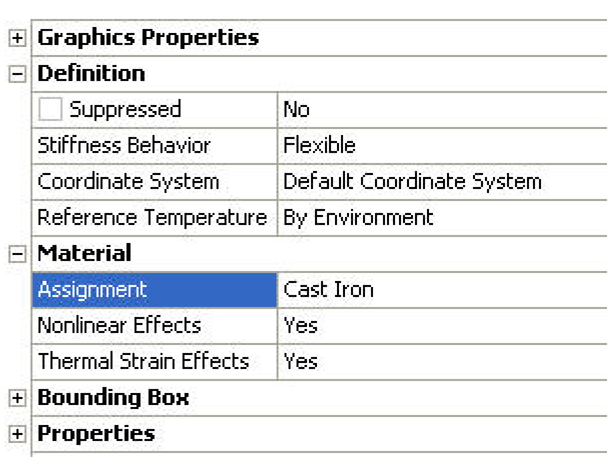


Basic Exercises – Maxwell Link with ANSYS Mechanical

- **Setup Geometry**
  - **Suppress Unwanted Objects**
    - Expand the Project tree for **Geometry** under Model
    - Select all bodies apart from **Disk,** right click on the Bodies and select **Suppress**
    - This will keep only **Disk** for analysis and rest will be ignored



  - **Specify Material**
    - Select the **Disk** from the tree and goto Details View window
    - In Details View window,
      1. Material
         - Assignment: set to **Cast Iron**



Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	Cast Iron
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	

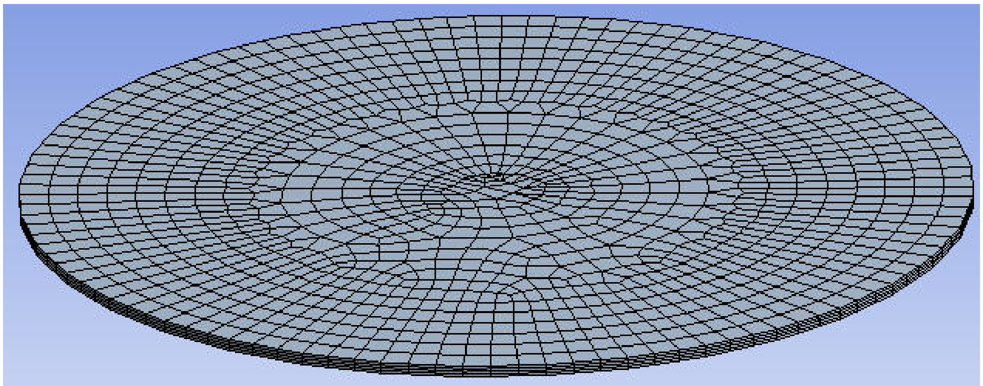
Basic Exercises – Maxwell Link with ANSYS Mechanical

Generate Mesh

- Mesh specifications are important from the data mapping perspective. The mesh should be sufficiently refined in order to have good accuracy in mapped heat losses from Maxwell to ANSYS Mechanical
- **To Set Mesh Parameters**
  - Right click on **Mesh** tab from specification tree and select ***Insert > Method***
  - In Details View window,
    1. Geometry: select the body **Disk**
    2. Method: **Sweep**
    3. Sweep Num Divs: **5**

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled
Target	Program Controlled
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	5
Sweep Bias Type	No Bias
Element Option	Solid

  - This will ensure four elements across the thickness of the disk and give the refinement close to the mesh we have in Maxwell
- **To Generate Mesh**
  - Right click on **Mesh** tab and select **Generate Mesh**



Map Data from Maxwell

- **To Map Heat Loss Data from Maxwell**
  - Right click on the **Imported Load (Maxwell3DSolution)** and select ***Insert > Heat Generation***
  - In Details View window,
    1. Geometry: Select the body **Disk**
  - Right click on the tab **Imported Heat Generation** and select **Import Load**
  - Heat losses calculated in Maxwell be mapped to the mesh in ANSYS Mechanical
  - A Summery of mapped heat losses is shown below Imported Heat Generation tab. Scaling factor shown in this summery should be close to **1** to ensure correct data mapping. If not, mesh needs to be refined in important regions

Exporting Volume Loss Density...

Object	Total Loss	Scaling Factor
Disk	270.385W	0.968471

Specify Convective Boundary

- **To Specify Convective Boundary**
  - Right click on **Steady State Thermal** tab from the specification tree and select ***Insert > Convection***
  - In Details View window,
    1. Geometry:
       - Change Selection Filter to **Faces**
       - Right click in Graphic window and select **Select All**
       - Press **Apply** in details view window
    2. Film Coefficient: **10 W/m²°C**

Scope	
Scoping Method	Geometry Selection
Geometry	38 Faces
Definition	
ID (Beta)	106
Type	Convection
Film Coefficient	10. W/m²·°C (ramped)
Ambient Temperature	22. °C (ramped)
Suppressed	No
Fluid Flow (Beta)	No

Create Temperature Plot

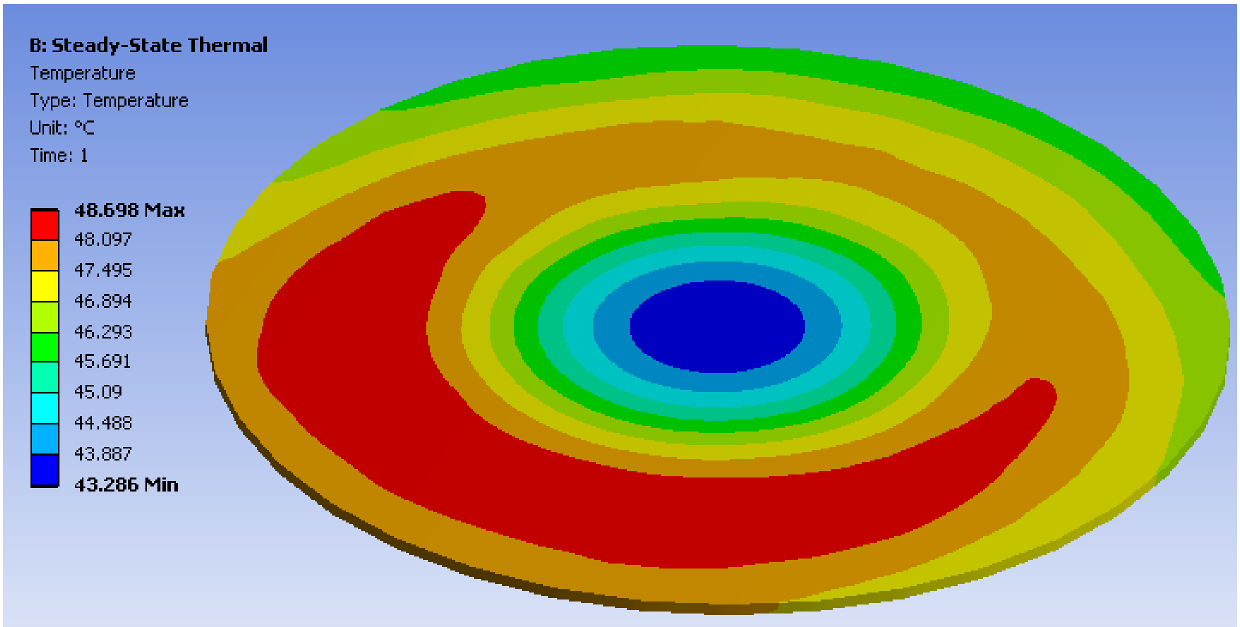
- **To Create Temperature Plot for Disk**
  - Right click on **Solution** tab from specification tree and select ***Insert > Thermal > Temperature***

Solve

- **To Run the Solution**
  - Right click on **Solution** tab from specification tree and select **Solve**

Temperature Plot

- **To View Temperature Plot**
  - Select the **Temperature** plot from specification tree under **Solution**
  - Temperature distribution on **Disk** will be displayed in graphic window



Large Motion – its Quick Implementation Using the Maxwell 3D Transient Solver

- Maxwell Transient is able to consider *interactions* between *transient electromagnetic fields* and *mechanical motion* of objects.
- Maxwell Transient (with motion) includes d*B* /d*t* arising from mechanically moving magnetic fields in space, i.e. moving objects. Thus, effects coming from so-called motion induced currents can be considered.
- In Maxwell rotational motion can occur around one single motion axis.
- This paper represents a quick start to using *rotational motion*. It will exercise rotational motion in Maxwell 3D using a rotational actuator (experimental motor) example.
- Subsequent papers will demonstrate *rotational motion* in more depth, *non-cylindrical rotational motion* using a relays example, as well as *translational motion* which a solenoid application will serve as an example for.
- The goal of these papers is solely to show and practice working with large motion in Maxwell. It is neither the goal to simulate real-world applications, nor to match accurately measured results, nor will these papers show in detail how to setup and work with other Maxwell functionality. Please refer to the corresponding topics.

Quickstart – Rotational Motion Using a Rotational Actuator Example

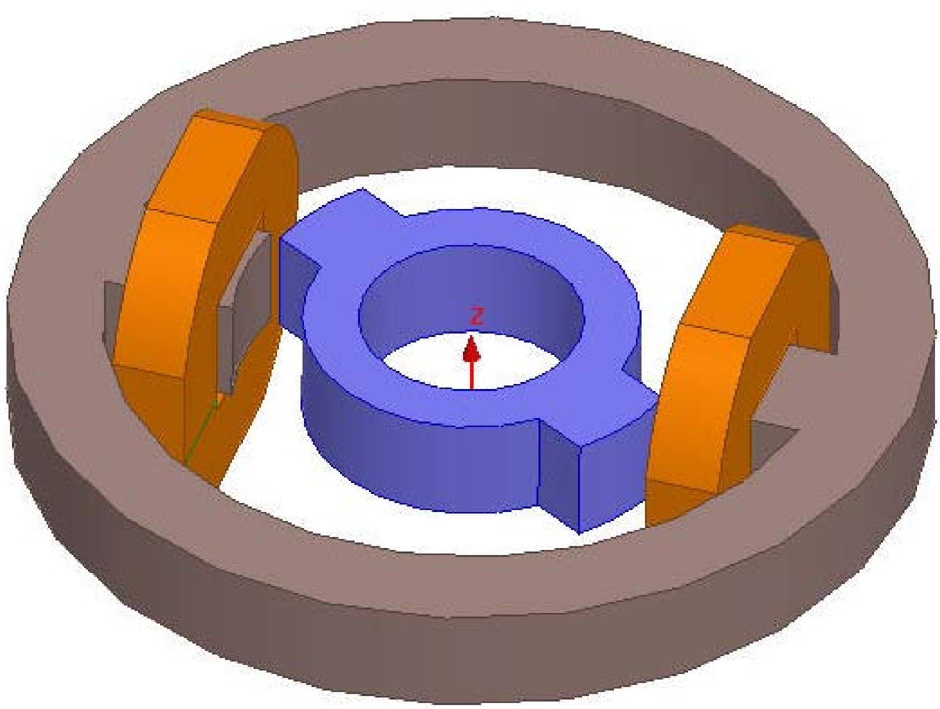
- Maxwell Transient with large motion is a set of advanced topics. Users should have thorough knowledge on Maxwell fundamentals as well as Maxwell Transient (without motion) prior to approaching large motion. If necessary, please consult the proper training papers, help files, manuals, and application notes.
- We will exercise the following in this document:
  - Create a new or read in an existing *rotational actuator model* – to serve as an experimental testbench for large motion
  - Prepare and adapt this existing actuator model to our needs
  - Apply large motion to the rotational actuator
    - Create the *band* object
    - Setup rotational motion
    - Mesh

Basic Exercises – Transient – Large Motion - Rotational

- Perform basic large motion tests
  - *"Large Rotational Standstill"* test
  - *"Large Rotational Constant Speed"* test
  - *"Large Rotational Transient Motion"* test
    - Compute magnetic rigidity and mechanical natural frequency
    - Estimate timestep for transient solver
- Make a *field animation* with large motion

Open Input File

- **To Open the file**
  - Select the menu item ***File > Open***
  - Browse to the file **Ex_9_13_Large_Motion_Rotational.mxwl** and open it
  - File should look as shown below



Setup and Verify the Electromagnetic Part

- Prior to employing large motion, the electromagnetic part of the model should work correctly. Users are well advised not to setup a complex model completely at once and then try to simulate. Rather, they should work in steps. Especially in cases where the model includes eddy current effects, external circuits, and large motion, extra steps should be taken to verify the correctness of the setup for each individual property. After that, all properties can be considered together.
- We use stranded windings with constant current (to generate a fixed stator flux vector around which Rotor1 will oscillate later). Also, eddy effects will be excluded.
- Perform a test simulation on the electromagnetic part alone. If desired, play with various excitations, switch eddy effects in Stator1 and Rotor1 on and off, vary material properties, etc. For each test check the electromagnetic fields for correctness.
- Refer to the corresponding topics on materials, boundaries, excitations, meshing, transient simulations without motion, and post processing.
- If the electromagnetic part without motion effects yielded correct results, make sure to re-apply the same model setup as elaborated at the previous page

Rotational Large Motion – The Maxwell Approach

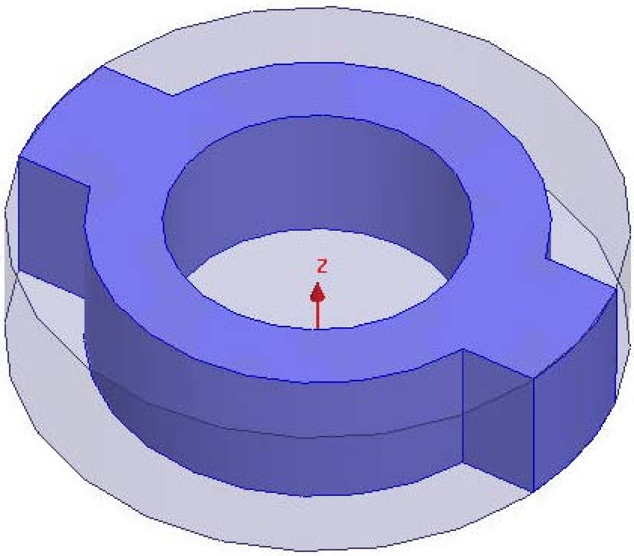
- Maxwell separates moving from non-moving objects.
- All moving objects must be enclosed by one so-called *band* object.
- For rotational motion, the *band* object must be cylindrical with segmented outer surface, i.e. a regular polyhedron.
- Maxwell considers all moving objects (inside the *band*) to form one single moving object group.
- *Constant Speed* mode:
  - If the model is setup to operate in constant speed mode (see below), Maxwell will not compute mechanical transients.
  - However, changing magnetic fields owing to speed ω_m, i. e. dB/dt effects are included in the field solution.
- *Mechanical Transient* mode:
  - In case inertia was specified, Maxwell will compute the motion equation in each time step.

$$\boxed{J_m \cdot \mathrm{d}^2\varphi_m(t) / \mathrm{d}t^2 \;+\; k_D(t) \cdot \mathrm{d}\varphi_m(t) / \mathrm{d}t = T_\psi(t) \;+\; T_m(t)}$$

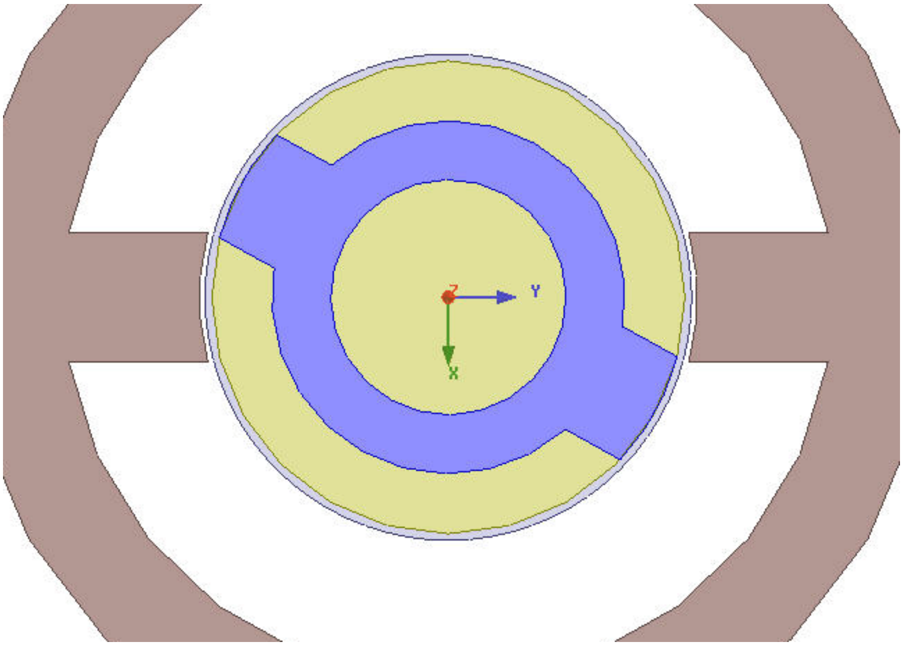
  - See Appendix A for a variable explanation.

Setup Model

- First, let's examine the moving parts to comply with Maxwell's conventions:
  - All moving objects can be separated from the stationary objects and can be combined to one single rotating group. All moving objects be considered to perform the same cylindrical motion.
  - However, the *band* object has a hole that we have to fill first.
  - Because there is only one moving object, there is no immediate need to enclose it. But filling the hole can also be achieved by fully enclosing Rotor1 by an extra object. We will first do this.
- **Create Enclosure for Rotor**
  - Outer radius of Rotor is around 51.05 mm and height is 25.4 mm
  - Select the menu item ***Draw > Cylinder***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **51.05**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0** dY: **0**, dZ: **25.4**, Press the **Enter** key
  - Change the name of the resulting object to **Region_Inner** and material to **Vacuum**
  - **Note:** Instead of using coordinate entry field, users can directly pick vertices of **Rotor** to specify radius and height directly.



- **Create Band Object**
  - We want a regular polyhedron that fully encloses Region_Inner.
  - Outer surface segmentation should be between 1° and 5°, i. e. we will have between 360 and 72 outer surface segments.
  - The *band* object should preferably cut through the middle of the airgap, leaving about the same space to Rotor1 and Stator1. However, this is not a must.
  - Inner radius of the Stator is around 53.75 mm while outer radius of Rotor is around 51.05 mm
  - Thus middle position comes out to be 52.4 mm. So we will use 52.5 mm as radius of Band object
  - Select the menu item ***Draw > Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - X: **0**, Y: **0**, Z: **-12.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **52.5**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Using the coordinate entry fields, enter the height
       - dX: **0**, dY: **0**, dZ:**50**, Press the **Enter** key
    4. Number of Segments: **72**
  - Change the name of the object to **Band** and material to **Vacuum**



Setup1: Large Rotational Standstill

Assign Motion

- **To Specify Motion**
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Model > Motion Setup > Assign Band***
  - In Motion Setup window,
    - **Type** tab
      1. Motion Type: **Rotation**
      2. Rotation Axis: **Global:Z**
      3. Positive: ☑ **Checked**
    - **Data** tab
      1. Initial Position: **0 deg**
         - Thus, motion will start at $t = 0$ with the rotor position being as drawn. A $\varphi_{m0} \neq 0$ would start with Rotor1 rotated by φ_{m0} from the drawn position.
      2. Rotate Limit: ☐ **Unchecked**
    - **Mechanical** tab
      1. Consider Mechanical Transient: ☐ **Unchecked**
      2. Angular Velocity: **0 rpm**
    - Press **OK**
  - Now, we have setup “Large Rotation Standstill“ Positive magnetic torque is generated around the positive z-axis (global coordinate system)
  - A mesh Operation is automatically created after motion setup which will ensure refinement of mesh in the gap region between Band and Stator/Rotor

Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup window,
    1. Stop Time: **20 ms**
    2. Time Step: **5 ms**
    3. Press **OK**

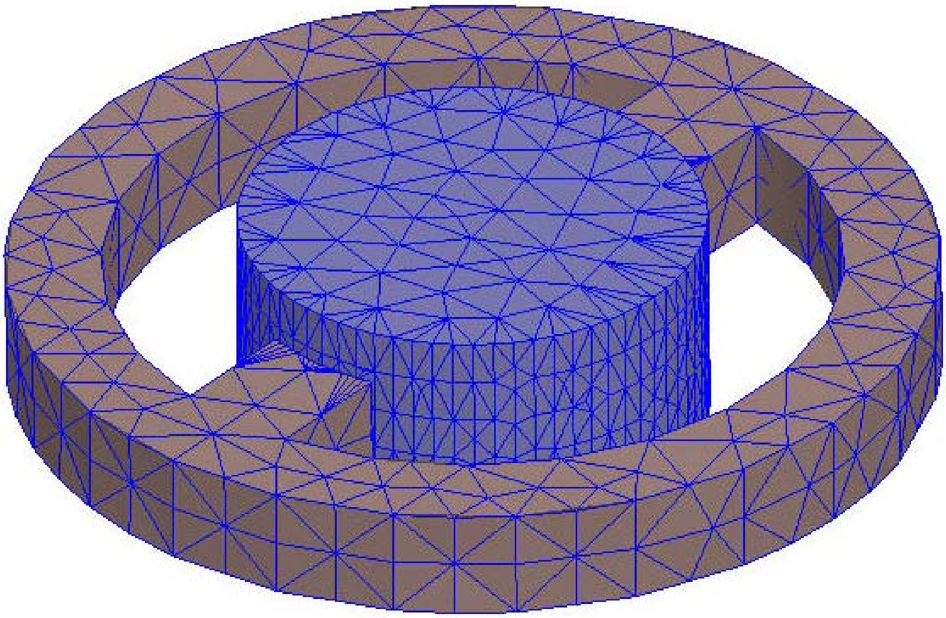
Mesh Operations

- Meshing is a very critical issue with respect to simulation speed and accuracy. For here, we will apply a rather coarse mesh only, by which the solver will just yield satisfactory results.
- **To Assign Mesh Operations on Band**
  - For torque computation, the most critical areas are the airgap and its immediate proximity. Thus, the *band* mesh is crucial for accurate results.
  - Our Band1 is 50 mm high, one segment is about 4.5 mm wide.
  - We will apply a length based mesh on the surface and inside of Band1. Tetrahedral edge length will be set to 20 mm. We will then see a mesh length of about 10 mm on the outer surface. This will do for these tests.
  - Select the object **Band** from the history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Band_Length**
    2. Restrict Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **20 mm**
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**
- **To Assign Mesh Operations on other Objects**
  - Press **Ctrl** and select the objects **CoilA** and **CoilB**
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Coils_Length**
    2. Restrict Length of Elements: ☑ **Checked**
    3. Restrict the Number of Elements: ☐ **Unchecked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**
  - **Note:** Simultaneously selecting CoilA and CoilB will try to assign 1000 tetrahedrals to both objects, i. e. about 500 to each

Basic Exercises – Transient – Large Motion - Rotational

- In Similar way specify Mesh operations of other objects as specified below
- Region_Inner
  1. Name: **Region_Inner_Length**
  2. Maximum Number of Elements: **2000**
- Rotor
  1. Name: **Rotor_Length**
  2. Maximum Number of Elements: **2000**
- Stator
  1. Name: **Stator_Length**
  2. Maximum Number of Elements: **2000**
- Region
  1. Name: **Region_Length**
  2. Maximum Number of Elements: **2500**

- **To Create Mesh**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Apply Mesh Operations***
  - Select all the objects except Region and select the menu item ***Maxwell 3D > Fields > Plot Mesh***



Run Solution

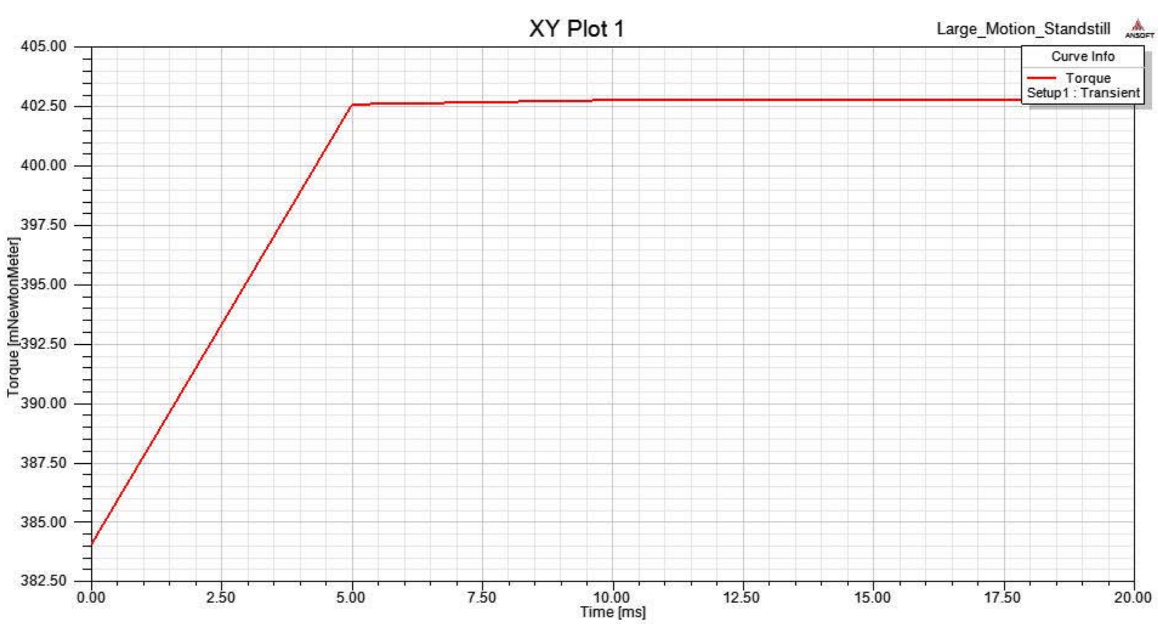
To Run the Solution

- Select the menu item *Maxwell 3D > Analyze All*

Results

Create Plot for Torque

- Select the menu item *Maxwell 3D > Results > Create Transient Report > Rectangular Plot*
- In Reports window,
  1. Solution: **Setup1: Transient**
  2. Parameter: **Moving1**
  3. X: **Default**
  4. Category: **Torque**
  5. Quantity: **Torque**
  6. Press **New Report**
- Report should show a constant torque value of **0.4Nm**



Basic Exercises – Transient – Large Motion - Rotational

Setup2 : Large Rotational Constant Speed

- We will now operate the rotational actuator at a very slow constant speed.
- Remember, there is only one magnetic excitation present in the model – namely constant coil current with stranded windings. Alternatively, Rotor1 could have been assigned permanent magnet properties. Eddy effects have been switched off for all objects.
- We can now use Transient with Large Motion to monitor *cogging torque* effects.

Copy Design

- **To Craete a Copy of Design**
  - Select the tab **Large_Motion_Standstill** from the Project manager window, right click and select **Copy**
  - Right click on the Project Name in Project Manager window and select **Paste**
  - Change the name of the design to **Large_Motion_ConstantSpeed**

Modify Rotation Specifications

- **To Modify Rotation**
  - Expand the tree for **Model** from Project Manager window
  - Double click on **MotionSetup1** to open Motion Setup window

  - In Motion Setup window,
    - **Data** tab
      1. Change Initial Position to **-61 deg**
    - **Mechanical** tab
      1. Change Angular Velocity to **1 deg_per_sec**
    - Press **OK**
  - Rotor as drawn has a -29° offset. This is taken to be the zero position for the transient solver. By giving an extra -61°, positive rotation of 1 °/s starts at -90°.

Modify Solution Setup

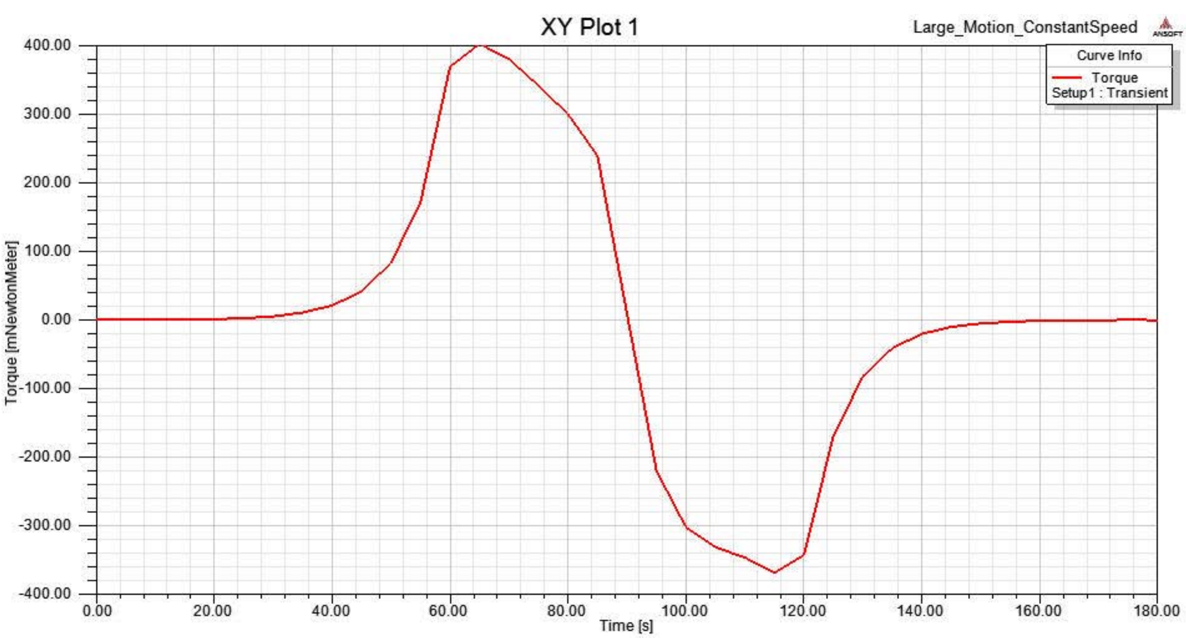
- **To Modify Solver Setup**
  - Expand the tree for **Analysis** from Project manager window
  - Double click on **Setup1** to open Solve Setup window
  - In Solve Setup window,
    1. Change Stop Time to **180 s**
    2. Chnage Time Step to **5 s**
    3. Press **OK**
  - By rotating at a speed of 1 °/s 180 s long, Rotor will move 180°, i. e. from -90° to +90°, at 5°/step.

Run Solution

- **To Run the Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***

Results

- **To View Plot**
  - Expand the tree for **Results** from Project Manager window
  - Double click on **Torque** plot that was created previously



Setup3 : Large Rotational Transient Motion

- We will now operate the actuator as a one-body oscillator.
- Inertia will be specified as well as some damping.
- We can expect Rotor to oscillate around the stator flux axis (y-axis) at some natural frequency f_0, which can be approximated as:

$$f_0 = \frac{1}{2\pi}\sqrt{\frac{c_\psi}{J}}$$

- J in kgm^2 is the total moment of inertia acting on Rotor.
- c_ψ in Nm/rad is the magnetic rigidity. As an analogy it can be understood as a mechanical spring spanned between Rotor and Stator, whose force coming from the magnetic field.
- We can roughly calculate rigidity c from the cogging torque function (stable limb):

$$c_\psi = \frac{\Delta T_\psi}{\Delta \varphi_m} \approx \frac{400\ \mathrm{mNm}}{\mathrm{rad}(10°)} = 2.3\ \mathrm{Nm/rad}$$

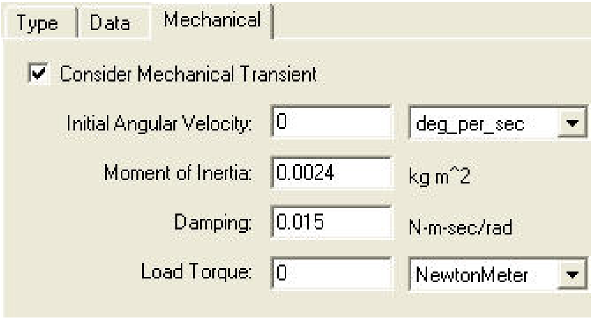
- Assuming inertia J = 0.0024 kgm^2, an approximated f_0 = 5 Hz results.
- This is sufficient for estimating the necessary timestep as far as mechanical oscillations are regarded.

Copy Design

- **To Craete a Copy of Design**
  - Select the tab **Large_Motion_ConstantSpeed** from the Project manager window, right click and select **Copy**
  - Right click on the Project Name in Project Manager window and select **Paste**
  - Change the name of the design to **Large_Motion_MechTransient**

Modify Rotation Specifications

- **To Modify Rotation**
  - Expand the tree for **Model** from Project Manager window
  - Double click on **MotionSetup1** to open Motion Setup window
  - In Motion Setup window,
    - **Data** tab
      1. Change Initial Position to **0 deg**
    - **Mechanical** tab
      1. Consider Mechanical Transient: ☑ **Checked**
      2. Initial Angular Velocity: **0 deg_per_sec**
      3. Moment of Inertia: **0.0024 Kgm^2**
      4. Damping: **0.015 N-m-sec/rad**
      5. Load Torque: **0 NewtonMeter**
    - Press **OK**
  - This causes 15 mNm resistive torque at 1 rad/s speed
  - We thus expect oscillation between -29° and +29° (w. r. t. stator flux axis) at f_0 < 5 Hz with damped amplitudes



Modify Solution Setup

- **To Modify Solver Setup**
  - Expand the tree for **Analysis** from Project manager window
  - Double click on **Setup1** to open Solve Setup window
  - In Solve Setup window,
    1. Change Stop Time to **0.5 s**
    2. Chnage Time Step to **0.01 s**
    3. Press **OK**
  - From f_0, we can expect a >200 ms cycle. At 10 ms timestep we will sample one cycle >20 times.

Run Solution

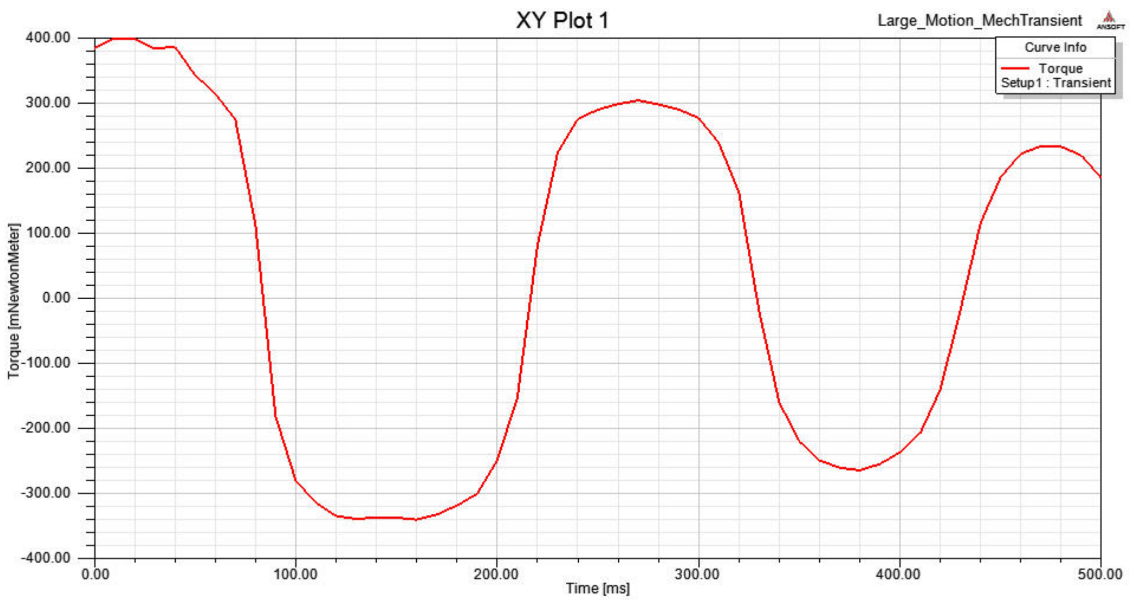
To Run the Solution

- Select the menu item *Maxwell 3D > Analyze All*

Results

To View Torque Plot

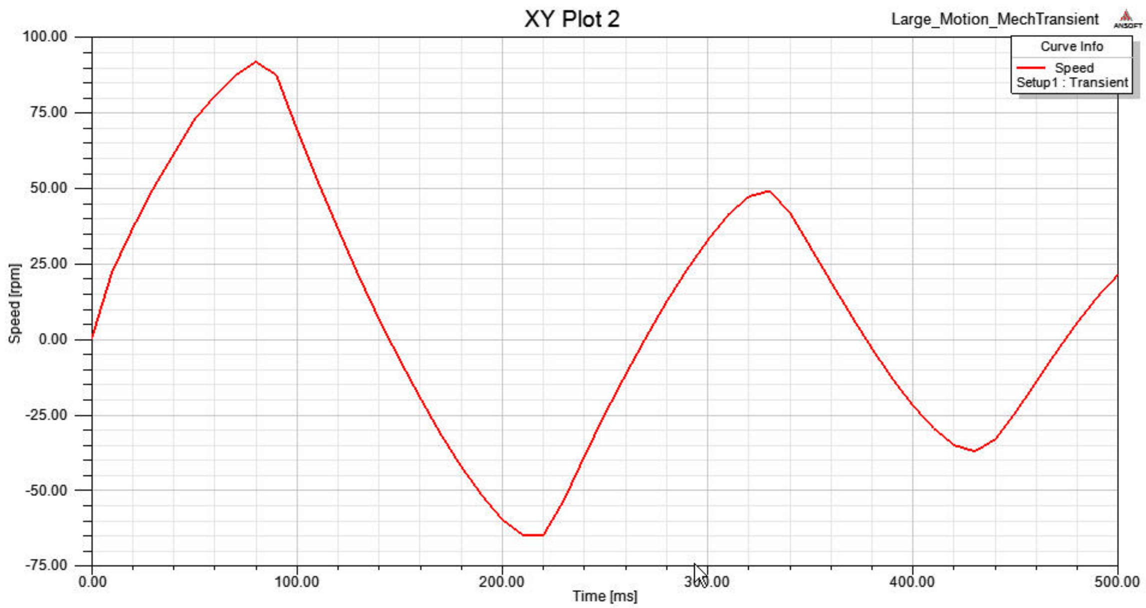
- Expand the tree for **Results** from Project Manager window
- Double click on **Torque** plot that was created previously



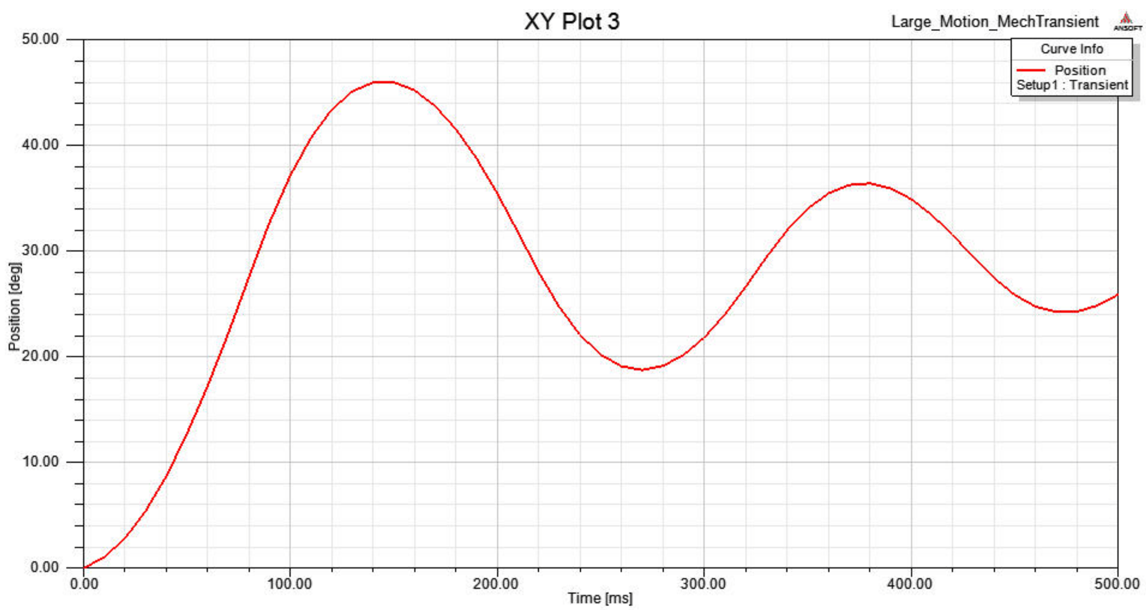
To Create Plots for Speed and Position

- Select the menu item *Maxwell 3D > Results > Create Transient Report > Rectangular Plot*
- In Reports window,
  1. Solution: **Setup1: Transient**
  2. Parameter: **Moving1**
  3. X: **Default**
  4. Category: **Speed**
  5. Quantity: **Speed**
  6. Press **New Report**

- Plot for Speed should look as below



- In Reports window,
  1. Change Category to **Position**
  2. Quantity: **Position**
  3. Press **New Report**



Appendix A: Variable Explanation:

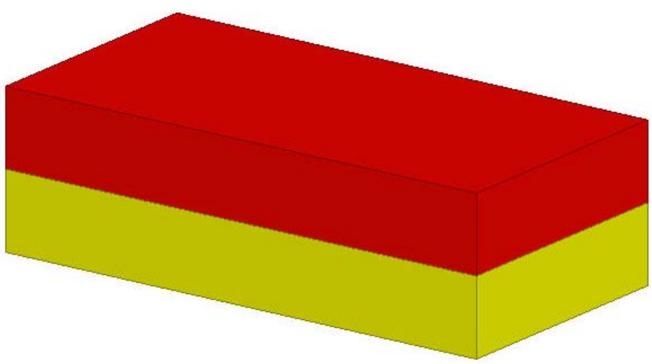
$\varphi_m(t)$	Mechanical angular position in rad (angles can also be given in degrees).
φ_{m0}	Initial φ_m in rad. Note that the *drawn rotor position* is considered as $\varphi_{m0} = 0$.
$\mathrm{d}\varphi_m(t) / \mathrm{d}t$, $\omega_m(t)$	Mechanical angular speed in rad/s.
ω_{m0}	Initial ω_m in rad/s.
$\mathrm{d}^2\varphi_m(t) / \mathrm{d}t^2$	Mechanical angular acceleration in rad/s^2.
J_m	Moment of inertia in kg·m^2. This is the total inertia acting on the moving object group. If extra inertia needs to be included (i. e. inertia not geometrically modeled), just add this to J_m.
$k_D(t)$	Damping koefficient in Nm·s/rad. For k_D = 1 Nm·s/rad, resistive torque of 1 Nm would be generated if the moving parts turn at 1 rad/s. k_D can be a function of t, ω_m, or φ_m.
T_ψ	Magnetically generated torque in Nm.
T_m	Mechanical extra torque in Nm, this can be a constant or a function of t, ω_m, or φ_m. Note, that a positive T_m value will accelerate rather than brake.
t	The current simulation time in s.

Introduction on the Electric Transient Solver

- This note introduces the Electro Transient solver based on a simple example. This solver is meant to solve the transient electric field with current flowing in "real" dielectrics before they reach electrostatic equilibrium.
- Practical applications for an electric transient solver range from low frequency biomedical applications to fine aspects of electric conduction in poor conductors, both insulators and semi-conductors. Charge relaxation is often of interest. Other application areas are distribution of electric fields in LED arrays, study of (transient) electric fields in insulators, effects related to non-linear conductivity materials used to control electric field stress in high voltage applications.

Maxwell's Capacitor

- In this example, we want to determine the transient electric field in the two layers of "real" dielectric material between two plates connected to a voltage source. Material properties in the two layers of dielectric are uniform within each layer but different between the two layers. We do not need to draw the plates; we just need to draw the two media regions.



Create Design

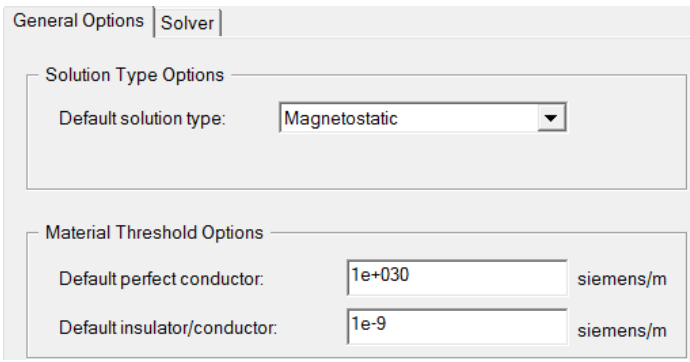
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Electric > Electric Transient**
    2. Click the **OK** button

Define Settings

- **To set Maxwell Settings**
  - Select the menu item *Tools > Options > Maxwell 3D Options*
  - In Maxwell 3D Options window,
    1. General Options tab
       - Default insulator/conductor: **1e-9 siemens/m**
    2. Click the **OK** button



Create Geometry

- **Create Box_Bottom**
  - Select the menu item *Draw > Box*
    1. Using the coordinate entry fields, enter the box position
       - X: **-1**, Y: **-2**, Z: **-0.6**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **4**, dZ: **0.6**, Press the **Enter** key
  - Change the name of the Object to **Box_Bottom**
- **Create Box_Top**
  - Select the object **Box_Bottom** from the history tree
  - Select the menu item *Edit > Duplicate > Along Line*
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0**, dY: **0**, dZ: **0.6**, Press the **Enter** key
    3. Total Number: **2**
    4. Press **OK**
  - Change the name of the Object to **Box_Top**

Create Materials

- **To Create Material for Box_Bottom**
  - Select the object **Box_Bottom**, right click and select **Assign Material**
  - In Select Definition window,
    1. Select **Add Material**
    2. In View/Edit Material window,
       - Material Name: **Mat_Bottom**
       - Relative Permittivity: **5**
       - Bulk Conductivity: **1e-7 siemens/m**
       - Select **OK**
- **To Create Material for Box_Top**
  - Select the object **Box_Top**, right click and select **Assign Material**
  - In Select Definition wondow,
    1. Select **Add Material**
    2. In View/Edit Material window,
       - Material Name: **Mat_Top**
       - Relative Permittivity: **4**
       - Bulk Conductivity: **2e-8 siemens/m**
       - Select **OK**

Assign Excitations

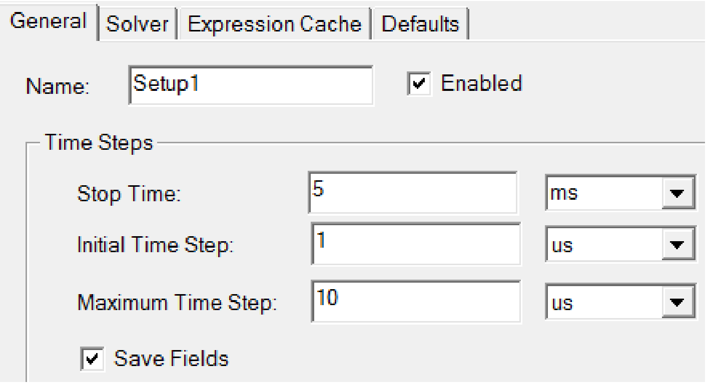
- We apply the voltage to the plates. We did not draw the plates, because we can apply a voltage to the top and bottom part of the regions.
- **Assign Excitation for Box_Top**
  - Select the menu item ***Edit > Select > Faces***
  - Select the top face of the object **Box_Top**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Set Value to **if(Time<0, 0, if(Time<2.5e-3,1,0))**
    2. Press **OK**
  - Select the bottom face of the object **Box_Bottom**
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Sink***
  - In Sink Excitation window,
    1. Press **OK**

Assign Mesh Operations

- **To Assign Mesh operations**
  - Select the menu item ***Edit > Select > Objects***
  - Press **Ctrl** and select the objects **Box_Bottom** and **Box_Top**
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict length of Elements: ☑ **Checked**
    2. Maximum Length of Elements: **0.55 mm**
    3. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Stop Time: **5 ms**
       - Initial Time Step: **1us**
       - Maximum Time Step: **10 us**
       - Save Fields: ☑ **Checked**
    2. Click the **OK** button



Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

Create Objects for Plots

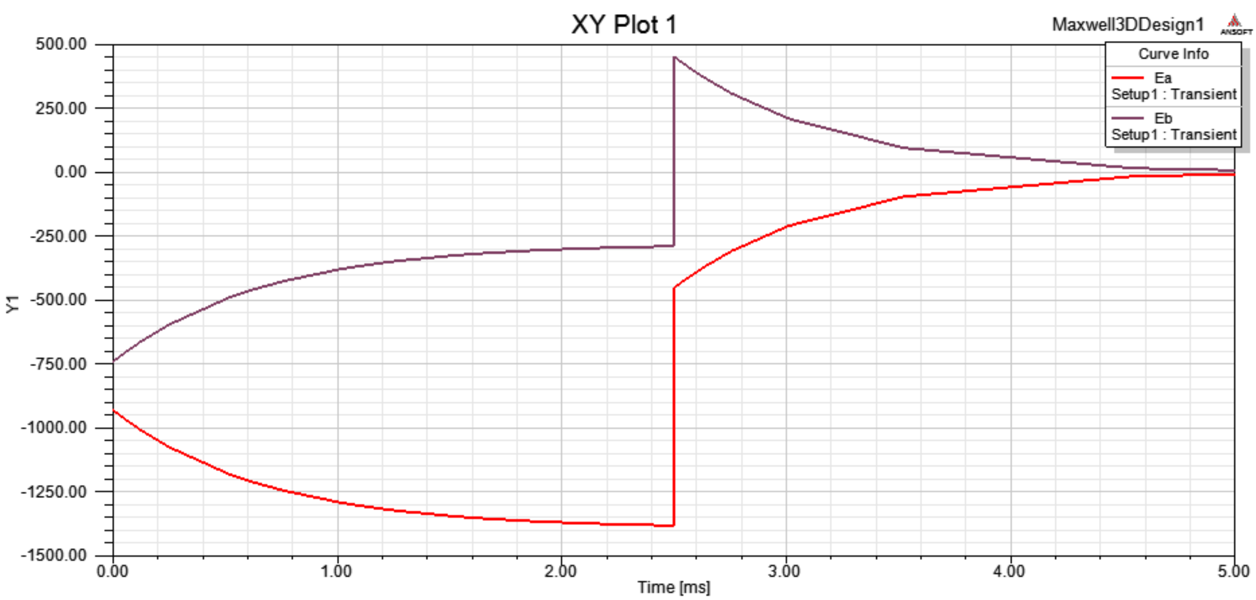
- **Create a Point**
  - Select the menu item ***Draw > Point***
    1. Using the coordinate entry fields, enter the point position
       - X: **0**, Y: **0**, Z: **0.3**, Press the **Enter** key
- **Create another Point**
  - Select the menu item ***Draw > Point***
    1. Using the coordinate entry fields, enter the point position
       - X: **0**, Y: **0**, Z: **-0.3**, Press the **Enter** key

Create Parameters for Plot

- **To Create Parameters**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity > E**
    2. Select **Vector > Scal? > ScalarZ**
    3. Select **Input > geometry**
       - Select **Point**
       - Select **Point1** from the list
       - Press **OK**
    4. Select **Output > Value**
    5. Press **Add**
    6. Specify name of the parameter as **E_a** and press **OK**
    7. Select **Input > Quantity > E**
    8. Select **Vector > Scal? > ScalarZ**
    9. Select **Input > geometry**
       - Select **Point**
       - Select **Point2** from the list
       - Press **OK**
    10. Select **Output > Value**
    11. Press **Add**
    12. Specify name of the parameter as **E_b** and press **OK**
    13. Press **Done**

Create Plot

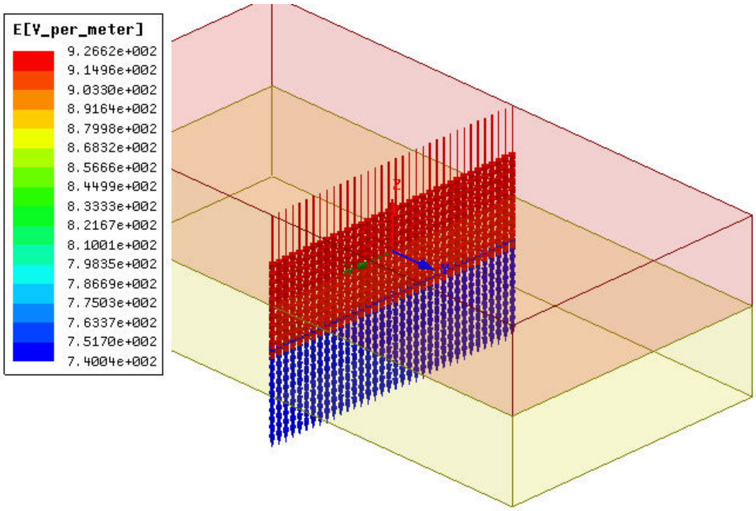
- **To Create Plot**
  - Select the menu item *Maxwell 3D > Results > Create Field Report > Rectangular Plot*
  - In Report window,
    - X: **Default**
    - Category: **Calculator Expressions**
    - Quantity: **E_a**
    - Select **New Report**
    - Change Quantity to **E_b**
    - Select **Add Trace**



Plot E_Vector on a Plane

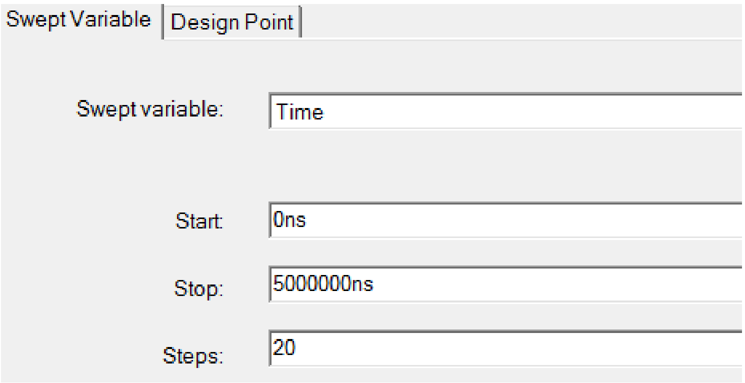
- **To Plot E_Vector**
  - Expand the tree for **Planes** from history tree and select the plane **Global:XZ**
  - Select the menu item *Maxwell 3D > Fields > Fields > E > E_vector*
  - In Create Field Plot window,
    1. Instrinsic Variables **>**Time:**1e-6 sec**
    2. Press **Done**

- **Modify Plot Attributes**
  - Double click on the Legend to modify plot attributes
  - In the window,
    1. **Marker/Arrow** tab
       - Arrow Options
         1. Size : move to right to increase arrow length
    2. **Plot** tab
       - Vector Plot
         1. Spacing: move to left to reduce space between arrows
    3. Press **Apply** and **Close**



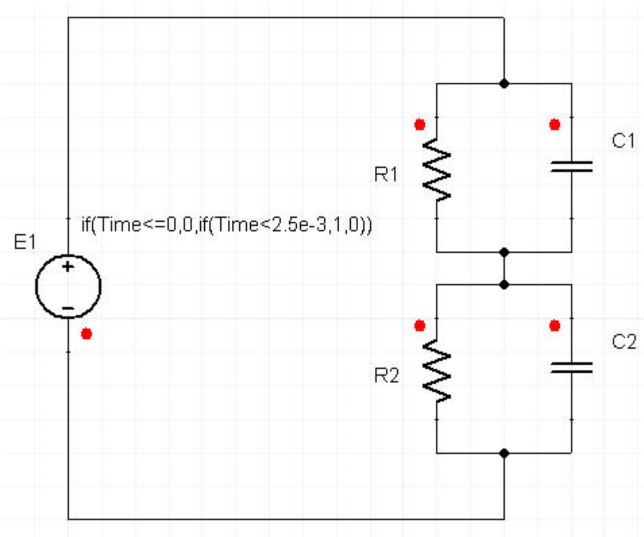
Animate Plot

- **To Animate Plot**
  - Expand the tree for **Field Overlays** from Project Manager window
  - Select the plot **E_Vector1** from the list, right click and select **Animate**
  - In Setup Animation window, set start time and Stop Time and press **OK**



System level Simulation

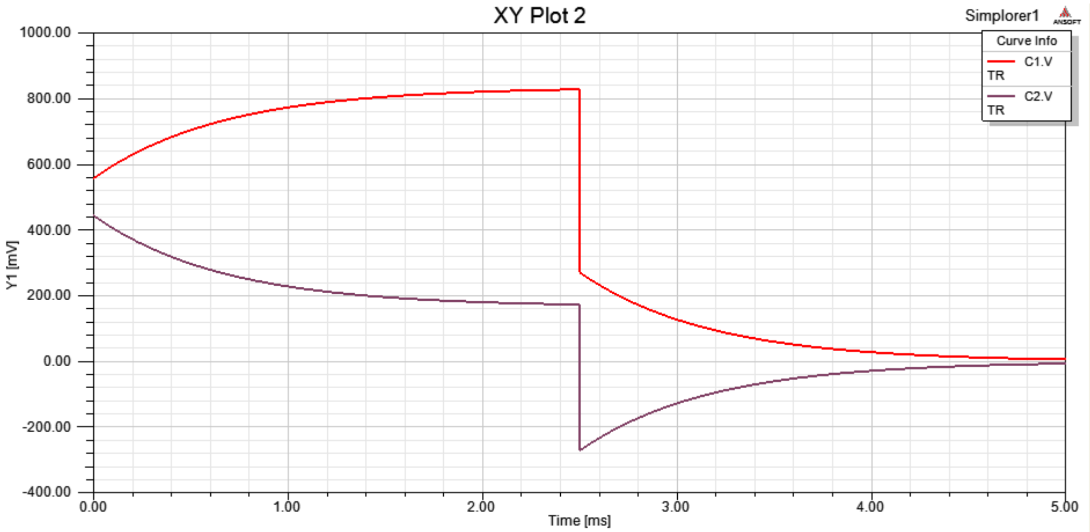
- Due to uniform thickness of the dielectric layer, the surface of separation between the two dielectrics can be assumed to be equipotential. Thus it becomes possible to replace the two dielectric layers with equivalent series/parallel RC circuits shown in the next page.



- Since the material and geometry information of the two dielectric layers are known, the resistances and capacitances can be calculated with the equations:

$$R = \frac{l}{\sigma A}\ ,\quad C = \frac{\varepsilon A}{l}$$

- Comparison of the voltages across the dielectric layers simulated in a circuit/system level simulator and calculated in Maxwell

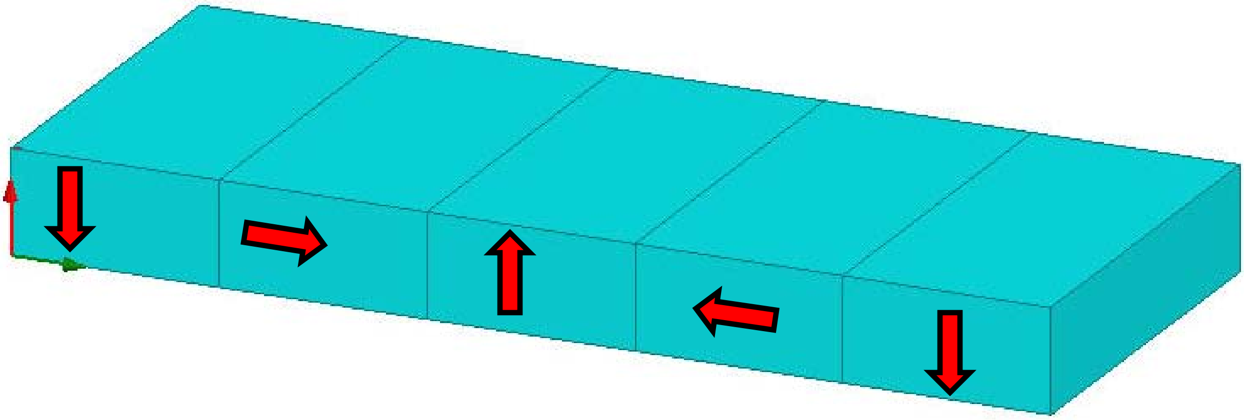


Hallbach Array

- This exercise will discuss how to set up a Hallbach array consisting of 5 magnets. This procedure is applicable for Magnetostatic and Transient Solvers.

Problem definition

- We are interested to solve the magnetic field around a Hallbach array consisting of 5 magnets. The magnetization direction of each magnet is shown below. The dimensions of each magnet are 5, 10, and 20 mm, respectively. The permanent magnet material is N27 (Br = 1.03T; Hc = 796 kA/m).



Create Design

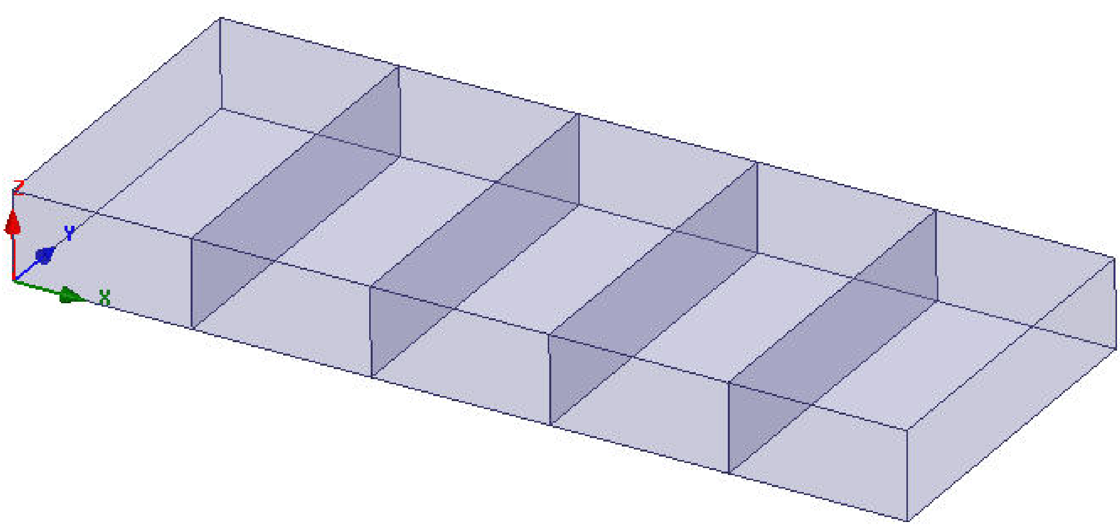
- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the  icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Magnetostatic**
    2. Click the **OK** button

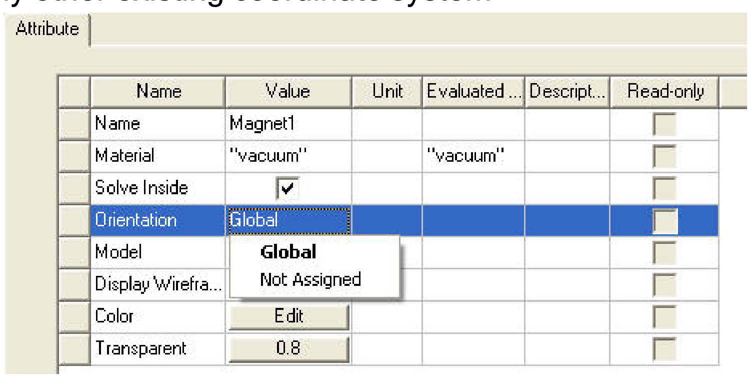
Create Geometry

- **Draw Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **10**, dY: **20**, dZ: **5**, Press the **Enter** key
  - Change the name of the resulting object to **Magnet1**
- **Duplicate Box**
  - Select the object **Magnet1** from the history tree
  - Select the menu item ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **10**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Total Number: **5**
  - Change the name of the resulting object to **Magnet2** through **Magnet5** in positive Y direction



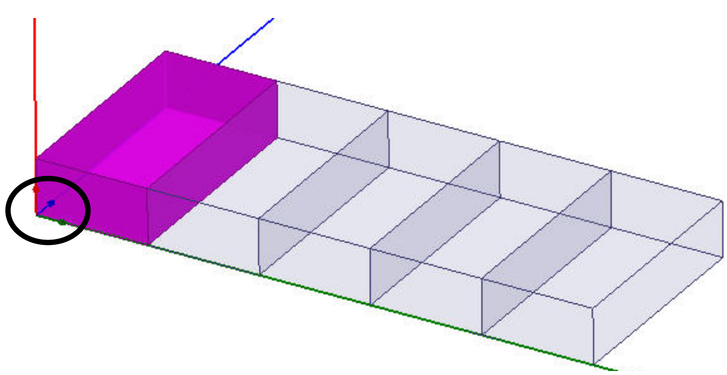
Check Orientation of Objects

- Each object in Maxwell is associated with certain coordinate system. This is called *Orientation* and it is specified under attributes for each object. The *Orientation* of a newly created (or imported) object is Global.
- **To Check Orientation of Magnet1**
  - Select the object **Magnet1** from the history tree, right click and select **Properties**
  - A Properties window will pop up where **Orientation** of the object is listed
  - Here **Magnet1** is orientated with respect to **Global** coordinate system
  - By left clicking on **Global,** we can change the orientation of the object to any other existing coordinate system



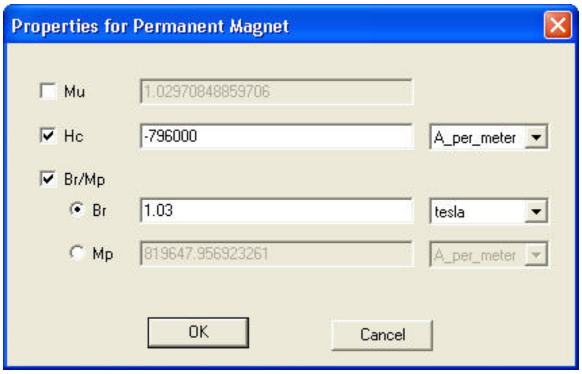
Name	Value	Unit	Evaluated ...	Descript...	Read-only
Name	Magnet1				☐
Material	"vacuum"		"vacuum"		☐
Solve Inside	☑				☐
Orientation	Global				☐
Model					☐
Display Wirefra...					☐
Color	Edit				☐
Transparent	0.8				☐

- **Another way to Check Orientation**
  - Select the menu item *Tools > Options > Modeler Options*
  - In Modeler Options window,
    1. **Display** tab
       - Show orientation of selected objects: ☑ **Checked**
    2. Press **OK**
  - This will enable display of coordinate system in small for each select object. The displayed coordinate system will the one to which object is oriented



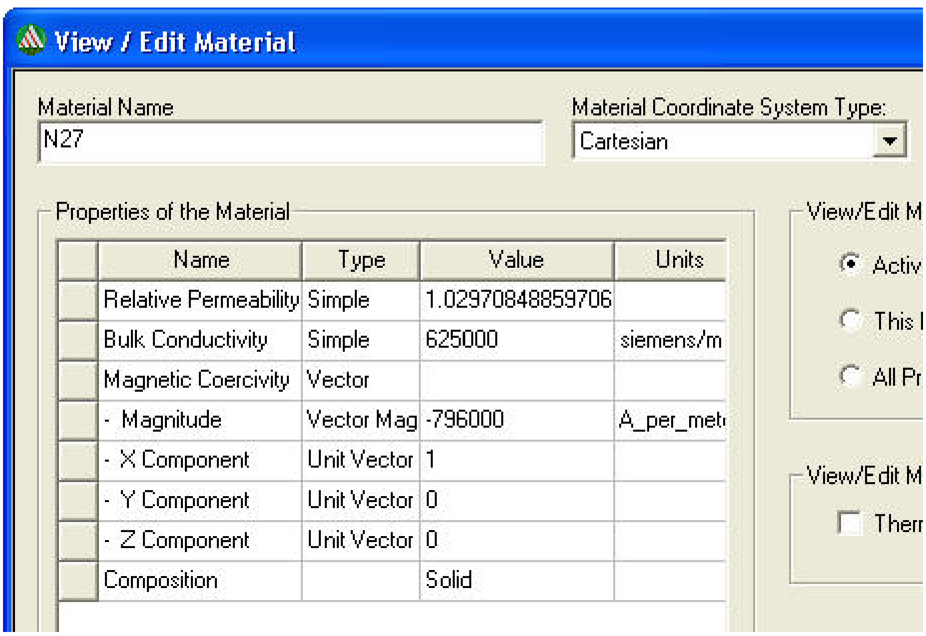
Define Permanent Magnet

- The magnet material is N27, which does not exist in the library. We have to create a new material with the properties of N27. For permanent magnets with the linear characteristic in the second quadrant (such as N27), the demagnetization curve is approximated with a line. This line is uniquely defined by specifying Remanence Br (1.03) and Coercive Field Hc (796 kA/m).
- **To Define Magnet Material**
  - Press **Ctrl** and select the objects **Magnet1** through to **Magnet5**
  - Right click and select **Assign Material**
  - In Select Definition window,
    1. Type **NdFe35** in the **Search by Name** field
    2. Select **Clone Material** button
  - In View/Edit Material window,
    1. Material Name: **N27**
    2. Select the tab Calculate Properties for **Permanent Magnet**
    3. In Properties for Permanent Magnet window,
       - Mu: ☐ **Unchecked**
       - Hc: ☑ **Checked**
         - Specify value of **-796000**
       - Br/Mp: ☑ **Checked**
       - Br: **1.03**
       - Press **OK**



    4. Note that the Relative Permeability (slope of the line) is automatically determined from the equation of the line defined by Br and Hc.
    5. Press **OK** to create material N27

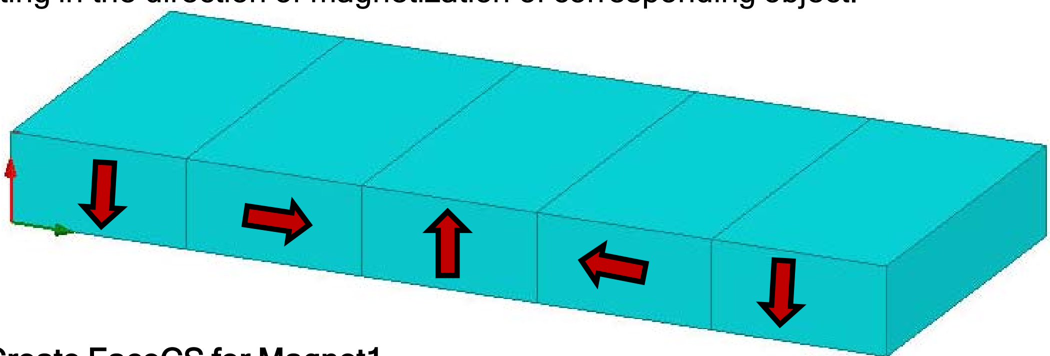
- The direction of magnetization is specified by a **unit vector** relative to the Coordinate System (CS) associated with the given object, that is relative to the ***Orientation*** of the object. If the ***Orientation*** of the object is **Global**, the unit vector will be specified relative to the Global CS. Maxwell also allows to specify the type of the Coordinate System (upper right corner). Thus Cartesian, Cylindrical and Spherical CS type can be defined. This means that if the ***Orientation*** of the object is **Global** and CS type **Cartesian**, the unit vector will be specified as ***X***, ***Y***, and ***Z*** relative to the Cartesian Global CS.
- **Hence, the right direction of magnetization is specified by the appropriate combination of object's Orientation, CS type and Unit Vector.**



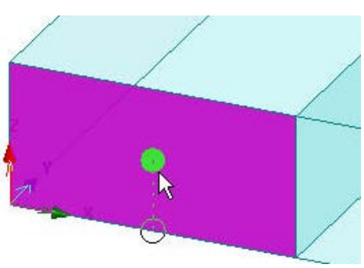
- For this particular magnet definition, the unit vector is **[1,0,0]**. This means that if the magnet remains oriented in the Global CS, the magnet will be magnetized in the Global X direction.
- In order to change this direction, we either change the unit vector or define a new coordinate system and associate the magnet with it. The X-axis of the new CS will have point in the direction of magnetization as the original (default) unit vector [1,0,0] in this case remains unchanged.
- The most advantageous is creation of FACE coordinate systems, which is discussed in the following.

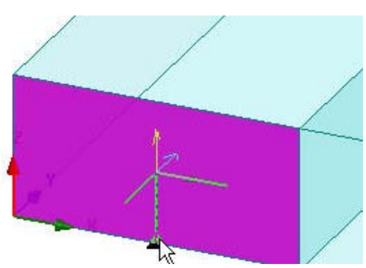
Create Orientation CS

- In this step, face coordinate systems are created in order to specify orientation of each magnet with respect to them.
- While creating FaceCS, it is ensured that X direction of the resulting CS is pointing in the direction of magnetization of corresponding object.

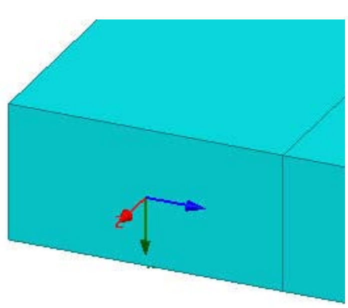


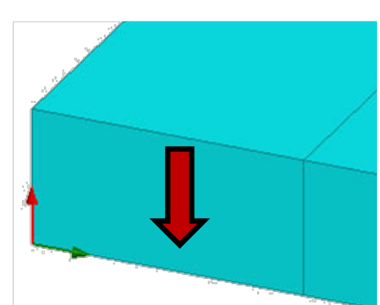
- **To Create FaceCS for Magnet1**
  - Select the menu item ***Edit > Select > Faces***
  - Select the face of **Magnet1** as shown in below image
  - Select the menu item ***Modeler > Coordinate System > Create > FaceCS***
  - For the origin of the new face CS, snap to the face center (cursor becomes circle) and click
  - For the X-axis (note that this should be oriented downwards in order to be aligned with the magnetization direction of magnet1) snap to the center of bottom edge (cursor becomes triangle) and click.



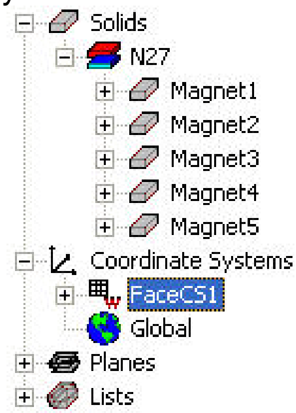


  - X-axis of the newly created face CS is now aligned with the direction of magnetization of **Magnet1**



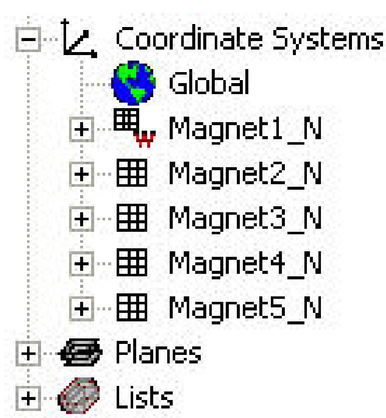


- Expand the history tree for Coordinate Systems and select **FaceCS1**

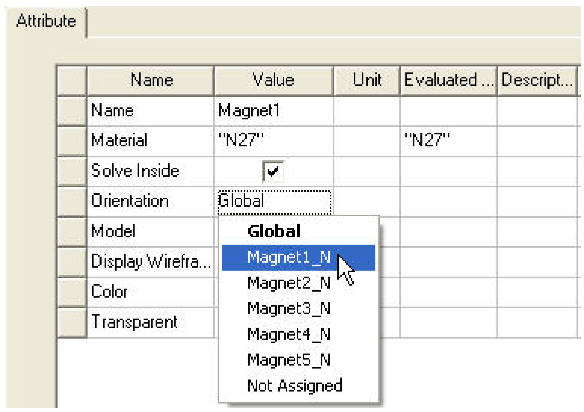


- Goto the properties window and change the name of the coordinate system as **Magnet1_N**
- Note also that small red W next to CS name signifies current working CS. Clicking on Global, Global CS again becomes current CS.

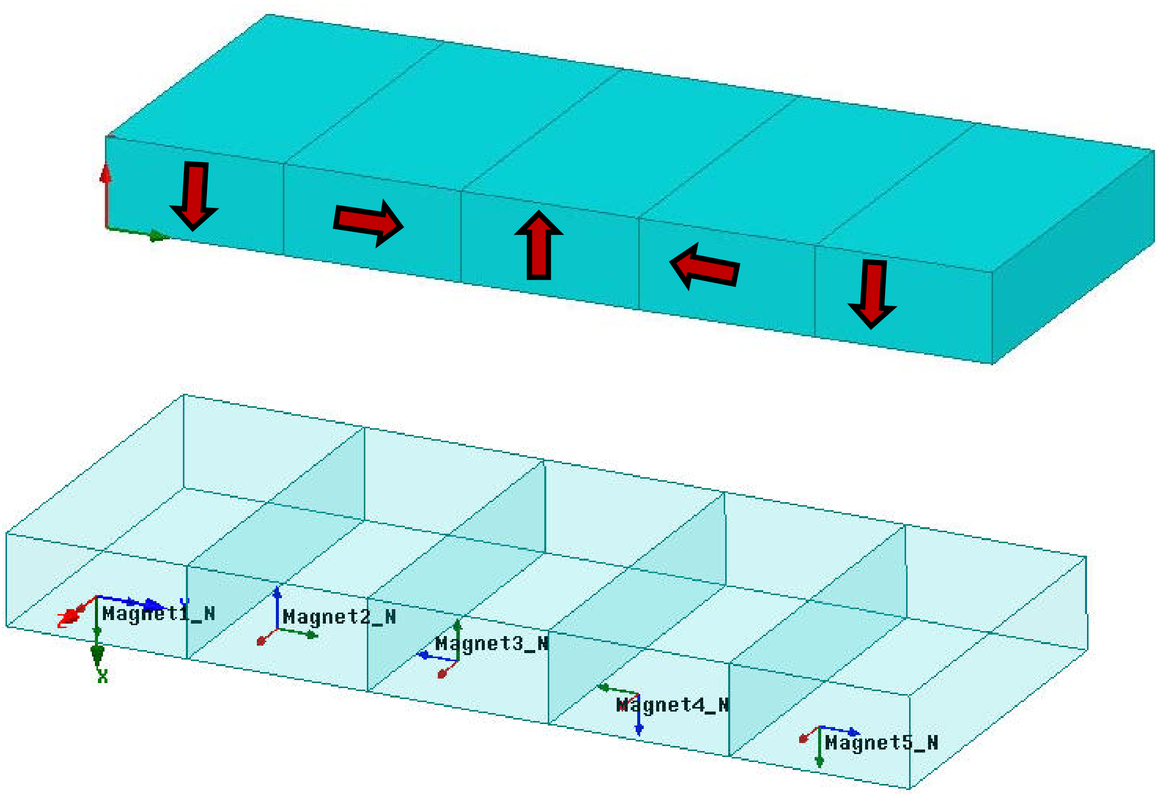
- The above procedure for face CS creation has to be repeated for each magnet. At the end we have 5 face CS:



- **Set Orientation CS for Objects**
  - Select the object **Magnet1** from the history tree
  - Right click and select **Properties**
  - Change the Orientation from Global to **Magnet1_N**

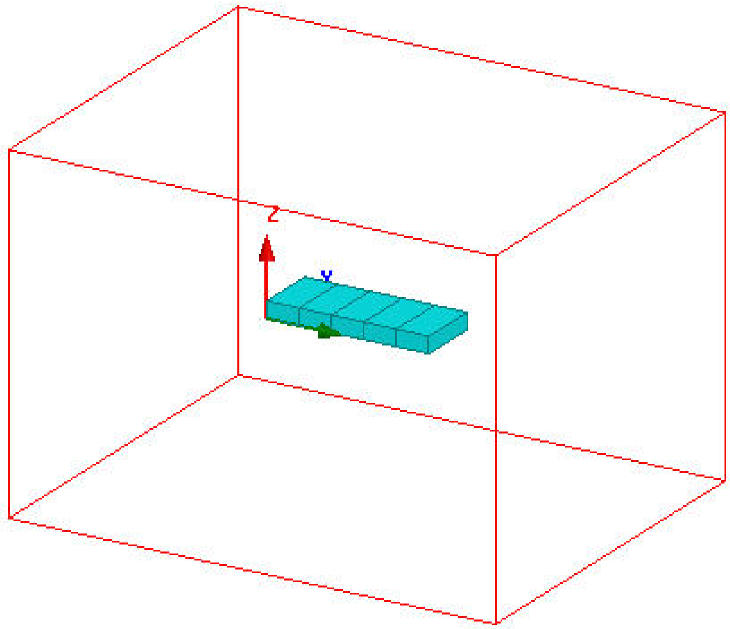


  - In similar way, set the orientation of each object with respect to their FaceCS



Define Region

- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Absolute Offset**
    3. Value: **50mm**
    4. Press **OK**



Analysis Setup

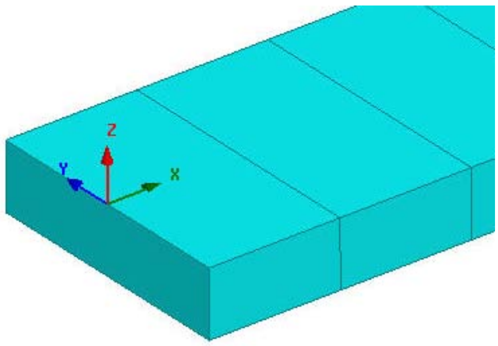
- **To create an analysis setup:**
  - Select the menu item *Maxwell 3D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Maximum Number of Passes: **20**
    2. Click the **OK** button

Analyze

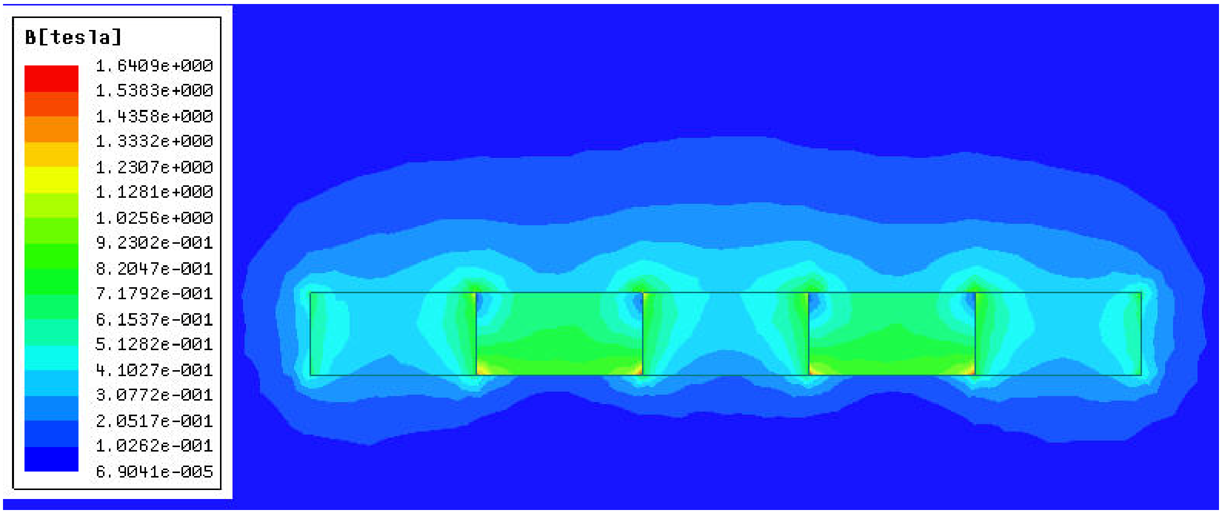
- **To start the solution process:**
  1. Select the menu item *Maxwell 3D > Analyze All*

Results

- We will plot the magnetic flux density on the magnets' middle plane. In order to access this plane, we have to create a new coordinate system such that one plane associated with this CS will be coincident with magnets' middle plane.
- **To Create CS**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Offset***
  - Snap cursor to any magnet's middle point
  - XZ plane of this new CS is coincident with magnets' middle plane



- **To Create Filed Plot**
  - Exapnd the history tree for **Planes**
  - Select the plane **RelativeCS1:XZ**
  - Select the menu item ***Maxwell 3D > Fileds >Fields > B > Mag_B***
  - In Create Field Plot window,
    1. Press **Done**



Basic Exercises – Electric Transient (High Voltage Line)

Overview

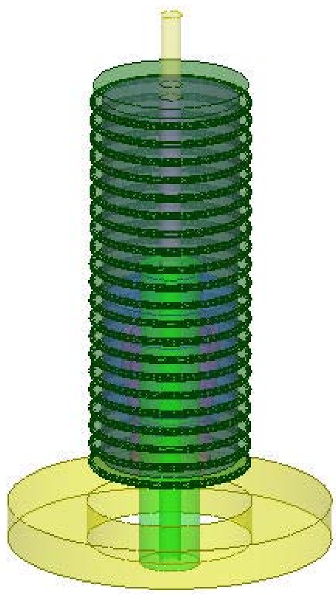
- This exercise explains how an electric transient case can be solved using Maxwell V15.
- The exercise uses the example of a high voltage electric line. The electric field between the line and the ground is simulated with Electric Transient Solver
- As the Electric Transient solver does not provide adaptive meshing, we will first solve the case using Electrostatic solver and import its adaptively refined mesh for use in Transient solution

Launch Maxwell

- **To Launch Maxwell**
  - To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 15.**

Open Input File

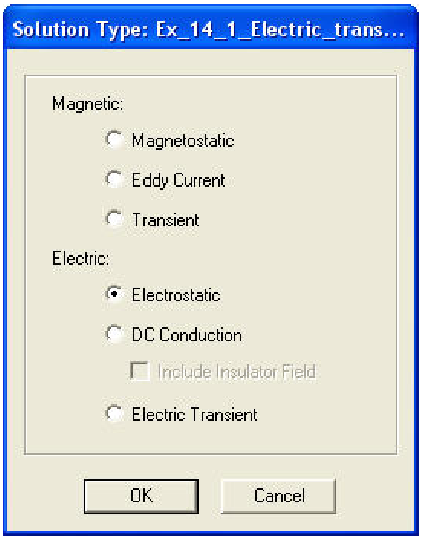
- **To open a file**
  - In Maxwell window, select the menu item ***File →Open***.
  - Browse to the file **"Ex_9_16_BasicElectricTransient_HighVoltageLine.mxwl"** and select **Open**
  - The file contains the geometry of the power line termination created in Maxwell as shown in below image. The geometry contains a copper conductor surrounded by insulating objects. The materials for all the object is already set in the file. Conductor and ground terminals are specified with copper material



Basic Exercises – Electric Transient (High Voltage Line)

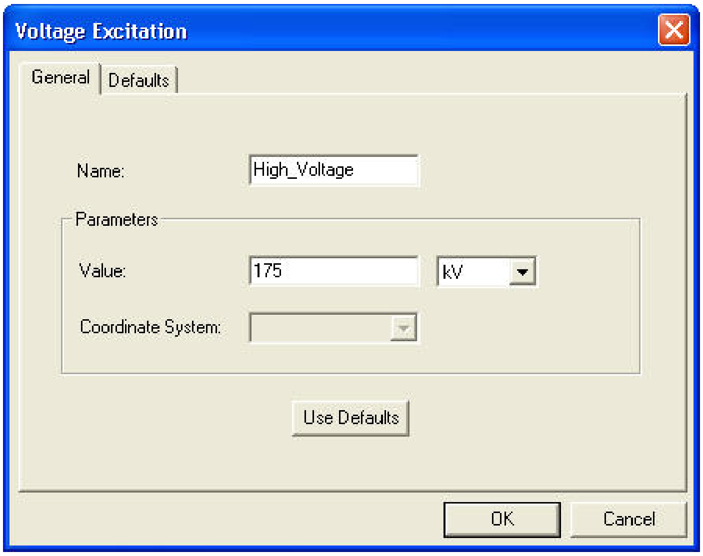
Set Solution Type

- The case will be solved with the Electrostatic solver to get an adaptively refined mesh. The refined mesh will be used for the transient solution
- **To set the solution type**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Electrostatic**
    2. Click the **OK** button



Create Excitations

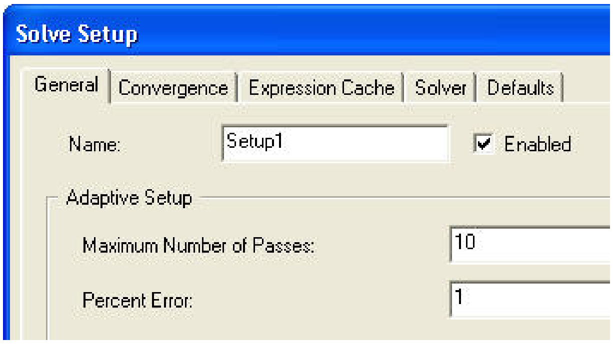
- **Specify excitation for the Conductor**
  - Select the object **Conductor** from the tree
  - Go to the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In the Voltage Excitation Window
    1. Name: **High_Voltage**
    2. Value: **175 kV**
    3. Click the **OK** button

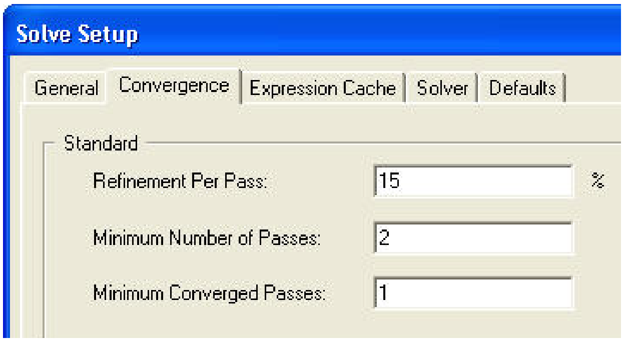


- **Specify excitation for Ground**
  - Select the object **Ground** from the history tree
  - Go to the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In the Voltage Excitation Window
    1. Name: **Ground**
    2. Value: **0 V**
    3. Click the **OK** button
- **Specify excitation for Shield**
  - Select the object **Ground** from the history tree
  - Go to the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In the Voltage Excitation Window
    1. Name: **Shield_Ground**
    2. Value: **0 V**
    3. Click the **OK** button

Analysis Setup

- **To Create an Analysis Setup**
  - Go to the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In the Solve Setup window
    1. **General** tab
       - Percentage Error: **1**
    2. **Convergence** tab
       - Refinement Per Pass: **15 %**
    3. Press **OK** to close



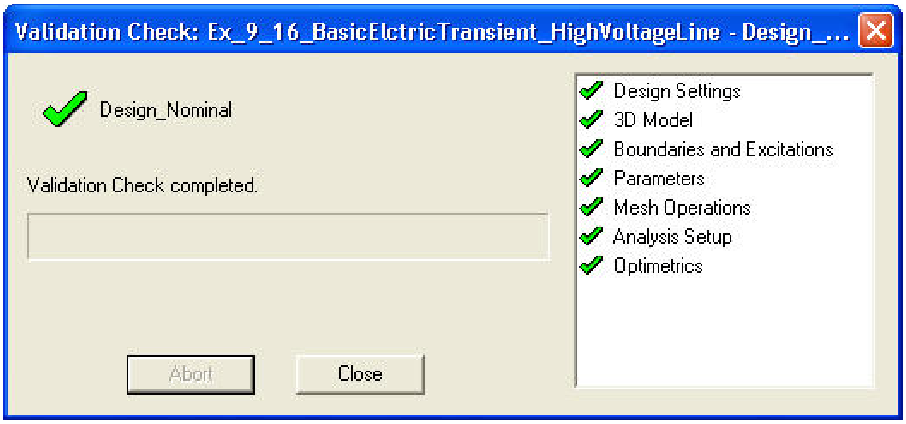


Save Project

- **To save the project:**
  - In an Ansoft Maxwell window, select the menu item ***File > Save As***.
  - From the **Save As** window, type the appropriate file name
  - Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



- **Note:** To view any errors or warning messages, use the Message Manager.

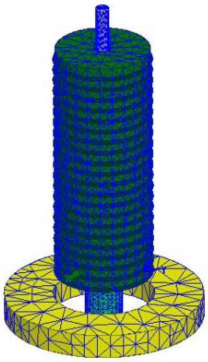
Analyze

- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

Basic Exercises – Electric Transient (High Voltage Line)

Plot Mesh

- **To Hide Region**
  - Press **Ctrl** and select the objects **InnerRegion** and **Region**
  - Select the menu item *View > Hide Selection > Active View*
- **To Plot Mesh**
  - Select the menu item *Edit > Select All Visible*
  - Select the menu item *Maxwell 3D > Fields > Plot Mesh*
  - In Create Mesh Plot window,
    - Press **Done**

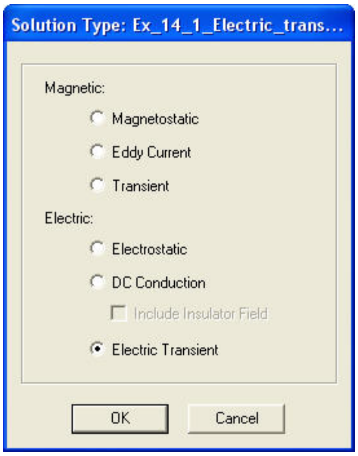


Copy Design

- **To Create a Copy of the Design**
  - Select the design **"Design_Nominal"** from the Project Manager tree, right click and select **"Copy"**
  - Select the tab **"Ex_9_16_BasicElectricTransient_HighVoltageLine"** in Project Manager tree, right click and select **"Paste"**
  - Rename the newly created design as **"Design_Transient"**

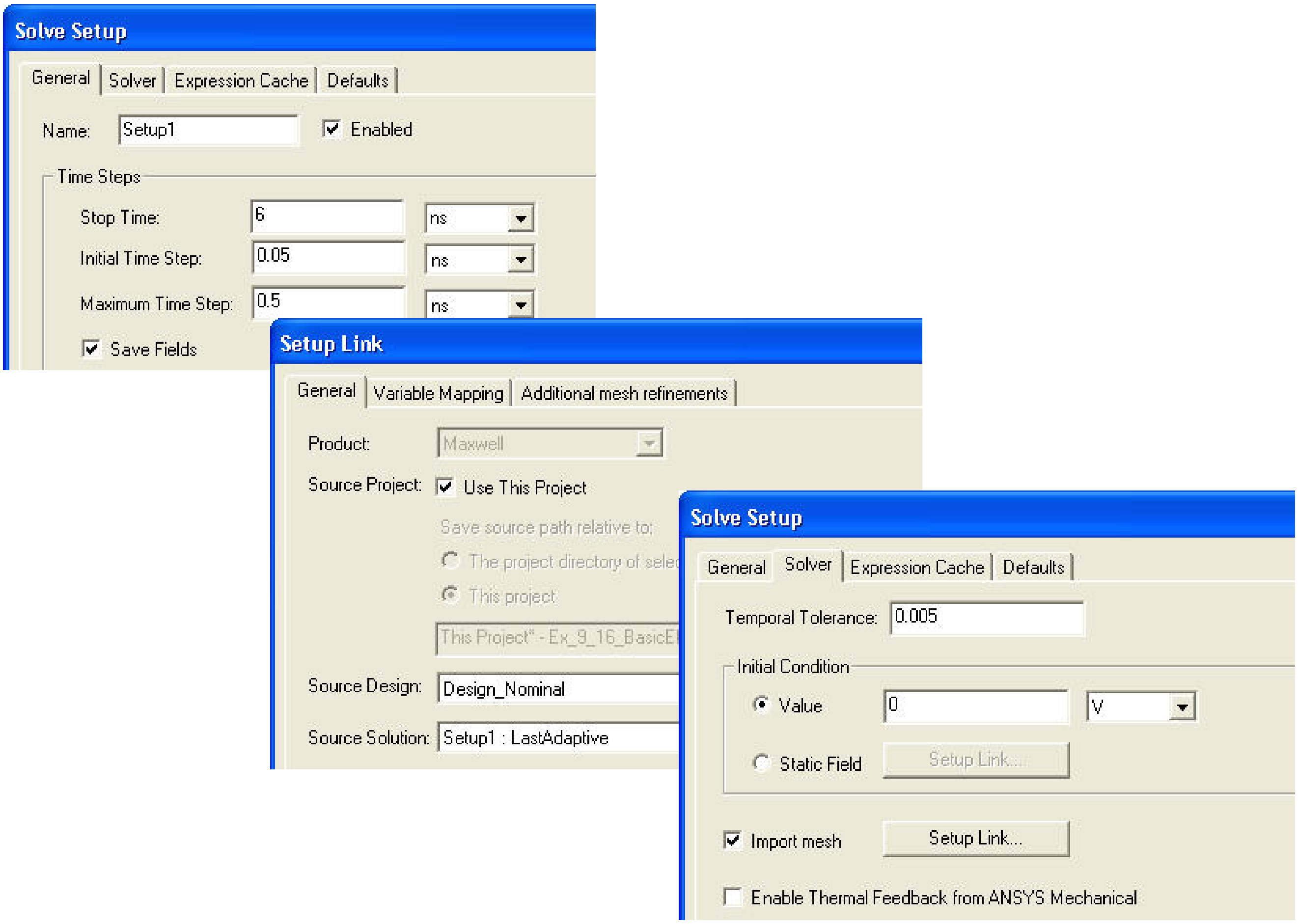
Set Transient Solution

- **To Set Transient Solution Type**
  - Double click on **"Design_Transient"** from the Project manager window to activate it
  - Select the menu item *Maxwell 3D > Solution Type*
  - In Solution Type window
    1. Change Solution Type to **Electric >Transient**
    2. Press **OK**



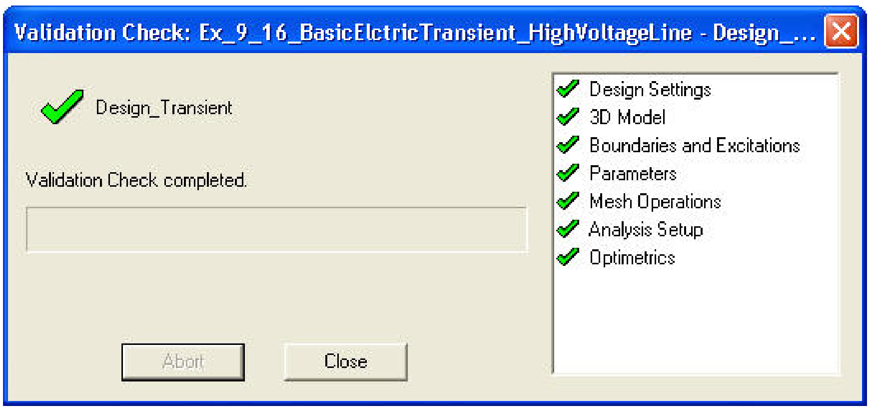
Transient Analysis Setup

- **To Create a Transient Analysis Setup**
  - Go to ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - In Solve Setup Window
    1. **General** Tab
       - Stop Time: **6 ns**
       - Initial Time Step: **0.05 ns**
       - Maximum Time Step: **0.5 ns**
    2. **Solver** Tab
       - Import mesh: ☑ **Checked**
       - In Setup Link Window
         1. Source Project: Use This Project ☑ **Checked**
         2. Source Design: **Design_Nominal**
         3. Press **OK**



Model Validation

- **To validate the model**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Analyze

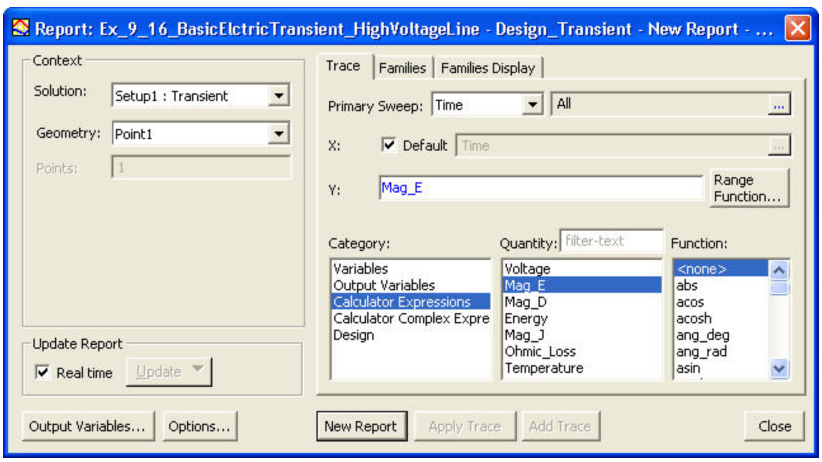
- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

Rectangular Plot

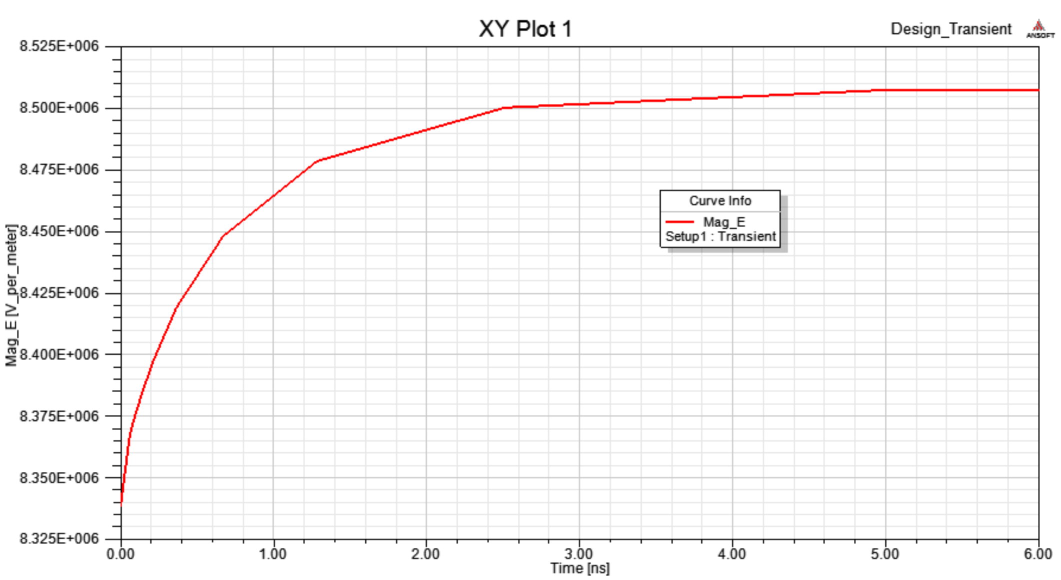
- **Create a point to plot magnitude of E on it**
  - Select the menu item ***Draw > Point***
    1. Using Coordinate Entry Field, Enter the point position
       - X: **0,** Y: **0,** Z: **-150,** Press **Enter**

Basic Exercises – Electric Transient (High Voltage Line)

- **To Plot Mag_E Vs Time**
  - Select the menu item ***Maxwell 3D > Results > Create Field Reports > Rectangular Plots***
  - In Report window,
    1. Geometry: **Point1**
    2. X : **Default**
    3. Category: **Calculator Expressions**
    4. Quantity: **Mag_E**
    5. Select **New Report**



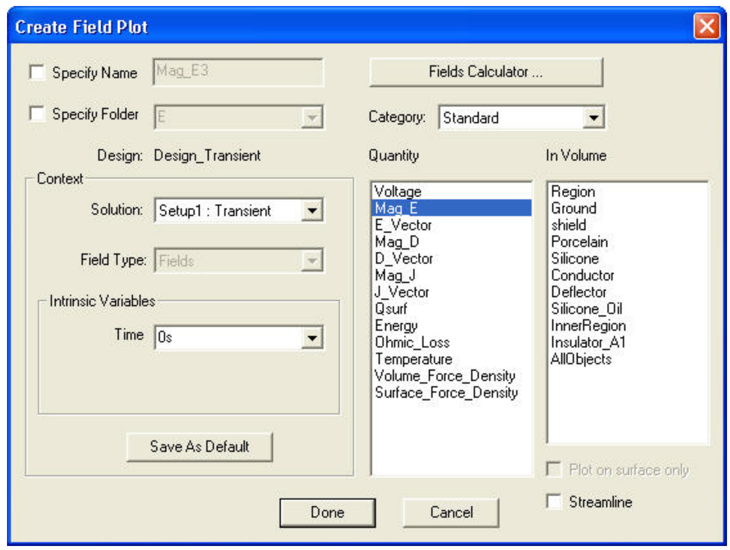
- Plot will be displayed on the screen
- From the plot, it is clear that the magnitude of E has reached steady value by 6ns and field has developed completely. Thus the solution need not to be run further



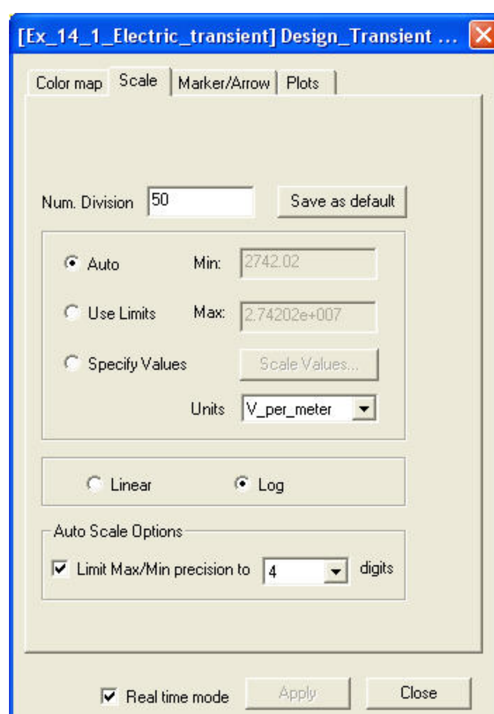
Basic Exercises – Electric Transient (High Voltage Line)

Mag_E Field Plot

- **Plot Mag_E at Time= 0s**
  - Select **Global: XZ** plane from the history tree
  - Go to ***Maxwell 3D > Fields > Fields > E > Mag_E***
  - In the Create Filed Plot window
    1. Time: **0 s**
    2. Select **Done**

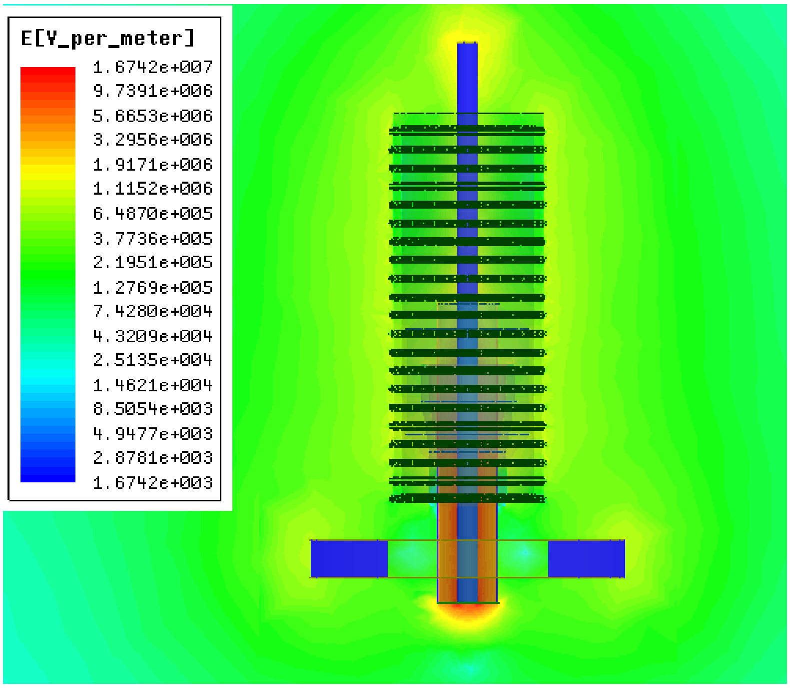


  - Similarly create a plot at **Time = 6e-9 s**
- **To Change the Plot Attributes**
  - Double click on the legend to modify plot
  - In the window
    1. **Scale** tab
    2. Num. Divisions: **50**
    3. Log: ☑ **Checked**
    4. Press **Apply** and **Close**

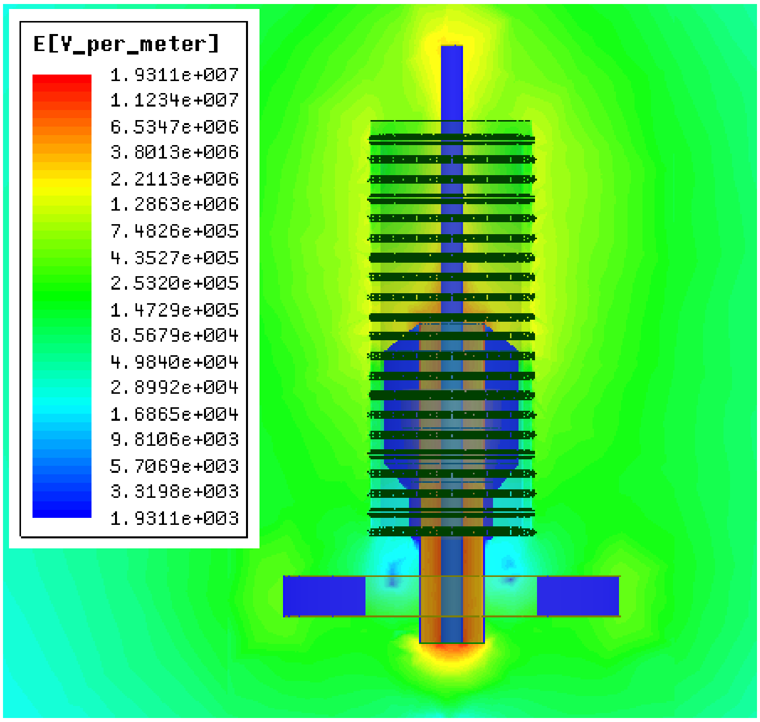


Basic Exercises – Electric Transient (High Voltage Line)

- A Field Plot will be displayed on screen as shown below



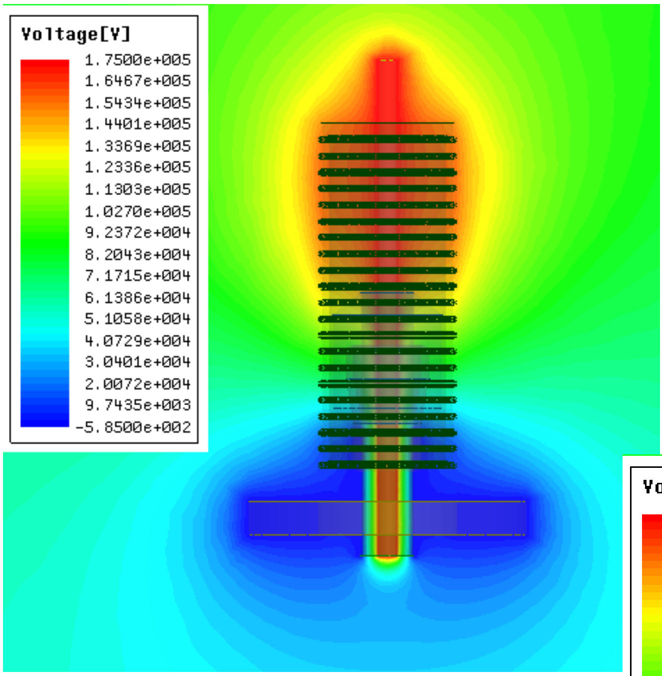
Plot at **Time = 0s**



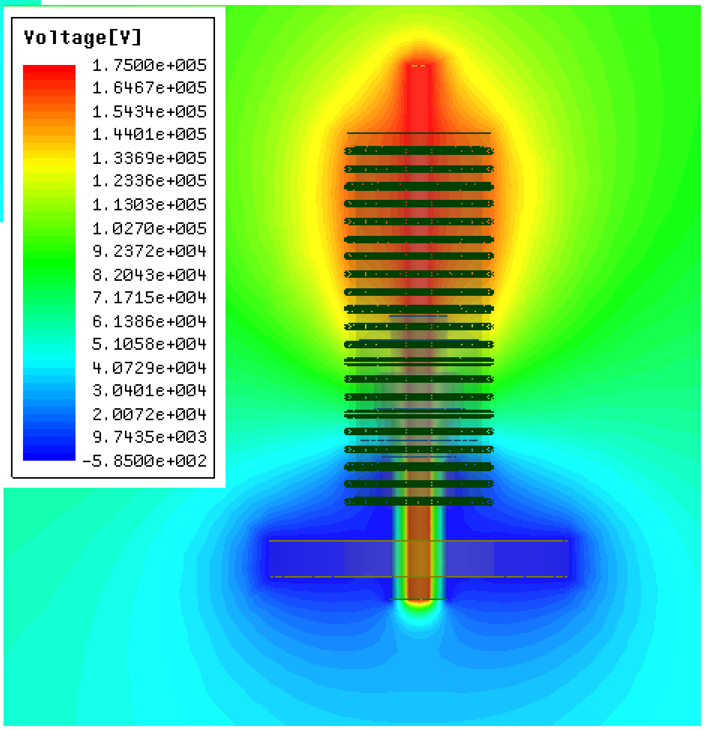
Plot at **Time = 6e-9s**

Voltage Field Plot

- **Plot Voltage at Time= 0s**
  - Select **Global: XZ** plane from the history tree
  - Go to ***Maxwell 3D > Fields > Fields > Voltage***
  - In the Create Filed Plot window
    1. Time: **0 s**
    2. Select **Done**
- Similarly create a plot of Voltage at **Time = 6e-9s**



Time = 0s



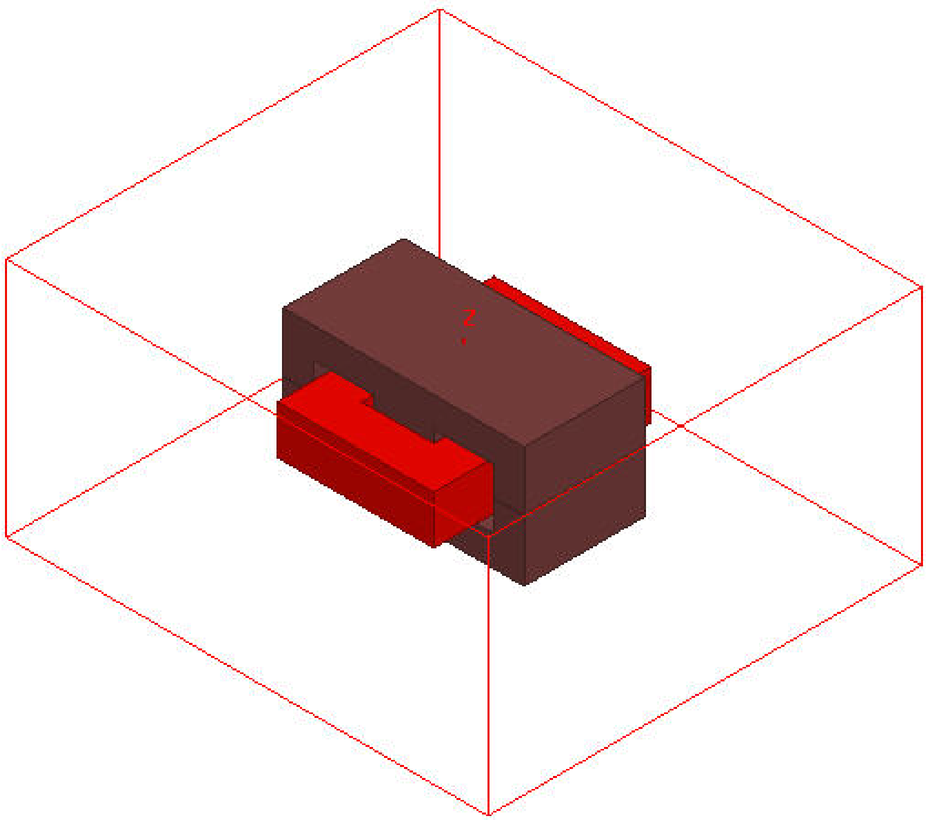
Time = 6e-9s

Chapter 10.0 – Optimetrics

Gapped Inductor

- This example describes how to create and optimize a gapped inductor for a particular value of inductance using the 3D Magnetostatic solver and Optimetrics in the ANSYS Maxwell 3D Design Environment.

- Two optimizations will be completed:
  1. Obtain the desired inductance by varying current
  2. Obtain the desired inductance by core gap



Example (Optimetrics) – Gapped Inductor

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **Box, Recatangle**
    - Boolean Operations: **Subtract, Unite, Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
    - **Optimetrics**
  - Results
    - **Inductance**
    - **Flux Density**

Example (Optimetrics) - Gapped Inductor

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

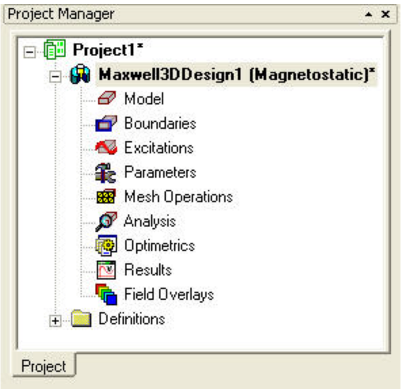
Setting Tool Options

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

Example (Optimetrics) - Gapped Inductor

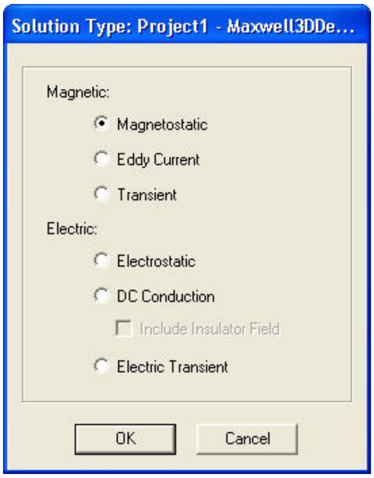
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button



Set Model Units

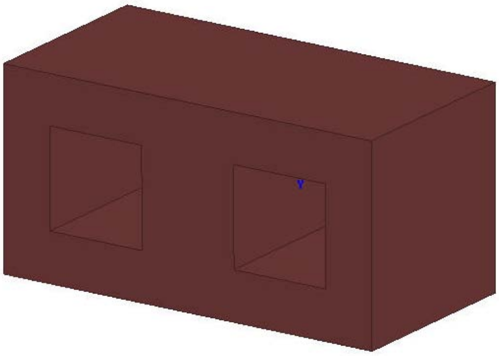
- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **mm**
    2. Click the **OK** button

Create EE- Core

- **Create a Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the position of box
       - X: **-10**, Y: **-20**, Z: **-10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **20**, dY: **40**, dZ: **20**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Core**
    2. Change the color of the object to **Gray**
    3. Change the material of the object to **"mu_metal"**
  - Select the menu item ***View > Fit All > Active View***
- **Create Core window**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the position of box
       - X: **-10**, Y: **-15**, Z: **-5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **20**, dY: **10**, dZ: **10**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Window**
- **Duplicate Window**
  - Select the object **Window** from the history tree
  - Select the menu item, ***Edit > Duplicate > Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - dX: **0**, dY: **20**, dZ: **0**, Press the **Enter** key
    3. Total Number: **2**
    4. Press **OK**

Example (Optimetrics) - Gapped Inductor

- **Subtract Windows from Core**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    1. Blank Parts: **Core**
    2. Tool Parts: **Window, Window_1**
    3. Press **OK**

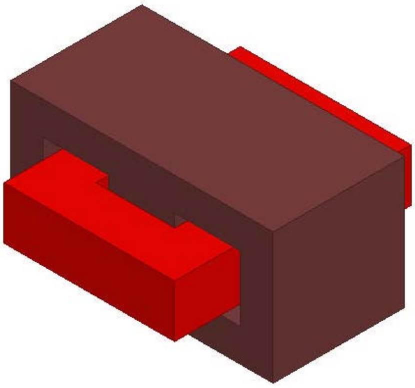


Create Coil

- **Create a Box**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the position of box
       - X: **-18**, Y: **-13**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **36**, dY: **26**, dZ: **8**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Coil**
    2. Change the color of the object to **Red**
    3. Change the material of the object to **"copper"**
- **Create Hole**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the position of box
       - X: **-11**, Y: **-6**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **22**, dY: **12**, dZ: **10**, Press the **Enter** key
- **Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Hole**

Example (Optimetrics) – Gapped Inductor

- **Subtract Hole from Coil**
  - Press **Ctrl** and select the objects **Coil** and **Hole** from history tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    1. Blank Parts: **Coil**
    2. Tool Parts: **Hole**
    3. Press **OK**



Create Coil Terminal

- **Create Section of coil**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Click the **OK** button

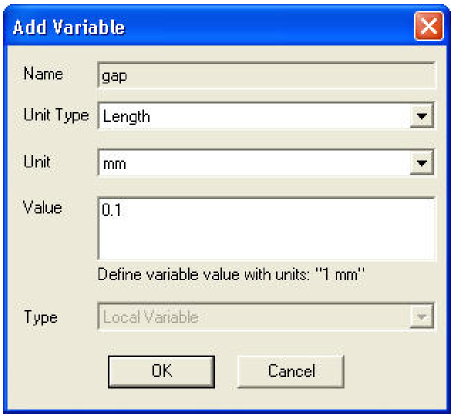
- **Change Attributes**
  - Select the object **Coil_Section1** from the tree and goto Properties window
    1. Change the name of the object to **Coil_Terminal**
- **Separate Sheets**
  - Select the sheet **Coil_Terminal**
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Delete Extra Sheets**
  - Select the sheet **Coil_Terminal_Seperate1** from the tree
  - Select the menu item ***Edit >Delete***

Create Core Gap

- **To Create Core gap**
  - Select the menu item ***Draw > Box***
    1. Using the coordinate entry fields, enter the position of box
       - X: **-15**, Y: **-25**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **30**, dY: **50**, dZ: **0.1**, Press the **Enter** key

Example (Optimetrics) - Gapped Inductor

- **Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Gap**
- **Parameterize Gap**
  - Expand the history tree for the object **Gap**
  - Double click on the command **CreateBox** from the history tree
  - In Properties window,
    1. For ZSize type **gap,** press **Tab** to accept the value
    2. In Add Variable window,
       - Unit Type: **Length**
       - Unit: **mm**
       - Value: **0.1**
       - Press **OK**

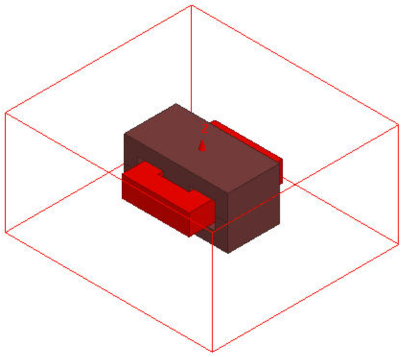


    3. Press **OK** to close Properties window
- **Subtract Gap from Core**
  - Press **Ctrl** and select the objects **Coil** and **Hole** from history tree
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    1. Blank Parts: **Core**
    2. Tool Parts: **Gap**
    3. Press **OK**
- **Separate Core**
  - It is necessary to unlump both core halves for inductance calculation
  - Select the object **Core** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - **Core_Separate1** will be created.
  - Rename the object as **Core2**

Define Region

Create Simulation Region

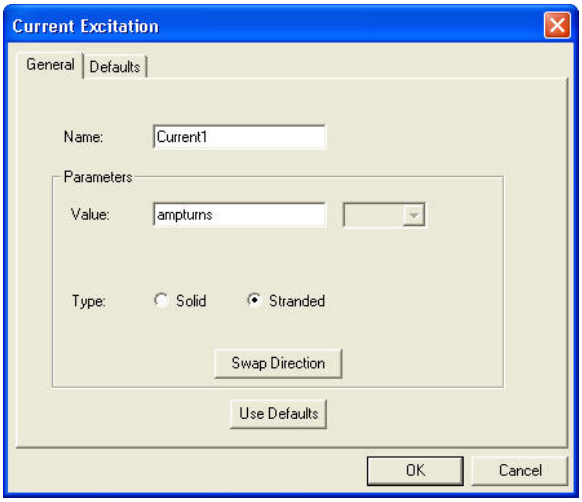
- Select the menu item *Draw > Region*
- In Region window,
  1. Pad all directions similarly: ☑ **Checked**
  2. Padding Type: **Percentage Offset**
  3. Value: **50**
  4. Press **OK**



Assign Excitations

To Assign Excitations

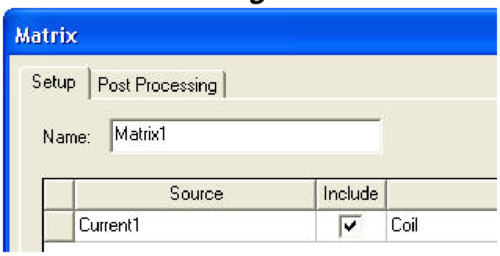
- Select the object **Coil_Terminal** from the history tree
- Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
- In Current Excitation window,
  1. Name: **Current1**
  2. Value: **ampturns**
  3. Type: **Stranded**
  4. Press **OK**
- In Add Variable window,
  1. Unit Type: **Current**
  2. Unit: **A**
  3. Value: **100**
  4. Press **OK**



Calculate Inductance

To Assign Inductance Calculation

- Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
- In matrix window,
  - Current1
    - Include: ☑ **Checked**
  - Press **OK**



Analysis Setup

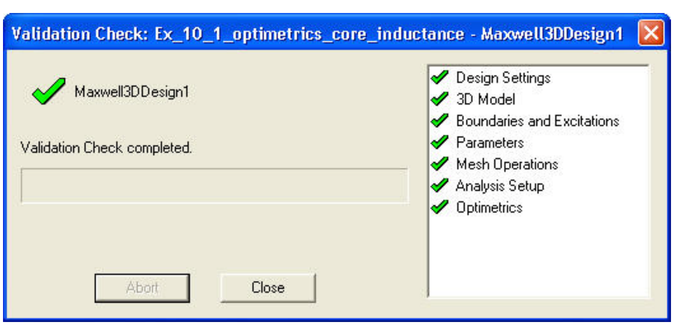
- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** tab
       - Maximum Number of Passes: **5**
    2. Press **OK**
- **Note:** Convergence will not be very good with specified settings. The settings are used just to demonstrate the optimetrics capabilities in faster way.

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: **Ex_10_1_Core_Inductance**
  3. Click the **Save** button

Model Validation

- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



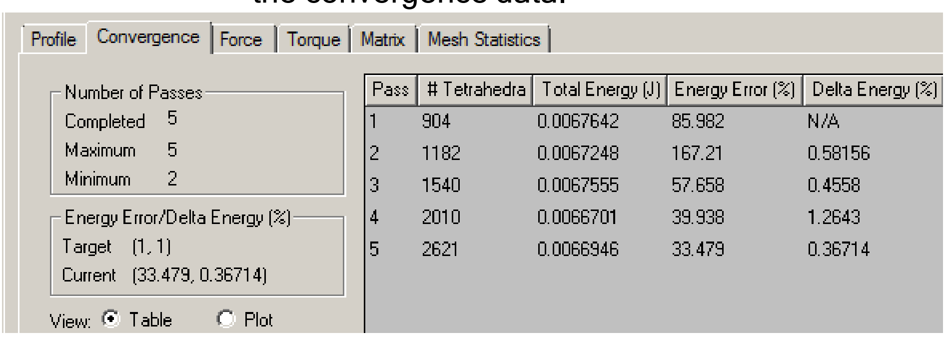
Note: To view any errors or warning messages, use the Message Manager.

Analyze

- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***

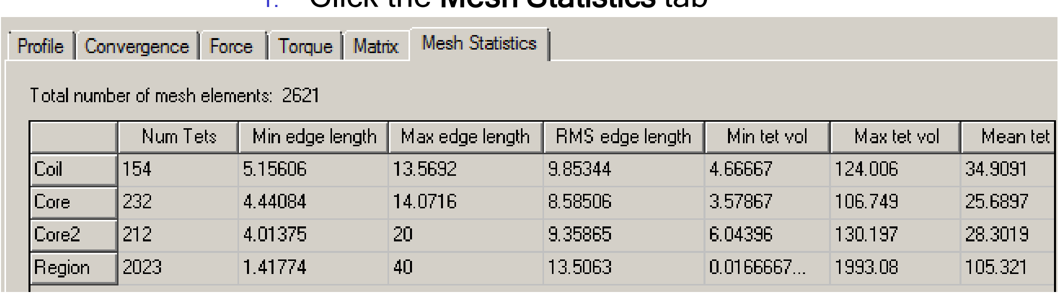
Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.



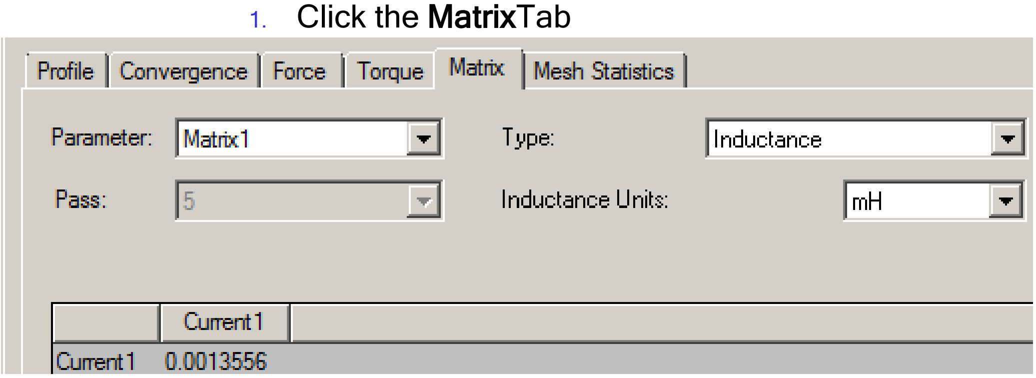
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	904	0.0067642	85.982	N/A
2	1182	0.0067248	167.21	0.58156
3	1540	0.0067555	57.658	0.4558
4	2010	0.0066701	39.938	1.2643
5	2621	0.0066946	33.479	0.36714

- To View Mesh information
  1. Click the **Mesh Statistics** tab



	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet
Coil	154	5.15606	13.5692	9.85344	4.66667	124.006	34.9091
Core	232	4.44084	14.0716	8.58506	3.57867	106.749	25.6897
Core2	212	4.01375	20	9.35865	6.04396	130.197	28.3019
Region	2023	1.41774	40	13.5063	0.0166667...	1993.08	105.321

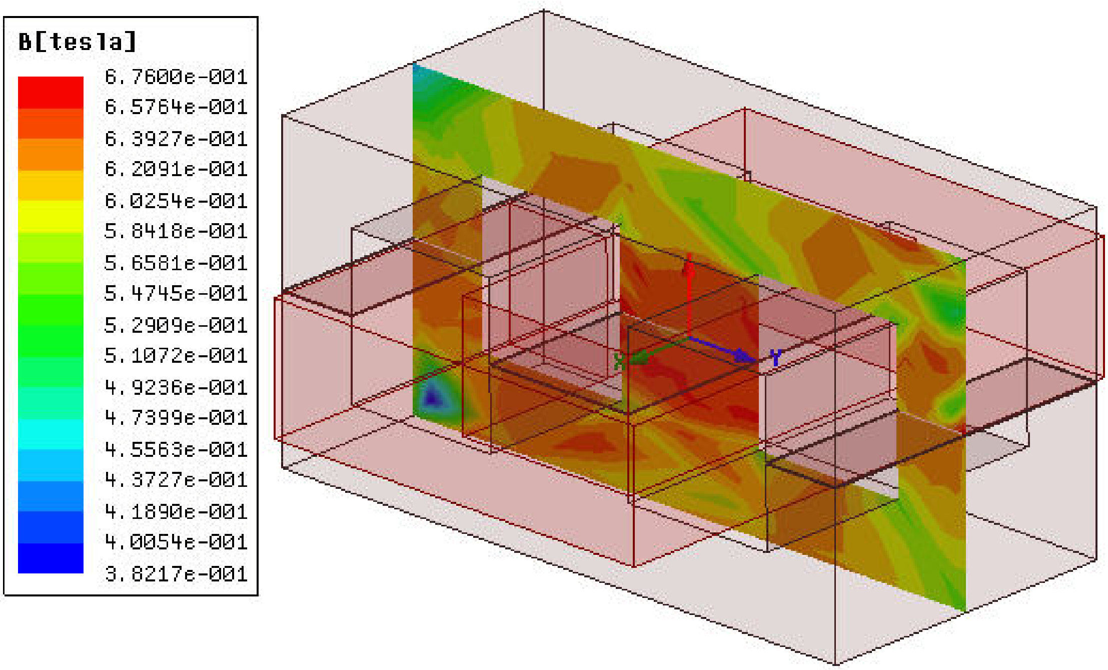
- To view the Inductance values:
  1. Click the **Matrix**Tab



	Current1
Current1	0.0013556

Plot Flux Density in the core

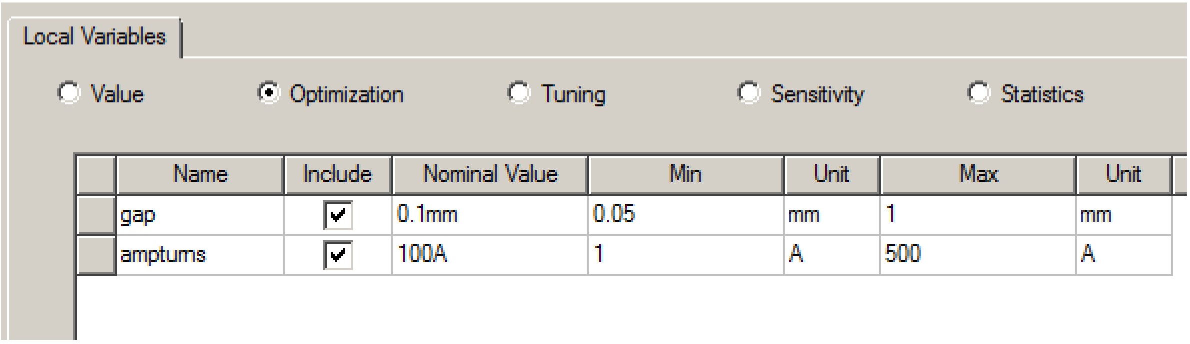
- **To Hide Region**
  - Select the object **Region** from the history tree
  - Select the menu item ***View > Hide Selection > Active View***
- **To Plot Flux Density**
  - Expand the history tree for Planes and select **Global: YZ**
  - Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    1. Quantity: **Mag_B**
    2. In Volume: Press **Ctrl** and select **Core** and **Core2**
    3. Press **Done**



- **Note:** This field plot is for the nominal design case with gap = 0.1mm and ampturns = 100A.

Optimetrics Setup and Solution

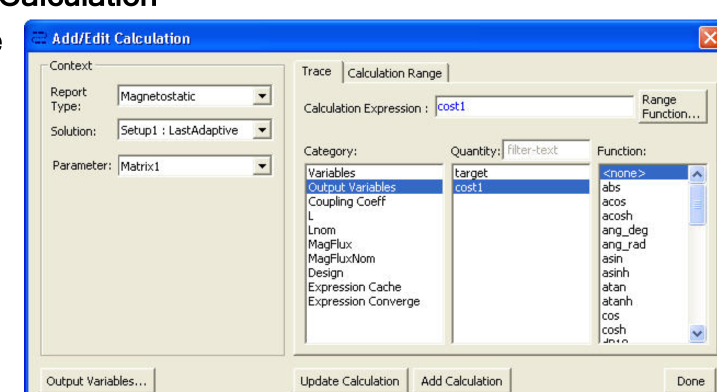
- It is possible to optimize current, material properties such as permeability, or any geometric dimension such as gap in order to obtain the specified inductance. For this optimization, the current will be varied to obtain the desired inductance of 0.8uH. Later, the gap will be varied to obtain the desired inductance
- **Specify Parametric Variables**
  - Select the menu item ***Maxwell 3D > Design properties***
  - In Properties window,
    1. Optimization: ☑ **Checked**
    2. gap:
       - Include: ☑ **Checked**
       - Min : **0.05 mm**
       - Max : **1 mm**
    3. ampturns:
       - Include: ☑ **Checked**
       - Min : **1 A**
       - Max : **500 A**
    4. Press **OK**



- **Setup an Optimization Analysis**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Optimization***
  - In Setup Optimization window,
    1. Optimizer: **Sequential Nonlinear Programming**
    2. Max. No. of Iterations: **20**
    3. Select **Setup Calculations**
    4. In Add/Edit calculations window, select **Output Variables**
    5. In Output variables window,
       - Name: **target**
       - Expression: **8e-7 H**
       - Select **Add**
       - Without closing window, set name to **cost1**
       - Parameter: **Matrix1**
       - Expression: **abs(target - L(Current1,Current1))/target/1H**

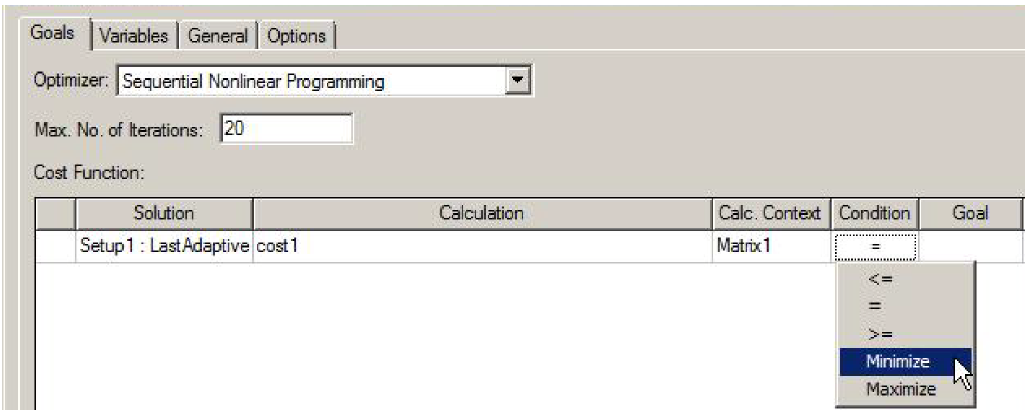
    Note: In the same Output Variable window, enter cost function. This unit-less cost function is the absolute value of the percent difference between the desired inductance (target) and the calculated inductance (L(Current1,Current1)). **Cost1** will approach zero as the calculated inductance approaches the desired inductance.Use ***Insert Into Expression*** buttons to help build this equation.

       - Press **Add** and **Done**
    6. In Add/Edit Calculations window,
       - Parameter : **Matrix1**
       - Category: **Output Variables**
       - Quantity: **cost1**
       - Select **Add Calculation**
       - Press **Done**

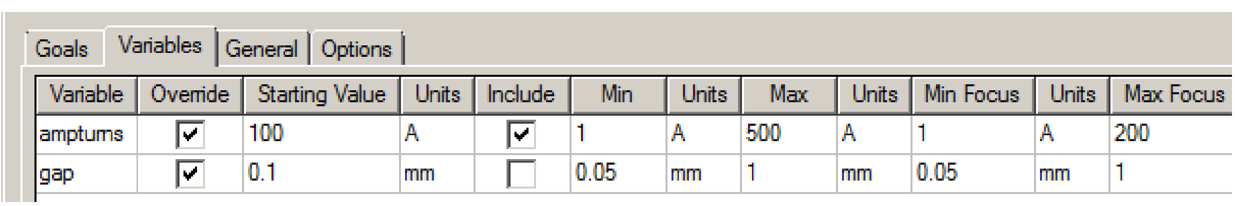


Example (Optimetrics) - Gapped Inductor

7. Parameter **cost1** will be added to Setup Optimization window.
8. Set Condition for cost1 to **Minimize**

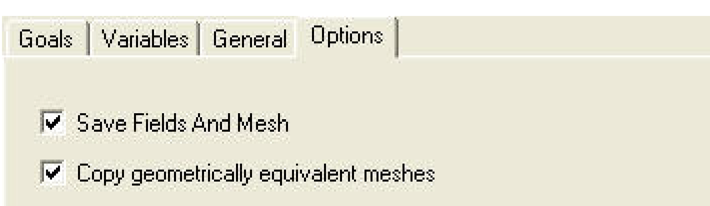


9. Change the tab to **Variables**
10. For the variable **ampturns**
    - Override: ☑ **Checked**
    - Include: ☑ **Checked**
    - Max Focus: **200**
11. For the variable **gap**
    - Override: ☑ **Checked**
    - Include: ☐ **Unchecked**



Variable	Override	Starting Value	Units	Include	Min	Units	Max	Units	Min Focus	Units	Max Focus
ampturns	☑	100	A	☑	1	A	500	A	1	A	200
gap	☑	0.1	mm	☐	0.05	mm	1	mm	0.05	mm	1

12. Change the tab to **Options**
    - Save Fields And Mesh: ☑ **Checked**
    - Copy geometrically equivalent meshes: ☑ **Checked**



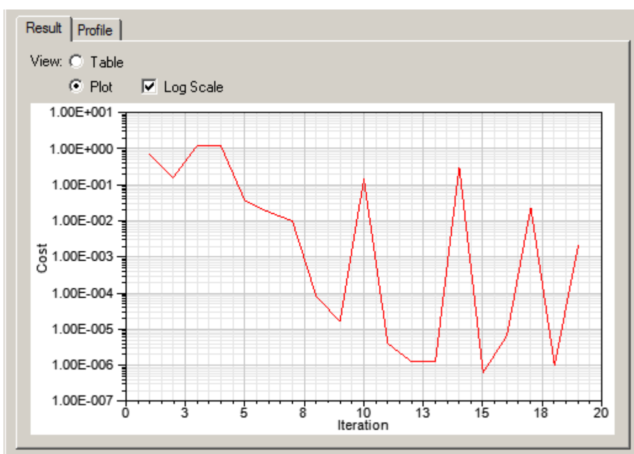
13. Press **OK** to close the window.

Solve Optimization Analysis

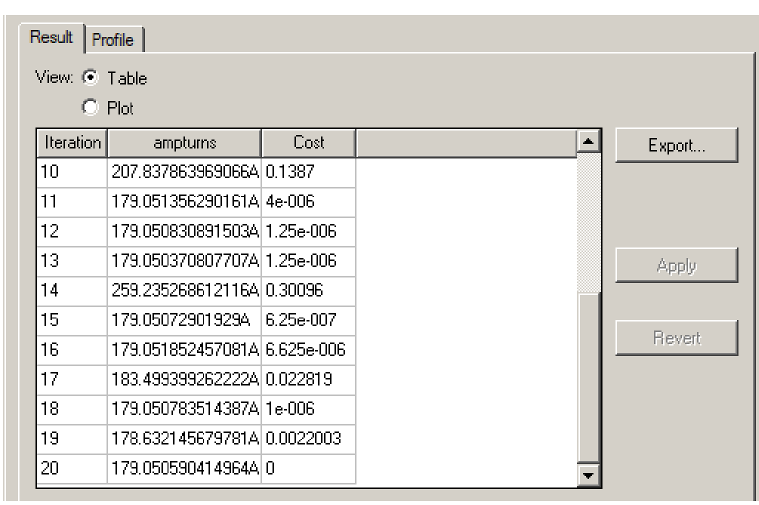
- **To Solve Optimization Analysis**
  - Select the menu item ***Maxwell 3D > Analyze All***

View Optimetrics Results

- **To View Results**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Optimetrics Results***
  - In Post Analysis Display window,
    1. Log Scale: ☑ **Checked**
  - Plot Display should look as below



  - Change View to **Table** to see actual values



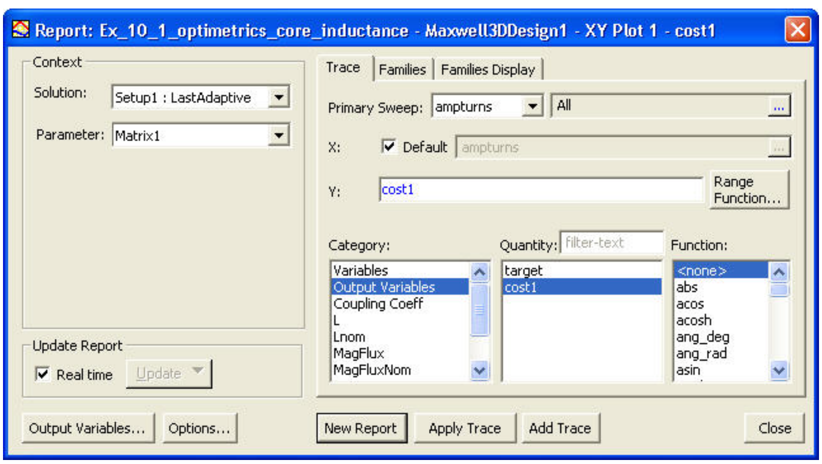
Iteration	ampturns	Cost
10	207.837863969066A	0.1387
11	179.051356290161A	4e-006
12	179.050830891503A	1.25e-006
13	179.050370807707A	1.25e-006
14	259.235268612116A	0.30096
15	179.05072901929A	6.25e-007
16	179.051852457081A	6.625e-006
17	183.499399262222A	0.022819
18	179.050783514387A	1e-006
19	178.632145679781A	0.0022003
20	179.050590414964A	0

  - Press **Close**

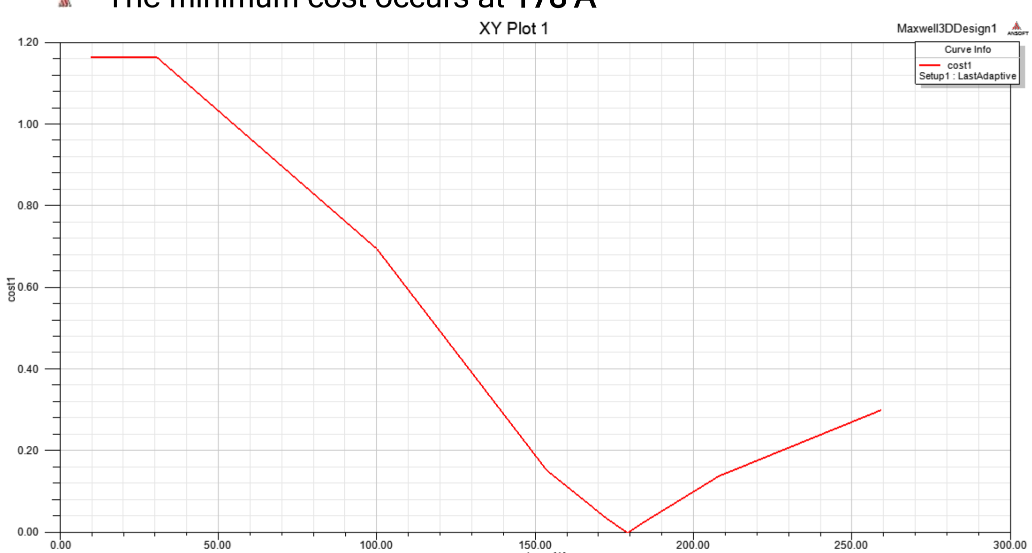
Example (Optimetrics) – Gapped Inductor

Create Plot of Cost vs Ampturns

- **To create a report:**
  - Select the menu item *Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot*
  - In Report window,
    1. Parameter: **Matrix1**
    2. Primary Sweep: **ampturns**
    3. Category: **Output Variables**
    4. Quantity: **cost1**
    5. Select **New Report**
    6. Press the **Close** button



- The minimum cost occurs at **178 A**



Second Optimetrics Setup and Solution

- For the second optimization, the core gap will be varied to obtain the desired inductance of 0.8uH with ampturns = 100A.

Setup Optimization Analysis

- **To Setup an Optimization Analysis**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Optimization***
  - In Setup Optimization window,
    1. Optimizer: **Sequential Nonlinear Programming**
    2. Max. No. of Iterations: **20**
    3. Select **Setup Calculations**
    4. In Add/Edit Calculations window,
       - Parameter : **Matrix1**
       - Category: **Output Variables**
       - Quantity: **cost1**
       - Select **Add Calculation**
       - Press **Done**
    5. Set Condition for cost1 to **Minimize**
    6. Change the tab to **Variables**
    7. For the variable **gap**
       - Include: ☑ **Checked**
    8. For the variable **ampturns**
       - Starting value: **100 A**
       - Include: ☐ **Unchecked**
    9. Change the tab to **Options**
       - Save Fields And Mesh: ☑ **Checked**
    10. Press **OK**

Goals | Variables | General | Options

Variable	Override	Starting Value	Units	Include	Min	Units	Max	Units	Min Focus	Units	Max Focus	Units
ampturns	☑	100	A	☐	1	A	500	A	1	A	500	A
gap	☐	0.1	mm	☑	0.05	mm	1	mm	0.05	mm	1	mm

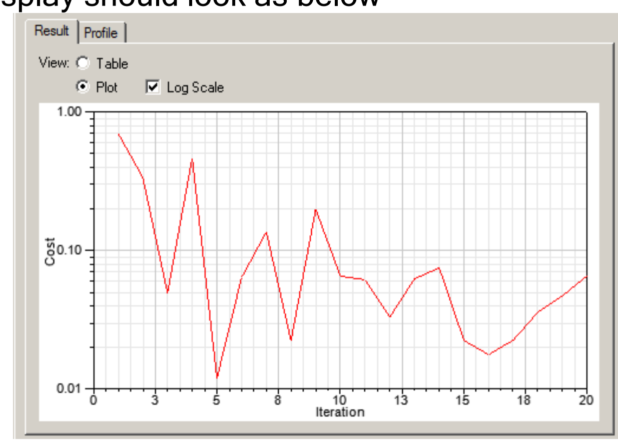
Example (Optimetrics) – Gapped Inductor

Solve Optimization Analysis

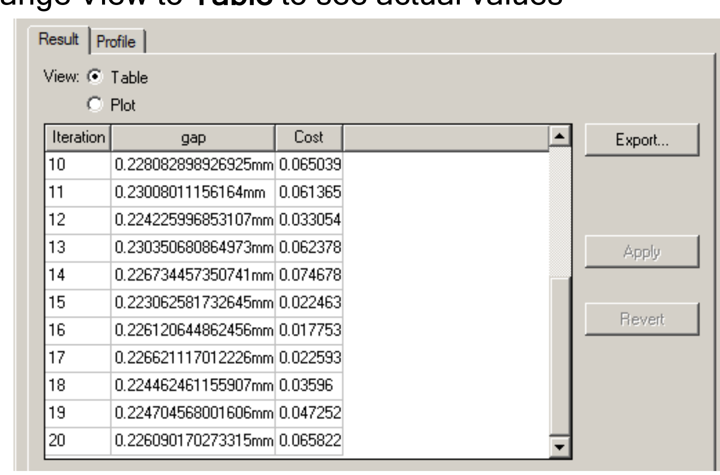
- **To Solve Optimization Analysis**
  - Exand the Project Manager tree to see **Optimetrics**
  - Right click on **Optimization Setup2** and select **Analyze**

View Optimetrics Results

- **To View Results**
  - Right click on the tab **Optimization Setup2** from Project Manager tree and select **View Analysis Results**
  - In Post Analysis Display window,
    1. Log Scale: ☑ **Checked**
  - Plot Display should look as below



  - Change View to **Table** to see actual values

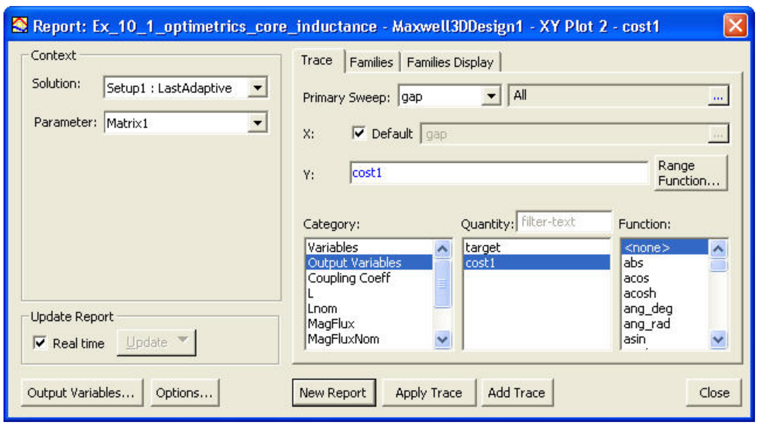


Iteration	gap	Cost
10	0.228082898926925mm	0.065039
11	0.23008011156164mm	0.061365
12	0.224225996853107mm	0.033054
13	0.230350680864973mm	0.062378
14	0.226734457350741mm	0.074678
15	0.223062581732645mm	0.022463
16	0.226120644862456mm	0.017753
17	0.226621117012226mm	0.022593
18	0.224462461155907mm	0.03596
19	0.224704568001606mm	0.047252
20	0.226090170273315mm	0.065822

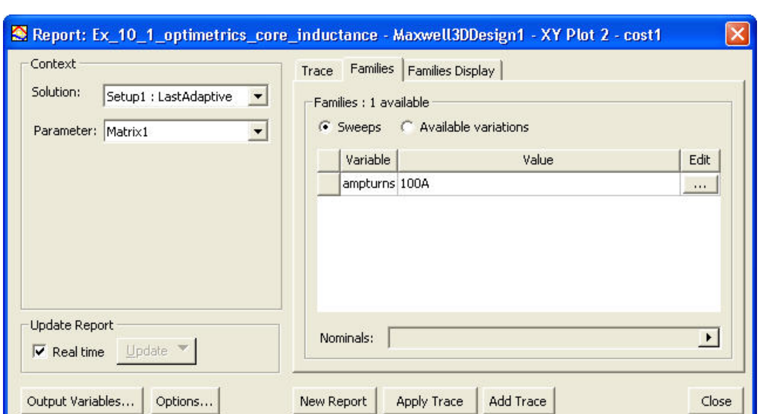
  - Press **Close**

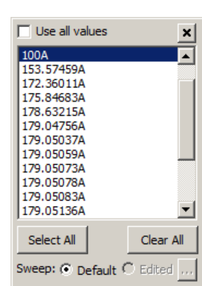
Create Plot of Cost vs gap

- **To create a report:**
  - Select the menu item ***Maxwell 3D > Results > Create Magnetostatic Report > Rectangular Plot***
  - In Report window,
    1. Parameter: **Matrix1**
    2. Primary Sweep: **gap**
    3. Category: **Output Variables**
    4. Quantity: **cost1**



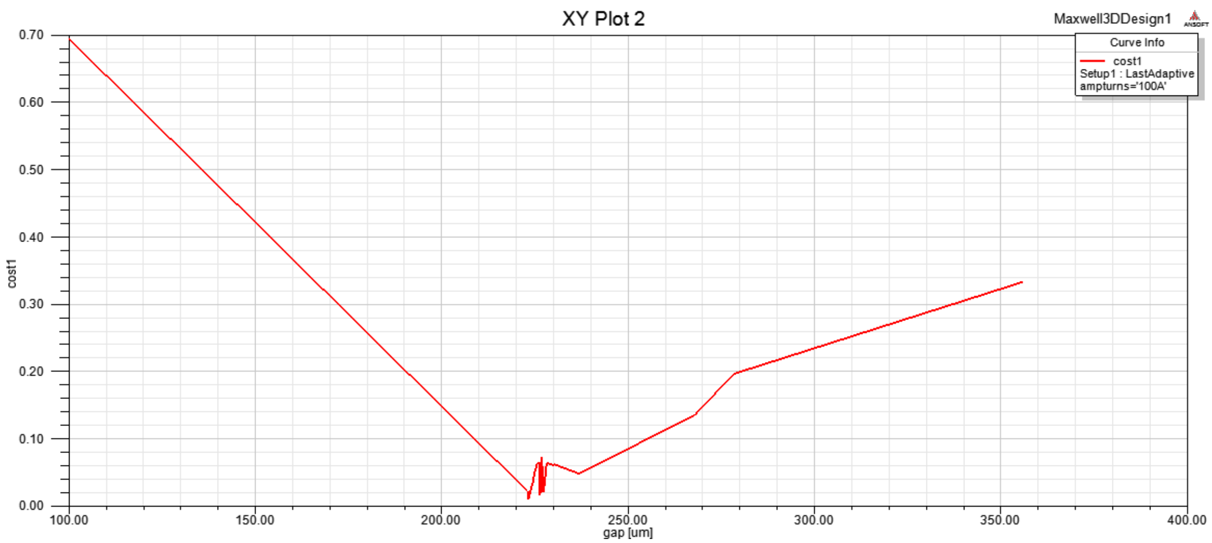
5. Change tab to **Families**
6. Under **Sweeps,** for variable **ampturns,** Select **Edit**
   - Use all values: ☐ **Unchecked**
   - Select the value **100 A**
   - Select **Close**
7. Select **New Report**
8. Press the **Close** button



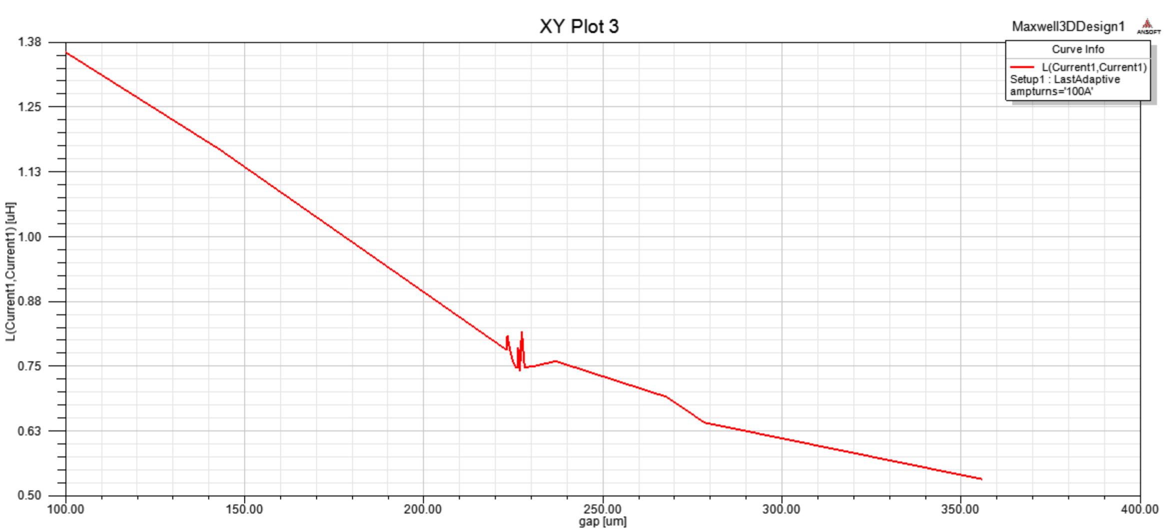


Example (Optimetrics) – Gapped Inductor

- Minimum value of cost occurs at around **195 μm**

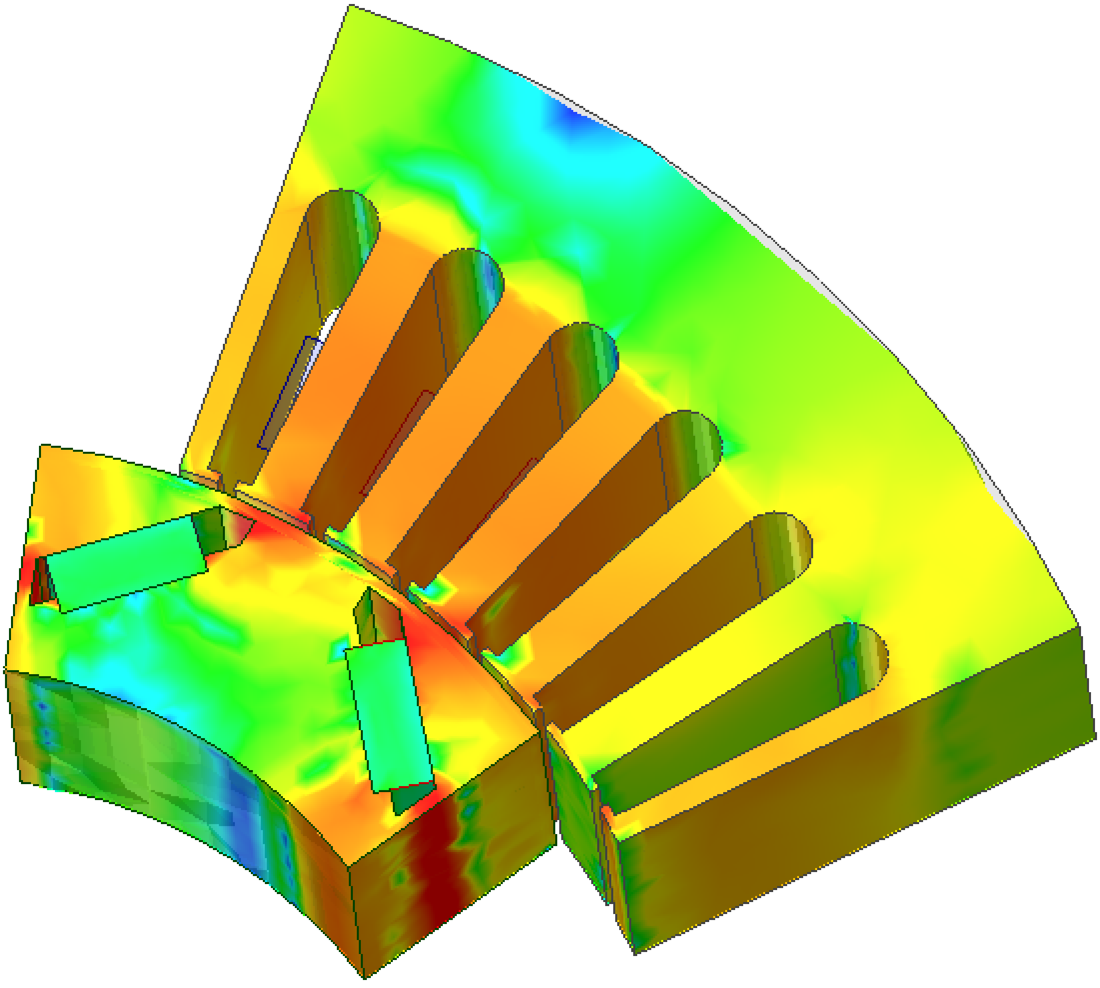


- In similar way, plot Inductance Vs Gap
  - From the plot it can be seen that target value of 0.8 uH for inductance is achieved at a gap of **220 μm**



Chapter 11.0 – Motors

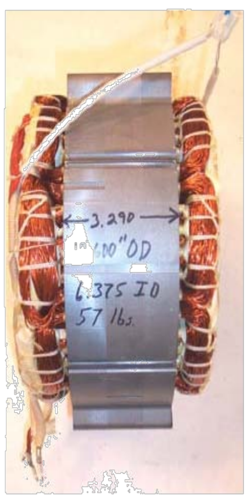
Study of a Permanent Magnet Motor with MAXWELL 3D: Example of the 2004 Prius IPM Motor



Study of a Motor

- The Electro Mechanical software package provided by Ansoft enables extensive motor simulation. This application note details the simulaton of a motor with Maxwell3D. We will cover static and transient simulations.
- This application note will use the 2004 Toyota Prius motor as basis. It is a 8-pole permanent magnet motor with embedded magnets. The single layer windings are made of 3 phases. The stator has 48 slots. This motor is public, we therefore have the full set of parameters. We will also use Oak Ridge National Laboratory testing results in this note.

Note: This application has not been done with the collaboration of Toyota



- References:
  - **Report on Toyota/Prius Motor Torque Capability, Torque Property, No-Load Back EMF, and Mechanical Losses,**
    - J. S. Hsu, Ph.D., C. W. Ayers, C. L. Coomer, R. H. Wiles
    - Oak Ridge National Laboratory
  - **Report on Toyota/Prius Motor Design and manufacturing Assessment**
    - J. S. Hsu, C. W. Ayers, C. L. Coomer
    - Oak Ridge National Laboratory
  - **Evaluation of 2004 Toyota Prius Hybrid Electric Drive System Interim Report**
    - C. W. Ayers, J. S. Hsu, L. D. Marlino, C. W. Miller,G. W. Ott, Jr.,C. B. Oland
    - Oak Ridge National Laboratory

Overview of the Study:

- Getting Started
  - Launching Maxwell
  - Setting Tool Options
  - Opening a new project
  - Set Model Units
- Creating the 3D Model
  - Create the Stator
  - Create the Rotor
  - Create the Magnets
  - Create the Windings
- Reducing the size of the 3D Model
- Material properties of the motor
  - Permanent Magnets characterization
  - Steel definition
- Applying Master/Slave Boundary Condition
- STATIC ANALYSIS
  - Full Load Study
  - Apply Excitations
  - Apply Mesh Operations
  - Apply Torque computation
  - Inductance computation
  - Analyze Setup
  - Analyze
  - Postprocessing

Overview of the Study (Cont'd)

- DYNAMIC ANALYSIS
  - Create Current Terminals
  - Motor Excitation
  - Create Parameters for excitations
  - Create Windings
  - Add Band object
  - Mesh Operations
  - Assign Movement
  - Add an Analysis Setup
  - Solve the problem
  - Post Processing
  - Parametric Study

Getting Started

Launching Maxwell

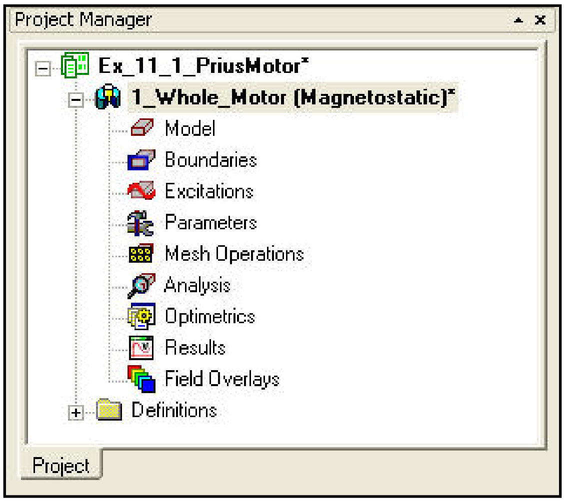
1. To access Maxwell, click the Microsoft **Start** button, select **Programs**, and select **Ansoft** and then **Maxwell 15.0** and **Maxwell15.0**

Setting Tool Options

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

Opening a New Project

- **To open a new project:**
  - In an Maxwell window, click the icon on the Standard toolbar, or select the menu item ***File > New***.
  - Right mouse click on the project name, then select the menu item Rename. Change the project name to **Ex_11_1_PriusMotor**
  - Select the menu item ***Project > Insert Maxwell Design***, or click on the icon
  - Right mouse click on Maxwelldesign1 and select Rename. Change the name to **1_Whole_Motor**



Set Model Units

- **To Set Model Units**
  - Select the menu item ***Modeler > Units.***
    1. Select Units*: mm (millimeters)*
    2. Press **OK**

Creating the 3D Model

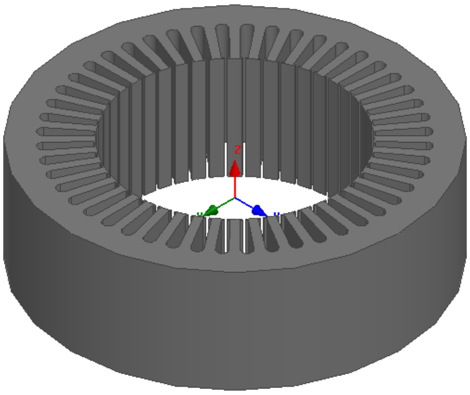
- Maxwell has number of **User Defined Primitives** for motor parts. These primitives can describe all the main parts of motors.

- **Create the Stator:**
  - A User Defined Primitive will be used to create the stator
  - Select the menu item ***Draw > User Defined Primitive > Syslib > Rmxprt > SlotCore***
  - Use the values given in the panel below to create the stator

Command

Name	Value	Unit	Evaluated Value	Description
DiaGap	161.9	mm	161.9mm	Core diameter on gap side, DiaGa...
DiaYoke	269.24	mm	269.24mm	Core diameter on yoke side, DiaYo...
Length	83.82	mm	83.82mm	Core length
Skew	0	deg	0deg	Skew angle in core length range
Slots	48		48	Number of slots
SlotType	2		2	Slot type: 1 to 6
Hs0	1.03	mm	1.03mm	Slot opening height
Hs01	0	mm	0mm	Slot closed bridge height
Hs1	0	mm	0mm	Slot wedge height
Hs2	29.5	mm	29.5mm	Slot body height
Bs0	1.93	mm	1.93mm	Slot opening width
Bs1	5	mm	5mm	Slot wedge maximum width
Bs2	8	mm	8mm	Slot body bottom width, 0 for parall...
Rs	8	mm	8mm	Slot body bottom fillet
FilletType	5		5	0: a quarter circle; 1: tangent conn...
HalfSlot	0		0	0 for symmetric slot, 1 for half slot
SegAngle	0	deg	0deg	Deviation angle for slot arches (10...
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 100: region.

- **Change Attributes**
  - Change the name of the object to **Stator**
  - Change the color of the object to **Gray**

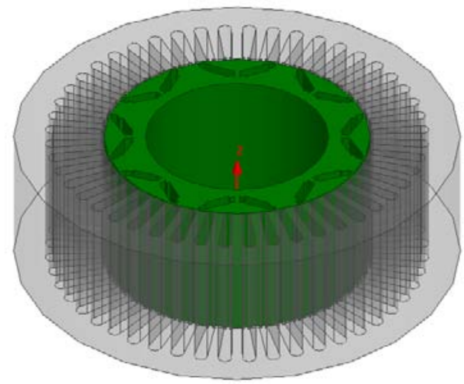


- **Create the Rotor**
  - A User Defined Primitive will be used to create the rotor
  - Select the menu item ***Draw > User Defined Primitive > Syslib > Rmxprt > IPMCore***
  - Use the values given in the below to create the rotor

Parameters | Info

Name	Value	Unit	Evaluated Value	Description
DiaGap	160.4	mm	160.4mm	Core diameter on gap si...
DiaYoke	110.64	mm	110.64mm	Core diameter on yoke ...
Length	83.82	mm	83.82mm	Core length
Poles	8		8	Number of poles
PoleType	3		3	Pole type: 1 to 6.
D1	157.44	mm	157.44mm	Limited diameter of PM ...
O1	3	mm	3mm	Bottom width for separa...
O2	7.28	mm	7.28mm	Distance from duct bott...
B1	4.7	mm	4.7mm	Duct thickness
Rib	14	mm	14mm	Rib width
HRib	3	mm	3mm	Rib height (for types 3~5)
DminMag	4.5	mm	4.5mm	Minimum distance betw...
ThickMag	6.48	mm	6.48mm	Magnet thickness
WidthMag	32	mm	32mm	Total width of all magn...
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 1: magnets; 2: ...

- **Change Attributes**
  - Change the name of the object to **Rotor**
  - Change the color of the object to **Green**

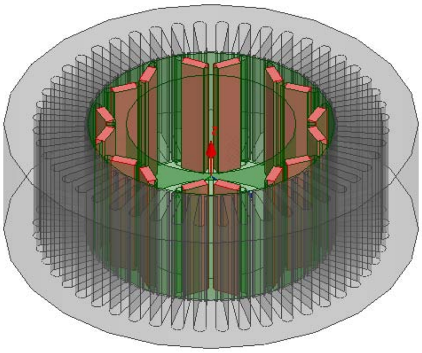


- **Create the Magnets**
  - The same User Defined Primitive can be used to create the magnets, but with different parameters. UDPs can be computed to generate different topologies.
  - Select the menu item ***Draw > User Defined Primitive > Syslib > Rmxprt > IPMCore***
  - Use the values given in the panel below to create the magnets

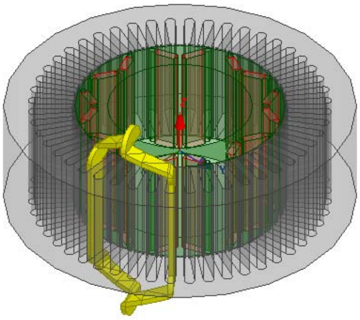
Command

Name	Value	Unit	Evaluated Value	Description
DiaGap	160.4	mm	160.4mm	Core diameter on gap side, or oute...
DiaYoke	110.64	mm	110.64mm	Core diameter on yoke side, or inn...
Length	83.82	mm	83.82mm	Core length
Poles	8		8	Number of poles
PoleType	3		3	Pole type: 1 to 6.
D1	157.44	mm	157.44mm	Limited diameter of PM ducts
O1	3	mm	3mm	Bottom width for separate or flat-b...
O2	7.28	mm	7.28mm	Distance from duct bottom to shaft...
B1	4.7	mm	4.7mm	Duct thickness
Rib	14	mm	14mm	Rib width
HRib	3	mm	3mm	Rib height (for types 3~5)
DminMag	4.5	mm	4.5mm	Minimum distance between side m...
ThickMag	6.48	mm	6.48mm	Magnet thickness
WidthMag	32	mm	32mm	Total width of all magnet per pole
LenRegion	200	mm	200mm	Region length
InfoCore	1		1	0: core; 1: magnets; 2: ducts; 3: o...

- **Change Attributes**
  - Change the name of the object to **Magnet**
  - Change the color of the object to **Light Red**



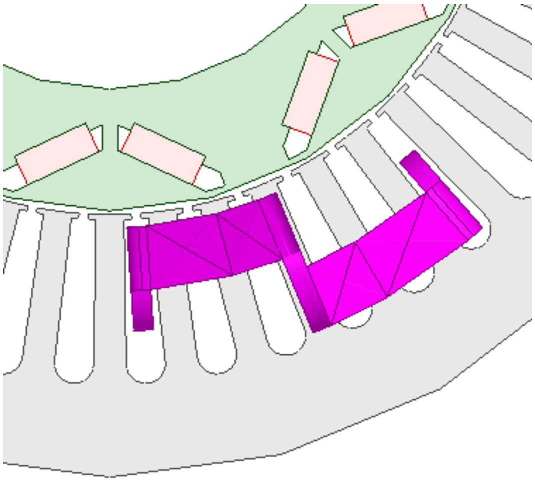
- **Create Windings**
  - A User Defined Primitive will be used to create the windings.
  - Select the menu item ***Draw > User Defined Primitive > Syslib > Rmxprt > LapCoil***
  - Use the values given in the panel below to create the coil
- **Change Attributes**
  - Change the material of the object to **Copper**
  - Change the color of the object to **Yellow**

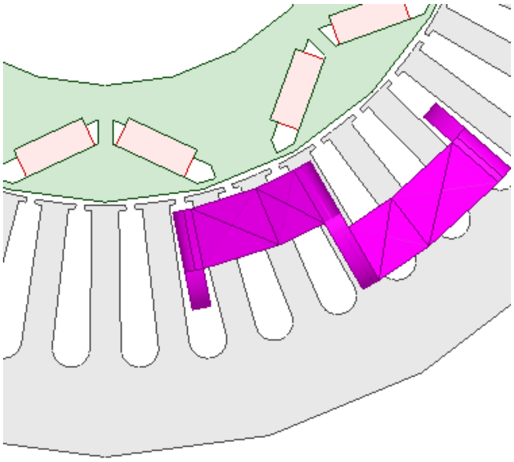


Parameters | Info

Name	Value	Unit	Evaluated Value	Description
DiaGap	161.9	mm	161.9mm	Core diameter on gap si...
DiaYoke	269.24	mm	269.24mm	Core diameter on yoke ...
Length	85	mm	85mm	Core length
Skew	0	deg	0deg	Skew angle in core len...
Slots	48		48	Number of slots
SlotType	2		2	Slot type: 1 to 7
Hs0	1.03	mm	1.03mm	Slot opening height
Hs1	0	mm	0mm	Slot wedge height
Hs2	29.5	mm	29.5mm	Slot body height
Bs0	1.93	mm	1.93mm	Slot opening width
Bs1	5	mm	5mm	Slot wedge maximum w...
Bs2	8	mm	8mm	Slot body bottom width,...
Rs	5	mm	5mm	Slot body bottom fillet
FilletType	0		0	0: a quarter circle; 1: ta...
Layers	1		1	Number of winding layers
CoilPitch	5		5	Coil pitch measured in s...
EndExt	0	mm	0mm	One-side end extended...
SpanExt	5	mm	5mm	Axial length of end spa...
SegAngle	15	deg	15deg	Deviation angle for end...
LenRegion	200	mm	200mm	Region length
InfoCoil	1		1	0: winding; 1: coil; 2: te...

- **Rotate Coil**
  - Select the object **LapCoil1** from the history tree
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **7.5 deg**
    3. Press **OK**



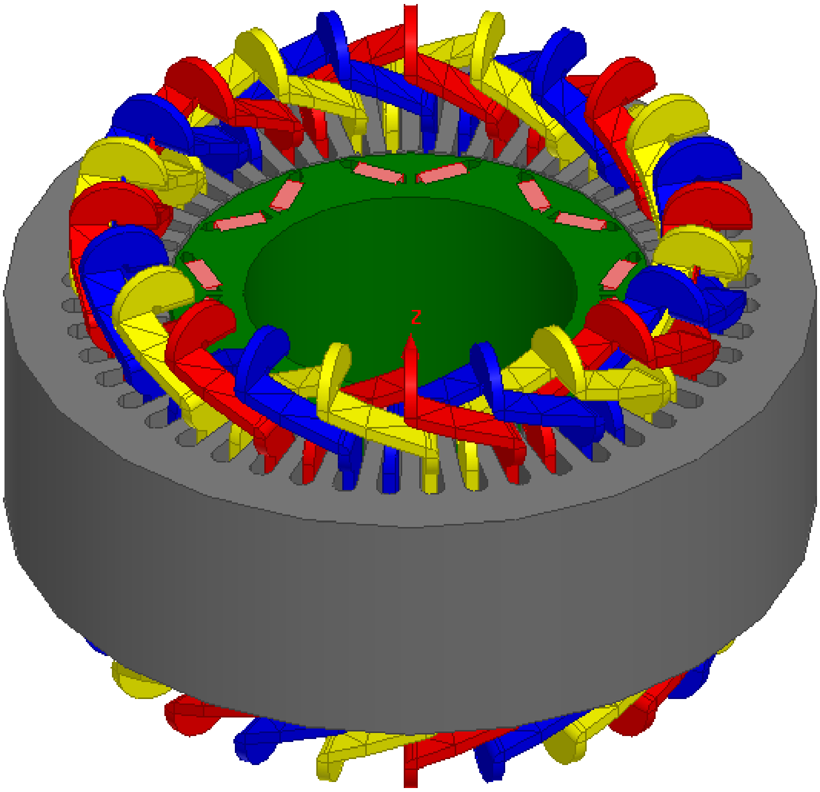


- **Duplicate Coil**
  - Select the object **LapCoil1** from the history tree
  - Select the menu item ***Edit > Duplicate > Around Axis***
  - In Duplicate Around Axis window,
    1. Axis: **Z**
    2. Angle : **15 deg**
    3. Total Number: **3**
    4. Press **OK**

- **Change Attributes**
  - Change the name of the objects LapCoil1, LapCoil1_1 and LapCoil1_2 to **PhaseA, PhaseC,** and **PhaseB** respectively
  - Change the color of the objects PhaseB and Phase C to **Blue** and **Red**

- **Duplicate Coils**
  - Press **Ctrl** and select the object **PhaseA, PhaseB** and **PhaseC** from the history tree
  - Select the menu item ***Edit > Duplicate > Around Axis***
  - In Duplicate Around Axis window,
    1. Axis: **Z**
    2. Angle : **45 deg**
    3. Total Number: **8**
    4. Press **OK**

- Motor geometry is complete



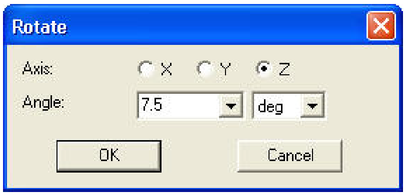
Copy Design

To Copy Design

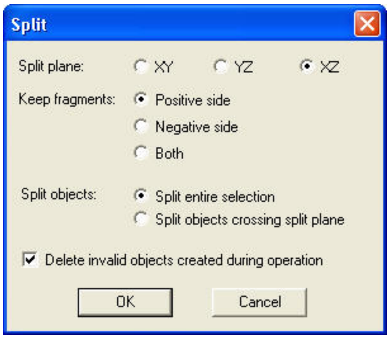
- Select the Maxwell design **“1_Whole_Motor”** from the Project Manager tree, right click and select **“Copy”**
- Select the project name from the Project manager tree, right click and select **“Paste”**
- Change the name of the design to **“2_Partial_Motor”**

Reduce Model Size

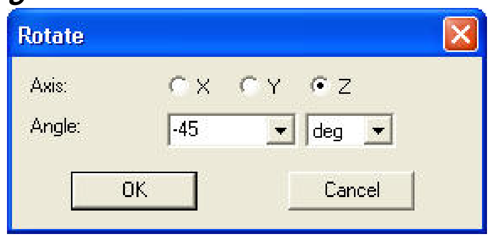
- We can take advantage of the topology of the motor to reduce the size of the problem. This motor has 8 pair of poles. We can only use one height of the motor. This is valid because the stator has:
  - 48 slots (8 is a divider of 48).
  - The 3-phase winding has also a periodicity of 45 degrees.
- From now on, the Maxwelldesign ‘*2_Partial_motor*’ will be used. We have saved a copy of the whole geometry as it will be used later for other studies.
- **Rotate Model**
  - Press **Ctrl** and select the object **Stator** and **all Coils**
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **7.5 deg**
    3. Press **OK**



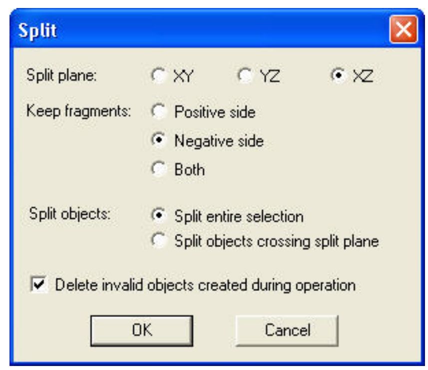
  - This will shift the winding before splitting the objects.
- **Split Model by XZ**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Split***
  - In Split window,
    1. Split Plane: **XZ**
    2. Keep fragments: **Positive Side**
    3. Split Objects: **Split entire Selection**
    4. Press **OK**



- **Rotate Model**
  - We have achieved half model now
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **-45 deg**
    3. Press **OK**

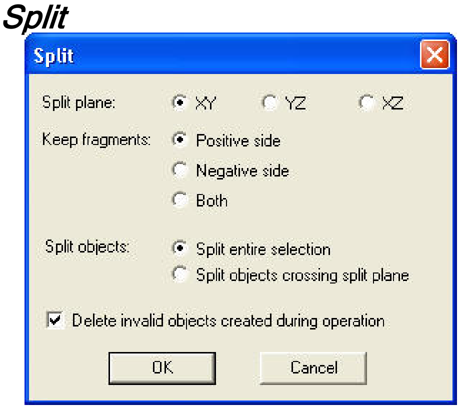


- **Split Model by XZ**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Split***
  - In Split window,
    1. Split Plane: **XZ**
    2. Keep fragments: **Negative Side**
    3. Split Objects: **Split entire Selection**
    4. Press **OK**

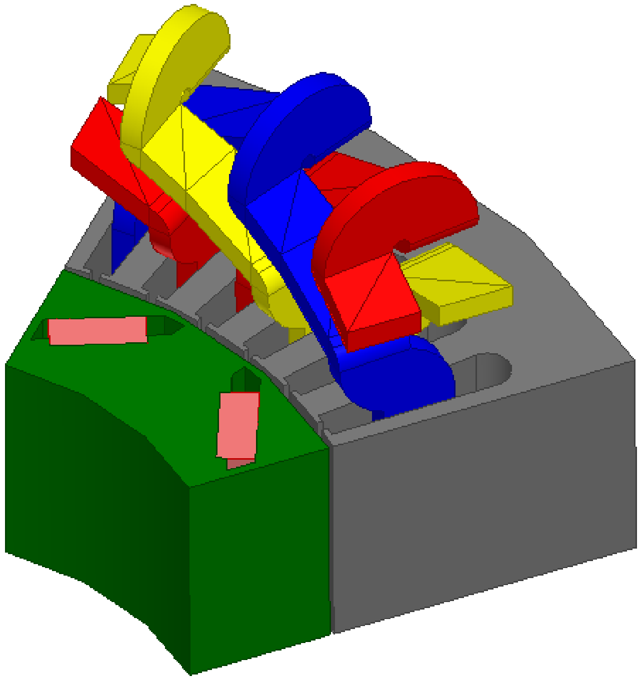


- **Rotate Model**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Edit > Arrange > Rotate***
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **45 deg**
    3. Press **OK**

- **Split Model by XY**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Split***
  - In Split window,
    1. Split Plane: **XY**
    2. Keep fragments: **Positive Side**
    3. Split Objects: **Split entire Selection**
    4. Press **OK**



- **Reduced Model**
  - The 3D model now looks like below



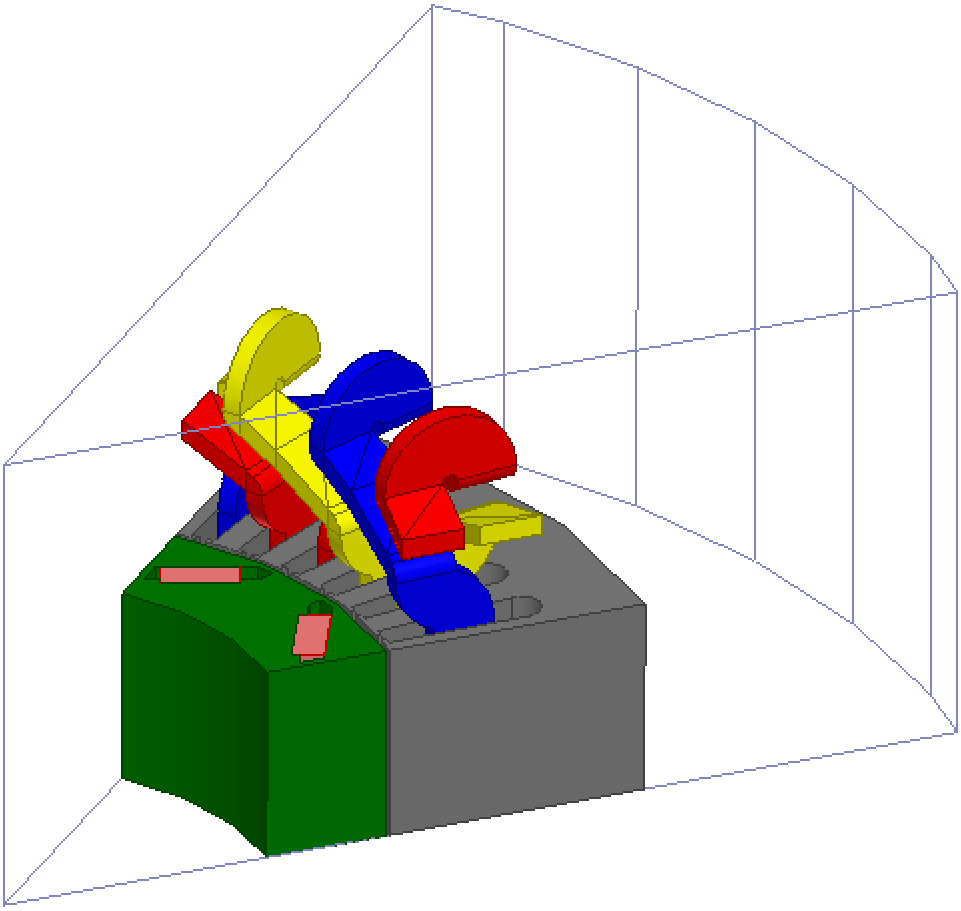
- **Change Attributes**
  - Change the name of the objects PhaseA and PhaseA_7 to **PhaseA1** and **PhaseA2**
  - Change the name of the objects PhaseB_7 to **PhaseB1**
  - Change the name of the object PhaseC and PhaseC_7 to **PhaseC1** and **PhaseC2**

Create Region

- **Create Rectangle**
  - Select the menu item ***Modeler > Grid Plane > XZ***
  - Select the menu item ***Draw > Rectangle***
    1. Using the coordinate entry field, enter the box position
       - **X: 0.0, Y: 0.0, Z: 0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the relative size of the box
       - **dX: 200.0, dY: 0.0, dZ: 100.0**, Press the **Enter** key

- **Sweep Rectangle**
  - Select the sheet **Rectangle1** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Axis: **Z**
    2. Angle of sweep: **45**
    3. Number of segments: **5**
    4. Press **OK**

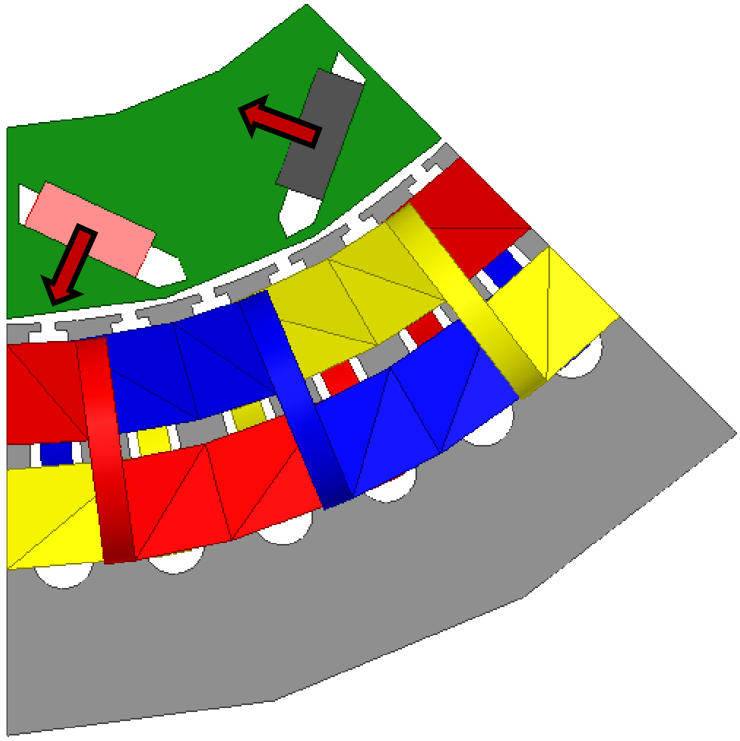
- **Change Attributes**
  - Change the name of the object to **Region**
  - Select Display Wireframe: ☑ **Checked**



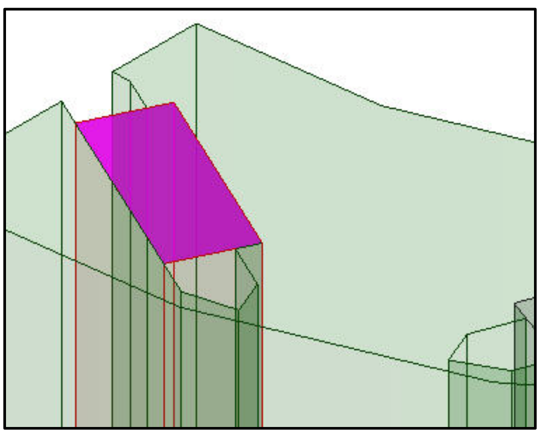
Material Properties for the Motor

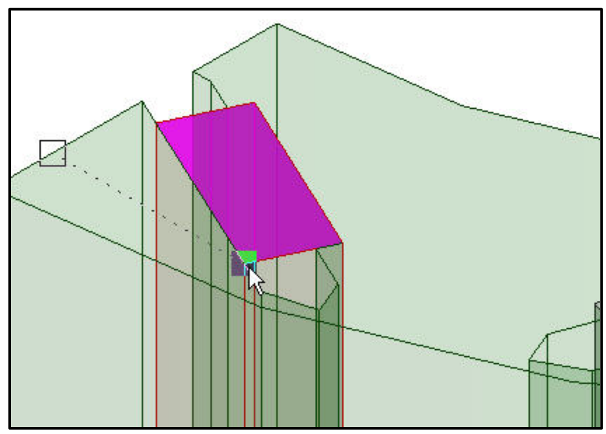
Permanent Magnet Characterization

- The Prius Permanent Magnets (PMs) are high-strength magnets.
- In order to define PMs magnetization orientation, we need to create separate objects for each magnet.
- **Separate Objects**
  - Select the object **Magnets** from the history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Change Attributes**
  - Change the name of the object Magnets to **PM1**
  - Change the name of the object Magnets_Separate1 to **PM2**
- Since the magnets will rotate, the orientation cannot be given through fixed coordinate systems (CS). The use of face CS is required. Face CS are CS that are attached to the face of an object. When the object moves, the Face CS also moves along with the object.
- The Prius's PMs are oriented as shown below. Therefore, we will create a face CS for each magnet.

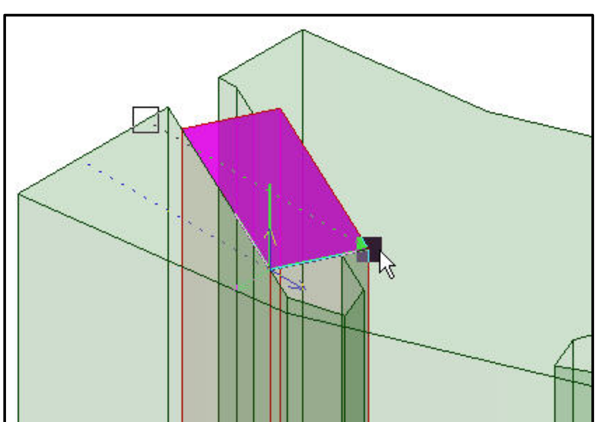


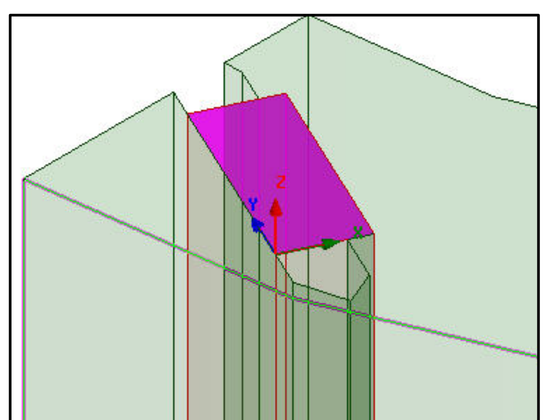
- **Hide All other Objects**
  - Select the menu item ***View > Visibility > Active View Visibility***
  - In the window,
    1. Disable the visibility for all objects except **Rotor, PM1** and **PM2**
    2. Press **Done**
- **To Create Face Coordinate System**
  - Select the menu item ***Edit > Select > Faces*** or press **“f”** from keyboard
  - Select the top face of the object **PM1** as shown in below image
  - Select the menu item ***Modeler > Coordinate System > Create > Face CS***
  - Select the vertex on the top face of PM1 to define origin



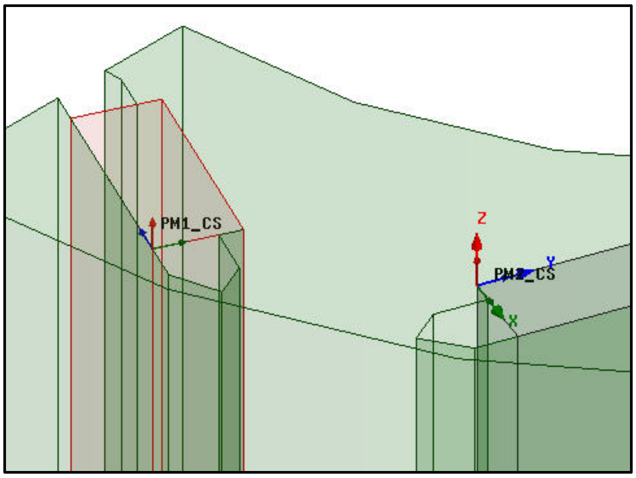


  - Select the vertex of the magnet as shown in below image to define X axis
  - A Face coordinate system will be created as shown in below image
  - Rename the FaceCS1 to **PM1_CS**

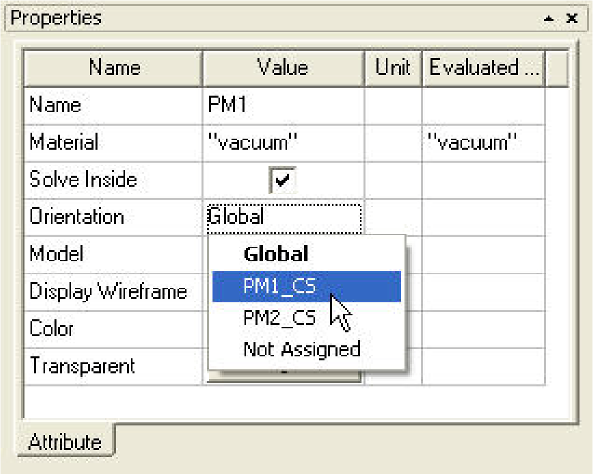




- Similarly Create a Face coordinate system for PM2 and name it as **PM2_CS.**
- Make sure to have the X axis looking toward the air gap

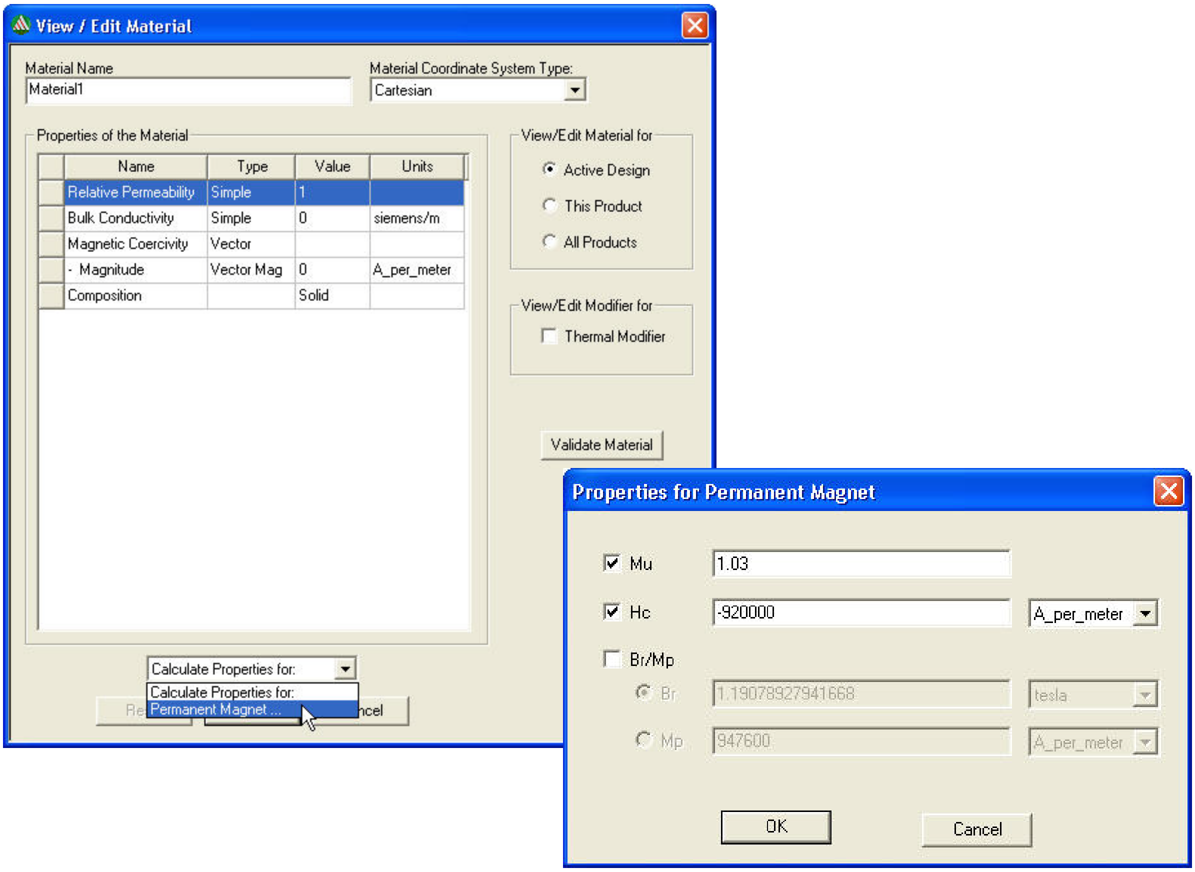


- Select the menu item ***Modeler > Coordinate System > Set Work CS***
    1. Select **Global**
- **Set Magnet Orientations**
    - Select the object **PM1** and goto the Properties window,
    - Change the orientation coordinate system from Global to **PM1_CS**

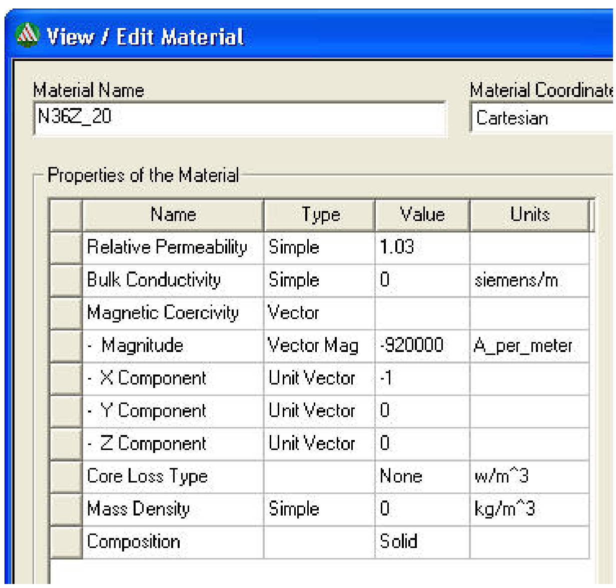


- Similarly change the orientation coordinate system for **PM2** to **PM2_CS**
- Select the menu item ***View > Visibility > Show All > Active View***
- Select the menu item ***Edit > Select > Objects*** or press **"O"** from keyboard

- **Create Material for PM1**
  - Select the object **PM1** from the history tree, right click and select **Assign Material**
  - In Select definition window, press the button **Add Material** to add new material
  - In View/Edit Material window,
    - A separate menu is available at the bottom of window to assign Magnet Properties
    - Set the option at the bottom Calculate Properties for **Permanent Magnet**
    - In Properties for Permanent Magnet window,
      1. Mu: **1.03**
      2. Hc: **-920000 A_per_meter**
      3. Press **OK**



- In View/Edit Material window,
  1. Material Name: **N36Z_20**
     - If the coordinate system *PM1_CS* is such that the **X axis** goes in the opposite direction of the air gap accordingly to the image below, change the X orientation to **-1** and 0 for the Y and Z components. If the X axis was in the opposite direction, you would need to enter 1 for the X component.
  2. Select **Validate Material**
  3. Press **OK**



View / Edit Material

Material Name: N36Z_20

Material Coordinate: Cartesian

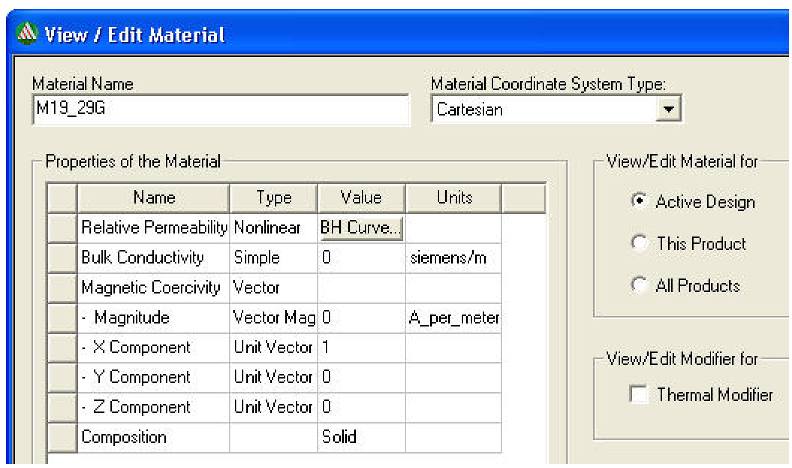
Properties of the Material

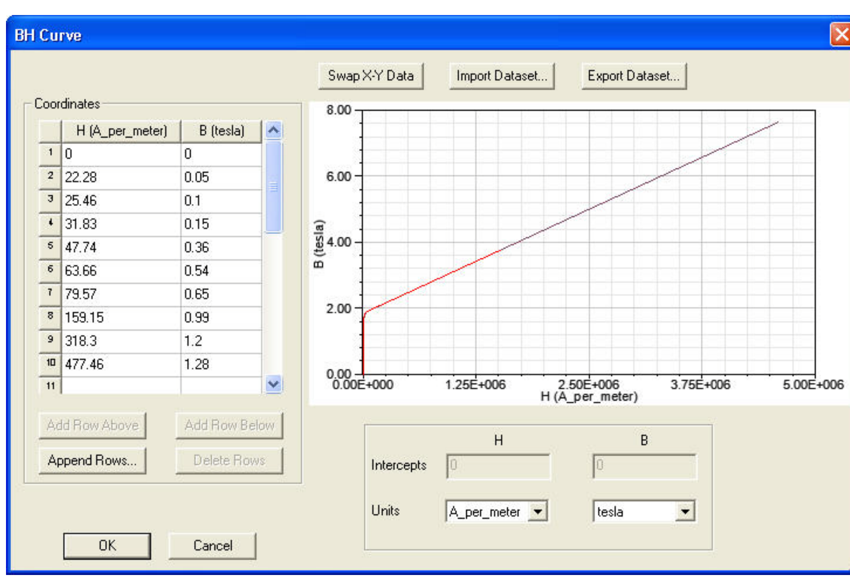
Name	Type	Value	Units
Relative Permeability	Simple	1.03	
Bulk Conductivity	Simple	0	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-920000	A_per_meter
- X Component	Unit Vector	-1	
- Y Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Core Loss Type		None	w/m^3
Mass Density	Simple	0	kg/m^3
Composition		Solid	

- **Material for PM2**
  - If the definition of *PM2_CS* is opposite ***with PM1_CS*** (X axis in the direction of the air gap), you can use the same material for ***N36Z_20*** for ***PM2***. If it is not the case, you can clone the material ***N36Z_20*** and change the orientation to be consistent with the ***PM2_CS*** axis.
  - Since we created PM2_CS in opposite to PM1_CS, we will use same material
  - Right click on the object **PM2** and select **Assign Material**
  - In Select Definition window,
    - Type **N36Z_20** in Search by Name field
    - Press **OK**

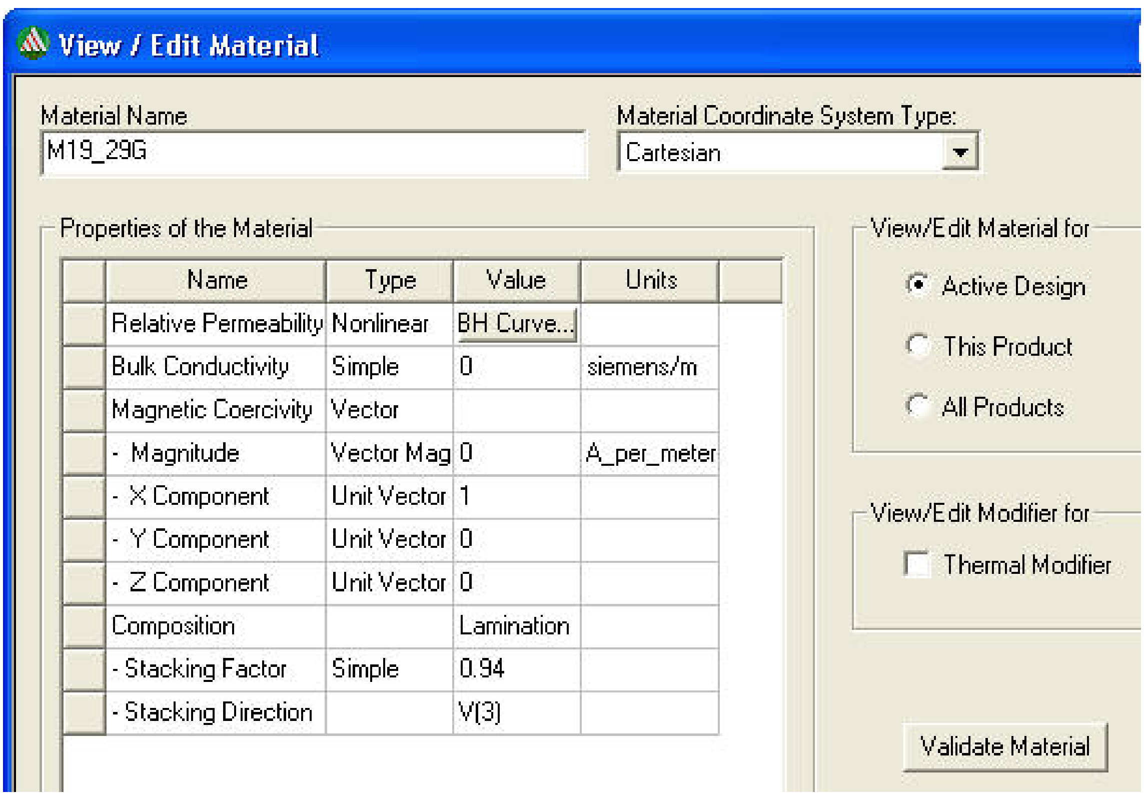
- **Material for Stator and Rotor**
  - Press **Ctrl** and select the objects **Stator** and **Rotor,** right click and select **Assign Material**
  - In Select Definition window, select **Add Material**
  - In View/Edit Material window,
    1. Material Name: **M19_29G**
    2. Relative Permeability
       - Type: **Nonlinear**
       - Press the button **BH Curve** that appears in Value field
       - In **BH Curve** window, enter that values for **B** and **H** as given in below table
       - Press **OK**

H	B
0	0
22.28	0.05
25.46	0.1
31.83	0.15
47.74	0.36
63.66	0.54
79.57	0.65
159.15	0.99
318.3	1.2
477.46	1.28
636.61	1.33
795.77	1.36
1591.5	1.44
3183	1.52
4774.6	1.58
6366.1	1.63
7957.7	1.67
15915	1.8
31830	1.9
111407	2
190984	2.1
350138	2.3
509252	2.5
560177.2	2.563994494
1527756	3.779889874



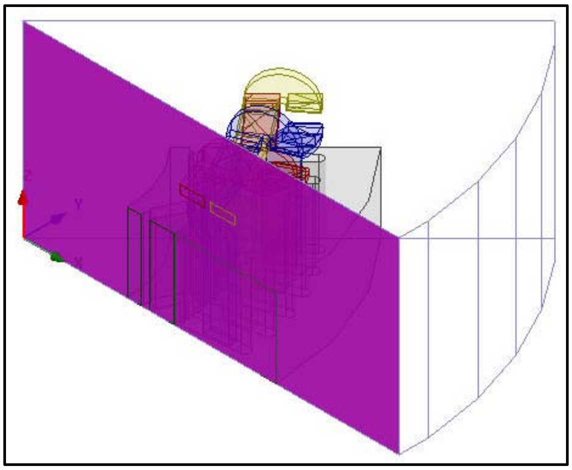


- In View/Edit Material window,
  1. Composition: change to **Lamination**
  2. Stacking Factor: **0.94**
     - The stacking factor is the proportion of steel in regards to insulating. Also we provide to Maxwell the lamination direction.
  3. Stacking Direction : Select **V(3)** (Z Direction)
     - We neglect the Eddy current in this example, therefore we leave the conductivity to 0.
  4. Press **Validate Material**
  5. Press **OK**

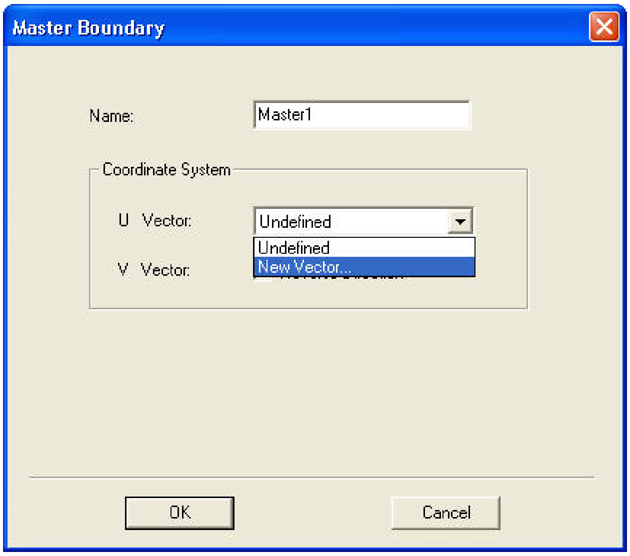


Applying Master / Slave Boundary

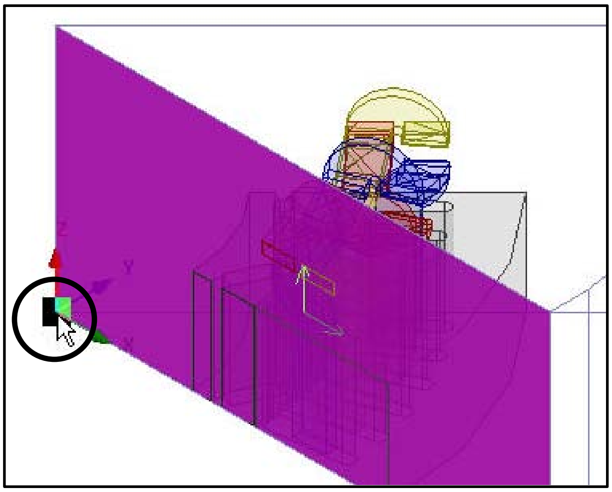
- The Master and Slave boundary condition takes advantage of the periodicity of the motor. Two planes are to be defined: the master and slave planes. The H-field at every point on the slave surface matches the (plus or minus) H-field at every point on the master surface.
- **To Assign Master Boundary**
  - Select the menu item ***Edit > Select > Faces***
  - Select the face of the **Region** that lies on Global:XZ Plane



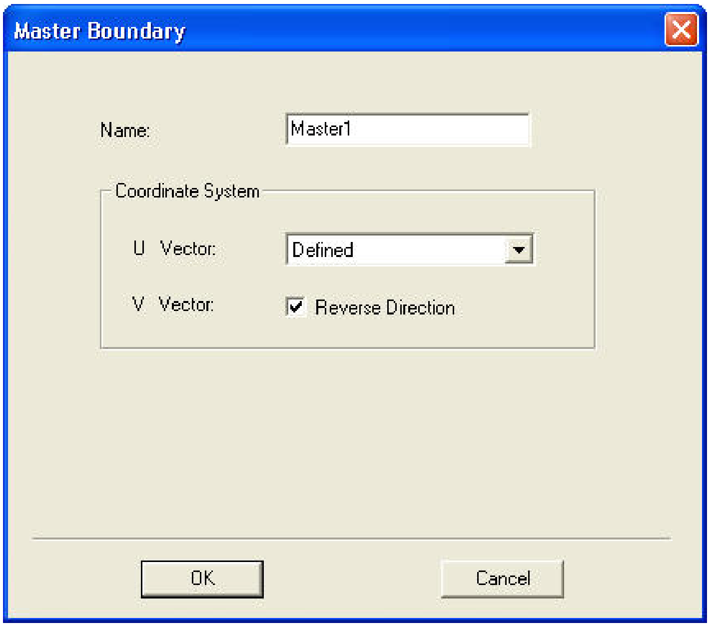
  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Master***
  - In Master Boundary window,
    1. U Vector: Select the option **New Vector**



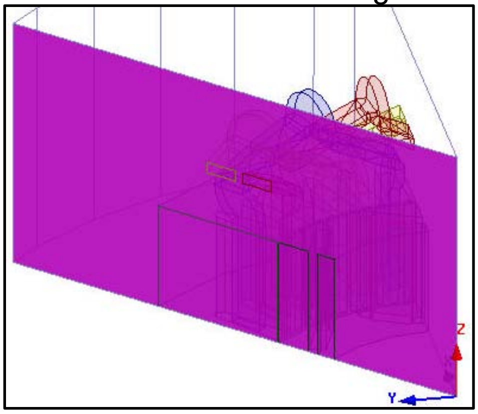
2. Select the vertex of selected face that lies on the Global origin to define first point for U Vector
3. Select the outer vertex of the selected face to define second point



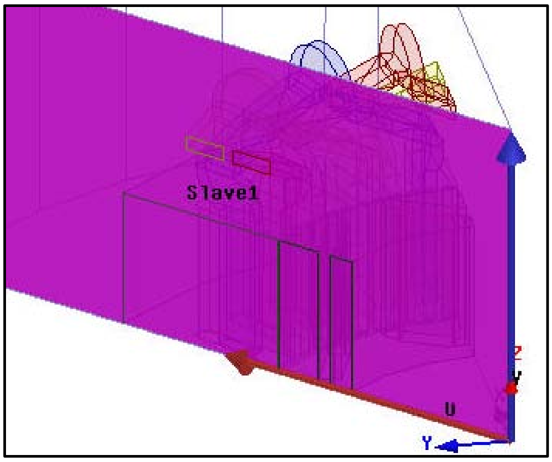
4. V Vector: Select **Reverse Direction**
5. Press **OK** to finish master definition

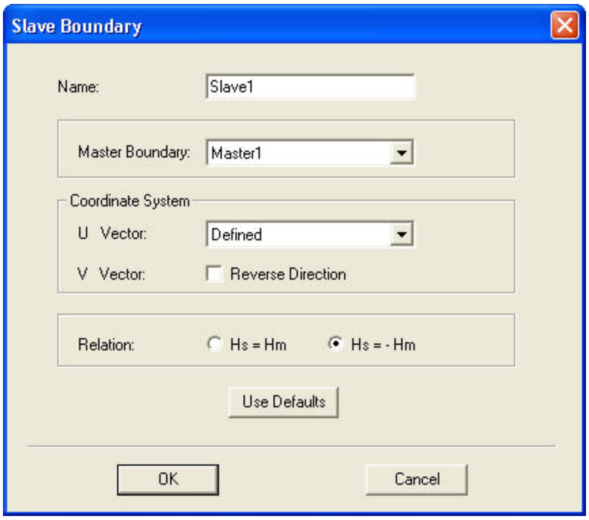


- **To Define Slave Boundary**
  - Select opposite vertical face of the region as shown in below image



  - Select the menu item ***Maxwell 3D > Boundaries > Assign > Slave***
  - In Slave Boundary window,
    1. Master Boundary: **Master1**
    2. U Vector: **New Vector**
       - Select the vertices using same method as for Master
    3. Relation: select **Hs = -Hm**
    - The model represents one pole out of height. Since we represent an odd number of poles, the condition at the slave surface is Slave = -Master





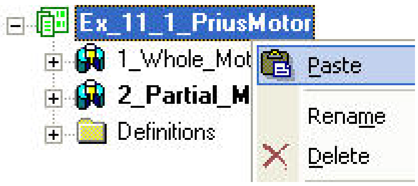
  - Note: The *XY* plane is a symmetry plane, but it is not necessary to enter a specific boundary condition as the natural boundary condition for a bounding plane is the odd condition (Flux tangential)

Static Analysis

- We will study the different static parameters of the motor.

Copy Design

- **To Copy Design**
  - Select the Maxwell Design **" 2_Partial_Motor"** from the Project Manager tree, right click and select **Copy**
  - Right click on Project Name and select **Paste**

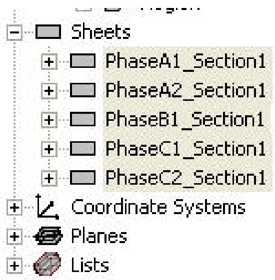


  - Rename the newly created design to **"3_Partial_Motor_MS"**
  - The first analysis that will be performed consists in computing the fields due to the permanent magnets and static currents applied .

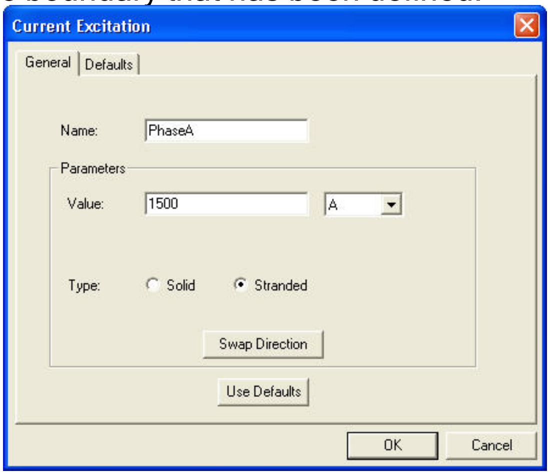
Apply Excitations

- The coils are partially represented in the model. We need to enter the current that flows in and out inside each coil. The excitation is realized through a balanced three phase system. For instance, in our example, we apply:
  - 1500 A to PhaseA
  - -750 A to PhaseB
  - -750 A to PhaseC.
- In the Magnetosatic solver, the sources are given in terms of currents. We do not need to model each turn at this stage; therefore we only enter the total current in each phase. The number of turns and the electrical topology are only taken into account for the inductances calculation.
- Current terminals need to be created to apply the loads. The terminals consist in 3D sheets that are normal to current directions. The 2D sheets are defined thanks to plane sections.

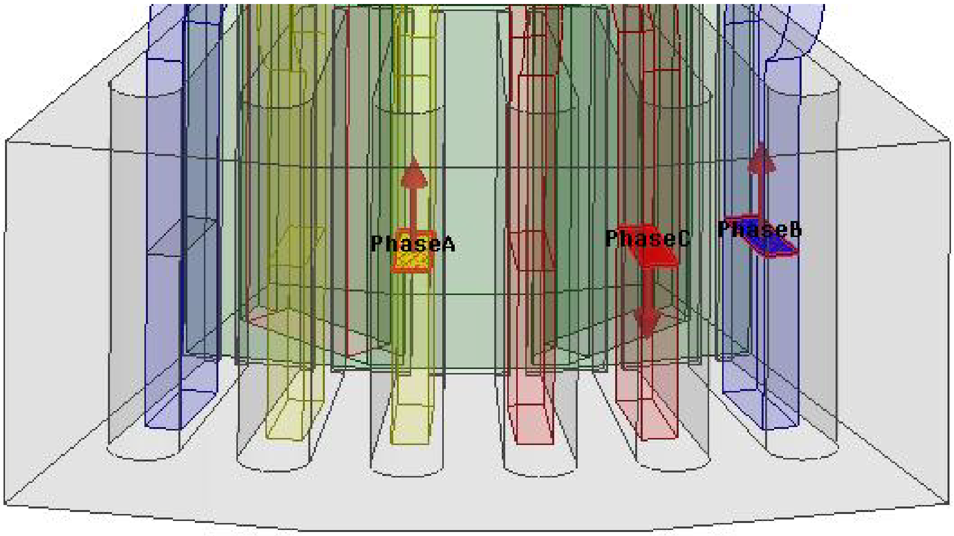
- **Create Coordinate System for Sectioning**
  - Select the menu item ***Modeler > Coordinate System > Create > Relative CS > Offset***
    1. Using Coordinate Entry Field, enter the origin
       - **X: 0.0, Y: 0.0, Z: 20.0**, Press the **Enter** key
- **Create Coil Terminals**
  - Press **Ctrl** and select all five coils from history tree
  - Select the menu item ***Modeler > Surface > Section***
  - In Section window,
    1. Section Plane: **XY**
    2. Press **OK**
  - The 2D sheets have been created
  - Select the menu item ***Modeler > Coordinate System > Set Work CS***
    1. Select **Global**



- **Assign Excitations for Phase A**
  - Select the sheet **PhaseA1_Section1** from history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseA**
    2. Value: **1500 A**
    3. Type: **Stranded**
    4. Direction: **Positive Z** (Use Swap Direction)
    5. Press **OK**
  - Maxwell will also apply 1500A to the coil PhaseA2 thanks to the Master/Slave boundary that has been defined.

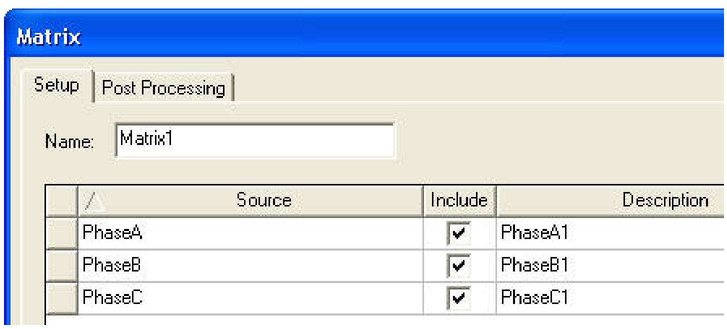


- **Assign Excitations for PhaseB**
  - The sheet PhaseB1_Section1 is made of two isolated sheets which needs to be separated
  - Select the sheet **PhaseB1_Section1** from history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
  - Select the sheet **PhaseB1_Section1** from history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseB**
    2. Value: **-750 A**
    3. Type: **Stranded**
    4. Direction: **Positive Z** (Use Swap Direction)
    5. Press **OK**
- **Assign Excitations for PhaseC**
  - Select the sheet **PhaseC1_Section1** from history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Current***
  - In Current Excitation window,
    1. Name: **PhaseC**
    2. Value: **-750 A**
    3. Type: **Stranded**
    4. Direction: **Negative Z**
    5. Press **OK**

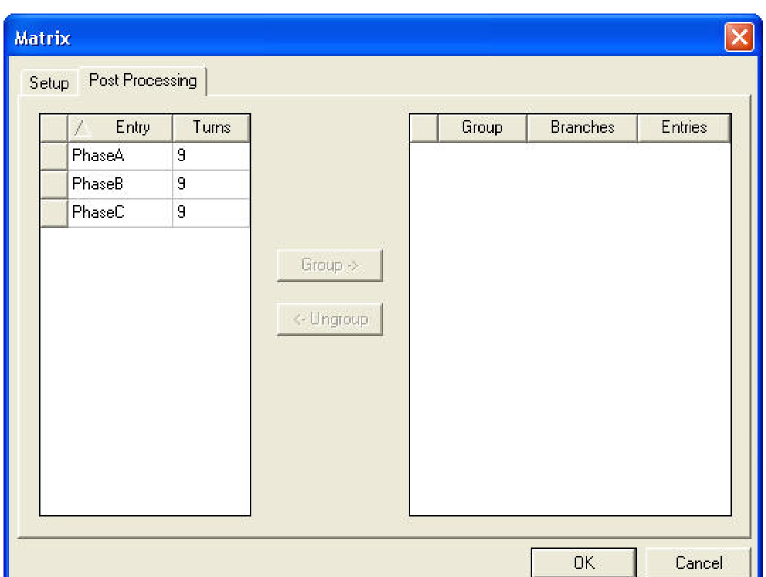


Set Parameters

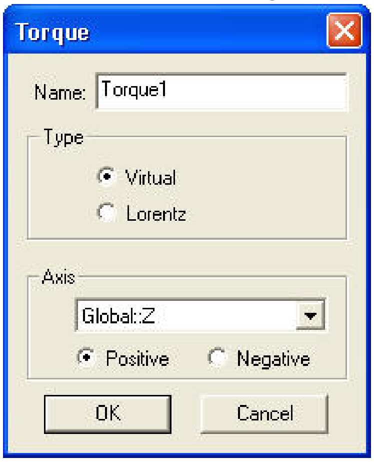
- We are interested by the inductances computation. The source set up is independent from the winding arrangement: we have only entered the corresponding amp-turns for each terminal. When looking at the inductances, we obviously need to enter the number of turns for the coils and also how the coils are electrically organized.
- **To Sect Inductance Calculations**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    1. Setup Tab
       - PhaseA, PhaseB and PhaseC:
         1. Include : ☑ **Checked**



    2. Post Processing Tab
       - Set Turns for all phases to **9**
       - Press **OK**

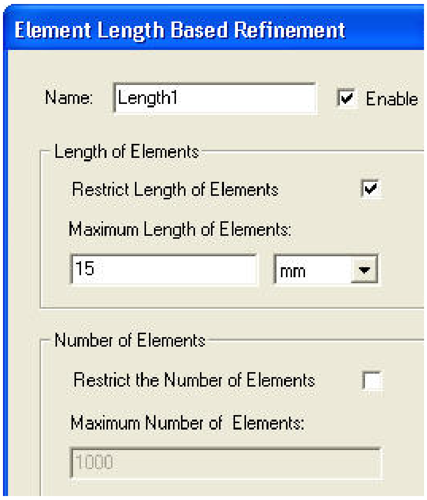


- **Assign Torque Computation**
  - Press **Ctrl** and select the objects **PM1, PM2** and **Rotor**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Torque***
  - In Torque window,
    1. Type: **Virtual**
    2. Press **OK**



Apply Mesh Operations

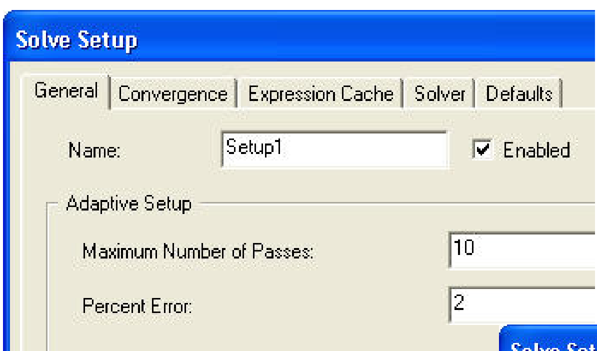
- The adaptive meshing is very effective, so it is not necessary to enter dedicated mesh operations. However, it is always a good idea to start with a decent initial mesh in order to reduce time computation since we know where the mesh needs to be refined for a motor. The non linear resolution will be faster with a small aspect ratios for the elements in the steel.
- **To Apply Mesh Operations**
  - Press **Ctrl** and select the objects **PM1, PM2, Stator** and **Rotor**
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Restrict length of Elements: ☑ **Checked**
    2. Maximum Length of Elements: **15 mm**
    3. Press **OK**

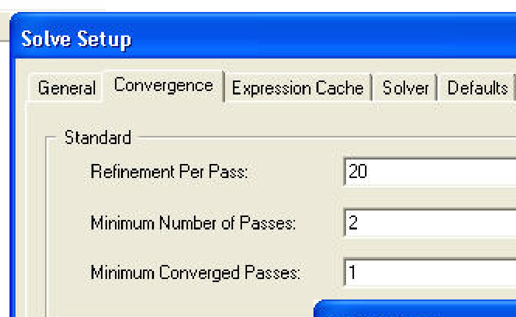


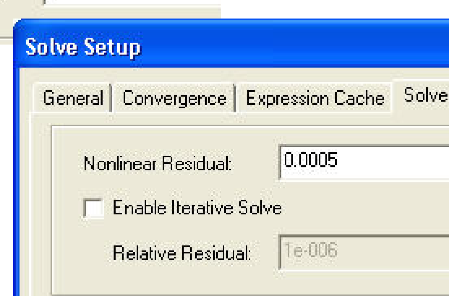
Analysis Setup

To Create Analysis Setup

- Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
- In Solve Setup Window,
  1. **General** Tab
     - Percentage Error: **2**
  2. **Convergence** tab
     - Refinement Per pass: **20 %**
  3. **Solver** Tab
     - Non-linear Residuals: **0.0005**
  4. Press **OK**







Analyze

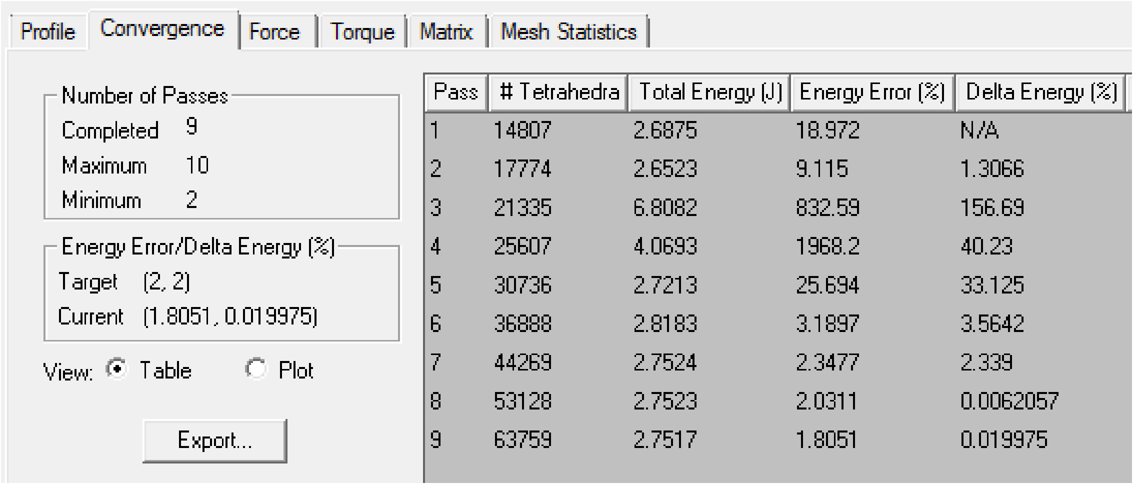
To Run the Solution

- Select the menu item ***Maxwell 3D > Analyze All***

Post Processing

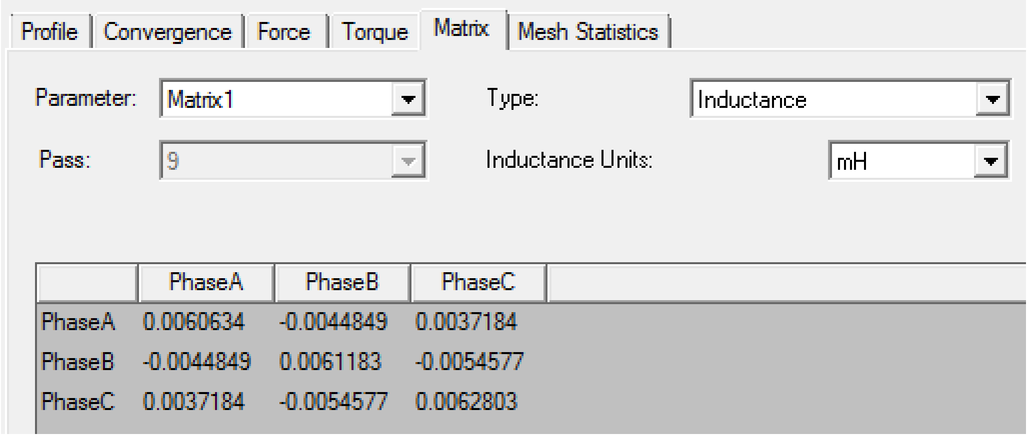
Solution Data

- Select the menu item ***Maxwell 3D > Results > Solution Data***
- To View Convergence
  1. Select **Convergence** tab



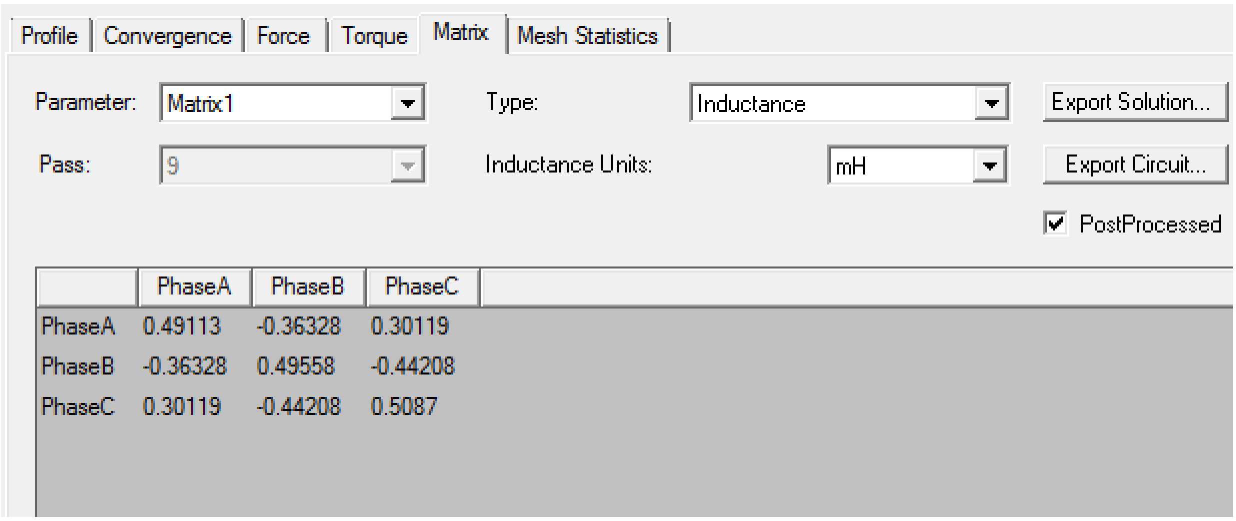
Pass	# Tetrahedra	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	14807	2.6875	18.972	N/A
2	17774	2.6523	9.115	1.3066
3	21335	6.8082	832.59	156.69
4	25607	4.0693	1968.2	40.23
5	30736	2.7213	25.694	33.125
6	36888	2.8183	3.1897	3.5642
7	44269	2.7524	2.3477	2.339
8	53128	2.7523	2.0311	0.0062057
9	63759	2.7517	1.8051	0.019975

- To View Inductance Values
  1. Select the **Matrix** tab

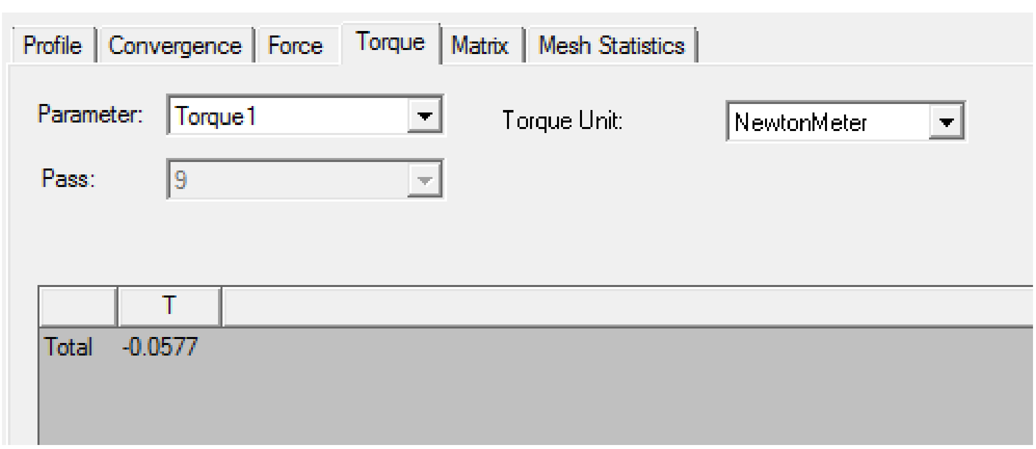


	PhaseA	PhaseB	PhaseC
PhaseA	0.0060634	-0.0044849	0.0037184
PhaseB	-0.0044849	0.0061183	-0.0054577
PhaseC	0.0037184	-0.0054577	0.0062803

- The Values shown are Per Turn Values
- To View Total inductance, check the button **PostProcessed**



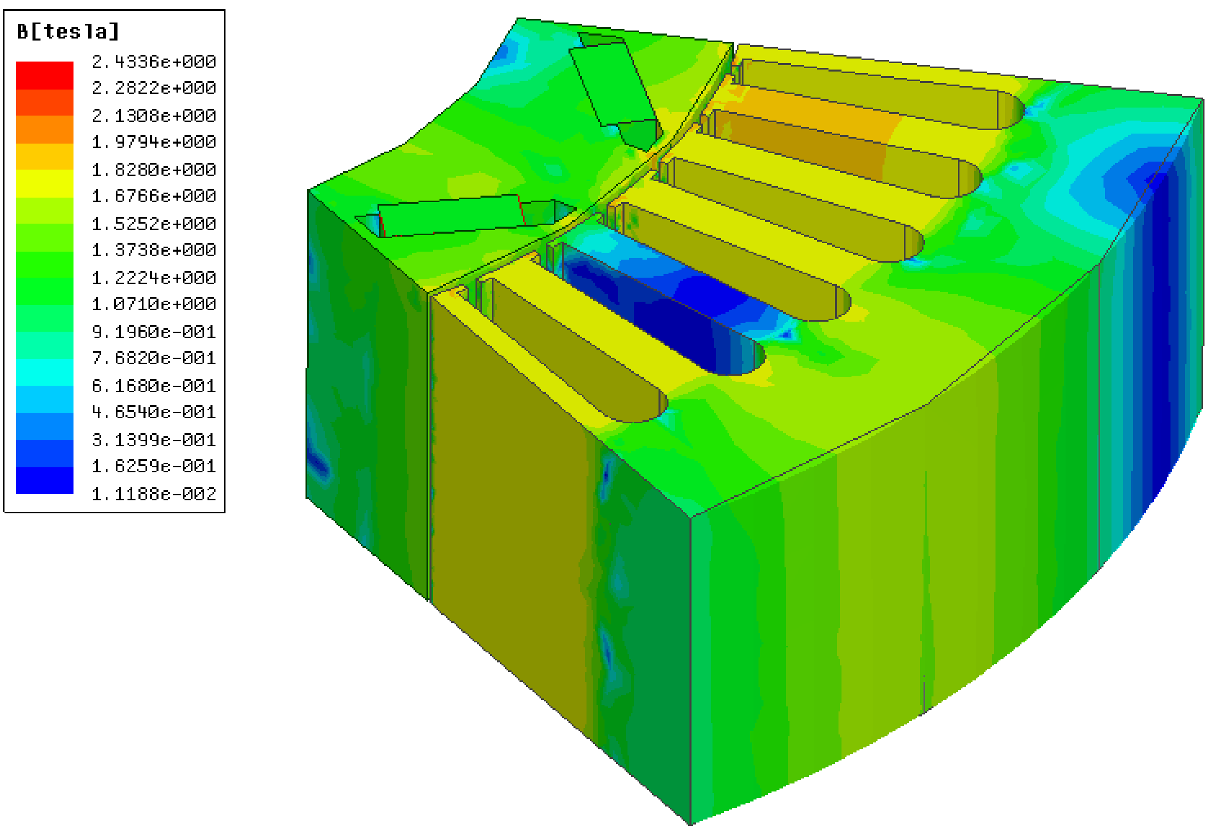
- To View Torque Values
  1. Select the **Torque** tab
  2. The torque for the full motor needs to be multiplied by 16. This gives around 8 N.m. In this case, we have not synchronized the position of the rotor poles with the winding currents, so we are far from the optimized excitation value to obtain a maximum torque. Different angles between the rotor and the stator would give different values.



Field Plots

Plot Magnetic Flux Density

- Press **Ctrl** and select **Region** and **all Coils** from history tree
- Select the menu item ***View > Visibility > Hide Selection > Active View***
- Select the menu item ***Edit > Select All Visible***
- Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
- In Create Field Plot window,
  1. Plot on surface only: ☑ **Checked**
  2. Press **Done**
- The steel is highly saturated in the ducks of the rotor as expected. This saturations appears just because of the magnets strengths.



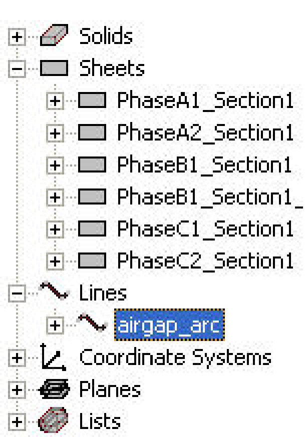
Vector Plot in Airgap

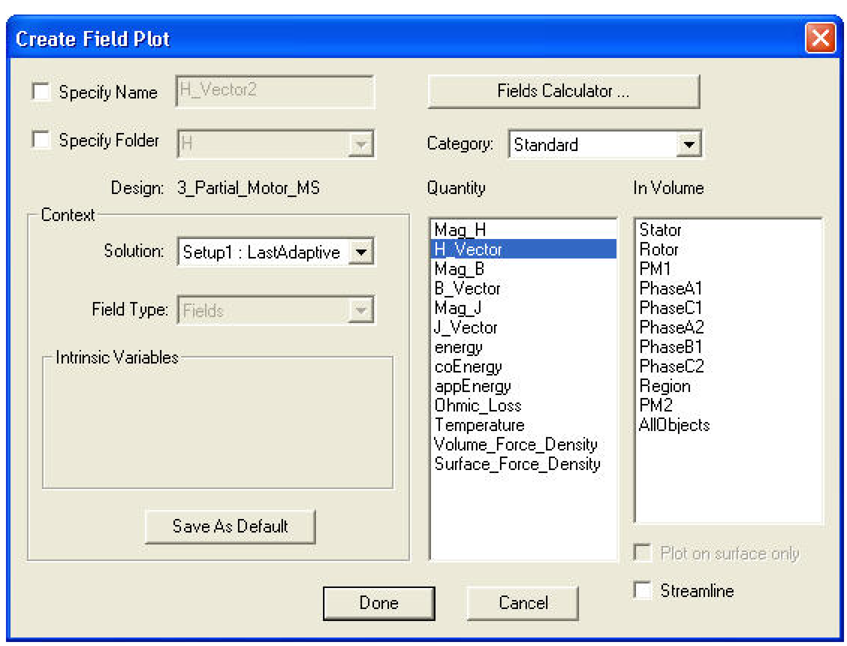
Create an Arc for Vector Plot

- Select the menu item *Draw > Arc > Center Point*
- A massage box appears asking if Geometry needs to be create d as a Non-model object. Press **Yes** to it
  1. Using Coordinate entry field, enter the center of arc
     - **X: 0, Y :0, Z: 20,** Press the **Enter** Key
  2. Using Coordinate entry field, enter the first point of arc
     - **X: 80.575, Y :0, Z: 20,** Press the **Enter** Key
  3. Using Coordinate entry field, enter the first point of arc
     - **X: 56.70996, Y : 56.70996, Z: 20,** Press the **Enter** Key
  4. To finish the arc, move the mouse on the drawing area, right mouse click, and select the menu entry done
- Rename the created Polyline to **airgap_arc**

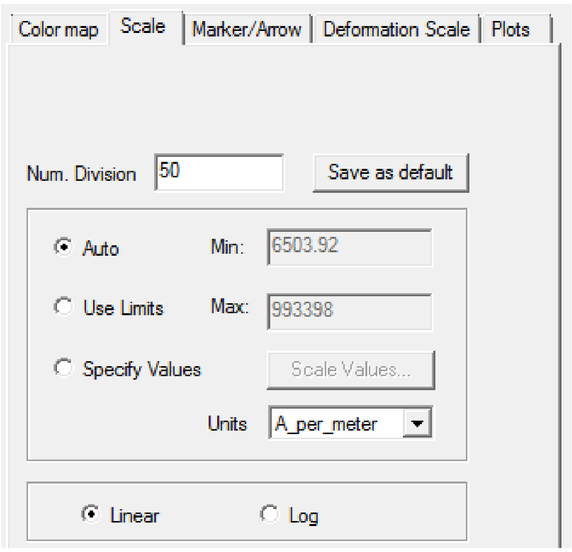
Plot H_Vector on Arc

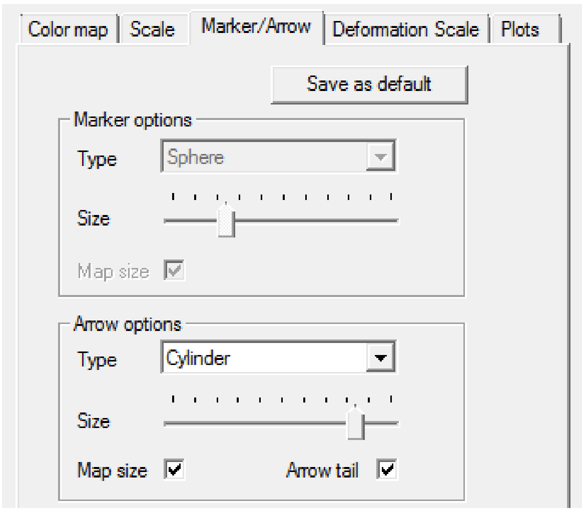
- Select the line **airgap_arc** from the history tree
- Select the menu item *Maxwell 3D > Fields > Fields > H > H_Vector*
- In Create Field Plot window,
  1. Press **Done**

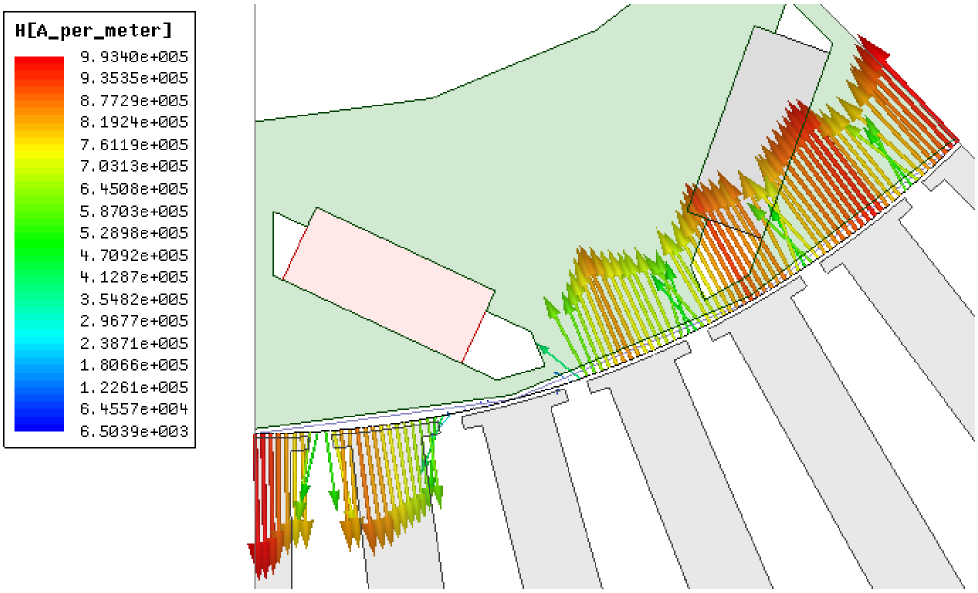




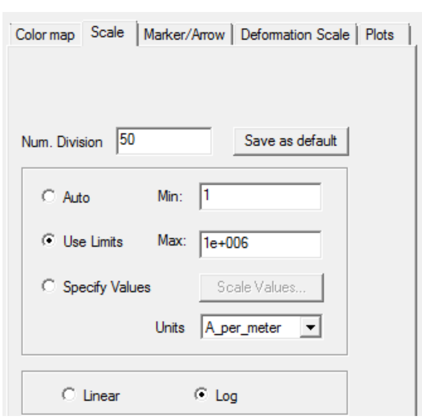
- **Modify Plot Attributes**
  - Double click on the plot Legend to modify its attributes
  - In the window,
    1. **Scale** Tab
       - Num. Division: **50**
    2. **Marker/Arrow** Tab
       - Size: **Set to appripriate value**
    3. Press **Apply** and **Close**

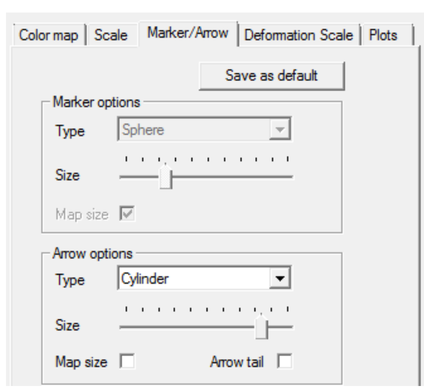


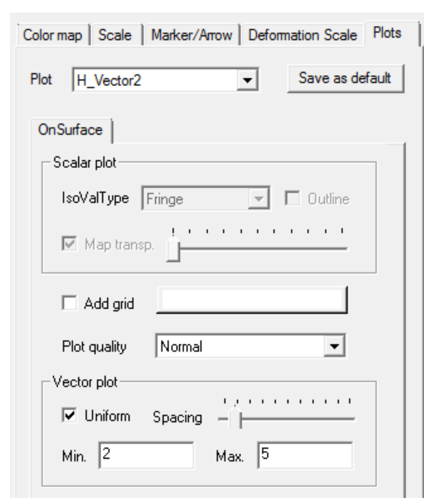




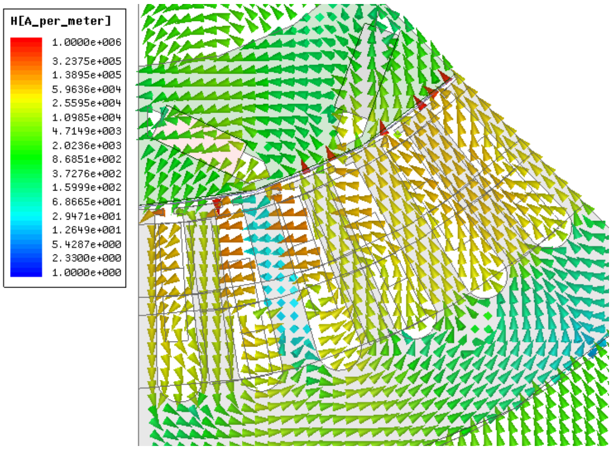
- **Plot H_Vector on XY Plane**
  - Select the Plane **Global:XY** from history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > H > H_Vector***
  - Press **Done**
- **Modify Plot Attributes**
  - In the window,
    1. **Scale** Tab
       - Select **User Limits**
         - Min: **1**
         - Max: **1e6**
       - Select **Log**
    2. **Marker/Arrow** Tab
       - Map Size: ☐ **Unchecked**
       - Arrow Tail: ☐ **Unchecked**
    3. **Plots** Tab
       - Plot: **H_Vector2**
       - Spacing: **One Space from Left**
         - Min: **2**
         - Max: **5**
    4. Press **Apply** and **Close**





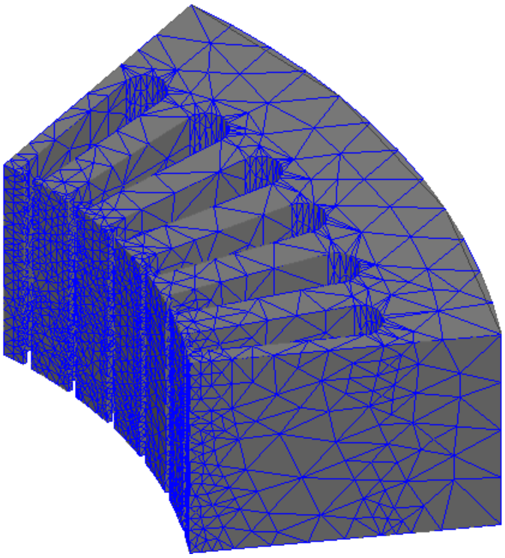


- Plot should look as below



Plot Mesh

- Maxwell uses an adaptive meshing process. The mesh is continuously improves pass after pass until converge is reached. It is always a good idea to plot the mesh in order to see where Maxwell has put elements.
- **To Plot Mesh on Stator**
  - Select the object **Stator** from history tree
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - Press **Done**
  - It is interesting to see that the mesh is extremely refined around the air gap, when the field changes rapidly whereas the field on the outside of the **Stator** is that refined since the field change is low.

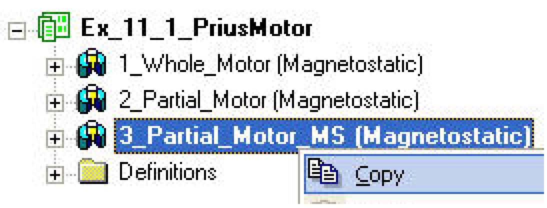


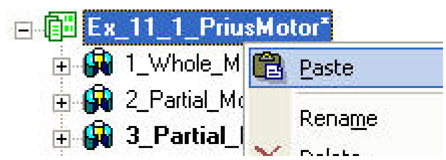
Dynamic Analysis

- We will study the transient characteristic of the motor.

Copy Design

- **To Copy Design**
  - Select the Maxwell Design **“ 3_Partial_Motor_MS”** from the Project Manager tree, right click and select **Copy**
  - Right click on Project Name and select **Paste**

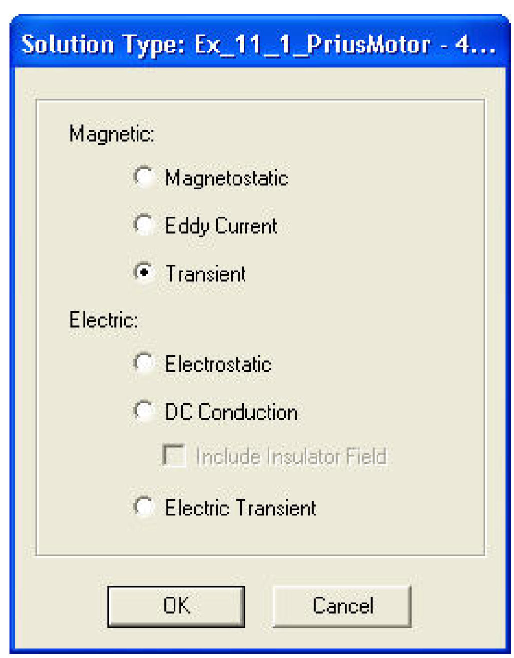




  - Change the name of the design to **“4_Partial_Motor_TR”**

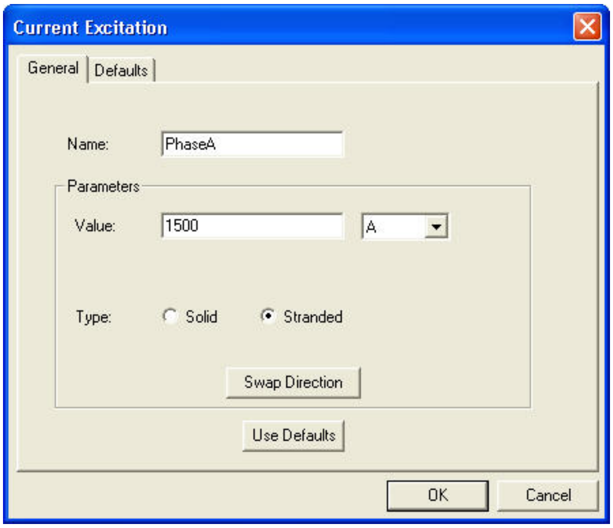
Change Analysis Type

- **To Set Transient Analysis**
  - Select the menu item ***Maxwell 3D > Solution Type***
    1. Select **Magnetic > Transient**
    2. Press **OK**

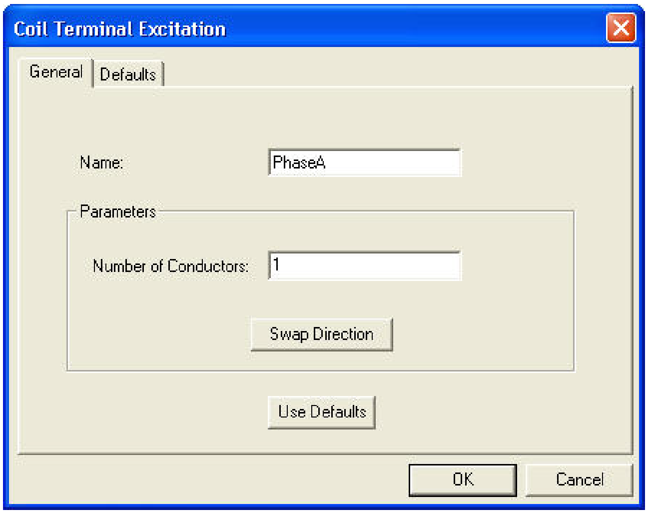


About Transient Solver

- The transient solver acts differently from the Magnetostatic solver mainly because:
  - There is not adaptive meshing. Since the relative position of objects changes at every time step, Maxwell does not re-mesh adaptively for obvious time saving. In transient analysis, we will build a good mesh valid for all the rotor positions.
  - The sources definition is different. In Magnetostatic, we were only interested in the total current flowing into conductor. In Transient, we use stranded conductors (the exact number of conductors is required for each winding) as the current or voltage can be an arbitrary time function. We need to create dedicated current terminals and windings.
- When changing of solver, Maxwell removes incompatible setups between Magnetostatic and Transient. For instance the Analysis setup is removed. The current excitations are incompatible, but Maxwell does keep the source definitions: the current definition is transformed as a coil Terminal excitation



Current Excitation (Magnetostatic)



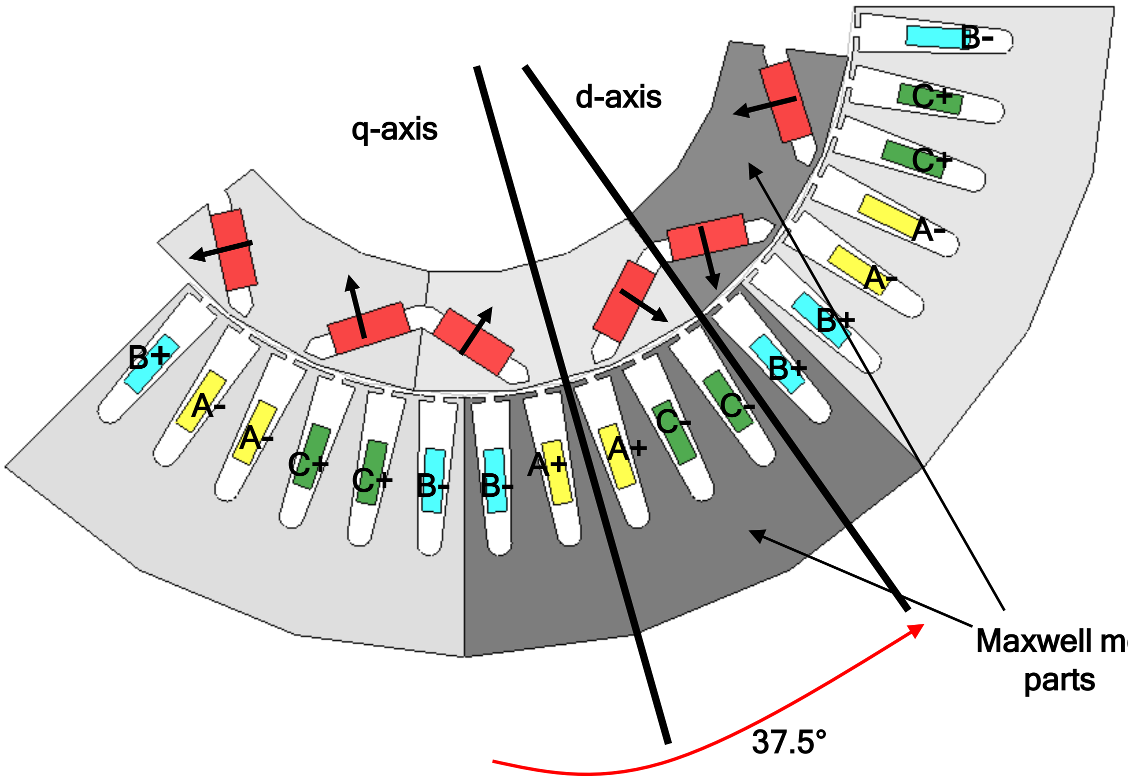
Coil Terminal (Transient)

Modify Coil Terminals

- **To Modify Coil Terminals**
  - Expand the Project Manager tree to view Excitations
  - Double click on the excitation **PhaseA** to modify it
  - In Coil Terminal Excitation window,
    1. Number of Conductor: **9**
    2. Press **OK**
  - Similarly change the number of coductors for **PhaseB** and **PhaseC** to **9**

Motor excitation

- The IPM motor is such that the rotor is in synchronism with the phase excitation. The excitation is such that the flux due to the permanent magnet is maximized in synchronization with the rotor movement.
- The excitation is a 3 phase balanced current. The phase sequence is A+C-B+
- At t=0, the A-phase has to be in the opposite axis to the d-axis. Therefore we have to move the initial position of the rotor by 30 deg such that the pole be aligned at the middle of A+A-



Create Parameters for excitations

- **To Create Parameters**
  - Select the menu item ***Maxwell 3D > Design Properties***
  - In Properties window, select **Add**
  - In Add Property window,
    - Name: **Poles**
    - Value: **8**
    - Press **OK**
  - Similarly add more properties as below
    1. Name: **PolePairs**
       - Value: **Poles/2**
    2. Name: **Speed_rpm**
       - Value: **3000**
    3. Name: **Omega**
       - Value: **360*Speed_rpm*PolePairs/60**
    4. Name: **Omega_rad**
       - Value: **Omega * pi / 180**
    5. Name: **Thet_deg**
       - Value: **20**
    6. Name: **Thet**
       - Value: **Thet_deg * pi /180**
    - **Note:** Do not specify unit to any of the above parameters
    7. Name: **Imax**
       - Value: **250**
       - Unit: **A**

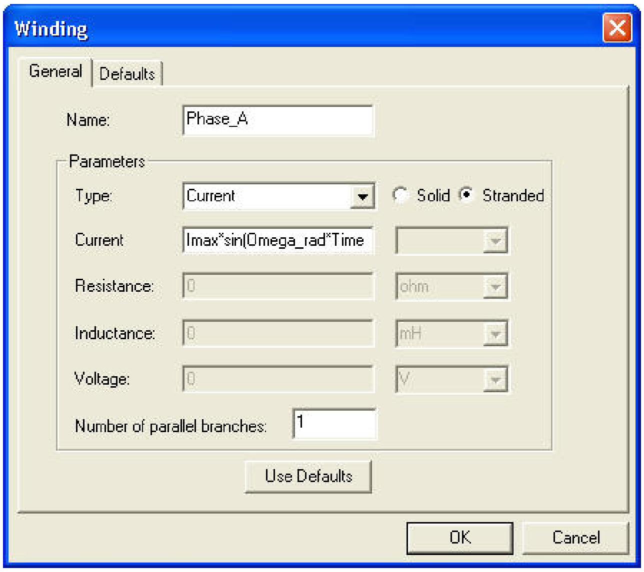
Local Variables

Value Optimization Tuning Sensitivity

Name	Value	Unit	Evaluated Value	Type
Poles	8		8	Design
PolePairs	Poles/2		4	Design
Speed_rpm	3000		3000	Design
Omega	360*Spee...		72000	Design
Omega_rad	Omega*pi/...		1256.63706143...	Design
Thet_deg	20		20	Design
Thet	Thet_deg*...		0.34906585039...	Design
Imax	250	A	250A	Design

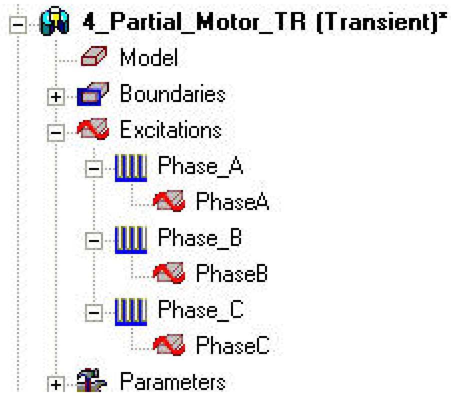
Create Windings

- The terminals are meant to define the excitation paths in and out of the model. The actual excitation is defined through the definition of windings. A winding needs to be defined for each parallel electrical excitation of the motor.
- The motor is excited with a balanced three phase connection. A sinusoidal excitation is applied. At each time step, the phases have a 120 degree shift. The load angle is also added.
- **To Create Winding for Phase A**
  - Select the menu item ***Maxwell 3D > Excitation > Add Winding***
  - In Winding window,
    1. Name: **Phase_A**
    2. Type: **Current**
    3. Check **Stranded**
    4. Current: **Imax*sin(Omega_rad*Time+Thet)**
       - **Time** is internally reserved variable for the current time.
    5. Press **OK**



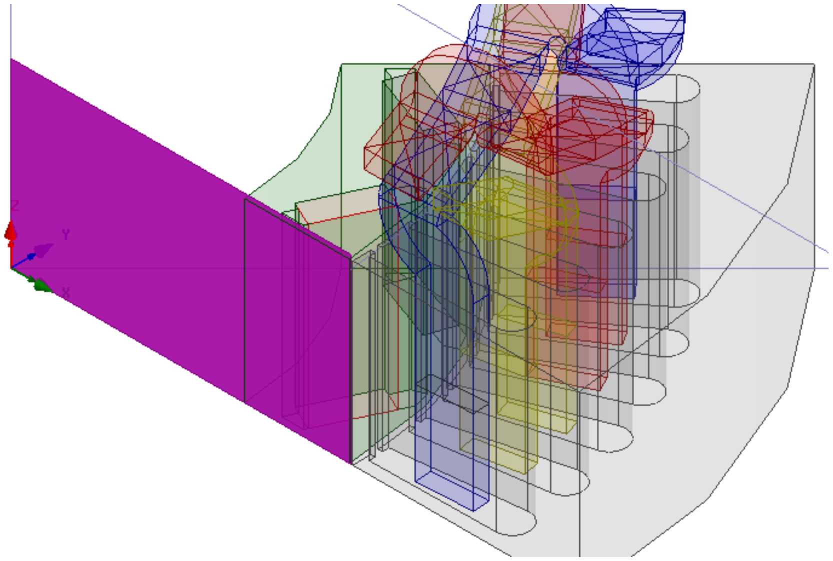
- **To Add Terminal to Winding**
  - Expand the Project Manager tree to view Excitations
  - Right click on winding **Phase_A** and select **Add Terminals**
  - Select the terminal **PhaseA** and select **Add**

- **Add Winding for Phase B and Phase C**
  - Similarly add Windings for Phase B and Phase C
  - Phase B
    1. Name: **Phase_B**
    2. Type: **Current**
    3. Check **Stranded**
    4. Current: **Imax*sin(Omega_rad*Time-2*pi/3+Thet)**
       - It is shift by -120 degrees from PhaseA.
    5. Press **OK**
    6. Add the terminal **PhaseB** to this winding
  - Phase C
    1. Name: **Phase_C**
    2. Type: **Current**
    3. Check **Stranded**
    4. Current: **Imax*sin(Omega_rad*Time+2*pi/3+Thet)**
       - It is shift by +120 degrees from PhaseA.
    5. Press **OK**
    6. Add the terminal **PhaseC** to this winding

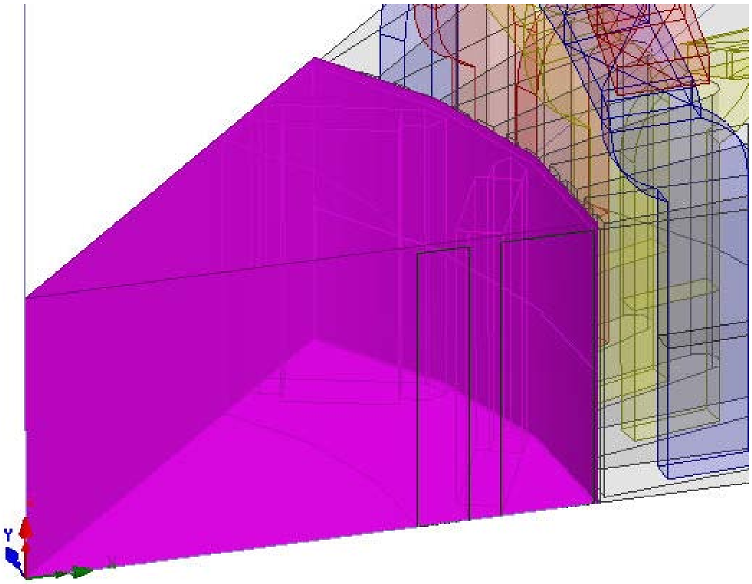


Create Band Object

- The moving parts (rotor and permanent magnets) need to be enclosed in an air object, the band. This will separated the moving part from the fixed part of the project. Some rules apply for the definition of the band object for motor applications:
  - The band object must be somewhat larger than the rotating parts in all directions (except at the boundaries)
  - An 'inner band' object must also be present: it has to enclose all the moving object inside the band object.
- To create the Band object, we use an rectangle on the plane XZ that will be swept around the Z axis to create a "Camembert" style volume.
- **To Create a Rectangle**
  - Select the menu item ***Modeler > Coordinate System > Set Work CS***
    - Select **Global** and press **Select**
  - Select the menu item ***Modeler > Grid Plane > XZ***
  - Select the menu item ***Draw > Rectangle***
    - The rotor radius is 80.2mm. The inner diameter of the stator 80.95mm. We pick the middle for Band object
    1. Using the coordinate entry field, enter the box position
       - **X: 80.575, Y: 0.0, Z: 0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the relative size of the box
       - **dX: -80.575, dY: 0.0, dZ: 43.0**, Press the **Enter** key
  - Change the name of the resulting sheet to **Band**

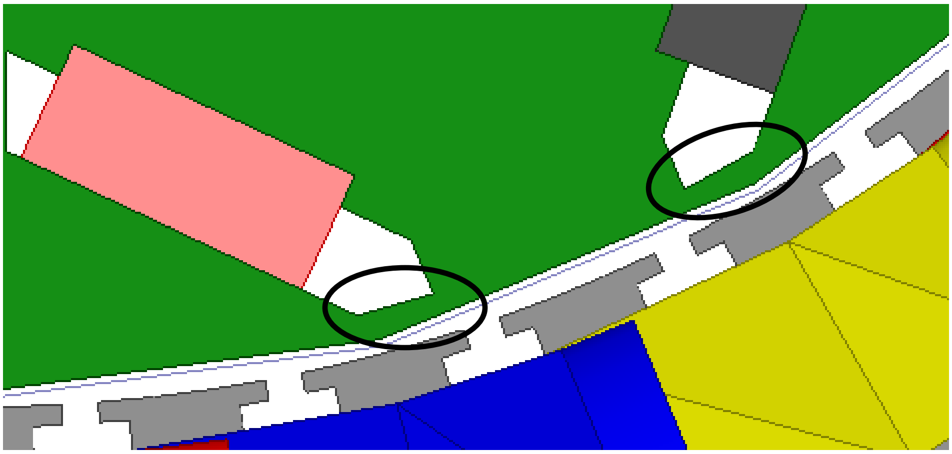


- **To Sweep Rectangle**
  - Select the sheet **Band** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Axis: **Z**
    2. Angle of sweep: **45**
    3. Number of segments: **0**
    4. Press **OK**



Mesh Operations

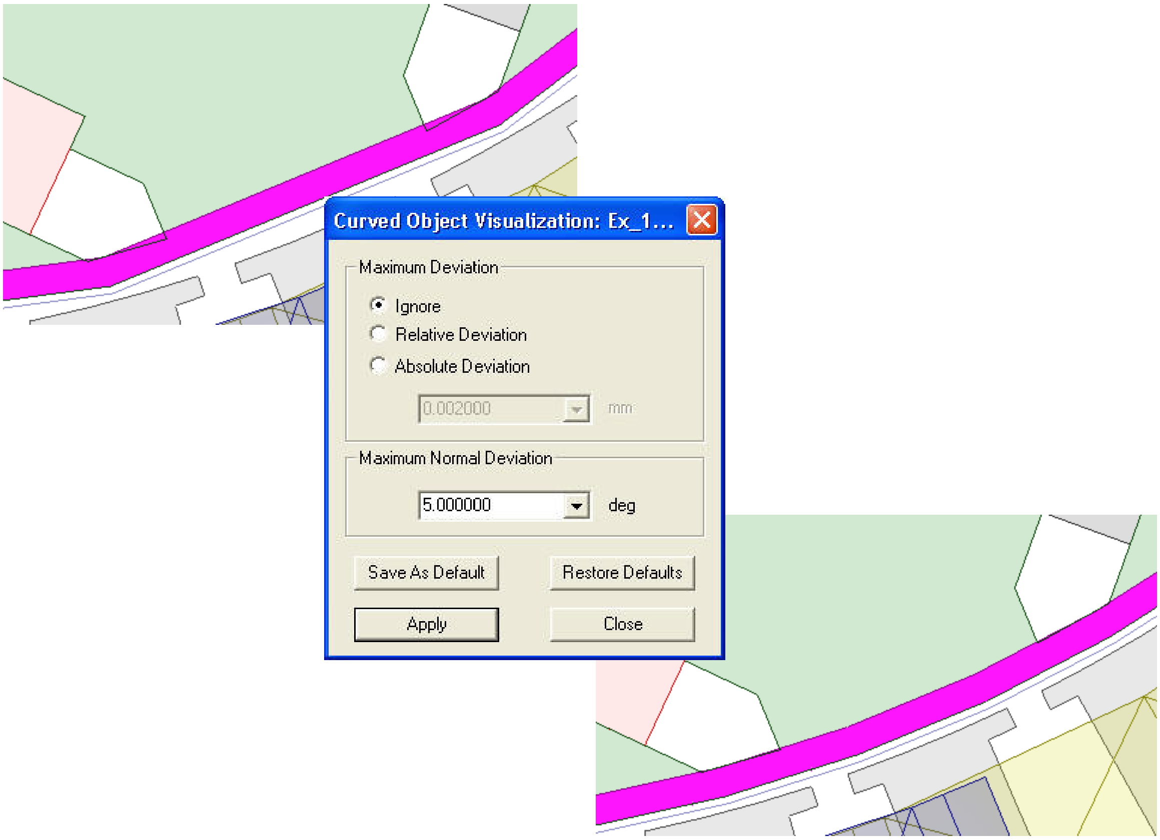
- The transient solver does not use adaptive meshing because this would require to refine the mesh at every time steps, leading to very high computation time. Using Mesh operations, we will define a decent mesh for the full transient simulation.
- Once the Motion Setup is a Mesh operation for Refinement in airgap region is automatically created. For the rest, we will define Mesh Operations
- **Mesh Operation for Rotor**
  - The Rotor is designed to be highly saturated around the permanent magnets, close to the air gap. It is required to have a good mesh density around this area.
  - To achieve this requirement, we create a couple of objects inside the rotor; then mesh operations will be applied to these objects in order to have a nice mesh around the ducts.



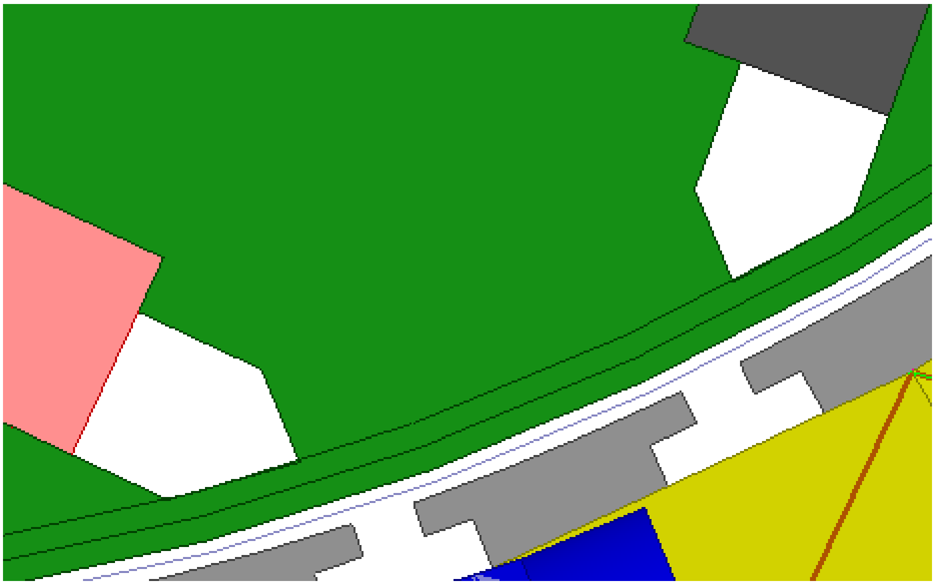
Highly saturated zones

- **Create Rectangle**
  - Make sure the Grid Plane is set to XZ plane
  - Select the menu item ***Draw > Rectangle***
    1. Using the coordinate entry field, enter the box position
       - **X: 78.72, Y: 0.0, Z: 0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the relative size of the box
       - **dX: 1.48, dY: 0.0, dZ: 41.91**, Press the **Enter** key
  - Name the resulting object as **Rotor2**

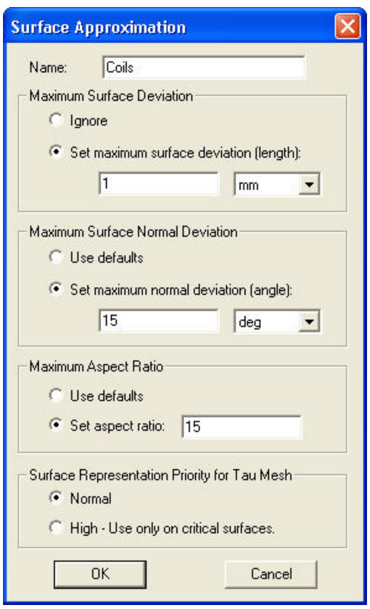
- **To Sweep Rectangle**
  - Select the sheet **Rotor2** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Axis: **Z**
    2. Angle of sweep: **45**
    3. Number of segments: **0**
    4. Press **OK**
  - Change the material property of **Rotor2** to **M19_29G**. Also, assign the same color and transparency as the object Rotor.
  - **Note**: since **Rotor2** is entirely inside **Rotor**, we do not need to apply Boolean operations.
  - **Note**: because of the finite number of pixels on the computer's screen, true surfaces are represented as facetted surfaces. Also, for the same reason, the object **Rotor2** *seems* to intersect with the ducts but this is not the case. You can modify the default visualization setting using: ***View > Render > Curved Object Visualization***

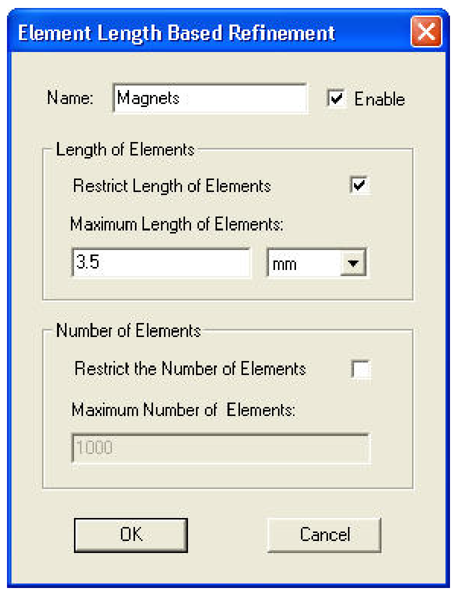


- **Create Rectangle**
  - Make sure the Grid Plane is set to XZ plane
  - Select the menu item ***Draw > Rectangle***
    1. Using the coordinate entry field, enter the box position
       - **X: 78.72, Y: 0.0, Z: 0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the relative size of the box
       - **dX: 1.48/2, dY: 0.0, dZ: 41.91**, Press the **Enter** key
  - Name the resulting object as **Rotor3**
- **To Sweep Rectangle**
  - Select the sheet **Rotor3** from the history tree
  - Select the menu item ***Draw > Sweep > Around Axis***
  - In Sweep Around Axis window,
    1. Axis: **Z**
    2. Angle of sweep: **45**
    3. Number of segments: **0**
    4. Press **OK**
  - Change the material property of **Rotor3** to **M19_29G**. Also, assign the same color and transparency as the object Rotor.
  - We now have created two layers between the ducts and the air gap.



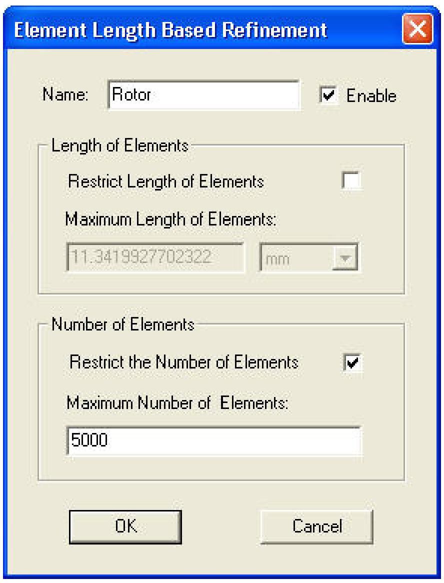
- **Assign Mesh Operation for Coils**
  - Delete existing mesh operations from the tree
  - Press **Ctrl** and select all **Coils** from history tree
  - Select the mesh item ***Maxwell 3D > Mesh Operations > Assign > Surface Approximation***
  - In Surface Approximation window,
    1. Name: **Coils**
    2. Set Maximum Surface Deviation (Length): **1mm**
    3. Set Maximum Normal Deviation (Angle) : **30 deg**
    4. Set Aspect Ratio: **15**
    5. Press **OK**

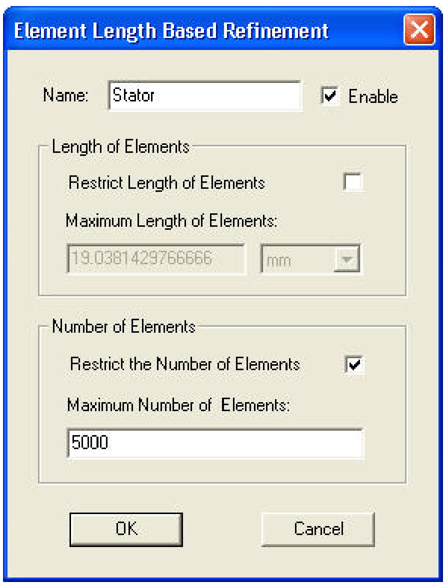




- **Mesh Operation for Magnets**
  - Press **Ctrl** and select the objects **PM1** and **PM2** from history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length based Refinement window,
    1. Name: **Magnets**
    2. Restrict the Length of Elements: ☑ **Checked**
    3. Restrict the Number of Elements: ☐ **Unchecked**
    4. Maximum Length of Elements: **3.5 mm**
    5. Press **OK**

- **Mesh Operation for Rotor**
  - Select the object **Rotor** from history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length based Refinement window,
    1. Name: **Rotor**
    2. Restrict the Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**



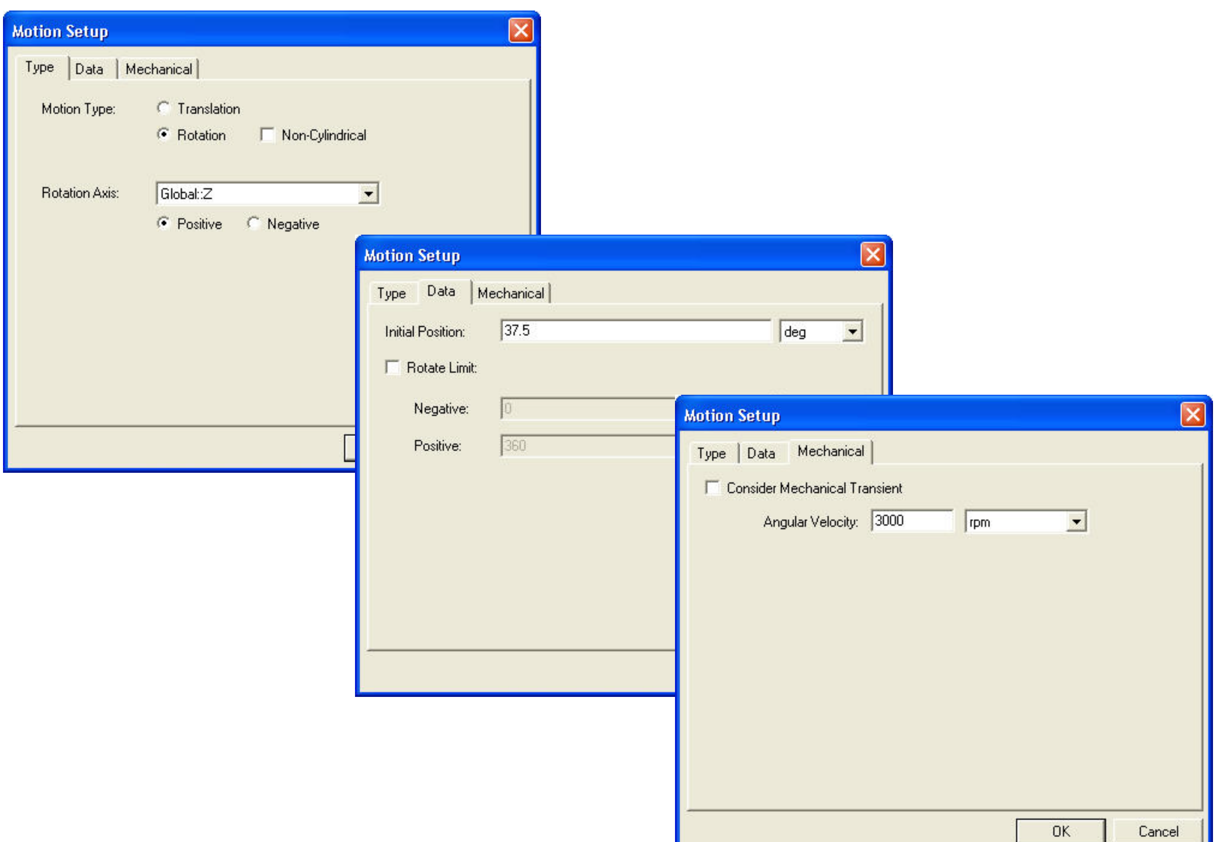


- **Mesh Operation for Stator**
  - Select the object **Stator** from history tree
  - Select the menu item ***Maxwell 3D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length based Refinement window,
    1. Name: **Stator**
    2. Restrict the Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **5000**
    5. Press **OK**

Assign Motion

To Assign Motion

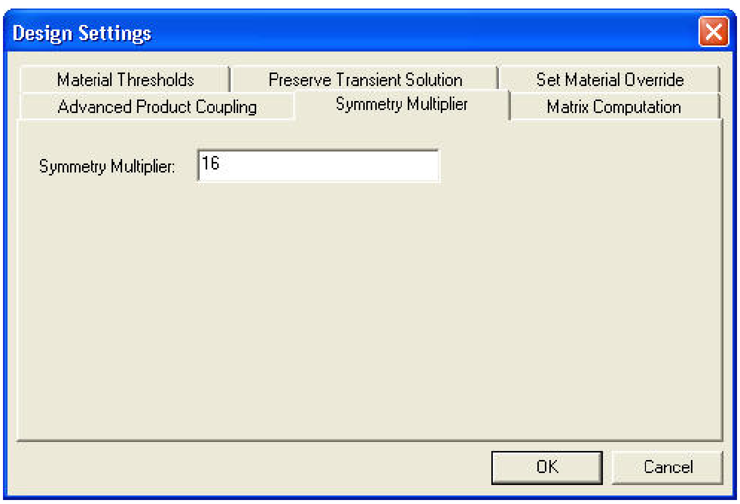
- Select the object **Band** from the history tree
- Select the menu item ***Maxwell 3D > Model > Motion Setup > Assign Band***
- In Motion Setup window,
  1. **Type** Tab
     - Motion Type: **Rotation**
     - Rotation Axis: **Global:Z**
     - Select **Positive**
  2. **Data** Tab
     - Initial Position: **37.5 deg**
     - The initial position of this synchronous motor is such that the A phase is opposite to the d-axis.
  3. **Mechanical** Tab
     - Angular Velocity: **3000 rpm**
  4. Press **OK**

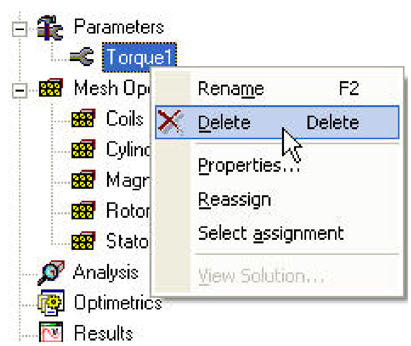


Set Symmetry Multiplier

To Set Symmetry Multiplier

- Select the menu item ***Maxwell 3D > Design Settings***
- In Design Settings window,
  1. **Symmetry Multiplier** tab
     - Symmetry Multiplier: **16**
  2. Since we model 1/16th of the motor (our model spans on 45° and the XY plane has a symmetry), The force, torque will be rescaled to take into account the full model.
- Since the Torque will be automatically computed, delete the Torque calculation that was set in Magnetostatic. The **Torque1** calculation is present in Parameters

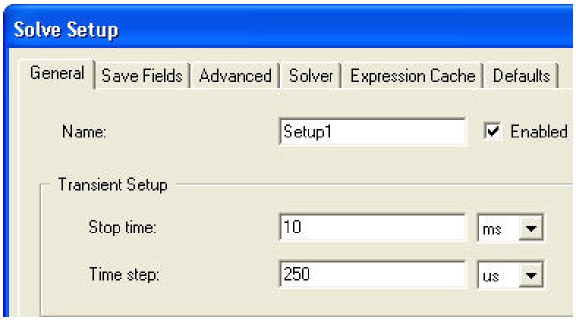


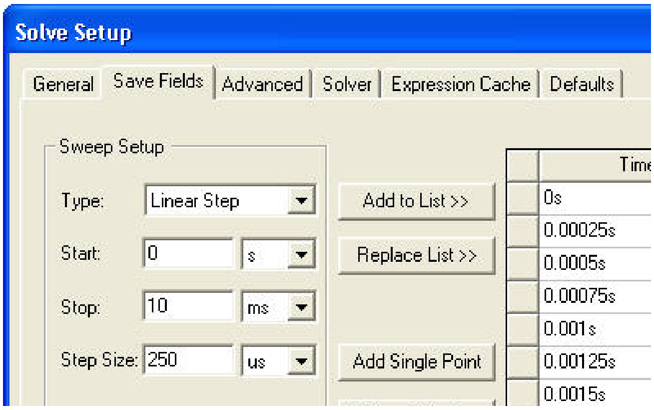


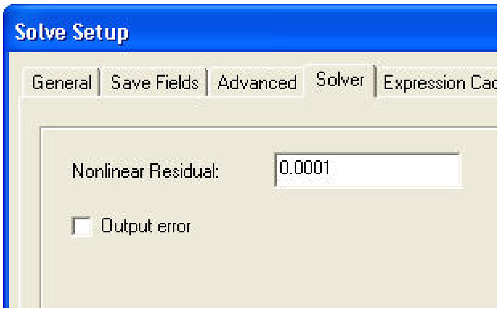
Solution Setup

To Create Solution Setup

- At 3000 rpm, a revolution takes 20ms (3000 rpm means 50 revolutions per second or 1/50 s for one revolution) . To achieve reasonable accuracy, we want to have a time step every 2 or 3 degrees. In this study, to have faster results, we use a time step of 250 us (thus every 4.5 degrees).
- Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
- In Solve Setup window,
  1. **General** tab
     - Stop Time: **10 ms**
     - Time Step: **250 us**
  2. **Save Fields** tab
     - Type : **Linear Step**
     - Start: **0 s**
     - Stop: **10 ms**
     - Step Size: **250 us**
     - Select **Add to List**
  3. **Solver** tab
     - Non-linear Residuals: **1e-4**
  4. Press **OK**

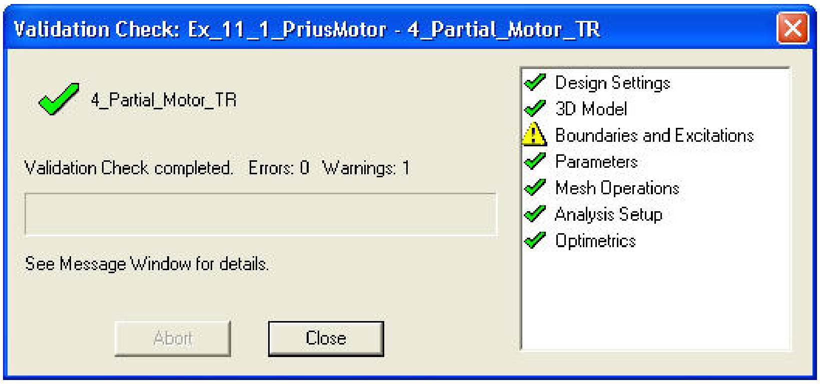






Validation Check

- **To Check the Validity of the Setup**
  - Select the menu item ***Maxwell 3D > Validation Check***



  - Maxwell checks the geometry, excitation definitions, mesh operations and so one. The model is validated but some Warnings are displayed in the message box:
    1. Boundary and Excitations lie on the same plane which is what we want (coils and the Master/Slave planes intersect)
    2. Eddy effect are not taken into account in our design which is what we decided

Solve

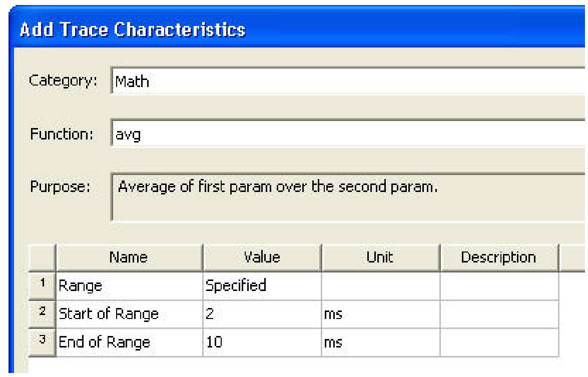
- **To Run Solution**
  - Select the menu item ***Maxwell 3D > Analyze All***
  - If you have several CPUs on your machine and the subsequent licensing, you can select in ***Tools > Options > Maxwell3D Options*** (Solver tab) the number of CPUs to use

Postprocessing

- The full simulation takes about one hour to solve. It is not necessary to wait for the end of the simulation to display results.
- Solve information appear in the profile of simulation
- **To View Solution Data**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
- Performance curves can be displayed during the simulation. They are updated at the end of each time steps.
- **To Create Torque Vs Time Plot**
  - Select the menu item ***Maxwell 3D > Results > Create Quick Reports***
  - In Quick report window,
    1. Select **Torque**
    2. Press **OK**

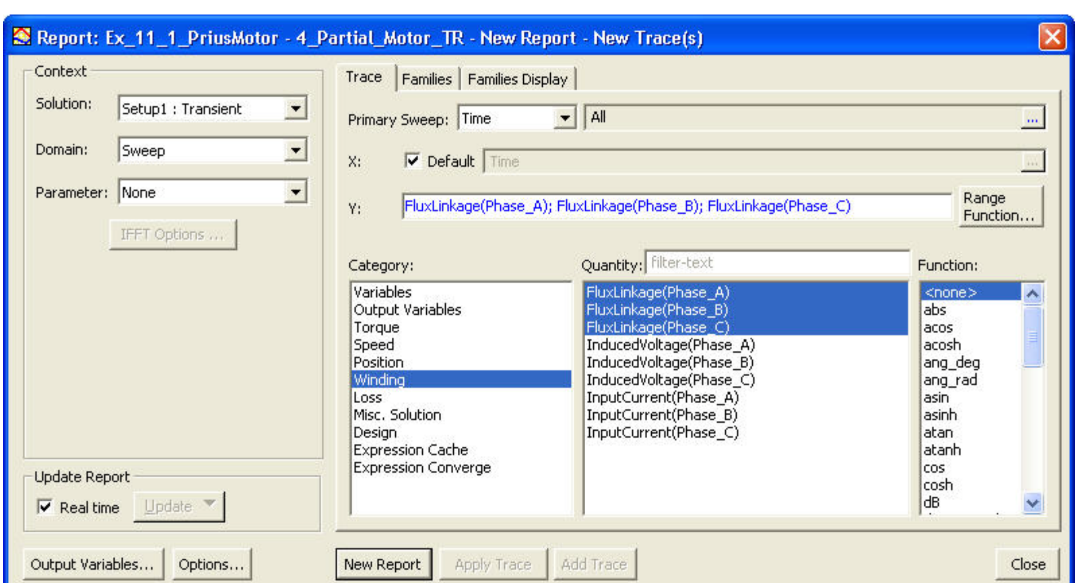
- We can see that there are a lot of ripples. The ratio between the torque and the torque ripples is almost 5 percent. This is due to the unique structure of the IPM motor (Internal Permanent Magnets). To limit the ripple, some manufacturers modify slightly the rotor shape around the magnets or add a second layer of internal magnets

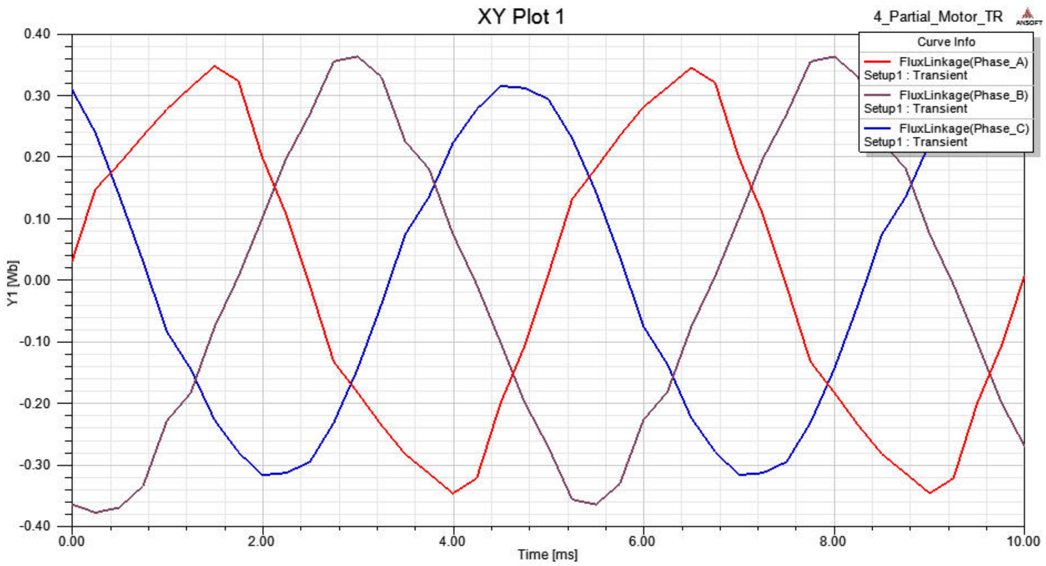
- **To Calculate Average Torque**
  - Right click on the Torque Plot and select ***Trace Characteristics > Add***
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **avg**
    - Range: **Specified**
    - Start of Range: **2 ms**
    - End of Range: **10 ms**
    - Press **Add**



  - The reported value of Torque is around **243 Nm**

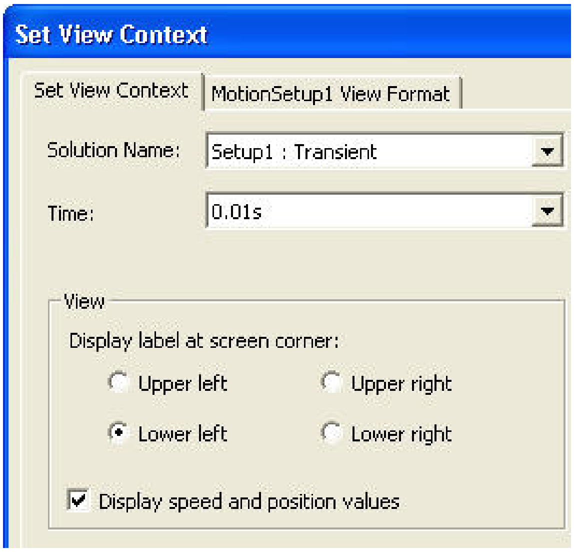
- **Plot Flux Linkages Vs Time**
  - Select the menu item ***Maxwell 3D > Results > Create Transient Report > Rectangular Plot***
  - In report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **FluxLinkage(PhaseA), FluxLinkage(PhaseB)** and **FluxLinkage(PhaseC)**
    - Select **New Report**



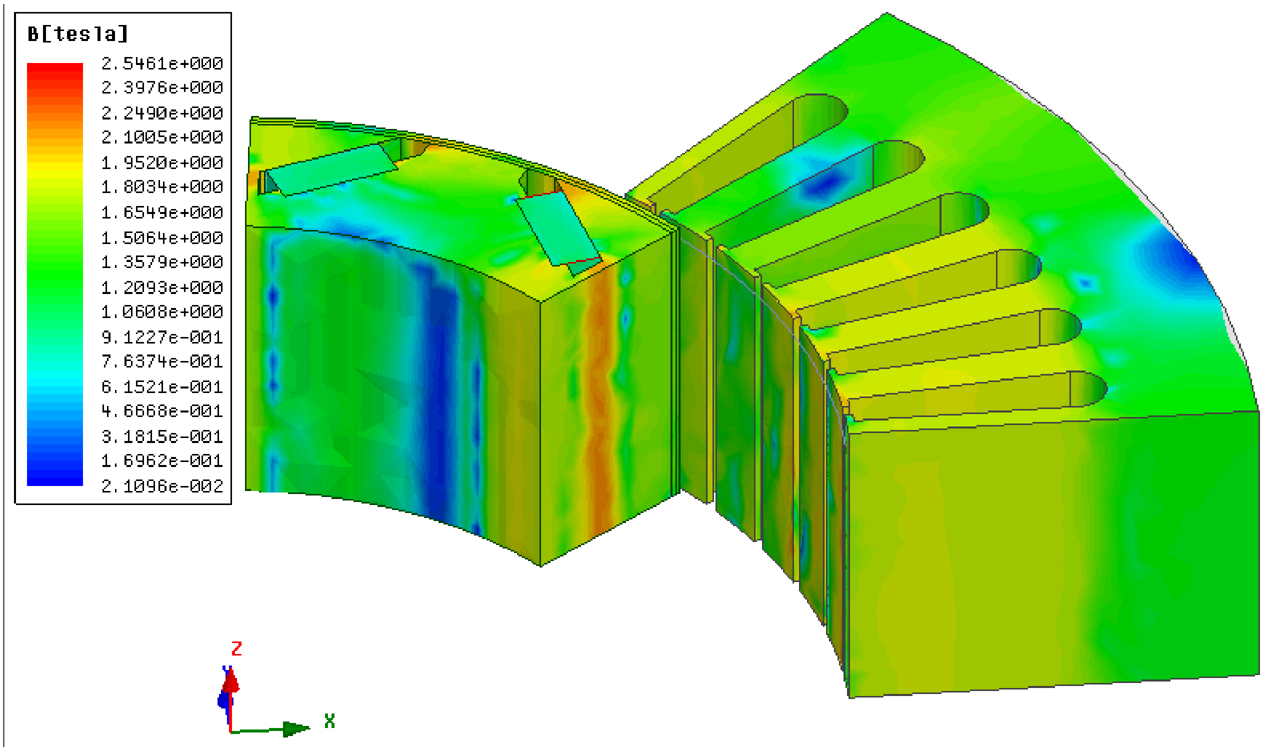


- The curves are not really smooth. The reason is that the mesh is certainly too coarse ; reducing the time step will also improve the smoothness of the curve

- **Plot Flux Density**
  - Select the menu item ***View > Set Solution Context***
  - In Set View Context window,
    1. Set Time to **0.01 s**
    2. Select **OK**



- Press **Ctrl** and select the objects **Rotor, Rotor2, Rotor3, PM1, PM2** and **Stator**
- Select the menu item ***Maxwell 3D > Fields > Fields > B > Mag_B***
- In Create Field Plot window,
  - Plot on surface only: ☑ **Checked**
  - Press **Done**
- The plot lets us see local saturation of the steel at this time step.

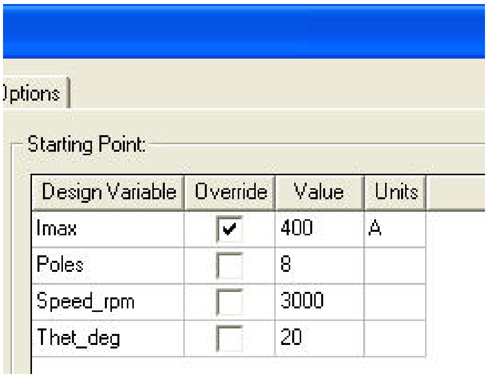


Parametric Study

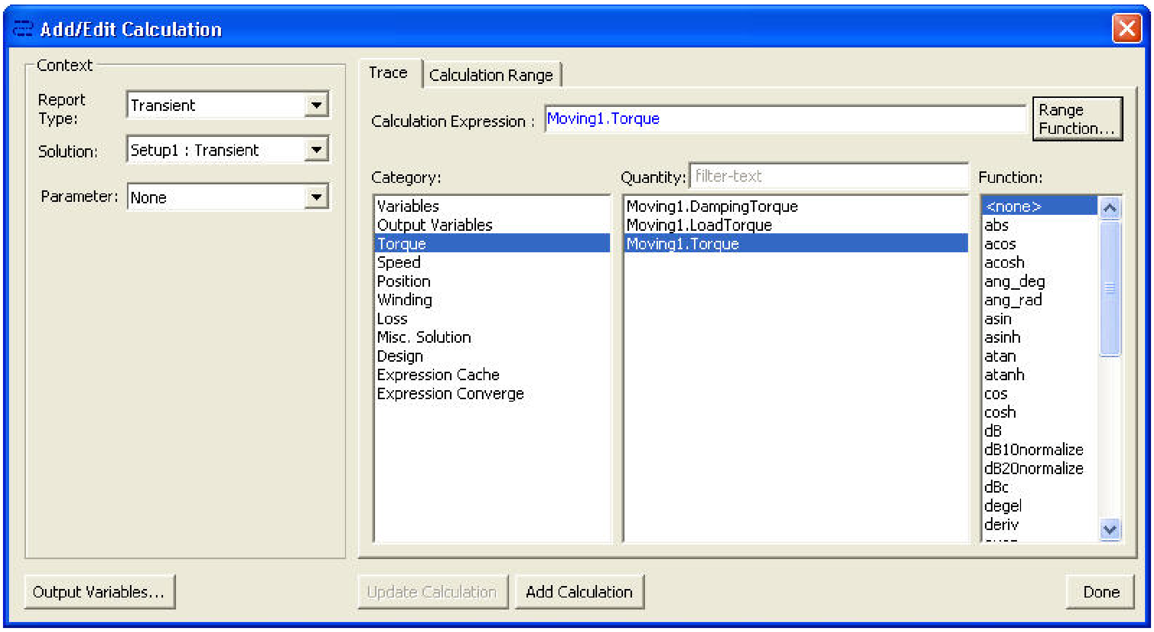
- The setup that has been solved was with a load angle of 20 deg. If the load angle is modified, the simulation has to be restarted.
- A parametric sweep will therefore take a very long time. We can propose two approaches:

  - Realize a Equivalent Circuit Extraction of the motor. This method requires the combination of parametric sweeps in magneto-static and the circuit simulator Simplorer. We will not discuss this method in this write-up.
  - Realize a parametric transient simulation. To cut the simulation time, the use of the Distributive Solve is necessary. This is the chosen method.
- **To Create a Parametric Analysis**
  - Select the menu item ***Maxwell 3D > Optimetrics Analysis > Add Parametric***
  - In Setup Sweep Analysis window, select **Add**
  - In Add/Edit window,
    1. Variable: **Thet_deg**
    2. Select **Linear Step**
    3. Start: **0 deg**
    4. Stop: **60 deg**
    5. Step: **15 deg**
    6. Select **Add**

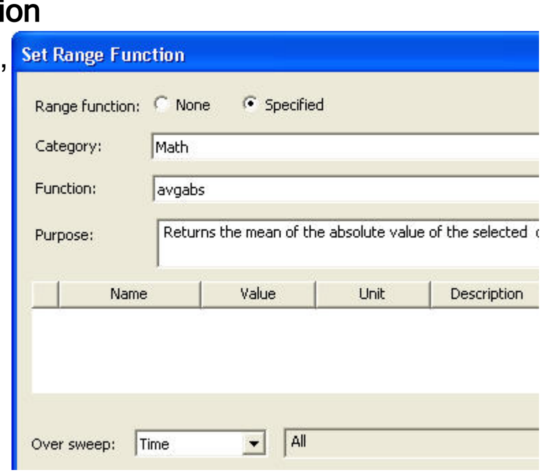
  - In Setup Sweep Analysis window, **General** Tab
    - This panel enables the user to change a design variable. For instance, if you wish to run the parametric sweep with a peak winding current of **400 A**, select the ***Override*** button, and change the current value.



- **Calculation** tab, Select **Setup calculation**
- In Add/Edit Calculation window,
  - Category: **Torque**
  - Quantity: **Moving1.Torque**



  - Select the button **Range Function**
  - In Set Range Function window,
    - Category: **Math**
    - Function: **avgabs**
    - Press **OK**
  - Select **Add Calculation**
  - Press **Done**
- Press **OK** to close

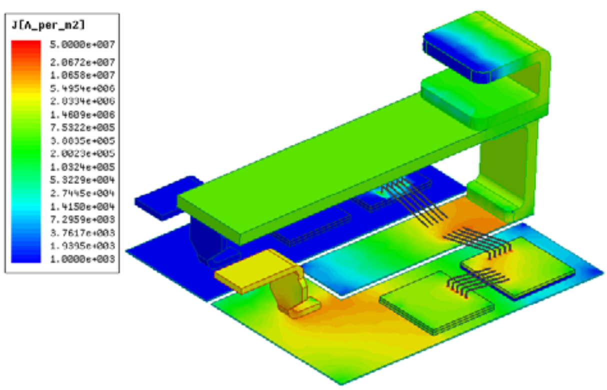


- **Run the parametric sweep.**
  - To run the sweep, select the ***Parametricsetup1***, right mouse click and select the menu item Analysis:
  - It will run with Distributed Analysis if the DSO (Distributed Solve) license is activated
  - It will run the sweep on a single computer otherwise

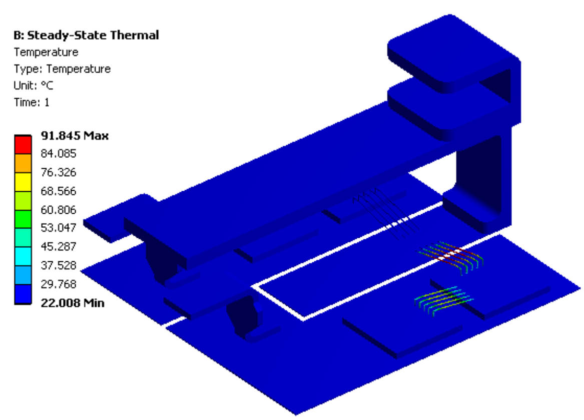
Chapter 12.0 – Multiphysics Coupling

Insulated Gate Bipolar Transistor (IGBT)

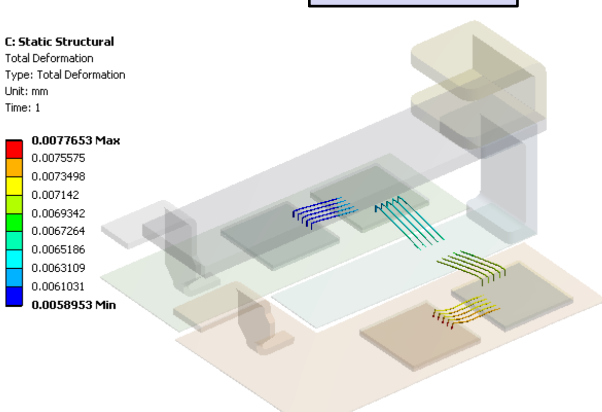
- Insulated Gate Bipolar Transistor is power semiconductor device used for high efficiency and fast switching in modern appliances such as electric cars, trains, variable speed refrigerators, air-conditioners, etc.
- These devices are designed for carrying high currents and many times lead to failures in Bondwires due to high thermal stresses.
- In this example, we will be modeling an IGBT device using Maxwell to get Heat losses and Lorentz forces in Bondwires and then transfer this data to ANSYS Mechanical to calculate Stresses on the wires and determine regions prone to failure.
- In addition, thermal conductivity of all components will be set as function of temperature. Temperature data from ANSYS Mechanical will be transferred back to Maxwell to calculate exact heat losses.
- Total geometry is a six leg IGBT system. To simplify the tutorial, we will be modeling only one leg in the example.



Maxwell



Steady State Thermal



Static Structural

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - Boundaries/Excitations
    - Current: **Stranded**
  - Analysis
    - **Magnetostatic**
    - **Steady State Thermal (ANSYS Mechanical)**
    - **Static Structural (ANSYS Mechanical)**
  - Results
    - **Force**
  - Field Overlays:
    - **J Field**
    - **Temperature**
    - **Equivalent Stress**
    - **Deformation**

Prerequisites

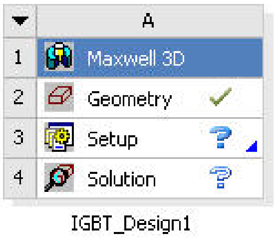
- **To Run this Tutorial**
  - ANSYS Workbench R13/R14 needs to be installed for performing this tutorial
  - ANSYS Mechanical licenses are needed
  - Maxwell needs to be integrated with ANSYS Workbench while installation

Launching ANSYS Workbench

- **To launch Workbench:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **ANSYS 14.0 > Workbench**

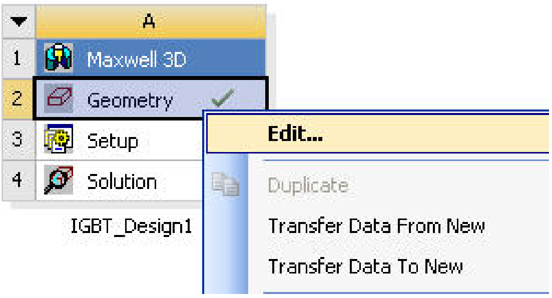
Open Maxwell Project

- **To Open Maxwell File**
  - In ANSYS Workbench Project window, select the menu item *File > Import*
  - Set file type to **Maxwell Project File (.mxwl)**
  - Locate the file **Ex_5_9_IGBT_One_Leg.mxwl** and Open it
  - Maxwell Analysis System will be automatically created as shown in below image



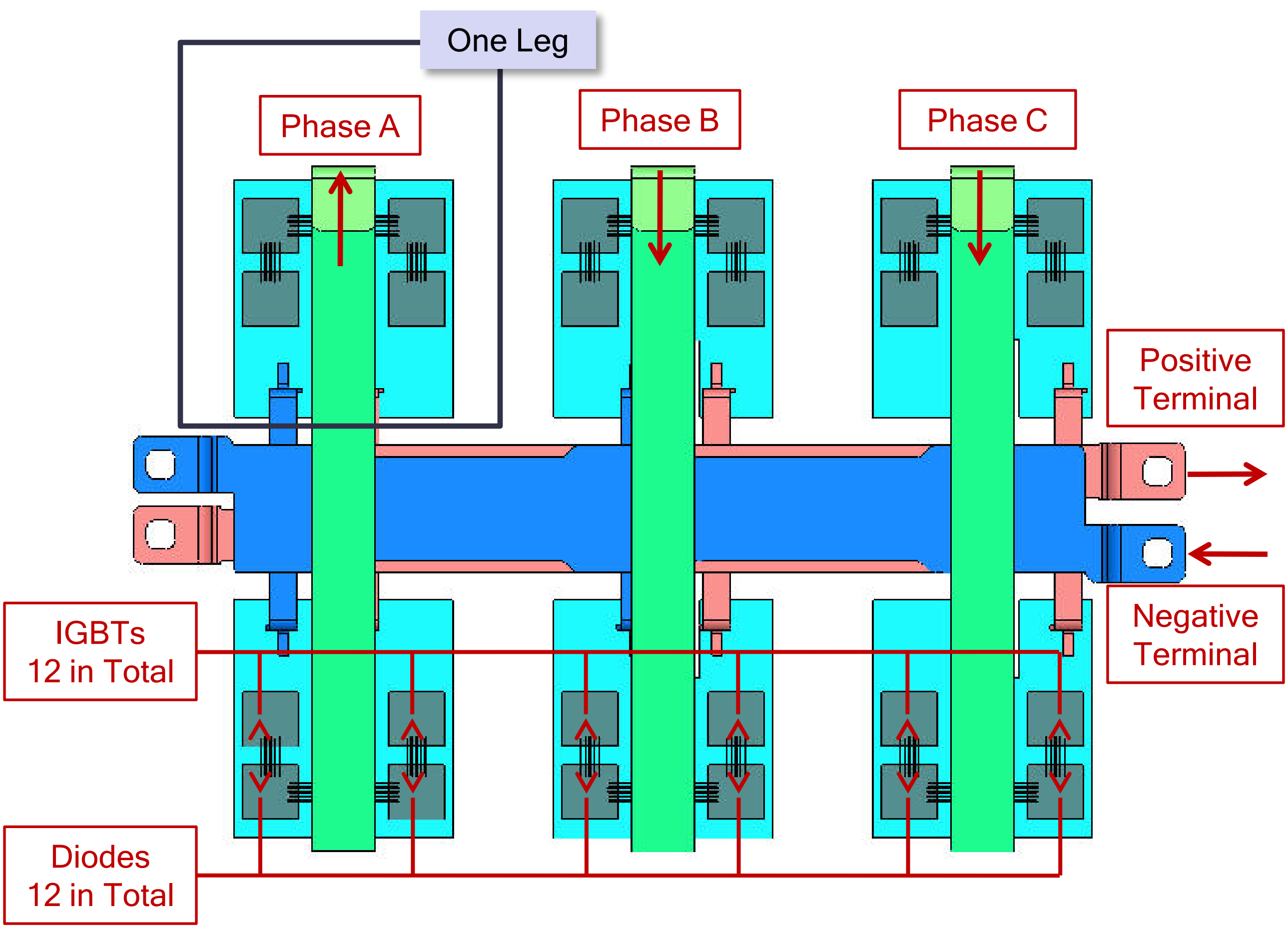
Launch Maxwell

- **To Launch Maxwell**
  - Right click on **Geometry** tab of Maxwell 3D Analysis system and select **Edit**



Description of Geometry

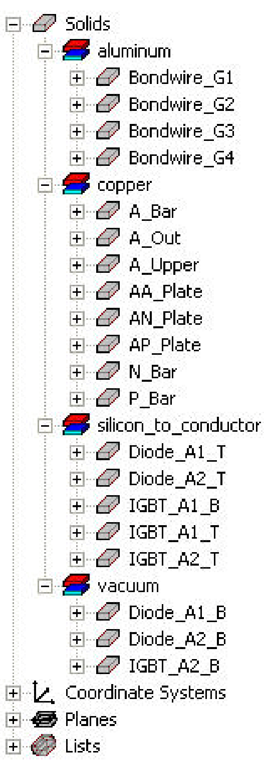
- **Six Leg IGBT Device**
  - Below shown is the image of a six leg IGBT device.
  - The device has three phase supply, Positive and Negative terminals as shown in image
  - There are 12 IGBT chips and 12 diodes, six on each side of terminal wires
  - Diodes and IGBT chips are connected with each other using Aluminum Bondwires
  - In this tutorial, we will be considering only one leg of geometry which is shown in rectangle in below image.

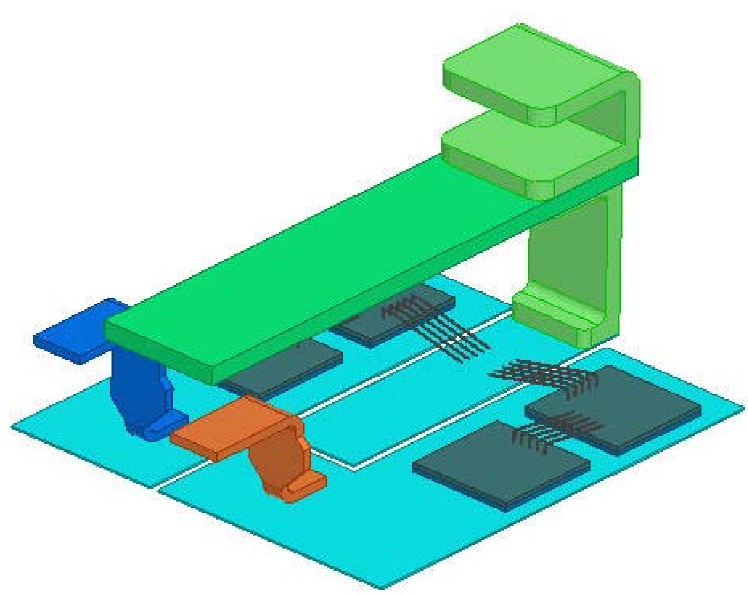


Input File Description

One Leg IGBT Geometry

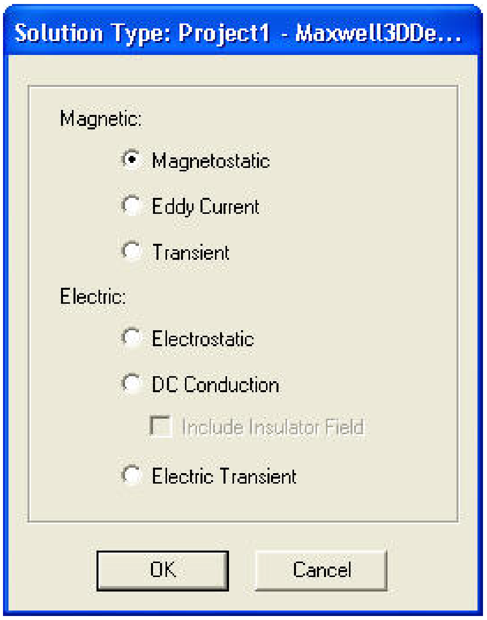
- Input file contains the geometry of one leg separated out of Six leg IGBT system. Names and Material definition to all parts is already done. Model tree should look as shown in the image.
- A_Bar, A_Out, A_Upper corresponds to Phase A of Supply. N_Bar corresponds to Negative terminal while P_Bar corresponds to the positive terminal.
- Diodes A1_B, A2_B and IGBT A2_B are in “OFF” conditions. They have been assigned with material as “vacuum” as they do not contribute to the field.
- Bondwires G1, G2, G3 and G4 will be the focus of this simulation as these thin wires carry very high currents and more likely to fail due to thermal stresses.





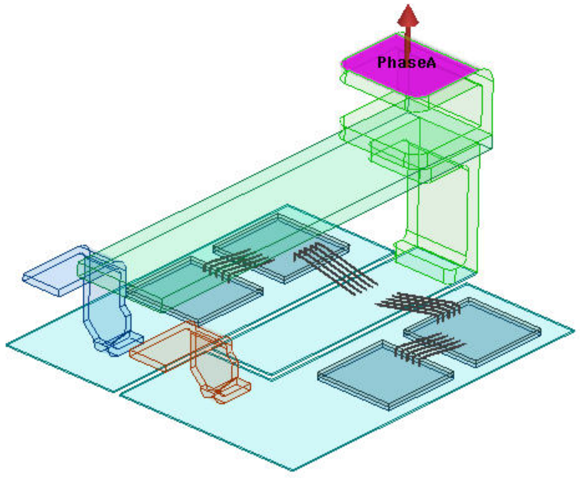
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Magnetostatic**
    2. Click the **OK** button

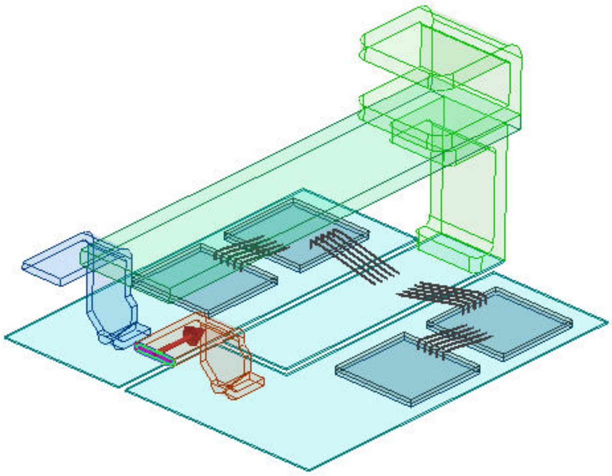


Assign Excitations

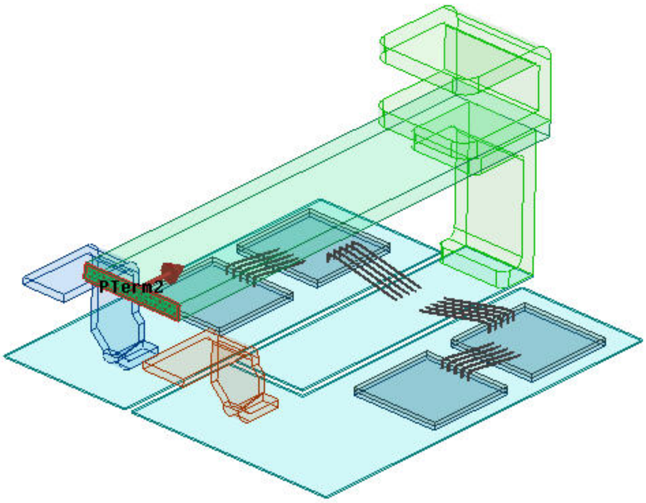
- **To Assign Phase Current**
  - Select the menu item ***Edit > Select > Faces*** or press **F** from the keyboard to change selection to faces
  - Select the face of **A_Out** as shown in image
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **PhaseA**
    2. Value: **40 A**
    3. Type: **Solid**
    4. Direction: **Positive Z**
    5. Press **OK**



- **To Assign Terminal Current to P_Bar**
  - Select the face of **P_Bar** as shown in image
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **PTerm**
    2. Value: **20 A**
    3. Type: **Solid**
    4. Direction: **Negative X**
    5. Press **OK**



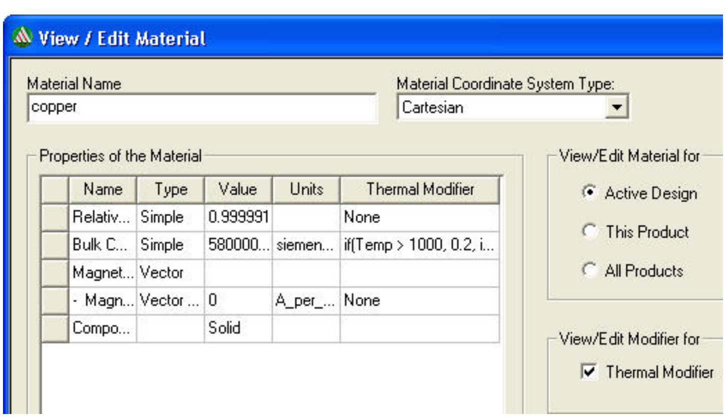
- **To Assign Terminal Current to A_Upper**
  - Select the face of **A_Upper** as shown in image
  - Select the menu item ***Maxwell 3D > Excitations > Assign >Current***
  - In Current Excitation window,
    1. Name: **PTerm2**
    2. Value: **20 A**
    3. Type: **Solid**
    4. Direction: **Negative X**
    5. Press **OK**

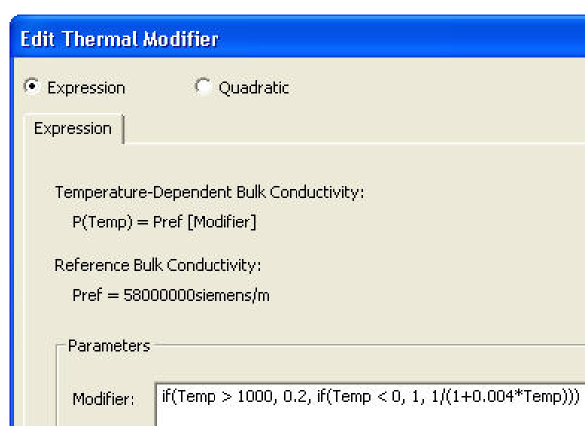


Set Temperature Dependent Material Properties

- **Note:** In this step, thermal conductivity of Copper and Aluminum will be set as function of temperature. At the start, temperature of all objects is considered to be at 22℃. After Solving for Temperature in ANSYS Mechanical, temperature data will be transferred back to Maxwell and solution will recalculated with new temperatures applied through ANSYS Mechanical.
- **To Set Thermal Modifier for Copper**
  - Right click on **copper** material tab from the history tree and select **properties**
  - In Select Definition window,
    1. Select option **View/Edit Material**
    2. In View/Edit Material window
       1. Thermal Modifier: ☑ **Checked**
       2. Bulk Conductivity:
          - Thermal Modifier: Select **Edit**
          - In Edit Thermal Modifier window,
          - Modifier: Set to

            if(Temp > 1000, 0.2, if(Temp < 0, 1, 1/(1+0.004*Temp)))
          - Press **OK**
       3. Press **OK**
    3. Press **OK**



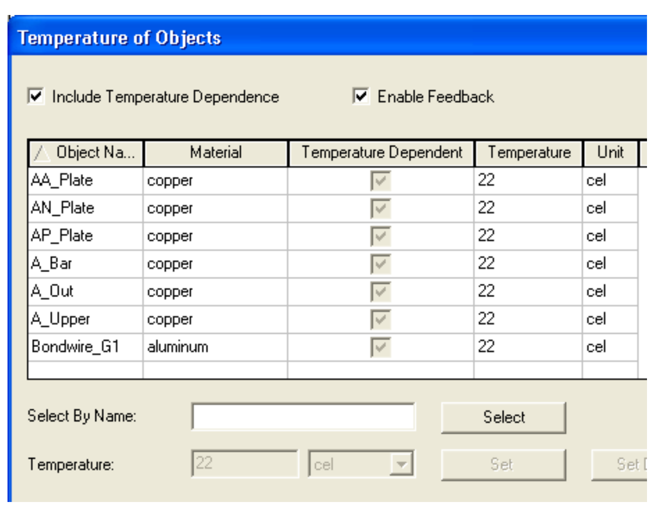


- **To Set Thermal Modifier for Aluminum**
  - Right click on **aluminum** material tab from the history tree and select **properties**
  - In Select Definition window,
    1. Select option **View/Edit Material**
    2. In View/Edit Material window
       1. Thermal Modifier: ☑ **Checked**
       2. Bulk Conductivity:
          - Thermal Modifier: Select **Edit**
          - In Edit Thermal Modifier window,
          - Modifier: Set to

          if(Temp > 1000, 0.2, if(Temp < 0, 1, 1/(1+0.004*Temp)))
          - Press **OK**
       3. Press **OK**
    3. Press **OK**

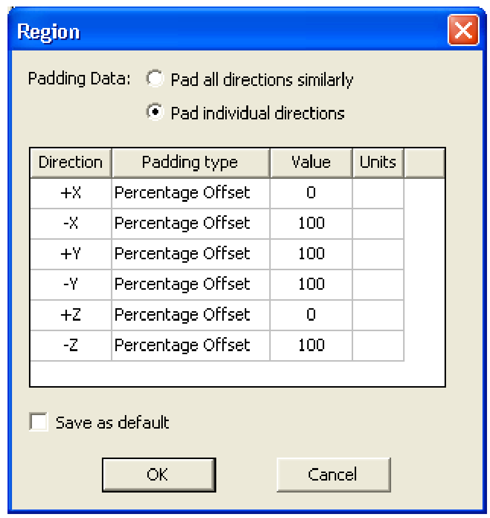
Set Object Temperature

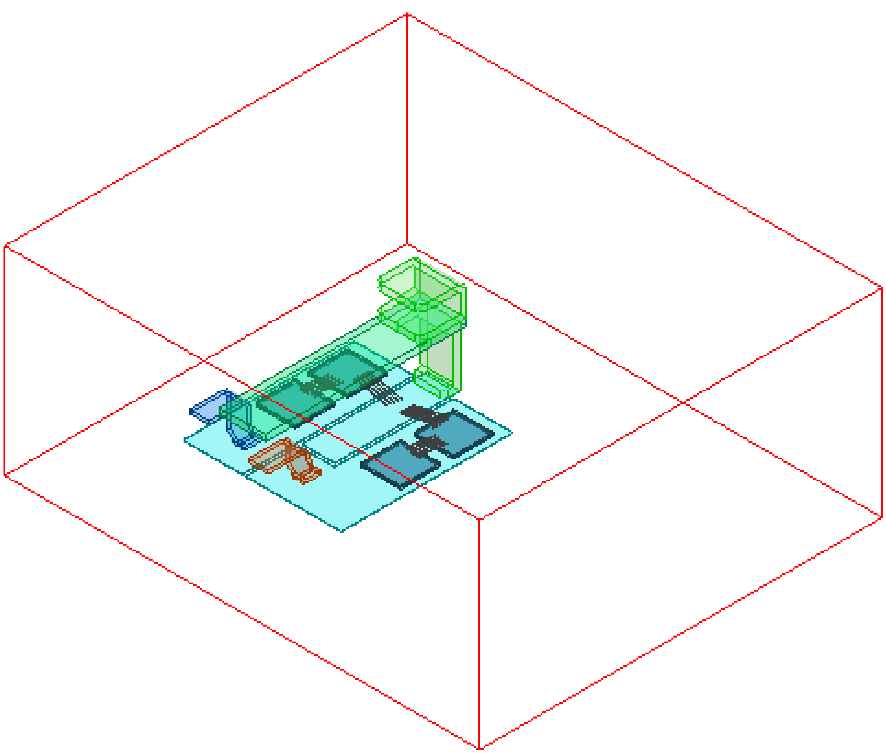
- **To Set Object Temperature**
  - Select the menu item ***Maxwell 3D > Set Object Temperature***
  - In Temperature of Objects window,
    1. Include Temperature Dependence: ☑ **Checked**
    2. Enable Feedback: ☑ **Checked**
    3. Ensure all objects are set to **22 cel**
    4. Press **OK**



Define Region

- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad individual directions: ☑ **Checked**
    2. +X = **0**
    3. -X = **100**
    4. +Y = **100**
    5. -Y = **100**
    6. +Z = **0**
    7. -Z = **100**
    8. Press **OK**



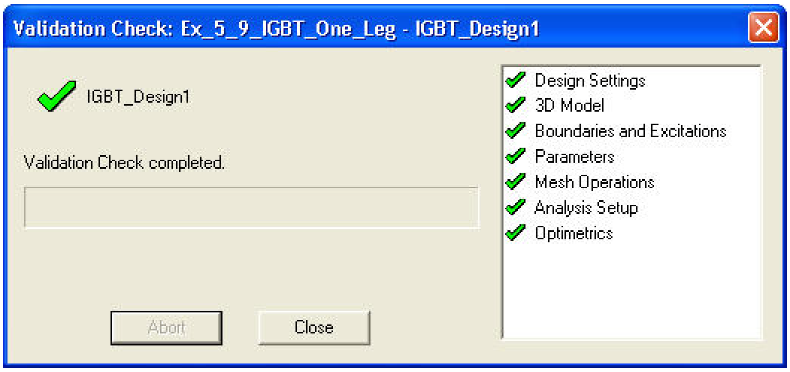


Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.2**
    2. **Convergence**Tab
       - Refinement Per Pass: **50**
    3. Click the **OK** button

Model Validation

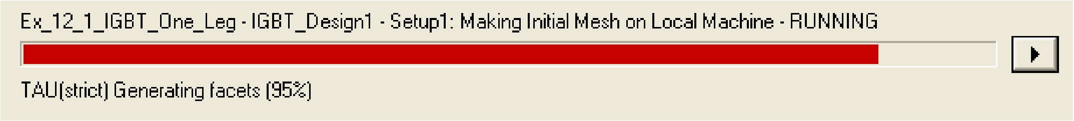
- **To validate the model:**
  - Select the menu item ***Maxwell 3D > Validation Check***
  - Click the **Close** button



Note: To view any errors or warning messages, use the Message Manager.

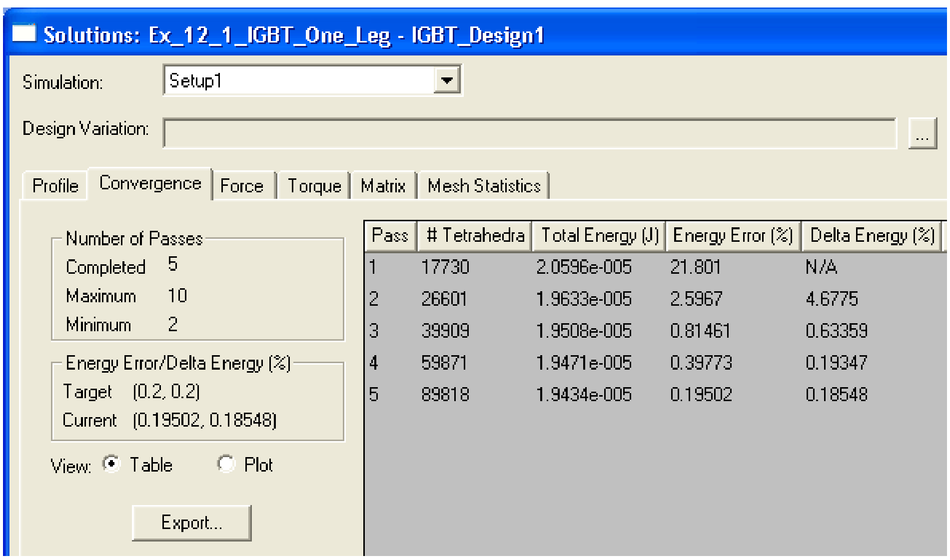
Analyze

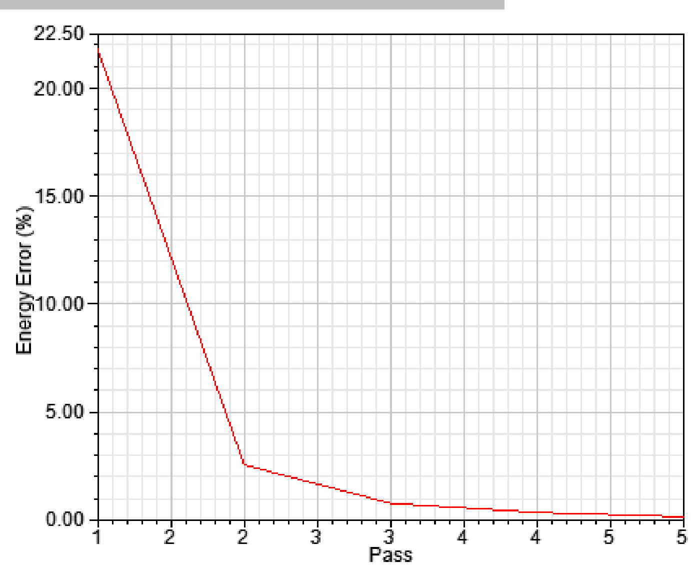
- **To start the solution process:**
  1. Select the menu item ***Maxwell 3D > Analyze All***



Solution Data

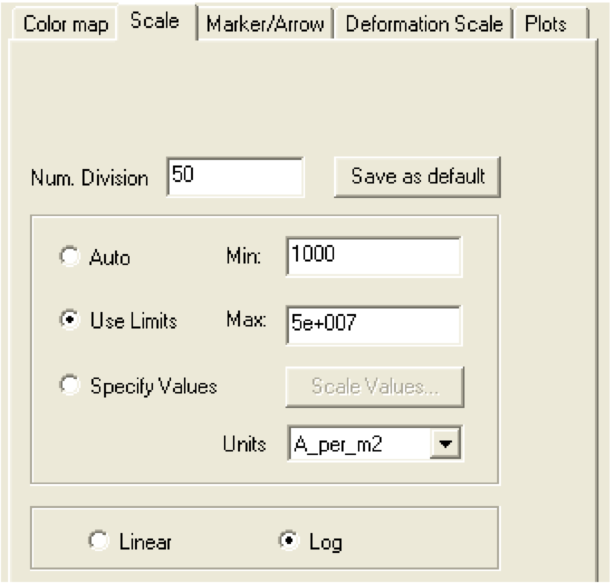
- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
      - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.



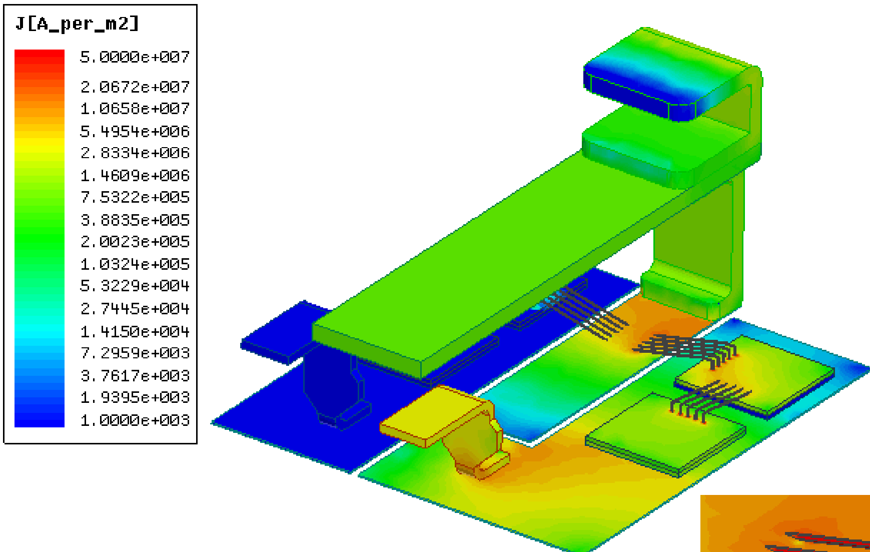


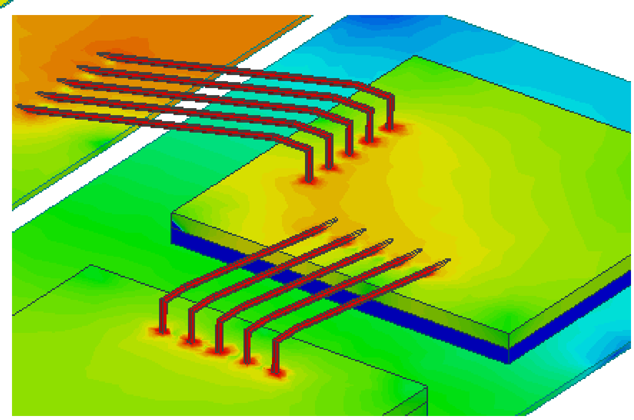
Mag_J Plot

- **Hide Region**
  - Select the menu item ***Edit > Select > Objects*** or press **O** from the keyboard to change selection to Objects
  - Select **Region** from the history tree
  - Select the menu item ***View > Visibility> Hide Selection > Active View***
- **Create Field Plot**
  - Select the menu item ***Edit > Select All Visible***
  - Select menu item ***Maxwell 3D > Fields > Fields > J > Mag_J***
  - In Create Field Plot window
    1. Plot on surface only: ☑ **Checked**
    2. Select **Done**
- **Modify Plot Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Scale** tab
       - Num. Division: **50**
       - User Limits: ☑ **Checked**
         1. Min: **1000**
         2. Max: **5e+7**
       - Log: ☑ **Checked**
    2. Press **Apply** and **Close**



- The plot of the Mag_J will be as below:
- Volume current density is high in thin sections of **Bondwire_G1** and **Bondwire_G2**





Exit Maxwell

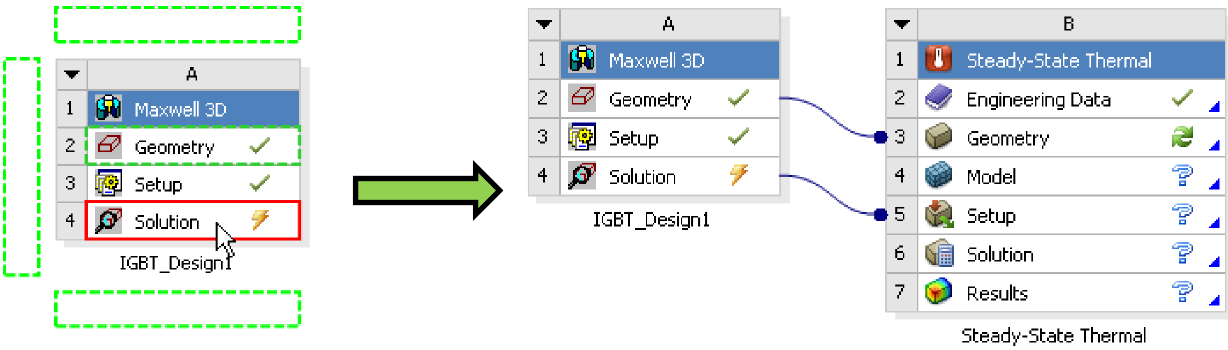
- **To Close Maxwell**
  - Select the menu item ***File > Close Desktop***

Save Project

- **Save Workbench Project**
  - To save the project, goto the Workbench project window and select menu item ***File > Save As***
  - Save the file with the name **Ex_12_1_IGBT_One_Leg**

Create Steady State Thermal System

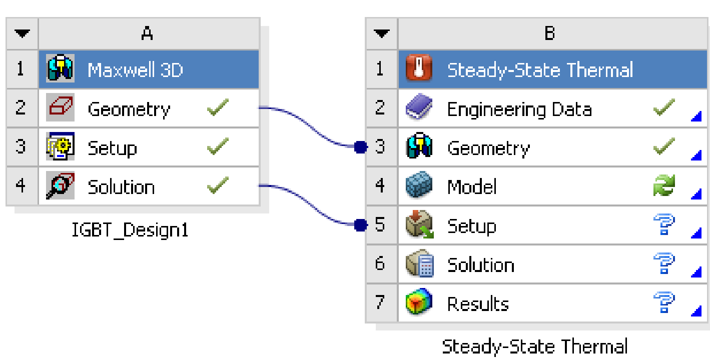
- **To Create a Steady State Thermal Analysis System**
  - Select **Steady State Thermal** system from the Analysis Systems list
  - Drag and drop it on the **Solution** tab of Maxwell 3D Analysis System
  - Drag and drop the geometry tab of Maxwell analysis system on the **Geometry** tab of the Steady State Thermal system



  - **Note:** By Dropping Steady State Thermal Analysis System onto Solution tab of Maxwell System we are establishing a link between **Solution** tab of Maxwell and **Setup** tab of Steady State thermal Analysis System. This will enable the data transfer from Maxwell to ANSYS Mechanical

Update Maxwell Cell

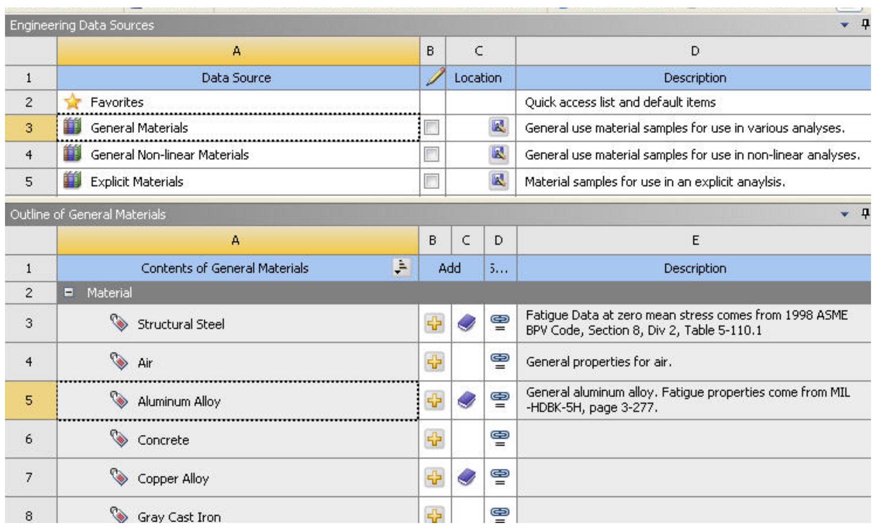
- **To Update Maxwell Solution**
  - Right click on **Solution** tab of Maxwell 3D Analysis System and select **update**
  - A tick mark will appear adjacent to Solution tab of Maxwell
  - Right click on **Geometry** tab of Steady-State Thermal system and select **Refresh**



Define Material Database

To Specify Material Data

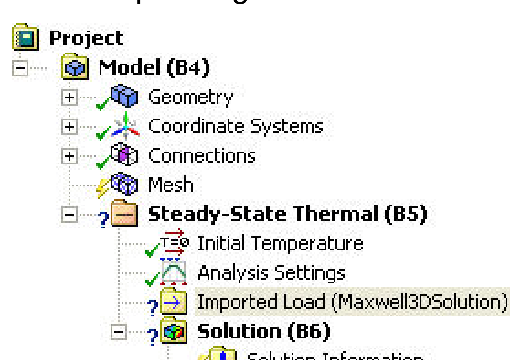
- Right click on **Engineering Data** tab and select **Edit**
- In Engineering Data window,
  1. Select the icon **Engineering Data Sources**
  2. From Engineering Data Sources, select the tab **General Materials**
  3. Press **Ctrl** and select materials **Aluminum Alloy, Copper Alloy** and **Silicon Anisotropic,** right click and select **Add**
  4. Select **Return to Project** to exit Engineering Data



Launch ANSYS Mechanical

To Launch ANSYS Mechanical

- Right click on **Model** tab of Steady State Thermal Analysis System and select **Edit**
- In ANSYS Mechanical window, a tab corresponding to Maxwell link will be automatically added

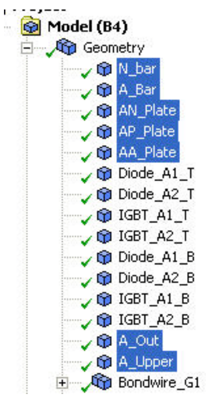


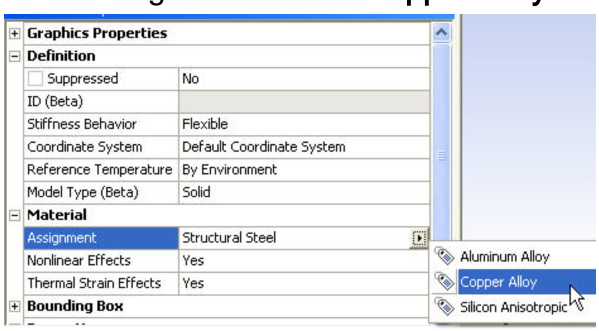
Set Units

- **To Set Unit System**
  - Select the menu item ***Units > Metric (mm, kg, N, s, mV, mA)***
  - Rest of the tutorial will be done in this unit system

Define Materials

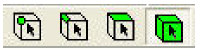
- **To Define Copper Material**
  - Expand the Geometry tab from the History tree
  - Press **Ctrl** and select the objects **A_Bar, P_Bar, AA_Plate, AP_Plate, AN_Plate, N_Bar, A_Out** and **A_Upper**
  - In Details View Window:
    1. Material:
       - Assignment: Set to **Copper Alloy**

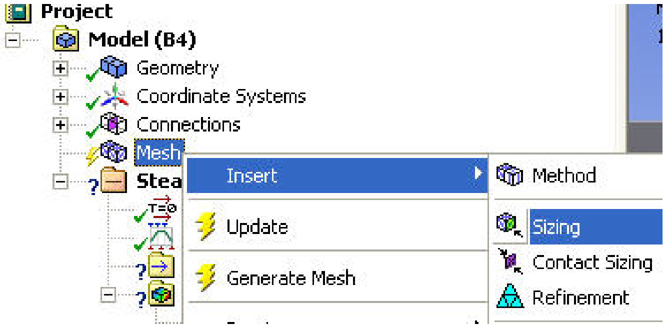


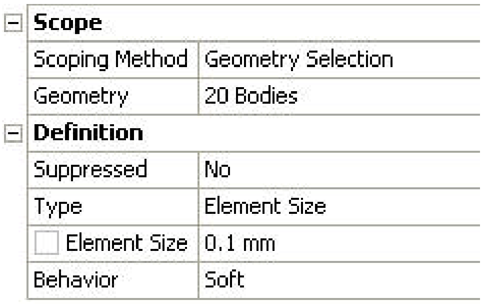


- **To Define Aluminum Material**
  - Press **Ctrl** and select the objects **Bonwire_G1, Bondwire_G2, Bondwire_3** and **Bondwire_G4**
  - In Details View Window:
    1. Definition:
       - Assignment: Set to **Aluminum Alloy**
- **To Define Silicon Material**
  - Press **Ctrl** and select the objects **Diode_A1_T, Diode_A1_B, Diode_A2_T, Diode_A2_B, IGBT_A1_T, IGBT_A1_B, IGBT_A2_T** and **IGBT_A2_B**
  - In Details View Window:
    1. Material:
       - Assignment: Set to **Silicon Anisotropic**
       - Non-linear Effects: **No**

Assign Mesh Parameters

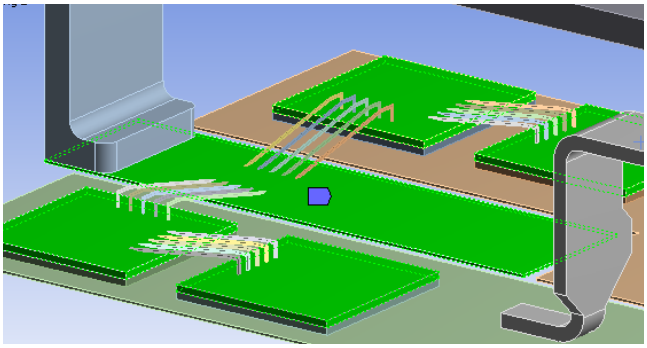
- **Note:** Mesh size variation from bondwires to the bodies attached with it can not be too large. In order to avoid this, we need to specify smaller mesh sizes to these bodies.
- **To Assign Mesh Parameters for Bondwires**
  - Right click on **Mesh** tab from the tree and select ***Insert > Sizing***
  - In Details of Sizing window,
    1. Geometry:
       - Change the selection filter to “Bodies” 
       - Press **Ctrl** and select all Bondwires from Graphic window
       - Press **Apply**
    2. Element Size: **0.1mm**





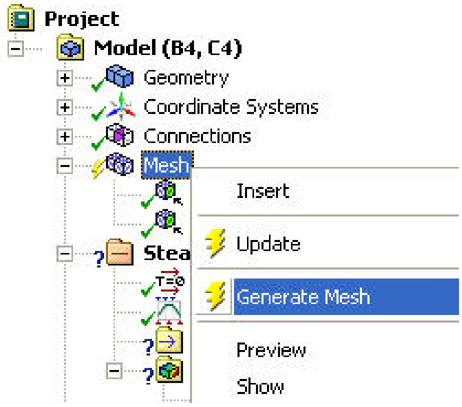
Scope	
Scoping Method	Geometry Selection
Geometry	20 Bodies
Definition	
Suppressed	No
Type	Element Size
Element Size	0.1 mm
Behavior	Soft

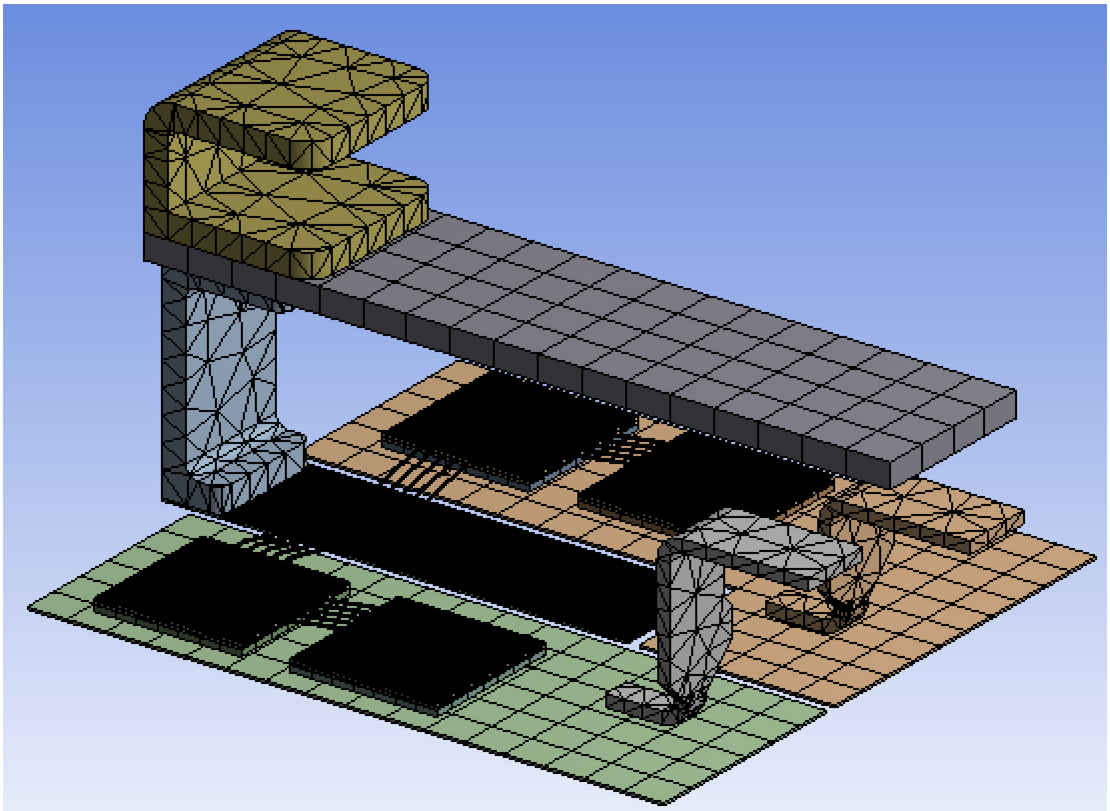
- **To Assign Mesh Parameters for Bondwires**
  - Right click on **Mesh** tab from the tree and select ***Insert > Sizing***
  - In Details of Sizing window,
    1. Geometry:
       - Press **Ctrl** and select the bodies **AA_Plate, Diode_A1_T, Diode_A2_T, IGBT_A1_T** and **IGBT_A2_T** from Graphic window
       - Press **Apply**
    2. Element Size: **0.25mm**



Generate Mesh

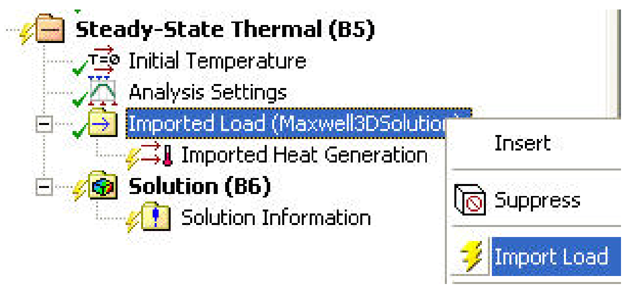
- **To Launch Meshing**
  - Right click on **Mesh** tab and select **Generate Mesh**
  - Mesh will be generated and displayed on the Graphic Window





Import Data from Maxwell

- **To Import Heat Generation from Maxwell**
  - Right click on the tab **Imported Load (Maxwell3DSolution)** and select ***Insert > Heat Generation***
  - In Details of Imported Heat Generation window,
    1. Geometry
       - Right click in graphic window and select **"Select All"**
       - Press **Apply**
  - Right click on the tab **Imported Load (Maxwell3DSolution)** and select **Import Load**
  - **Note:** An Import Load transfer summery will be created below "Imported Heat Generation". This lists error in data transfer due to variation of mesh in Maxwell and ANSYS Mechanical. The value of scale should be close to 1 in important regions.

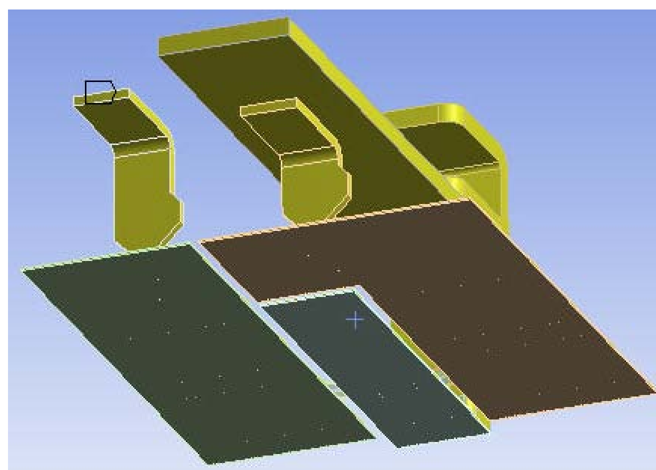


A_Out	0.0122011W	0.997543
A_Upper	0.0141301W	1.01533
Bondwire_G1	0.759776W	0.999037
Bondwire_G4	2.89867E-013W	0.997171
Bondwire_G2	0.969409W	1.00037
Bondwire_G3	6.56074E-006W	0.991267
P_Bar	0.0384212W	1.01268

Specify Convective Boundary

- **To Assign Convective Boundary**
  - Right click on **Steady State Thermal** tab from the tree and select ***Insert > Convection***
  - In Details of Convection window,
    1. Geometry:
       - Change the selection filter to "Faces"
       - Right click in graphic window and select **"Select All"**
       - Press **Ctrl** and select bottom faces as shown in below image to remove them from selection
       - Press **Apply**
    2. Film Coefficient: **1e-4 W/mm$^{2}\cdot$° C**

Scope	
Scoping Method	Geometry Selection
Geometry	543 Faces
Definition	
Type	Convection
Film Coefficient	1.e-004 W/mm²·°C (ramped)
Ambient Temperature	22. °C (ramped)
Suppressed	No

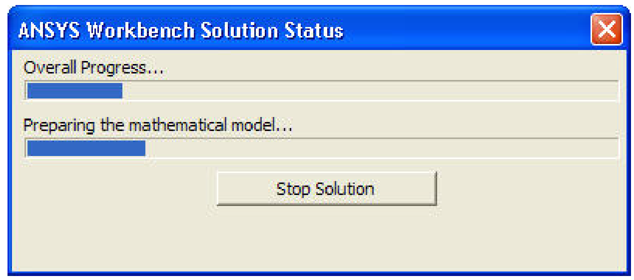


Create Temperature Field Plot

- **To Create Temperature Contours**
  - Right click on **Solution** tab from the tree and select ***Insert > Thermal > Temperature***

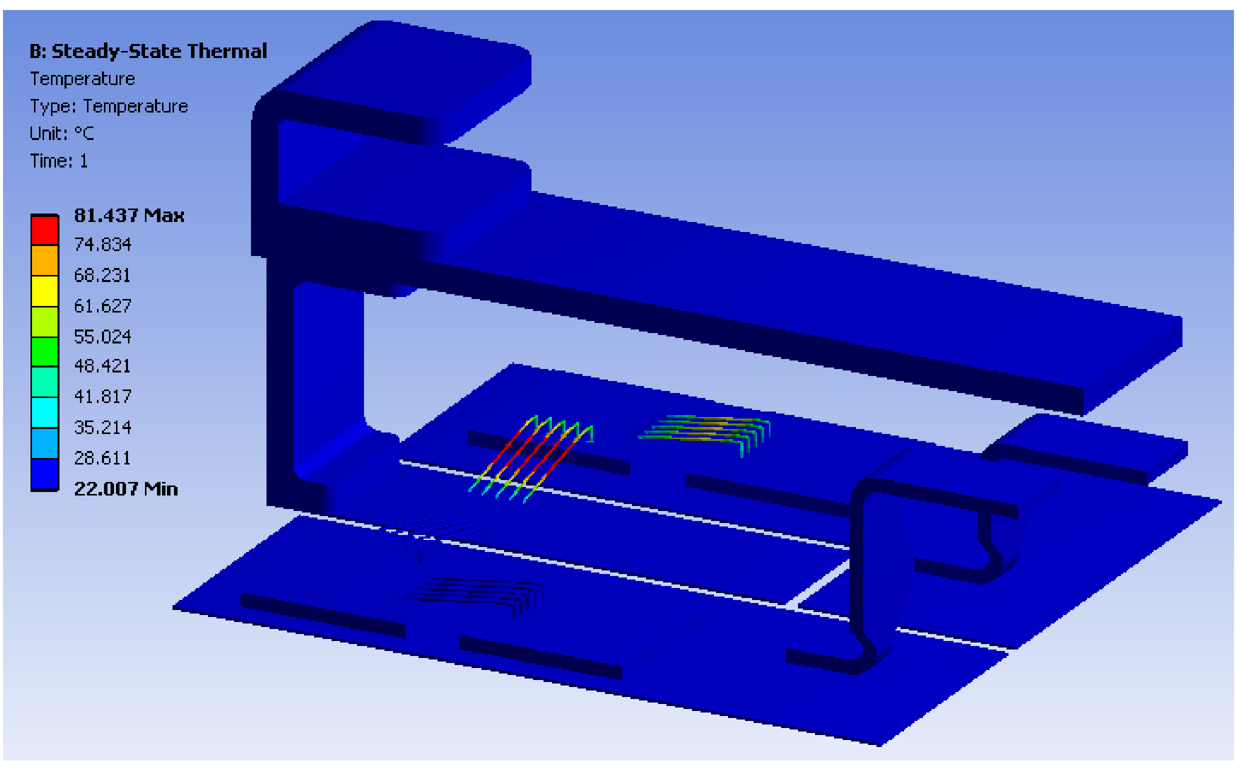
Solve Thermal Setup

- **To Run the Solution**
  - Right click on **Solution** tab and select **Solve**
  - Solution progress will be shown as below



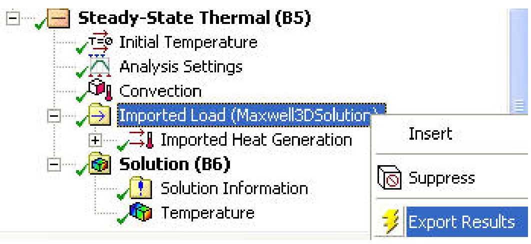
Temperature Contours

- Maximum Temperature is in Bondwires which is around **81°C**



Transfer Object Temperatures to Maxwell

- **To Transfer Temperature Data to Maxwell**
  - Right click on the tab **Imported Load (Maxwell3DSolution)** and select **Export Results**
  - Temperatures calculated by Steady State Thermal will be transferred back to Maxwell

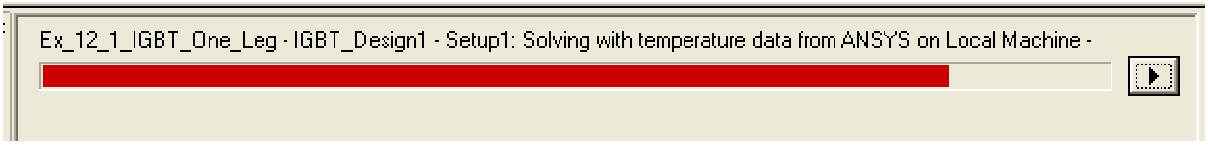


Exit Steady State Thermal

- **To Close ANSYS Mechanical**
  - Select the menu item ***File > Close Mechanical***

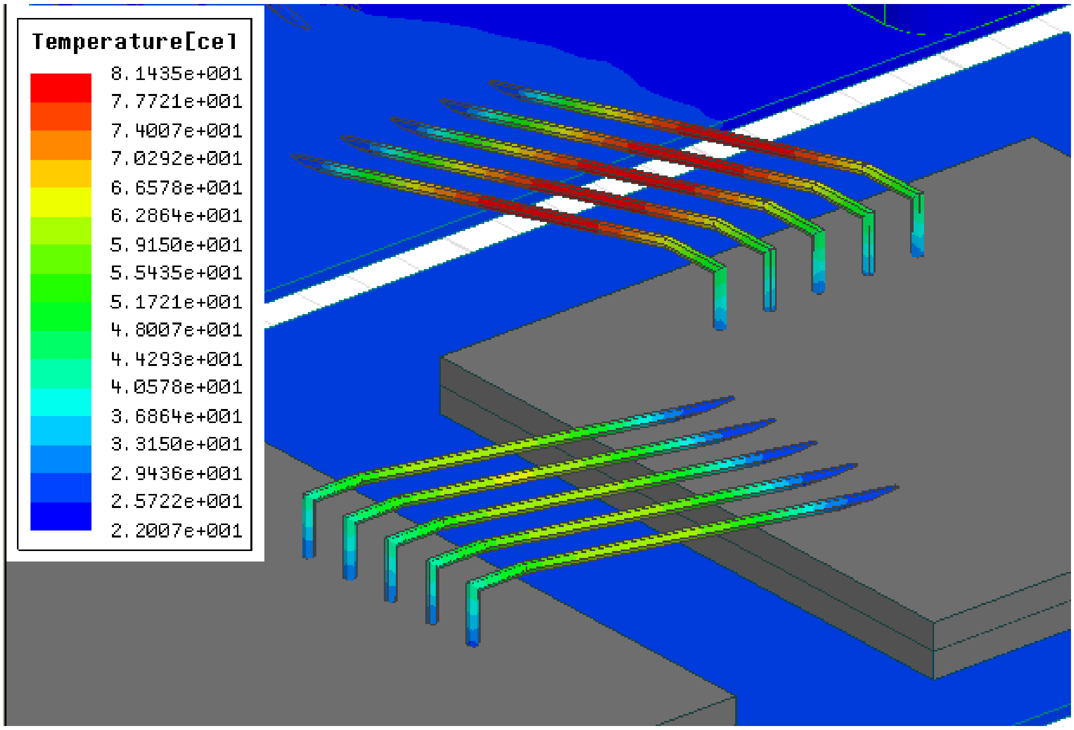
Solve Maxwell with Imported Temperatures

- **To Launch Maxwell**
  - Right click on **Solution** tab of Maxwell 3D analysis system and select **Edit**
- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***
  - Maxwell will read imported temperatures from ANSYS Mechanical and solve using imported temperature data



  - **Note:** Maxwell will now calculate heat losses based on imported temperatures which can be again applied back to Steady State Thermal Solver to get refined temperature distribution. These temperatures in turn can be applied to Maxwell. The process can be repeated until there is no significant change in Temperature results obtained

- **Temperature Distribution in Maxwell**
  - Temperature Filed plot can be can be created in Maxwell which will show imported temperatures
  - Select the menu item ***Edit > Select All Visible***
  - Select menu item ***Maxwell 3D > Fields > Fields > Other > Temperature***
  - In Create Field Plot window
    1. Plot on surface only: ☑ **Checked**
    2. Select **Done**

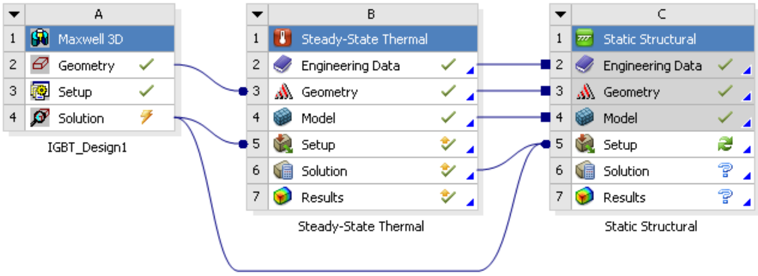


  - **Note:** As we applied temperature based properties only for Copper and Aluminum, plot is for the objects having those materials.

- **Close Maxwell**
  - Select the menu item ***File > Close Desktop***

Create Static Structural System

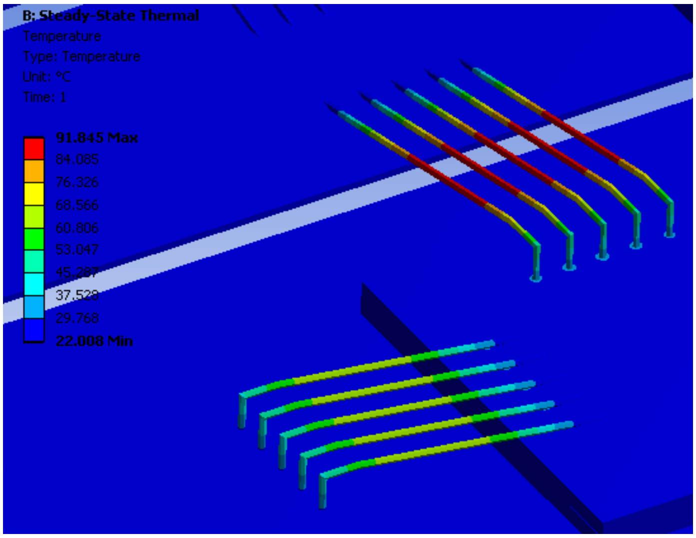
- **To Create Static Structural Analysis System**
  - Select **Static Structural** system from the Analysis Systems list
  - Drag and drop it on the **Solution** tab of Steady State Thermal Analysis System
  - Select **Solution** tab of Maxwell 3D Analysis system, drag and drop it on **Setup** tab of Static Structural Analysis System



  - **Note:** By dropping Solution tab of Maxwell onto Setup tab of Static Structural System, a link will be created between them. This will ensure transfer of Lorentz Forces from Maxwell to ANSYS Mechanical

Update Steady State Thermal Cell

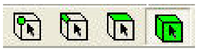
- **To Update Thermal Cell**
  - Right click on **Results** tab of Steady State Analysis System and select **update**
  - New temperature plot can be seem by opening Steady State Thermal window



Launch ANSYS Mechanical

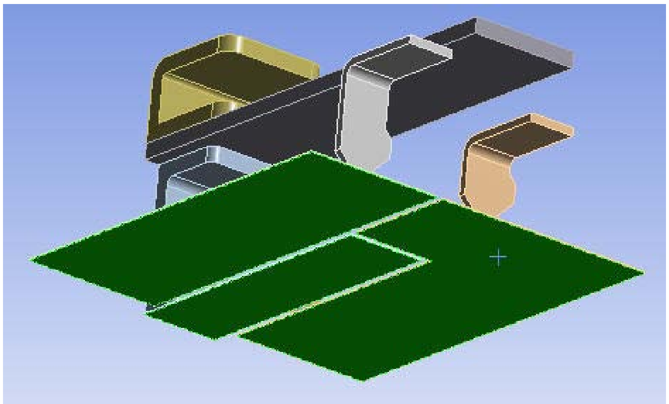
- **To Launch ANSYS Mechanical**
  - Right click on **Setup** tab of Static Structural Analysis System and select **Edit**

Import Forces from Maxwell

- **To Import Lorentz Forces from Maxwell**
  - Right click on the tab **Imported Load (Maxwell3DSolution)** and select *Insert > Body Force Density*
  - In Details of Body Force Density window,
    1. Geometry:
       - Change the selection filter to "Bodies" 
       - Press **Ctrl** and select all bondwires from the graphic window
       - Press **Apply**
  - Right click on the tab **Imported Load (Maxwell3DSolution)** and select **Import Load**

Specify Support

- **To Specify Supports**
  - Right click on **Static Structural** tab of tree and select *Insert > Frictionless Support*
  - In Details of Frictionless Support window
    1. Geometry:
       - Select bottom faces of geometry as shown in image
       - Press **Apply**

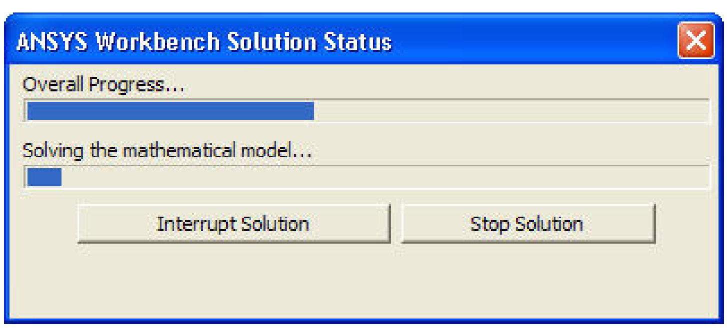


Create Field Plots

- **To Stress Field Plots for Bondwires**
  - Right click on **Solution** tab and select *Insert > Stress > Equivalent (von-Mise*
  - In Details of Equivalent Stress window
    1. Geometry:
       - Change the selection filter to “Bodies” 
       - Select all bondwires
       - Press **Apply**
- **To Deformation Field Plots for Bondwires**
  - Right click on **Solution** tab and select *Insert > Deformation > Total*
  - In Details of Total Deformation window
    1. Geometry:
       - Select all bondwires
       - Press **Apply**

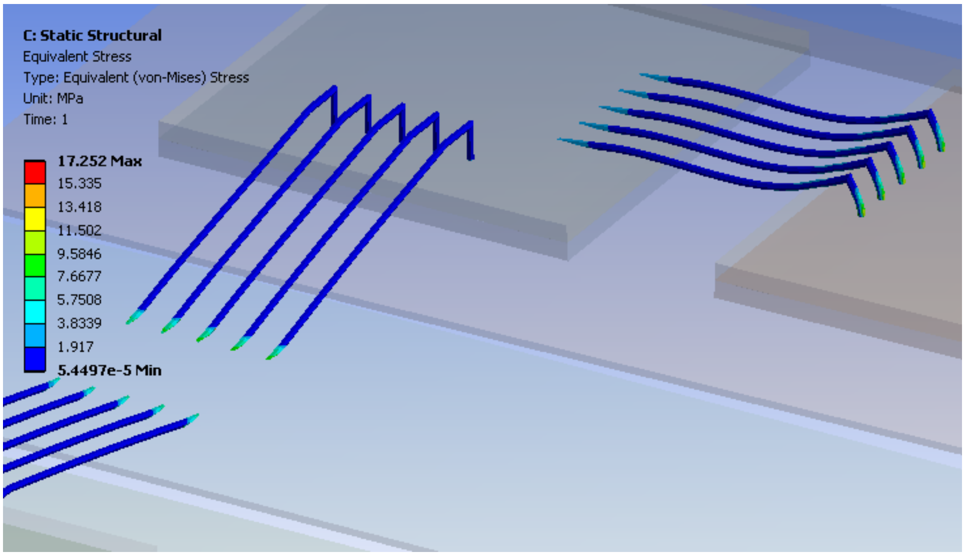
Solve Structural Setup

- **To Run the Solution**
  - Right click on **Solution** tab and select **Solve**
  - Solution progress will be shown as below



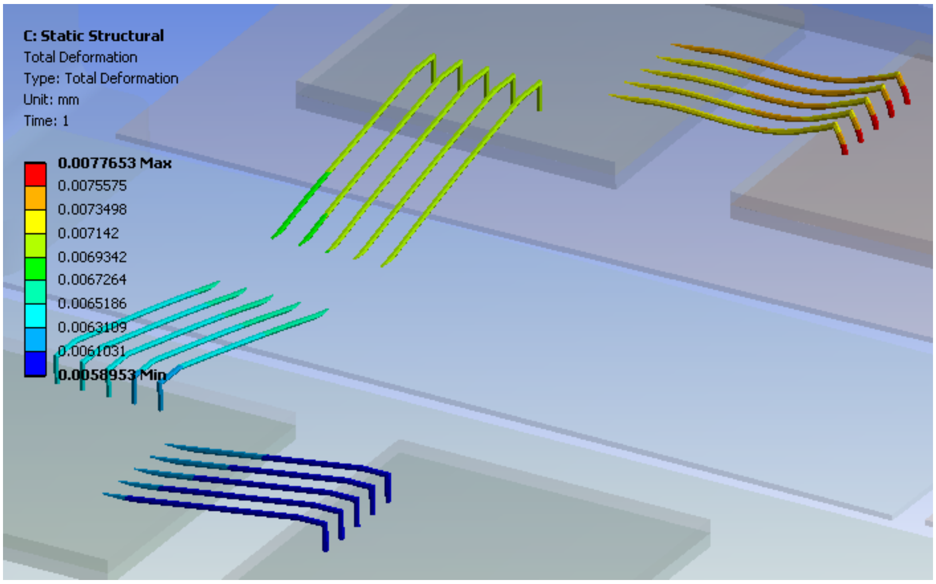
Field Plots

Equivalent Stress

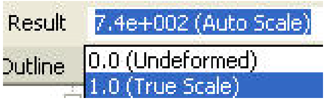


- Maximum stress resulting in wires can be compared with the material properties to predict the failure region.

Deformation



- **Note:** Deformation images are magnified. To view results to the scale Change the Result to **True Scale**

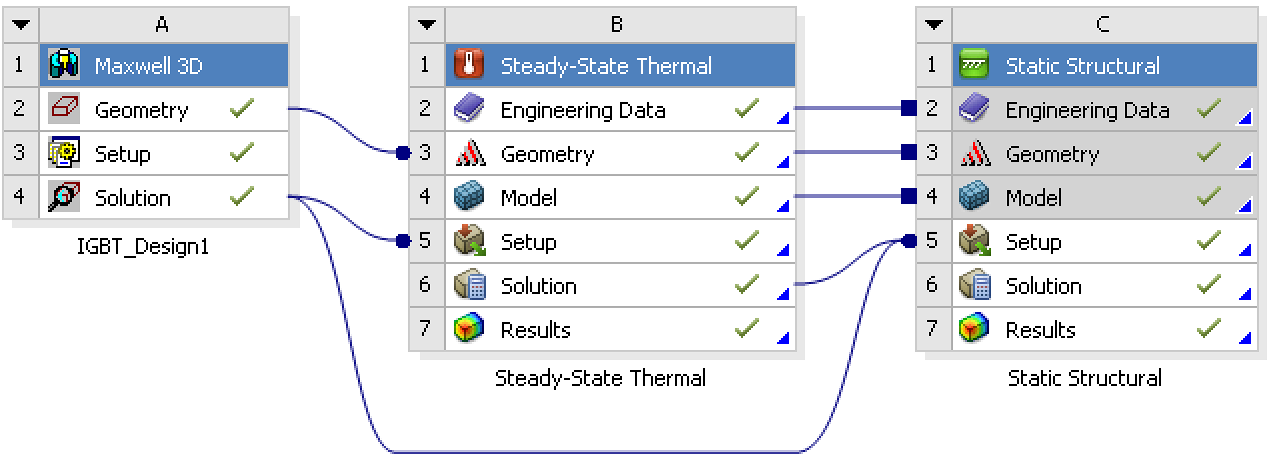


Close ANSYS Mechanical

- To Close ANSYS Mechanical
  - Select the menu item *File > Close Mechanical*

Save Workbench project

- To Save Project
  - Select the menu item *File > Save*



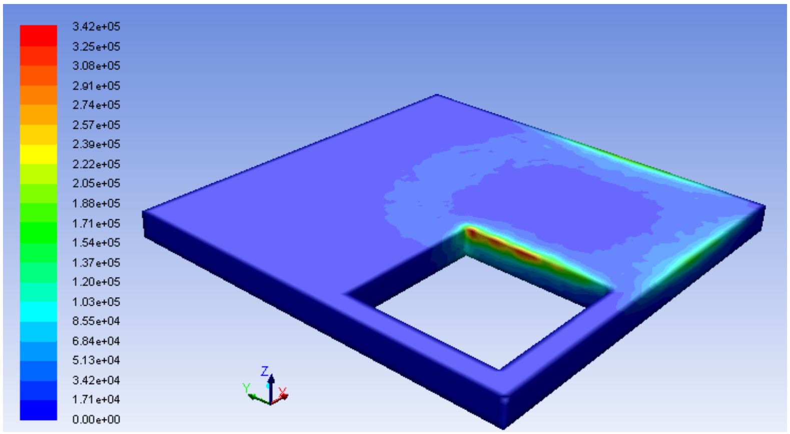
Multiphysics Coupling - Maxwell Eddy Current to FLUENT

Maxwell to FLUENT Coupling

- This workshop guides how heat loss data can be transferred from Maxwell 15 to FLUENT 13
- The example uses an eddy current solver of Maxwell to calculate heat losses which are then applied to FLUENT to simulate Natural Convection
- The workshop also guides users through basic FLUENT setup.
- This workshop is NOT intended to show how natural convection is simulated in FLUENT but rather gives basic information about FLUENT setup. For more details of FLUENT simulation, please refer FLUENT Users Guide

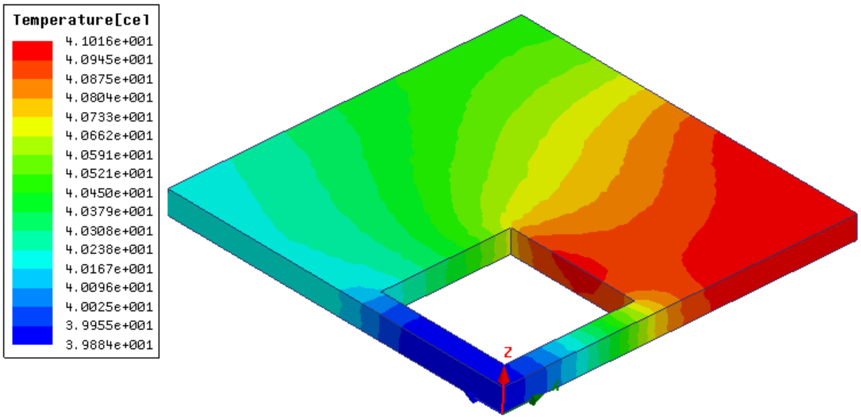
Prerequisites

- Maxwell V15 and FLUENT V14 licenses should be available to perform this tutorial
- ANSYS Workbench R14 should be installed



Heat Losses Mapped in FLUENT

Temperature In Maxwell

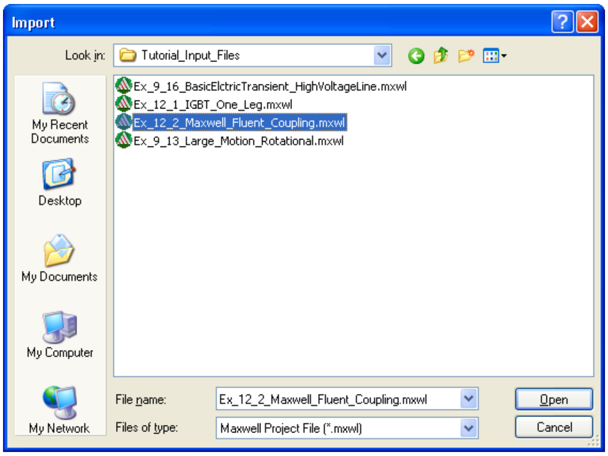


Launch ANSYS Workbench

- **To Launch Workbench**
  - Select the Microsoft **Start** button , select **All Programes > ANSYS 14.0 > Workbench 14.0**

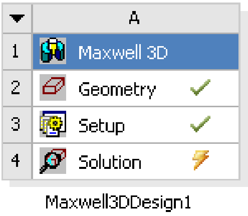
Import Maxwell Project File

- Since we are using an existing Maxwell project, we will import it into Workbench. Users can create a new Maxwell Analysis System to setup a new problem.
- **To Import Maxwell File**
  - Select the menu item ***File > Import***
  - Change the file type to **Maxwell Project File (*.mxwl)**
  - Browse to the location where tutorial input files are saved
  - Select the file **"Ex_12_2_Maxwell_Fluent_Coupling.mxwl"** and **Open** it.



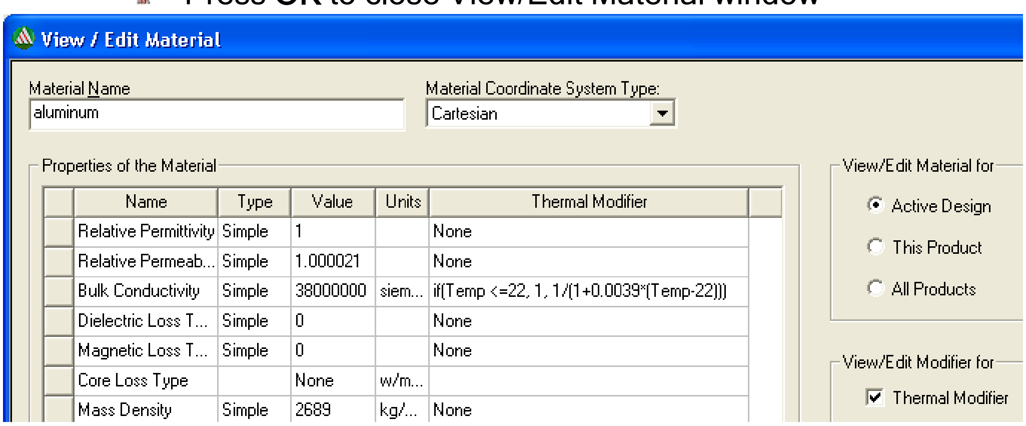
Launch Maxwell

- A Maxwell analysis system will be created as shown below.
- **To Lunch Maxwell**
  - Double click on the solution tab of Maxwell analysis system.
  - The project used here is from a Maxwell workshop. Detailed settings for the project can be found from the Maxwell workshop (6.2 Asymmetric Conductor). The starting point here is an already setup project.

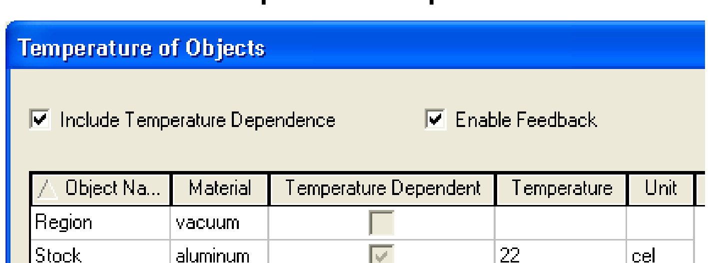


Set Temperature Dependence

- In this step we will set the conductivity of the aluminum plate as function of temperature. This will enable us to get the temperature from the CFD run and recalculate the losses based on temperature dependent properties.
- **To Set Temperature Dependent Properties**
  - Select the object **Stock** from the history tree, right click and select **Assign Material**
  - In Select Definition window, Select **View/ Edit Material**
  - In View/ Edit Material window,
    - Thermal Modifier: ☑ **Checked**
    - Goto the Thermal Modifier column for Bulk Conductivity and select **Edit**
    - In Edit Thermal Modifier window,
      - Set Modifier as : **if(Temp <=22, 1, 1/(1+0.0039*(Temp-22)))**
      - Press **OK**
    - Press **OK** to close View/Edit Material window



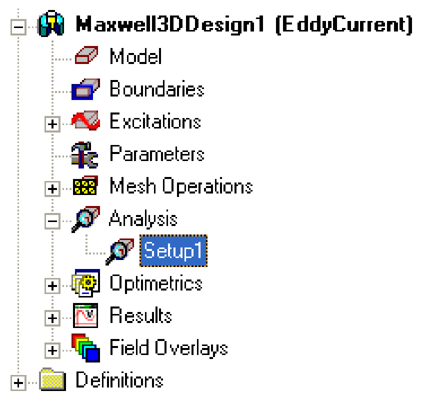
- **Enable Temperature Feedback**
  - Select the menu item ***Maxwell 3D > Set Object Temperature***
  - In the window,
    - Enable the options **"Include Temperature Dependence"** and **"Enable Feedback"**
    - Press **OK**

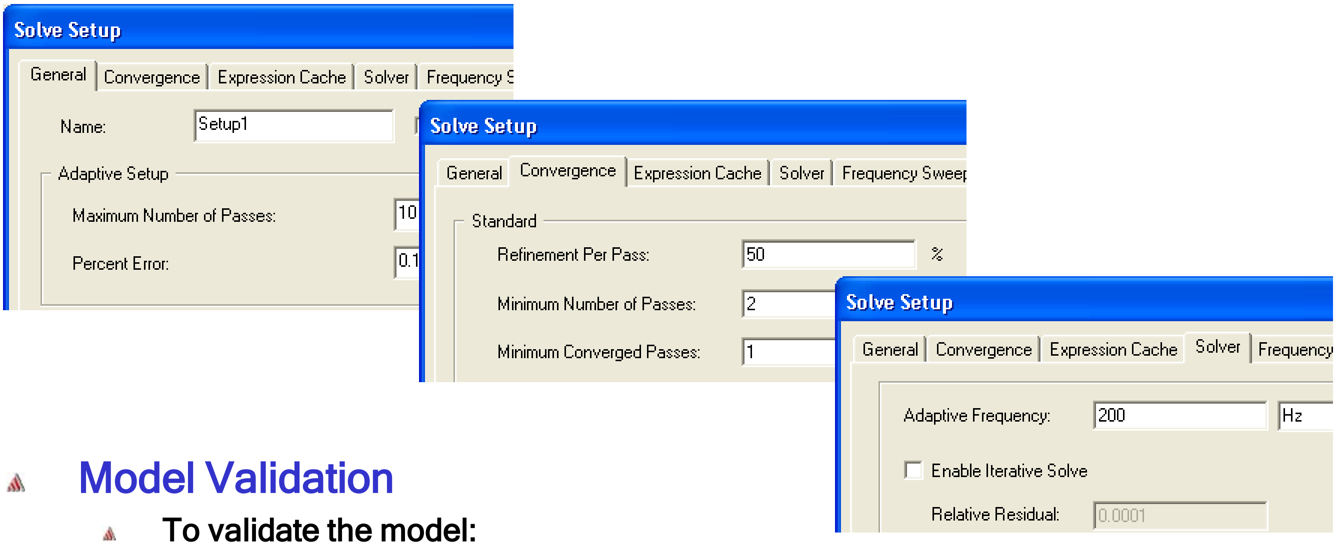


Multiphysics Coupling – Maxwell Eddy Current to FLUENT

Verify Maxwell Settings

- **To Verify Analysis Settings**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1**
  - In Solve Setup window
    1. **General** tab
       - Percentage Error : **0.1**
    2. **Convergence** Tab:
       - Refinement Per Pass: 50 %
    3. **Solver** tab:
       - Adaptive Frequency : **200 Hz**
    4. Click the **OK** button





Model Validation

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

Analyze

- **To start the solution process**
  - Select the menu item ***Maxwell 3D > Analyze All***

Calculate Ohmic Losses

- **To Calculate Ohmic Losses in Stock**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity >OhmicLoss**
    2. Select **Input >Geometry >Volume > Stock**
    3. Select **Scalar >** **integrate**
    4. Select **Output > Eval**
  - The Ohmic losses in Stock volume are around **8.30 Watts**
  - Click **Done** to exit

```
Scl : 8.30232504385782
Scl : Integrate(Volume(Stock), Ohmic-Loss)
```

Close Maxwell

- **To Close Maxwell window**
  - Select the menu item ***File > Close Desktop***

Save

- **To Save Workbench Project**
  - Return to Workbench Project window
  - Select the menu item ***File > Save***
  - Save the file with the name **"Ex_12_2_Maxwell_Fluent_Coupling.wbpj"**

Enable Temperature Feedback

- This Feature is a Beta option in Workbench R14.
- **To Enable this Feature**
  - Goto ***Tools > Options > Appearance*** and Enable the **Beta Options**

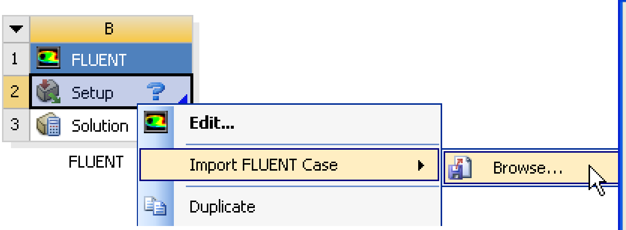
Create a FLUENT Component System

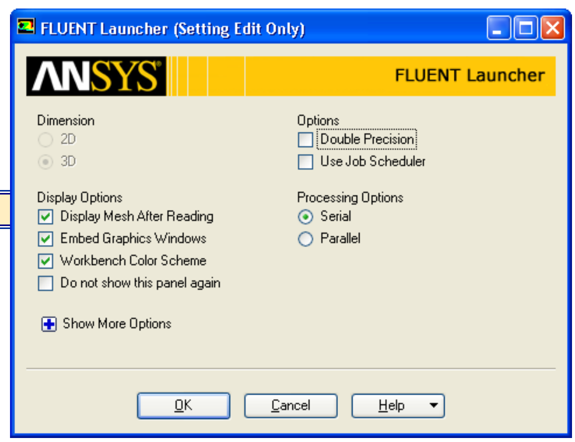
- **To Create a FLUENT Component System**
  - Expand the Toolbox for Component Systems and select a **Fluent** component system from it. Drag and drop the FLUENT System on Project page

Multiphysics Coupling - Maxwell Eddy Current to FLUENT

Load Mesh

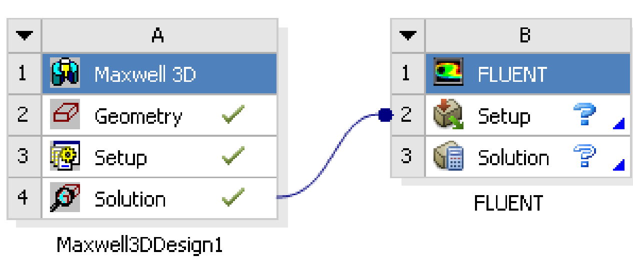
- A CFD mesh is needed for the FLUENT run. The purpose of this workshop is to show coupling steps and how it can be effectively used in mutiphysics simulations. Hence meshing steps are not shown. A mesh file has been provided in Tutorial Input files
- To Load Mesh
  - Right click on **Setup** tab of FLUENT Analysis system and select ***Import FLUENT Case > Browse***
  - Change the File type to FLUENT Mesh file
  - Browse to the location of the Tutorial Input Files and select the mesh file **"Ex_12_2_Maxwell_Fluent_Coupling.msh.gz"** and select **Open**
  - A FLUENT Launcher window will pop up as shown in below image. Press **OK** to it.
  - A FLUENT window will launch and mesh file will be imported into FLUENT





Build Links

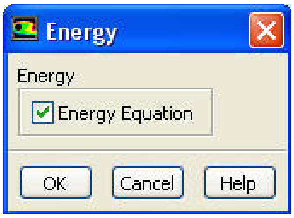
- To Create Links
  - Without closing FLUENT window, return to Workbench project page
  - Drag and drop **Solution** tab of Maxwell Analysis system onto the **Setup** tab of FLUENT system
  - Right click on **Solution** tab of Maxwell Analysis system and select **Update**
  - Right click on **Setup** tab of FLUENT System and select **Refresh**



FLUENT Setup

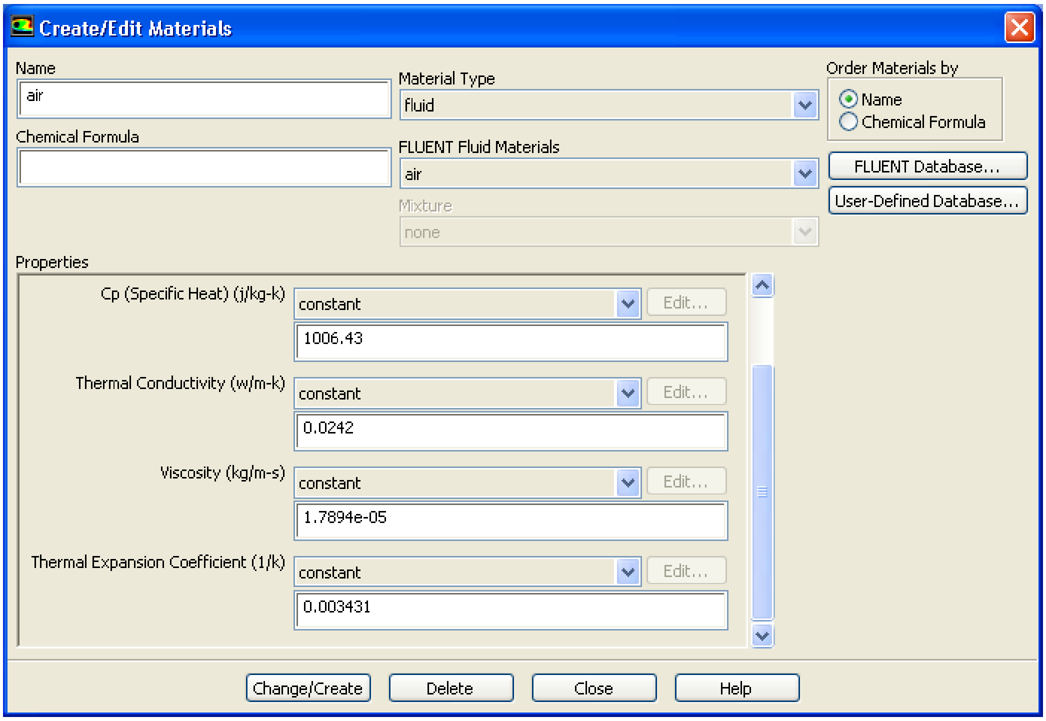
Set Thermal Solution

- Go to *Problem Setup > Models*, select **"Energy"** from the list and click **"Edit"**
- In Energy window,
  1. Energy Equation: ☑ **Checked**
  2. Press **"OK"** to exit

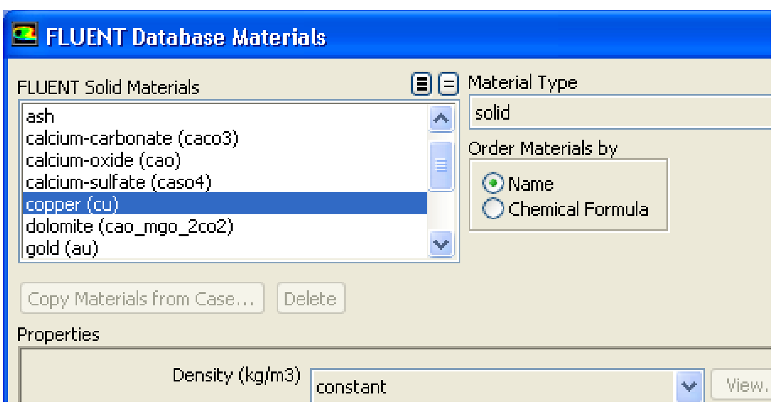


Set Fluid Material

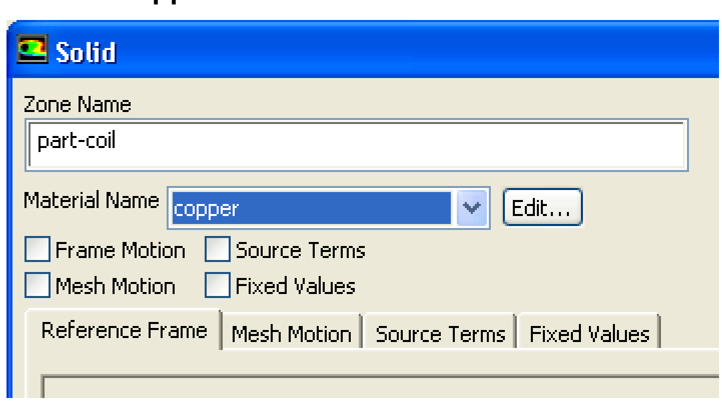
- Go to *Problem Setup > Materials >Fluid > air* and select **"Create/Edit"**
- Create/Edit Material window
  1. Change Density from Constant to **boussinesq**
  2. Set Density value to **1.225 kg/m^3**
  3. Set Thermal Expansion Coefficient to **0.00343 1/K**
  4. Select **"Change/Create"**
  5. Press **"Close"** to exit



- **Create Solid Material**
  - FLUENT applies **“Aluminum”** as default material for all solids.
  - We need to add copper material from material database
  - Go to ***Problem Setup > Materials***, select **“Solid”** from the Materials list and select **“Create/Edit”**
  - Create/Edit Material window
    - Select **FLUENT Datatbase**
    - In FLUENT Database Window
      1. Change the Material Type to **“Solid”**
      2. From FLUENT Solid Materials list, select **Copper(Cu)**
      3. Select **“Copy”** to add the material to project
      4. Select **“Close”** to exit FLUENT Material database
    - Select **Close** to exit Create/Edit Material window



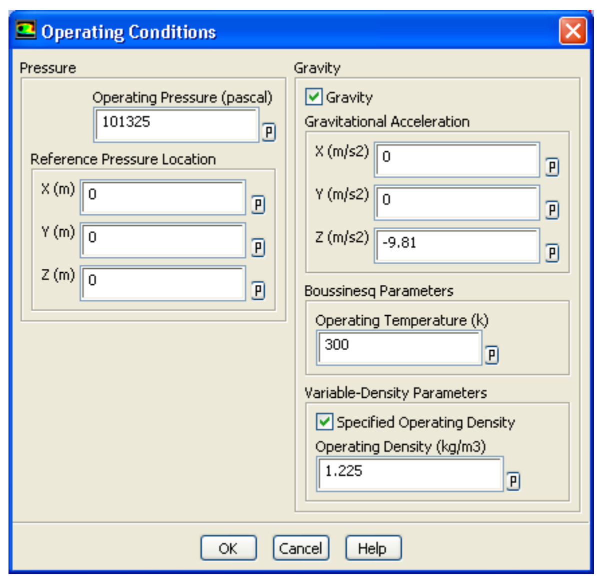
  - Go to *Problem **Setup > Cell Zone Conditions***, select the zone **“part-coil”** and select **“Edit”**
  - In Solid Window
    1. Change Material name to **“Copper”**
    2. Click **“OK”** to exit



Multiphysics Coupling – Maxwell Eddy Current to FLUENT

Set Operating Conditions

- **To Set Operating conditions**
  - Go to *Problem **Setup > Cell Zone Conditions*** and select the option **"Operating Conditions"**
  - In Operating Conditions Window
    1. Gravity: ☑ **Checked**
    2. Gravitational Acceleration:
       - X (m/s2) = 0
       - Y (m/s2) = 0
       - Z (m/s2) = **-9.81**
    3. Operating Temperature (K) = **300**
    4. Specified Operating Density: ☑ **Checked**
       - Operating Density (kg/m3) = **1.225**
    5. Select **"OK"** to exit



Initialize Solution

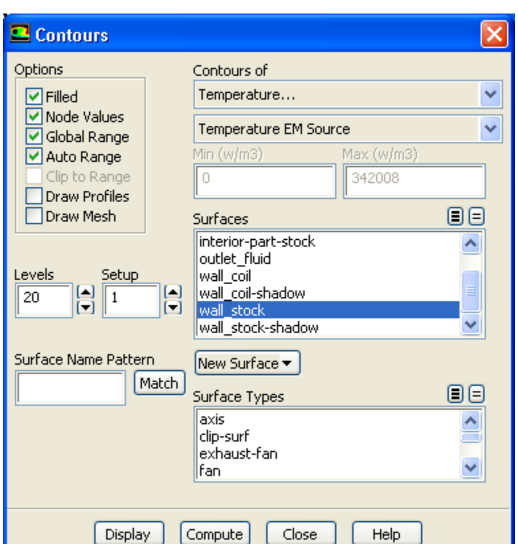
- **To Initialize the Solution**
  - Go to ***Solution > Solution Initialization***
  - Change the Initialization type to **Standard Initialization**
  - Press **Initialize**
  - Note: It is not necessary to initialize the solution before mapping the losses from Maxwell. If you initialize the solution after mapping losses, mapping will be redone automatically

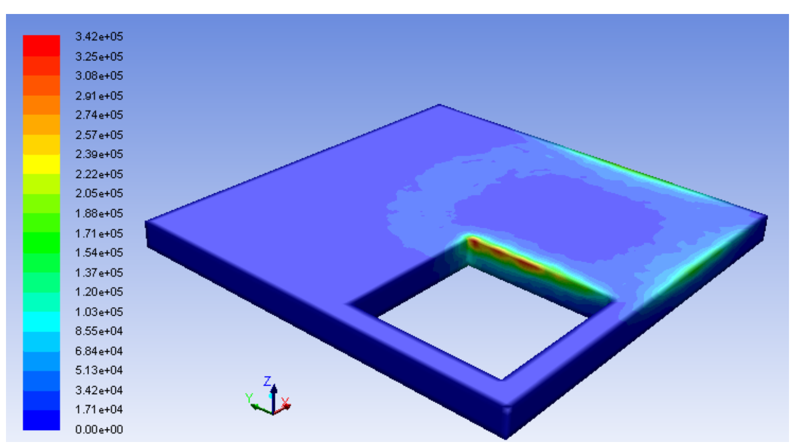
Map Losses from Maxwell

- **To Maps Losses from Maxwell**
  - Select the menu item *File > EM Mapping > Volumetric Energy Source*
  - In Maxwell Mapping window,
    - FLUENT Cell Zones: Select **part-stock**
    - Press **OK**
  - Mapped losses will be shown in transcript window. You can verify the losses mapped in FLUENT are same as reported in Maxwell

```
  Fluent generated 'Input_to_Ansoft.xml' file successfully

  Launching C:\Program Files\Ansoft\Maxwell15.0\Win64\maxwell.exe ....


Data assigned to selected Fluent solid zones ...

        Total loss on zone      7 is   8.302e+00 (Watt)

        Total loss is :  8.3023e+00 (Watt)
```

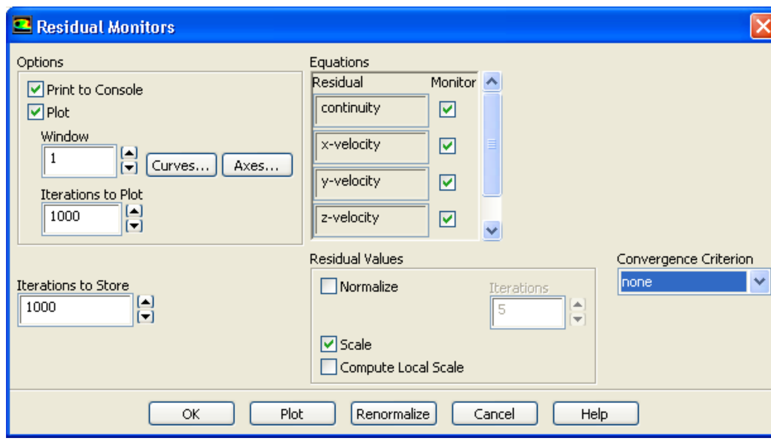
- **To Plot Losses**
  - Go to *Display > Graphics and Animations...* and double click **Contours**
  - In Contours window,
    - Filled: ☑ **Checked**
    - Contour of: **Teperature** > **Temperature EM Source**
    - Surfaces: **wall_stock**
    - Press **Display** and **Close**



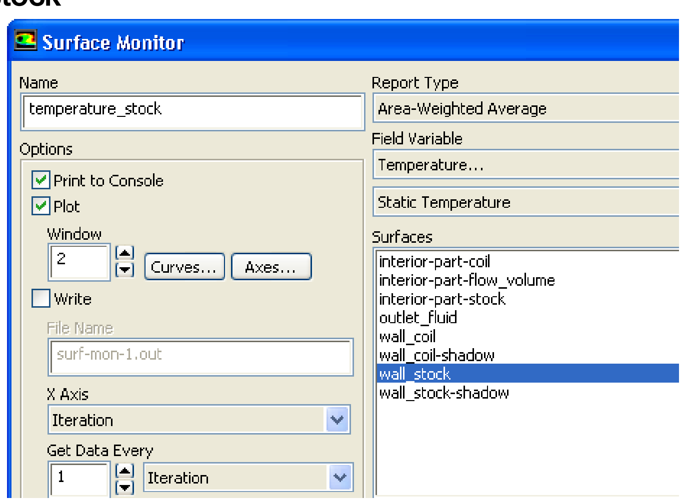


Set Residuals and Monitors

- **Turn off Residual Convergence**
  - It is advisable to turn off Residual check for first few iterations as Residuals might converge in first iteration itself
  - Go to ***Solution > Monitors***, select the option **Residuals-Print, Plot** and select **Edit**
  - In Residual Monitors window
    1. Change Convergence Criterion to **None**
    2. Press **OK** to exit



- **Set Surface Monitor**
  - Go to ***Solution > Monitors > Surface Monitors*** and select **Create**
  - In Surface Monitors Window
    1. Name: **temperature_stock**
    2. Report Type: **Area-Weighted Average**
    3. Field Variable: **Temperature, Static Temperature**
    4. Surfaces: Select **wall_stock**
    5. Options
       - Plot: ☑ **Checked**
    6. Press **OK** to exit

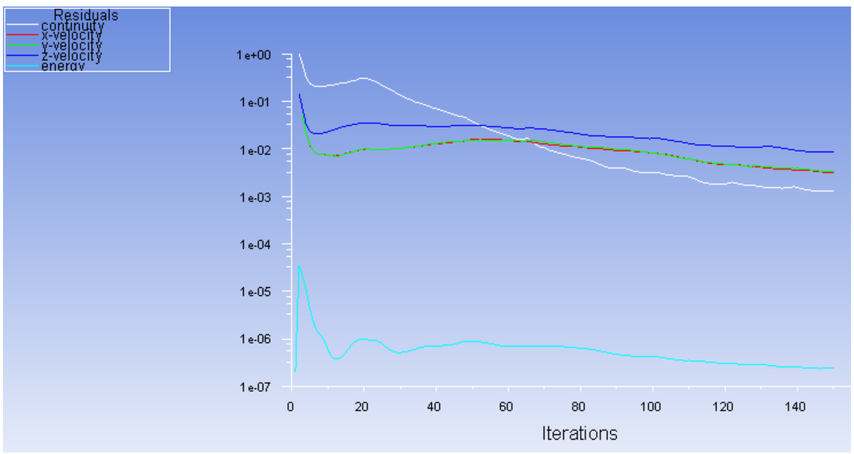


Multiphysics Coupling - Maxwell Eddy Current to FLUENT

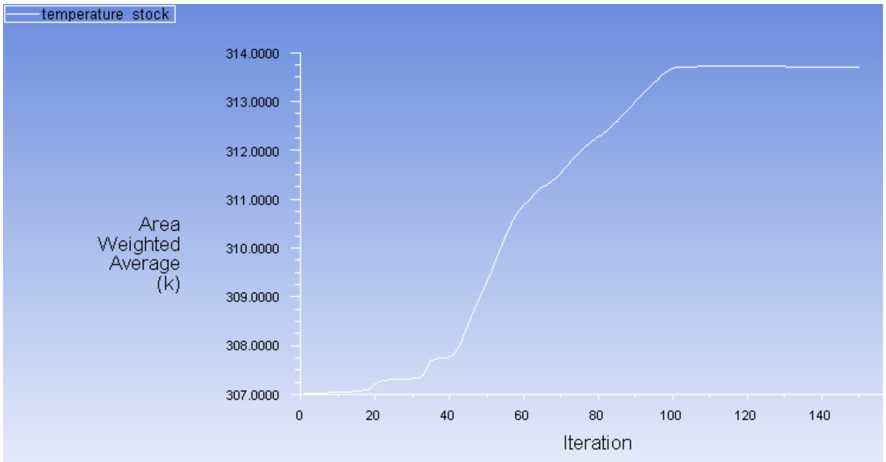
Run FLUENT Solution

- **To Run Fluent Solution**
  - Go to ***Solution > Run Calculation***
  - In Run Calculation Window
    1. Number of Iterations = **150**
    2. Select **Calculate** to run calculations
- FLUENT calculation will start
- Residual and Surface Monitor plots will be displayed on the screen
- Solution needs to be run until the monitor **temperature_stoc**k reaches a steady value

Residuals



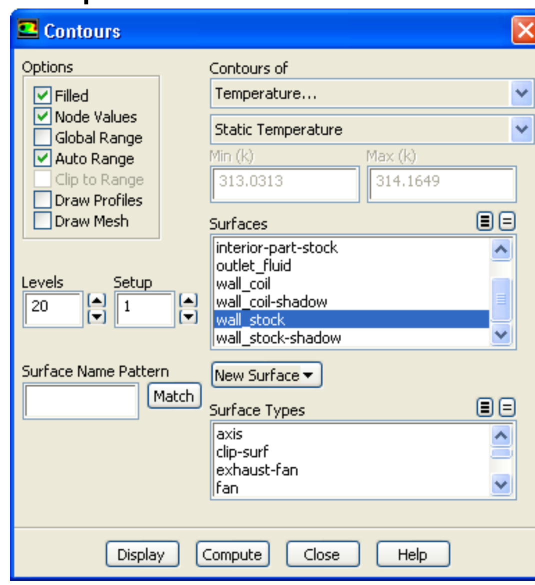
Temperature Monitor

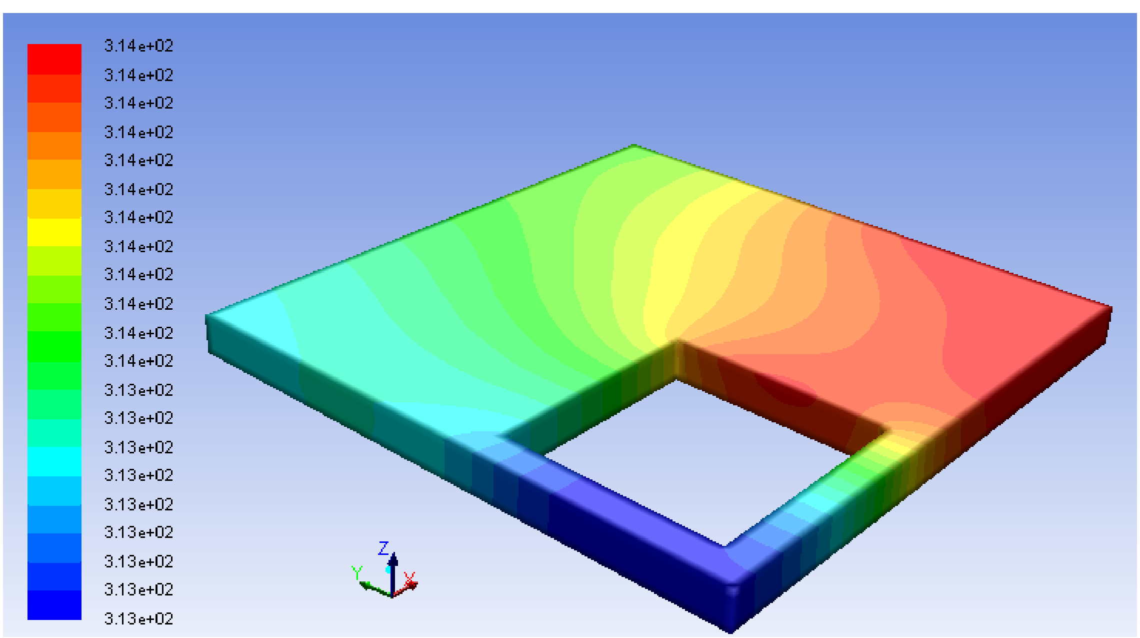


- **Temperature Feedback**
  - Once the FLUENT solution is completed, temperatures are sent to Maxwell automatically

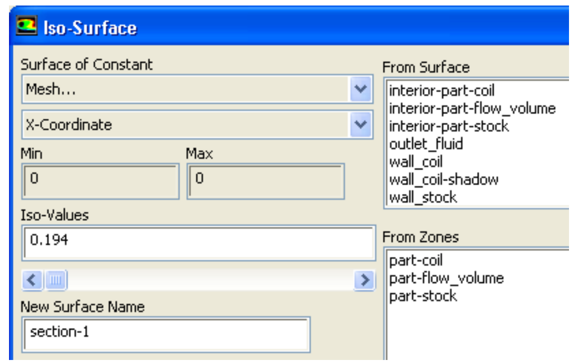
Postprocessing

- **Plot Temperature on Stock**
  - Go to ***Results > Graphics and Animations***, select **Contours** from the list and select **Set up**
  - In Contours Window
    1. Contours of : **Temperature > Static Temperature**
    2. Surfaces: Select **wall_stock**
    3. Options
       - Global Range: ☐ **Unchecked**
    4. Select **Display** to view contours

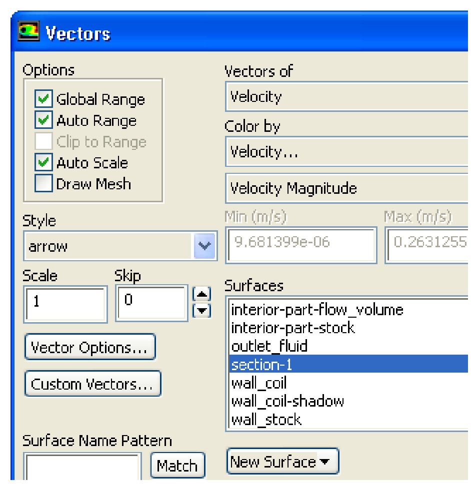


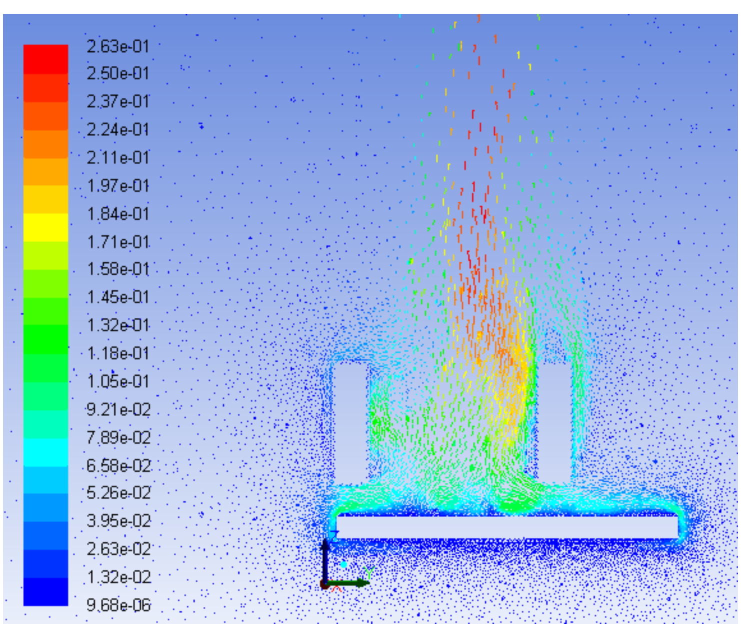


- **Create Iso-Surface**
  - From Menu items, select ***Surface > Iso-Surface***
  - In Iso-Surface window
    1. Surface of Constant : **Mesh > X-Coordinate**
    2. Iso-Values (m) : **0.194**
    3. New Surface Name : **section-1**
    4. Select **Create**
    5. Select **Close** to exit

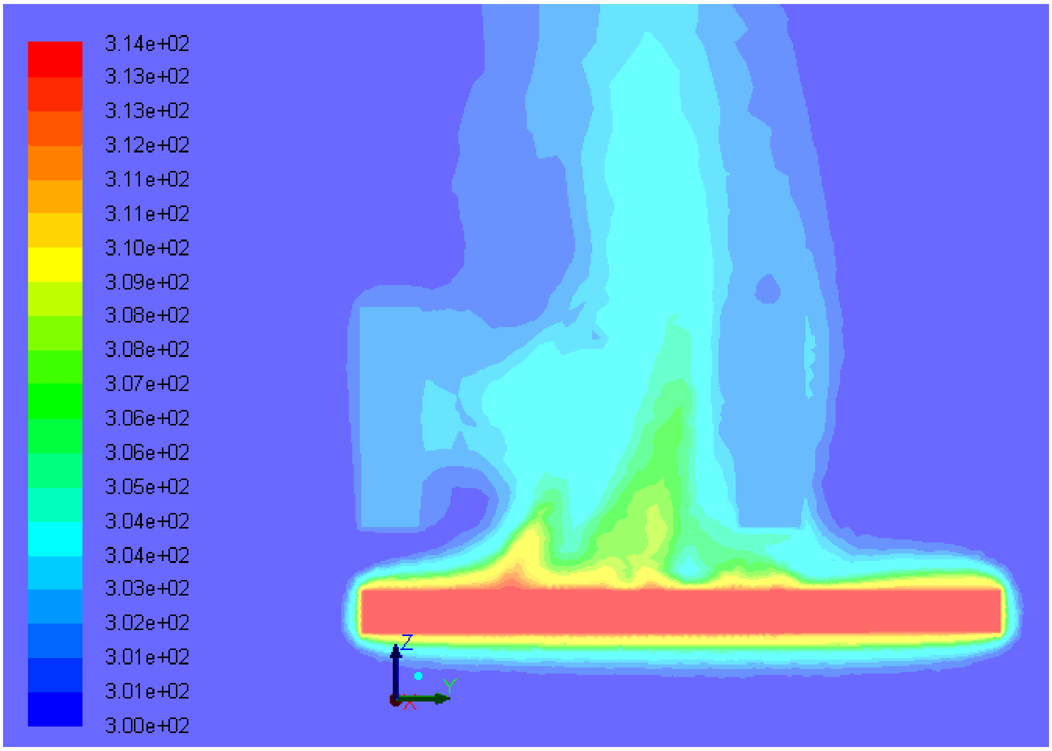


- **Plot Velocity Vectors**
  - Goto ***Results > Graphics and Animations***, select **Vectors** from the list and select **Set up**
  - In Vectors Window
    1. Surfaces: Select **section-1**
    2. Options
       - Global Range: ☐ **Unchecked**
    3. Select **Display** to view vectors





- **Temperature plot on section-1**
  - Using previous steps plot Static Temperature on section-1



Exit FLUENT

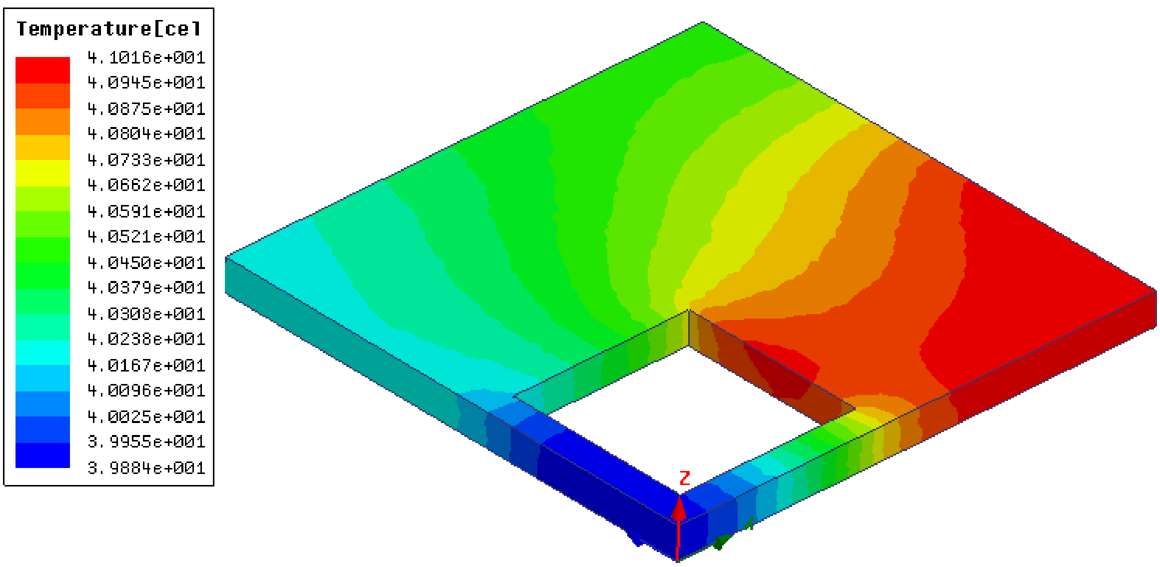
- **To Close FLUENT window**
  - Select the menu item ***File > Close FLUENT***

Maxwell Solution with Temperature Feedback

- **Run Maxwell with Temperature Feedback**
  - Double click on Solution tab of Maxwell system to launch Maxwell
  - In Maxwell, select the menu item ***Maxwell 3D > Analyze All***

Multiphysics Coupling – Maxwell Eddy Current to FLUENT

- **Plot Temperature in Maxwell**
  - Select the object **Stock** history tree
  - Select the menu item ***Maxwell 3D > Fields > Fields > Other > Temperature***
  - In Create Field Plot window,
    - Plot on Surface only: ☑ **Checked**
    - Press **Done**
  - Maximum Temperature on Stock reported by FLUENT was **314.16 °K** which comes out to be close to **41 °C** which is reported by Maxwell



- **Losses in Stock with Temperature Feedback**
  - Select the menu item ***Maxwell 3D > Fields > Calculator***
    1. Select **Input > Quantity >OhmicLoss**
    2. Select **Input >Geometry >Volume > Stock**
    3. Select **Scalar >** **integrate**
    4. Select **Output > Eval**

  - The losses calculated by Maxwell are **8.9 watts.** This can further be sent to FLUENT to calculate refined temperature
  - The loop can be continued until there is no significant change in temperature calculated by FLUENT or losses computed by Maxwell

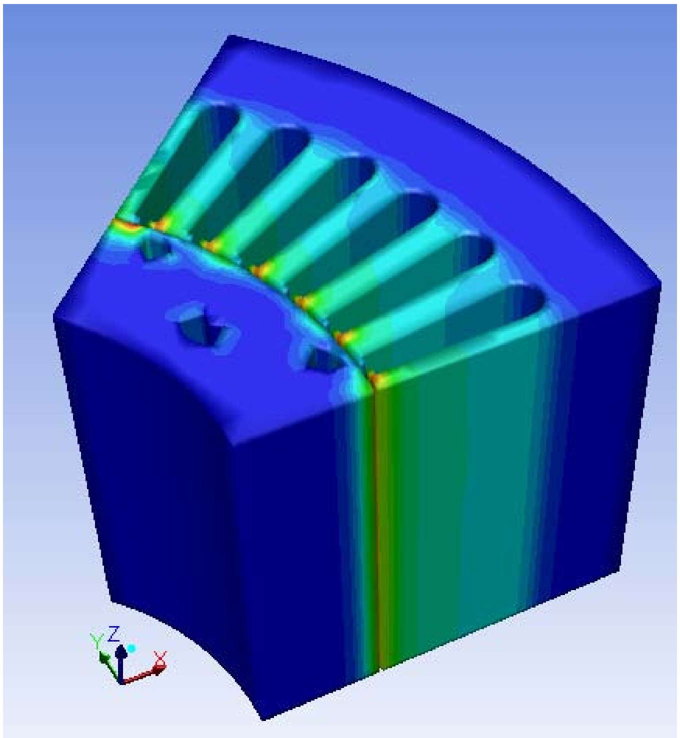
Multiphysics Coupling – Maxwell Transient to FLUENT

Maxwell Transient to FLUENT Steady State Coupling

- This workshop is intended to show how loss data from a Maxwell transient solution can be transferred to FLUENT steady state solver for temperature calculation
- The example used is a Prius motor. The motor is simulated in Maxwell 2D for electromagnetic losses and the thermal simulation is performed inside FLUENT 3D
- The Ohmic losses in the magnets and core losses in the stator and rotor are transferred from Maxwell 2D to a 3D mesh in FLUENT for temperature calculation
- Losses applied in FLUENT are time averaged values
- This workshop shows only steps related to transferring losses from Maxwell to FLUENT and does not contain details about Maxwell or FLUENT. Please refer to FLUENT Users' Guide for information on FLUENT Setup

Prerequisites

- Maxwell V15 and FLUENT V14 need to be installed in order to carry out this workshop
- ANSYS Workbench needs to be installed as a coupling interface



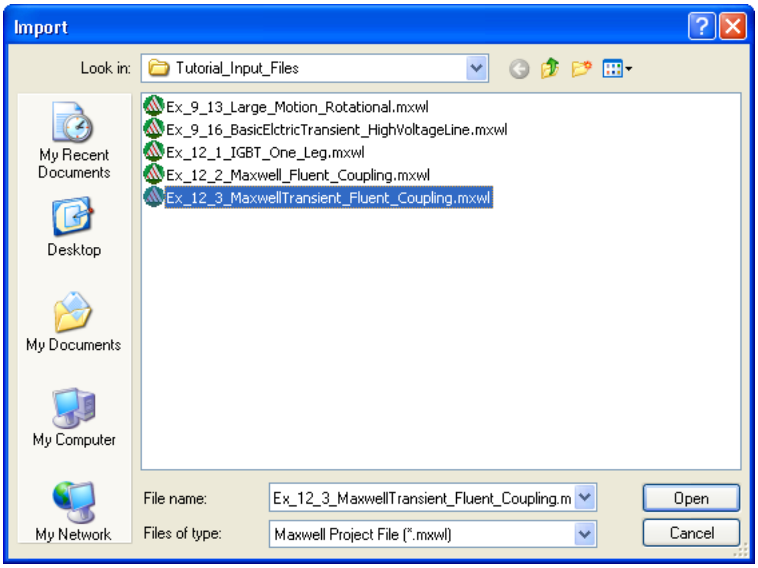
Multiphysics Coupling – Maxwell Transient to FLUENT

Launch ANSYS Workbench

- **To Launch Workbench**
  - Select the Microsoft **Start** button , select **All Programes > ANSYS 14.0 > Workbench 14.0**

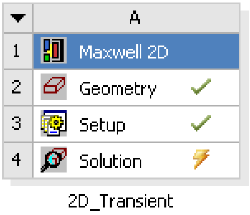
Import Maxwell Project File

- Since we are using an existing Maxwell project, we will import it into Workbench. Users can create a new Maxwell Analysis System to setup a new problem.
- **To Import Maxwell File**
  - Select the menu item ***File > Import***
  - Change the file type to **Maxwell Project File (*.mxwl)**
  - Browse to the location where tutorial input files are saved
  - Select the file **"Ex_12_3_MaxwellTransient_Fluent_Coupling.mxwl"** and **Open** it.



Launch Maxwell

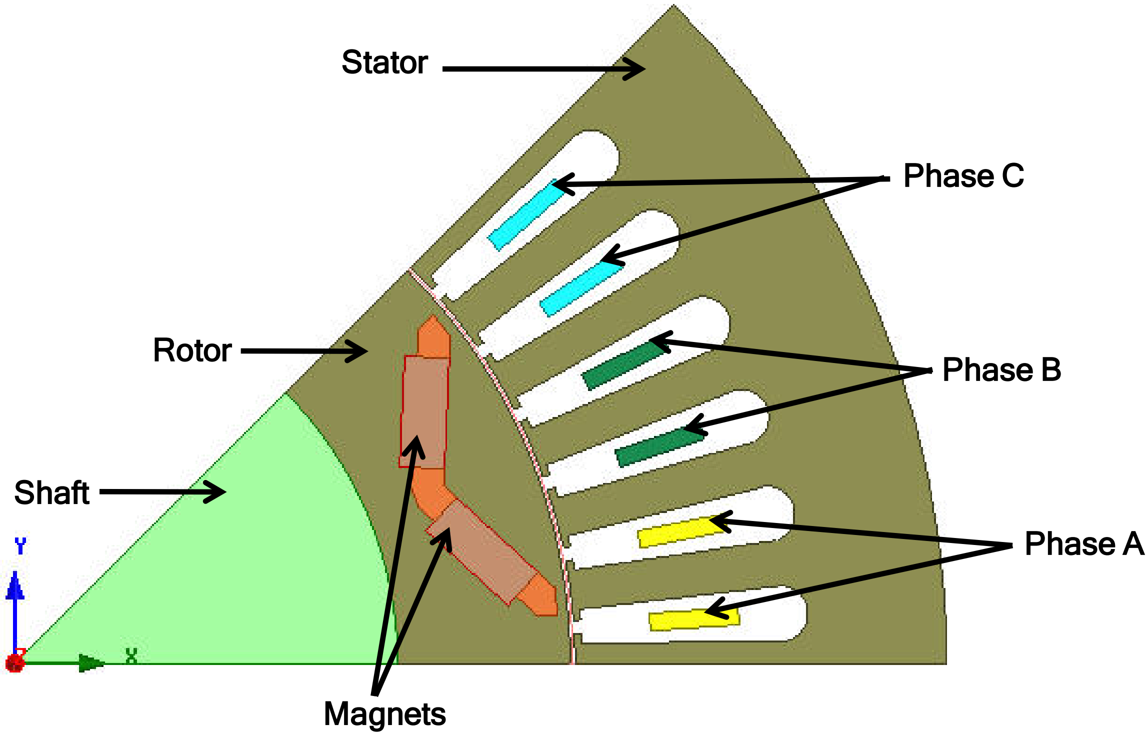
- A Maxwell analysis system will be created as shown below.
- **To Lunch Maxwell**
  - Double click on the solution tab of Maxwell analysis system.



Multiphysics Coupling – Maxwell Transient to FLUENT

About Input File

- Input file contains a 2D geometry of a Prius motor
- The setup of this motor has already been done
- To obtain more details about the setup please refer to Example 11.1 of Maxwell 2D Tutorials



Model Validation

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

Analyze

- **To start the solution process**
  - Select the menu item ***Maxwell 3D > Analyze All***

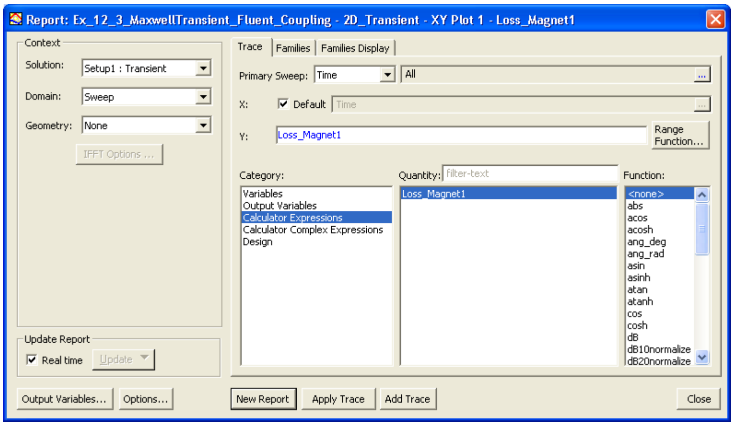
Plot Losses

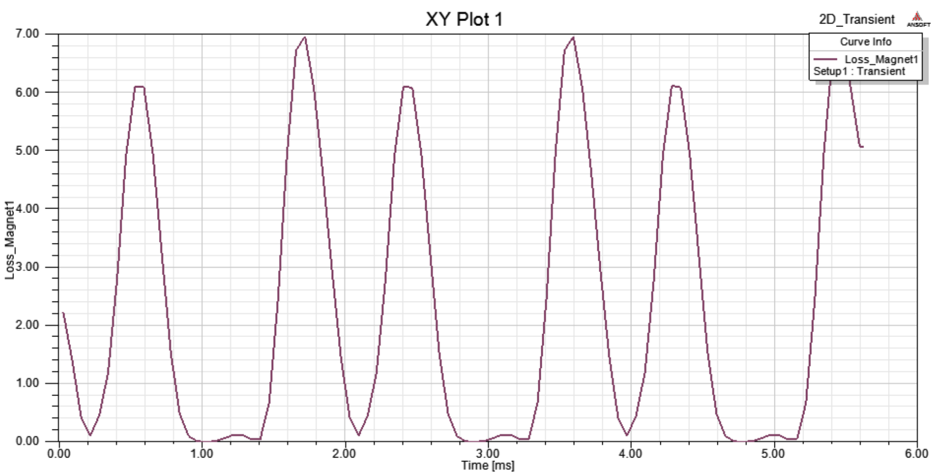
Create Parameter for Total Losses

- Select the menu item *Maxwell 2D > Fields > Calculator*
  1. Select **Input > Quantity >TotalLoss**
  2. Select **Input >Geometry >Surface > Magnet1**
  3. Select **Scalar >** **integrate**
  4. Select **Add**
  5. Specify the name of the expression as **Loss_Magnet1**
  6. Press **Done**

Plot Loss Vs Time

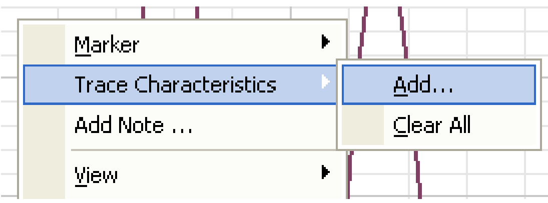
- Select the menu item *Maxwell 2D > Results > Create Field Reports > Rectangular Plot*
- In Report window
  1. Category: **Calculator Expressions**
  2. Quantity: **Loss_Magnet1**
  3. Select **New Report**
  4. Press **Close**

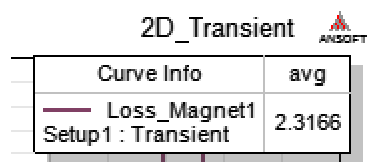




Multiphysics Coupling – Maxwell Transient to FLUENT

- **Calculate Time Average Losses**
  - Right click on the plot and select the option ***Trace Characteristics > Add***
  - In Add Trace Characteristics window,
    1. Category: **Math**
    2. Function: **avg**
    3. Range: **Full**
    4. Select **Add** and **Done**





  - The value displayed is around **2.3166 watts**
  - It shows time averaged Total losses
  - Since this is a 2D case, these losses are in watts per meter
  - We need to multiply this value with length of magnets in order to calculate Losses

- **Do the same for the stator and rotor.**

Close Maxwell

- **To Close Maxwell window**
  - Select the menu item ***File > Close Desktop***

Save

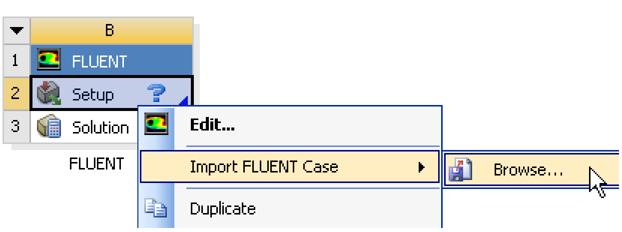
- **To Save Workbench Project**
  - Return to Workbench Project window
  - Select the menu item ***File > Save***
  - Save the file with the name **"Ex_12_3_MaxwellTransient_Fluent_Coupling.wbpj"**

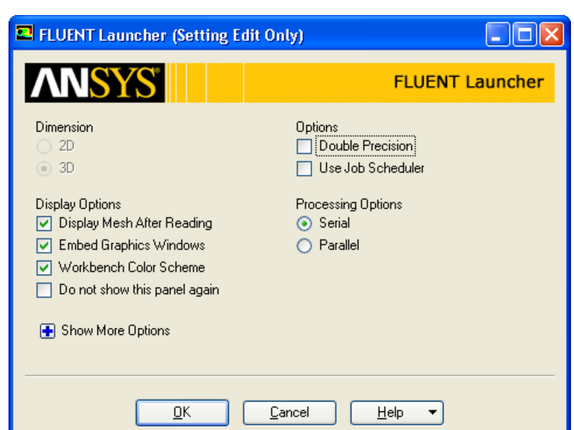
Create a FLUENT Component System

- **To Create a FLUENT Component System**
  - Expand the Toolbox for Component Systems and select a **Fluent** component system from it. Drag and drop the FLUENT System on Project page

Load Mesh

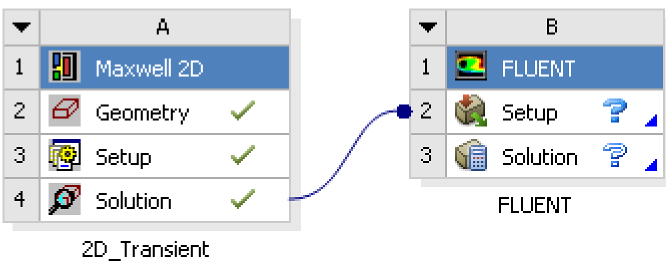
- A CFD mesh is needed for the FLUENT run. The purpose of this workshop is to show coupling steps and how it can be effectively used in mutiphysics simulations. Hence meshing steps are not shown. A mesh file has been provided in Tutorial Input files
- The mesh file contains only solid mesh. Users can bring Solid+Fluid mesh as well. The steps will be same as specified in the workshop.
- To Learn about meshing, please refer ANSYS Meshing Training manual
- **To Load Mesh**
  - Right click on **Setup** tab of FLUENT Analysis system and select ***Import FLUENT Case > Browse***
  - Change the File type to FLUENT Mesh file
  - Browse to the location of the Tutorial Input Files and select the mesh file **"Ex_12_3_MaxwellTransient_Fluent_Coupling.msh.gz"** and select **Open**
  - A FLUENT Launcher window will pop up as shown in below image. Press **OK** to it.
  - A FLUENT window will launch and mesh file will be imported into FLUENT





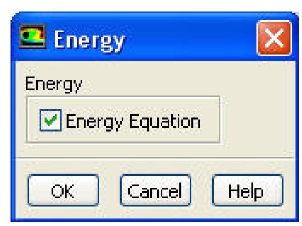
Build Links

- **To Create Links**
  - Without closing FLUENT window, return to Workbench project page
  - Drag and drop **Solution** tab of Maxwell Analysis system onto the **Setup** tab of FLUENT system
  - Right click on **Solution** tab of Maxwell Analysis system and select **Update**
  - Right click on **Setup** tab of FLUENT System and select **Refresh**



Set Thermal Solution

- **To Set Thermal Solution**
  - Go to ***Problem Setup > Models***, select **"Energy"** from the list and click **"Edit"**
  - In Energy window,
    1. Energy Equation: ☑ **Checked**
    2. Press **"OK"** to exit



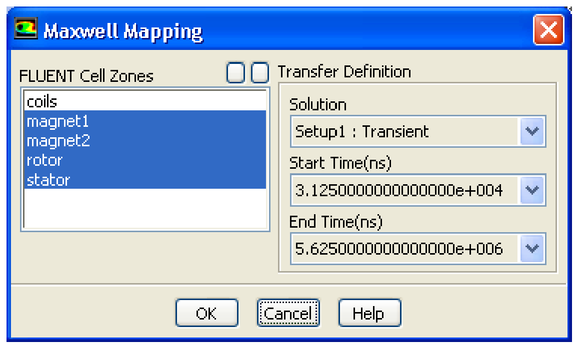
- **Note:** Thermal Solution needs to be **ON** for data mapping since mapped quantities will be Heat Losses. If the Energy is not ON in FLUENT, it will be turned ON automatically while mapping the data.

Initialize Solution

- **To Initialize the Solution**
  - Go to ***Solution > Solution Initialization***
  - Change the Initialization type to **Standard Initialization**
  - Press **Initialize**
  - **Note:** It is not necessary to initialize the solution before mapping the losses from Maxwell. If you initialize the solution after mapping losses, mapping will be redone automatically

Map Losses from Maxwell

- **To Maps Losses from Maxwell**
  - Select the menu item *File > EM Mapping > Volumetric Energy Source*
  - In Maxwell Mapping window,
    - FLUENT Cell Zones: Press **Ctrl** and select the zones **magnet1, magnet2, rotor** and **stator**
    - Start: **3.125e4**
    - Stop: **5.625e6**
    - Press **OK**

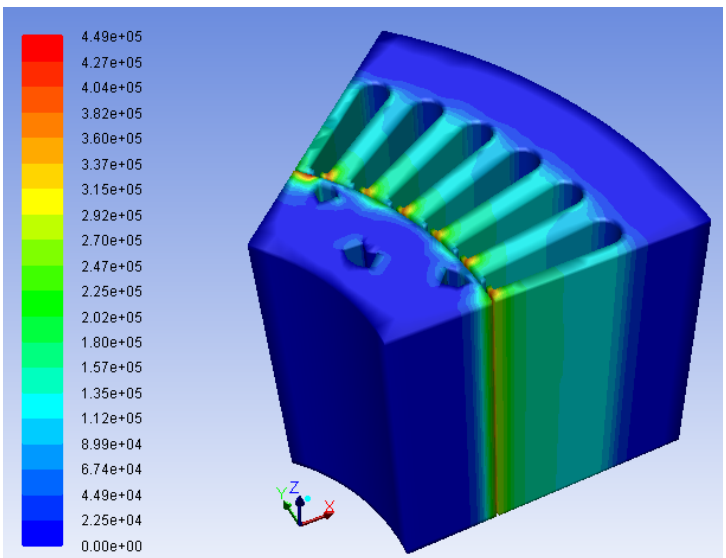


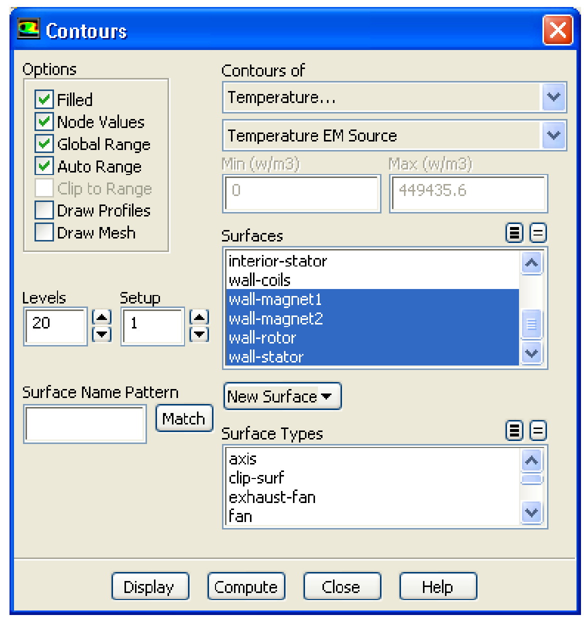
- **Mapped Losses**
- After data mapping, FLUENT will display the losses mapped to selected zones
- The losses reported for magnet1 and magnet2 are around **0.193 watts** for both

```
 Fluent generated 'Input_to_Ansoft.xml' file successfully

 Launching C:\Program Files\Ansoft\Maxwell15.0\Win64\maxwell.exe ....


Data assigned to selected Fluent solid zones ...

        Total loss on zone    12 is  1.928e-01 (Watt)
        Total loss on zone    13 is  1.938e-01 (Watt)
        Total loss on zone    11 is  2.345e+00 (Watt)
        Total loss on zone    10 is  1.617e+01 (Watt)

        Total loss is :  1.8903e+01 (Watt)
```

- If losses (per unit length value) calculated in Maxwell are multiplied with the length of the magnet, it should be close to the mapped losses reported by FLUENT
- Length of the magnets is **0.08382 meters** and losses calculated in Maxwell for magent1 is around **2.3166 watts** per meter
- This gives a value of **0.194 watts** which is close to the value reported in FLUENT
- It can be verified that losses for other regions are also correct.

- ## Plot Mapped Losses
  - **To Plot Losses**
    - Go to *Display > Graphics and Animations…* and double click **Contours**
    - In Contours window,
      - Filled: ☑ **Checked**
      - Contour of: **Teperature** > **Temperature EM Source**
      - Surfaces: **wall-magnet1, wall-magnet2, wall-rotor** and **wall-stator**
      - Press **Display** and **Close**

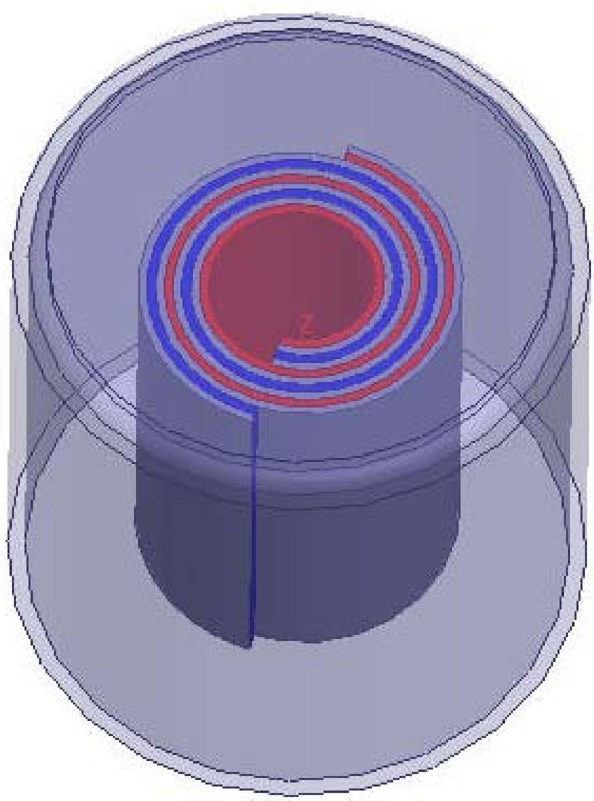




- ## The workshop focuses on the mapping process. For temperature calculation, the following is also important but not demonstrated in the workshop
  - A mesh for the fluid surrounding the solid regions is needed for a CFD calculation.
  - Winding losses are not mapped. Winding losses are actually the largest losses among all and must be supplied for realistic temperature calculation. For stranded windings, there is no spatial winding loss distribution and a constant value can be specified in FLUENT as a heat source term.

Compact Parallel Plate Capacitor

- This example uses the Electrostatic solver in the ANSYS Maxwell 3D Design Environment.
- It models an electrostatic parallel plate capacitor. The objective of this simulation is to determine the nominal capacitance and the static electrical stress. The electric fields will be visualized and the results coupled to ANSYS Mechanical for a structural analysis.
- The model consists of two parallel plates separated by a dielectric, coiled into a cylindrical package. The package has a diameter of 100mm and a height of 100mm.
- The application note covers the drawing of the model as well as the assignment of material properties and sources. The default Neumann boundaries on the outer region edges are adequate. The excitation consists of simple voltage sources. After the final solution is completed, plots are generated for voltage contours and electric field intensities.



Multiphysics Coupling – Compact Parallel Plate Capacitor

ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 3D Solid Modeling
    - Primitives: **User Defined Primitives, Cylinder**
    - Booleans: **Subtract**
  - Boundaries/Excitations
    - **Voltage**
  - Analysis
    - **Electrostatic**
    - **Static Structural**
  - Results
    - **Voltage**
    - **Stress**
  - Field Overlays:
    - **Voltage**
    - **Stress Distribution**

Multiphysics Coupling – Compact Parallel Plate Capacitor

Launching Maxwell

- **To access Maxwell:**
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

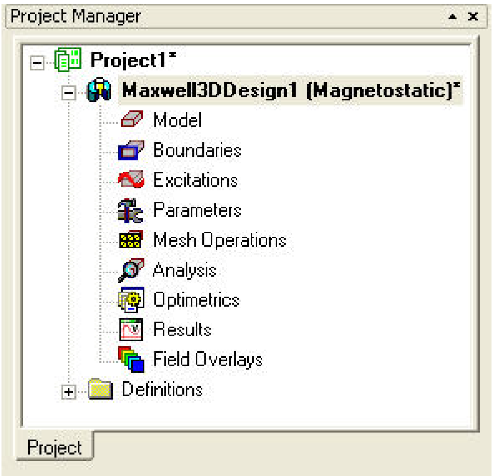
Setting Tool Options

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

Multiphysics Coupling – Compact Parallel Plate Capacitor

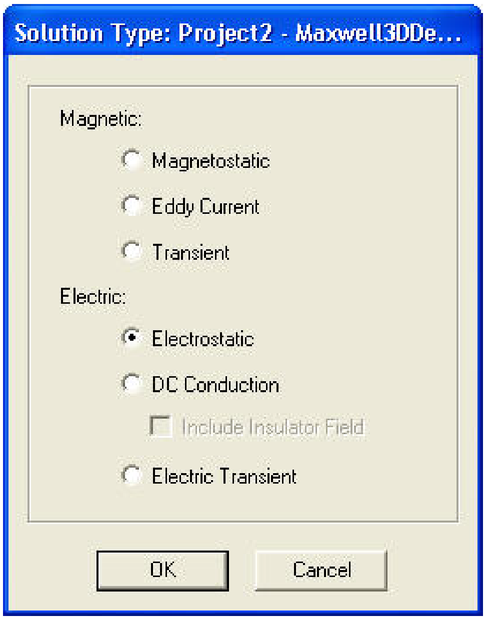
Opening a New Project

- **To open a new project:**
  - After launching Maxwell, a project will be automatically created. You can also create a new project using below options.
    1. In an Maxwell window, click the 🗋 On the Standard toolbar, or select the menu item ***File > New***.
  - Select the menu item ***Project > Insert Maxwell 3D Design***, or click on the icon



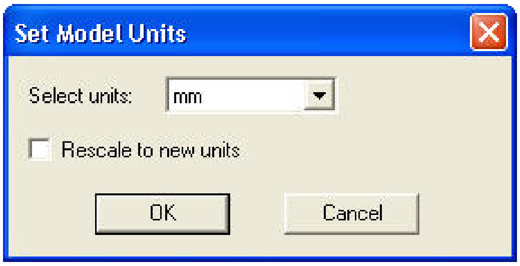
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 3D > Solution Type***
  - Solution Type Window:
    1. Choose **Electric > Electrostatic**
    2. Click the **OK** button



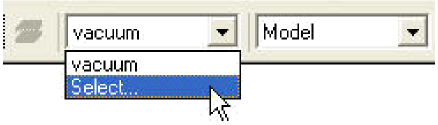
Set Model Units

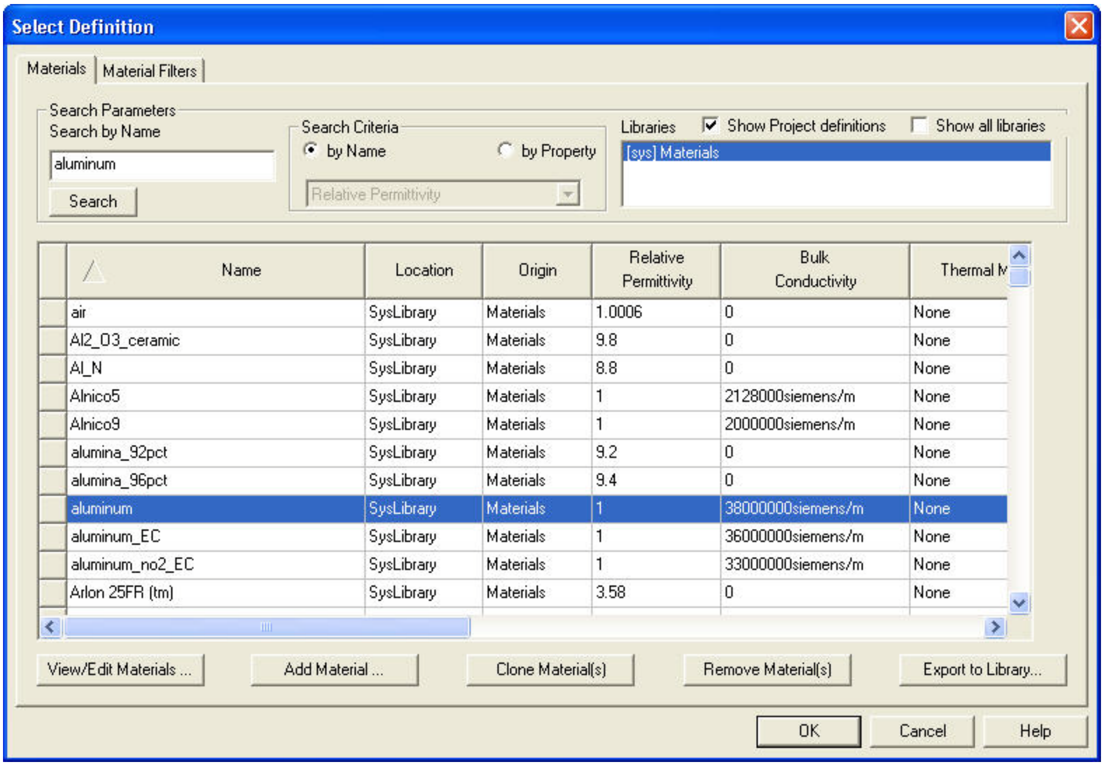
- **To Set the units:**
  - Select the menu item ***Modeler > Units***
  - Set Model Units:
    1. Select Units: **mm**
    2. Click the **OK** button



Set Default Material

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

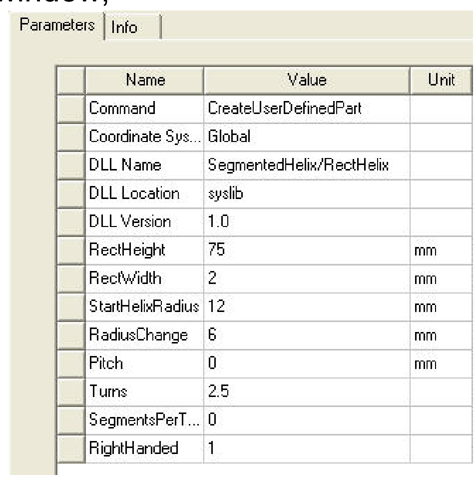




Create Coiled Electrodes

Create First Electrode

- Select the menu item ***Draw > User Defined Primitive > SysLib > SegmentedHelix > RectHelix***
- In User Defined Primitive Operation window,
  1. RectHeight: **75mm**
  2. RectWidth: **2mm**
  3. StartHelixRadius: **12mm**
  4. RadiusChange: **6mm**
  5. Pitch: **0 mm**
  6. Turns: **2.5**
  7. SegmentsPerTurn: **0**
  8. RightHanded: **1**



Parameters | Info

Name	Value	Unit
Command	CreateUserDefinedPart	
Coordinate Sys...	Global	
DLL Name	SegmentedHelix/RectHelix	
DLL Location	syslib	
DLL Version	1.0	
RectHeight	75	mm
RectWidth	2	mm
StartHelixRadius	12	mm
RadiusChange	6	mm
Pitch	0	mm
Turns	2.5	
SegmentsPerT...	0	
RightHanded	1	

To Change Attributes

- Select the resulting object from the history tree and goto Properties window
  1. Change the name of the object to **Electrode_A**
  2. Change the color of the object to **Red**

Create Second Electrode

- Using same steps as above create object with below parameters
  1. RectHeight: **75mm**
  2. RectWidth: **2mm**
  3. StartHelixRadius: **15mm**
  4. RadiusChange: **6mm**
  5. Pitch: **0 mm**
  6. Turns: **2**
  7. SegmentsPerTurn: **0**
  8. RightHanded: **1**

To Change Attributes

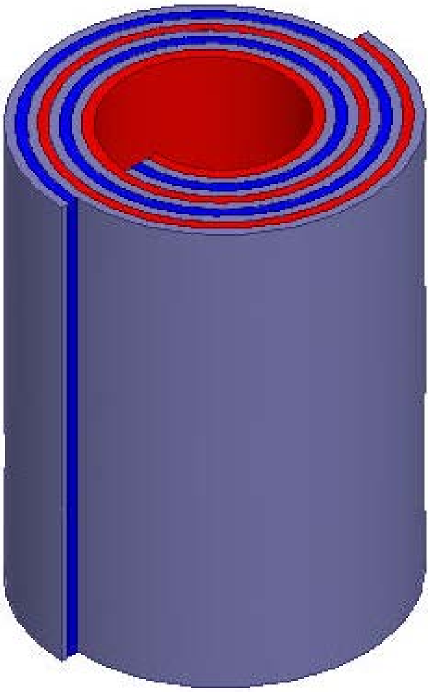
- Select the resulting object from the history tree and goto Properties window
  1. Change the name of the object to **Electrode_B**
  2. Change the color of the object to **Blue**

Set Default Material

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

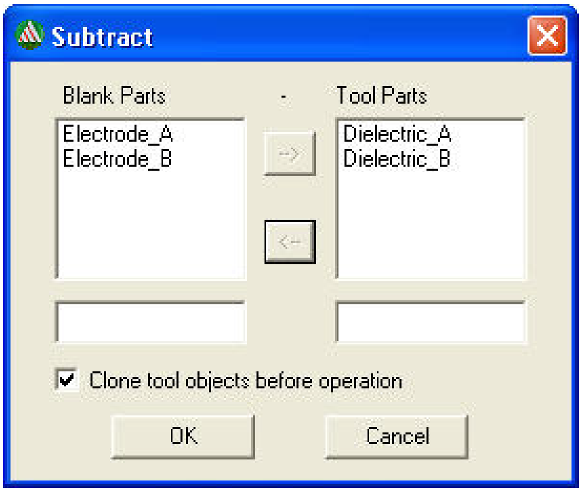
Create Coiled Dielectrics

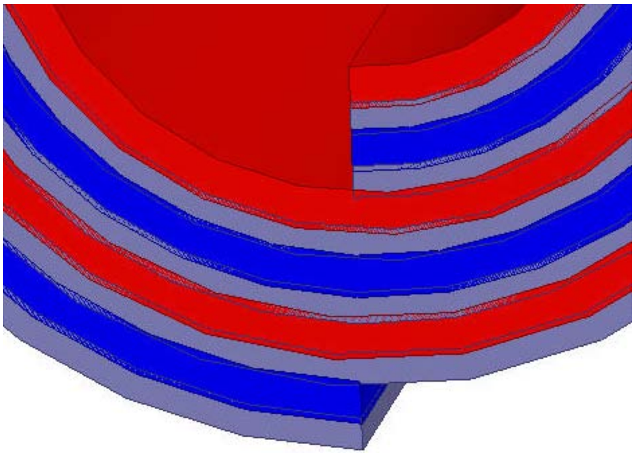
- **Create Dielectric Layers**
  - Using same steps as for electrodes, create first layer of dielectric with below parameters
  - **Dielectric_A**
    1. RectHeight: **75mm**
    2. RectWidth: **1.5mm**
    3. StartHelixRadius: **13.5mm**
    4. RadiusChange: **6mm**
    5. Pitch: **0 mm**
    6. Turns: **2.5**
    7. SegmentsPerTurn: **0**
    8. RightHanded: **1**
  - **Dielectric_B**
    1. RectHeight: **75mm**
    2. RectWidth: **1.5mm**
    3. StartHelixRadius: **16.5mm**
    4. RadiusChange: **6mm**
    5. Pitch: **0 mm**
    6. Turns: **2**
    7. SegmentsPerTurn: **0**
    8. RightHanded: **1**

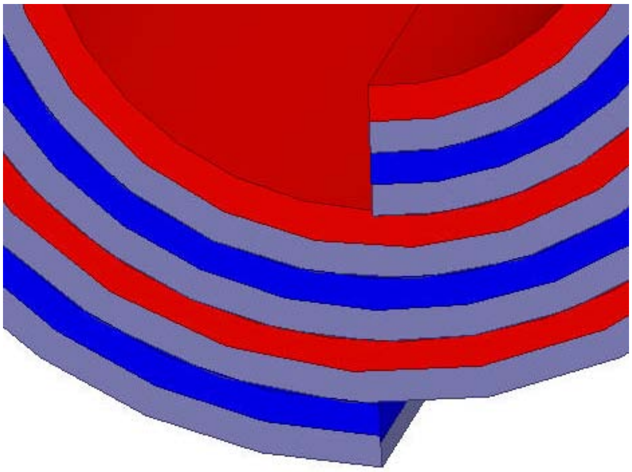


Fit the layers together

- The layers were purposely drawn with some overlap to ensure a snug fit. Subtract the overlapping areas to finish the layered plates
- **To Subtract Objects**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    - Blank Parts: **Electrode_A, Electrode_B**
    - Tool Parts: **Dielectric_A, Dielectric_B**
    - Clone tool objects before operation: ☑ **Checked**
    - Press **OK**

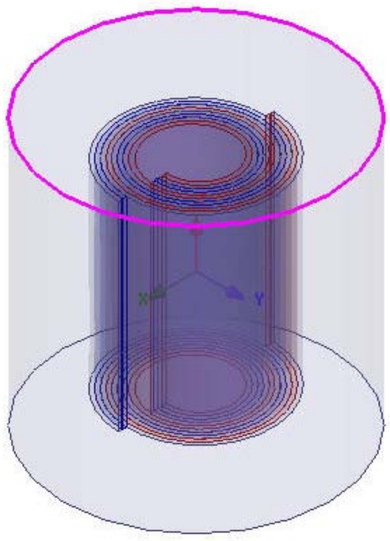


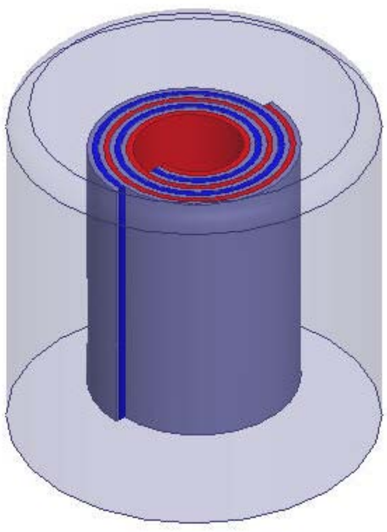




Create Container

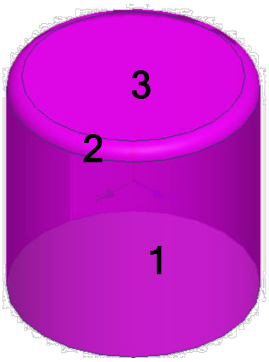
- **Create Cylinder**
  - Select the menu item ***Draw > Cylinder***
    1. Using Coordinate entry field, enter the center of base
       - X: **0,** Y: **0,** Z: **-50,** Press the **Enter** key
    2. Using Coordinate entry field, enter the radius and height
       - dX: **50,** dY: **0,** dZ: **100,** Press the **Enter** key
    3. Press **OK**
- **To Change Attributes**
  - Select the resulting object from the history tree and goto Properties window
    1. Change the name of the object to **Can**
    2. Change the material of the object to **Aluminum**
- **To Create Fillets**
  - Select the menu item ***Edit > Select > Edges*** or press **"E"** from the keyboard
  - Select the top edge of the cylinder as shown in image
  - Select the menu item ***Modeler > Fillet***
  - In Fillet Properties window,
    - Fillet Radius: **6mm**
    - Press **OK**
  - Select the menu item ***Edit > Select > Objects*** or press **"O"** from the keyboard to change selection back to objects





Create the Inner Fillings

- **Copy the object Can**
  - Select the object **Can** from the history tree
  - Select the menu item ***Edit > Copy*** followed by ***Edit > Paste***
- **Hide All Other Objects**
  - Select the menu item ***View > Visibility> Active View Visibility***
  - In Active View Visibility window,
    1. Check the visibility for **Can1** and uncheck for others
    2. Press **Done**
- **Create Inward**
  - Select the menu item ***Edit > Select > Faces*** or press **"F"** from keyboard
  - Press **Ctrl** and select the three faces of **Can1** as shown in image
  - Select the menu item ***Modeler > Surface > Move Faces > Along Normal***
    1. Set Distance to **-3mm**
    2. Press **OK**



- **To Change Attributes**
  - Select the object **Can1** from the history tree and goto Properties window
    1. Change the name of the object to **Innards**
    2. Change the material of the object to **Teflon**
  - Select the menu item ***View > Visibility > Show All > Active View***
- **Subtract Objects**
  - Press **Ctrl** and select the objects **Can** and **Innards**
  - Select the menu item ***Modeler > Boolean > Subtract***
  - In Subtract window,
    - Blank Parts: **Can**
    - Tool Parts: **Innards**
    - Clone tool objects before operation: ☑ **Checked**
    - Press **OK**

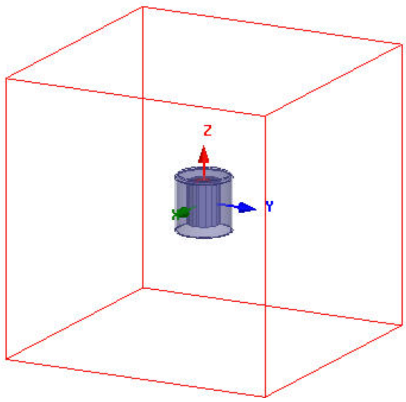
Multiphysics Coupling – Compact Parallel Plate Capacitor

Specify Excitations

- **Assign Excitations for Electrode_A**
  - Select the object **Electrode_A** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Voltage1**
    2. Value: **25 V**
    3. Press **OK**
- **Assign Excitations for Electrode_B**
  - Select the object **Electrode_B** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Voltage***
  - In Voltage Excitation window,
    1. Name: **Voltage2**
    2. Value: **0 V**
    3. Press **OK**
- **Assign Excitations for Can**
  - Select the object **Can** from the history tree
  - Select the menu item ***Maxwell 3D > Excitations > Assign > Floating***
  - In Voltage Excitation window,
    1. Name: **Floating1**
    2. Value: **0 C**
    3. Press **OK**

Create Solution Region

- **To Create Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Padding Data: **Pad all directions similarly**
    2. Padding type: **Percentage Offset**
    3. Value: **200**
    4. Press **OK**



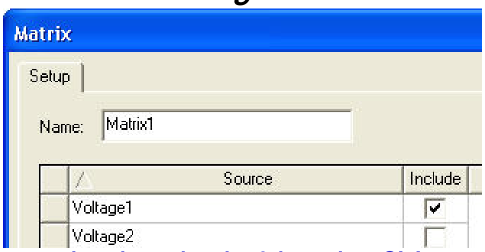
Multiphysics Coupling – Compact Parallel Plate Capacitor

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item ***Maxwell 3D > Analysis Setup > Add Solution Setup***
  - Solution Setup Window:
    1. Click the **OK** button to accept all default settings.

Set Capacitance calculation

- **To calculate Capacitance**
  - Select the menu item ***Maxwell 3D > Parameters > Assign > Matrix***
  - In Matrix window,
    - **Voltage1**
      - Include: ☑ **Checked**
    - Press **OK**



  - **Note:** Leaving Voltage2 unselected constrains it to be held at the 0V applied to it, making it the reference ground for our two plate capacitor

Save Project

- **To save the project:**
  - Select the menu item ***File > Save As***.
  - Save the file with the name: **Ex_8_2_Capacitor.mxwl**

Model Validation

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

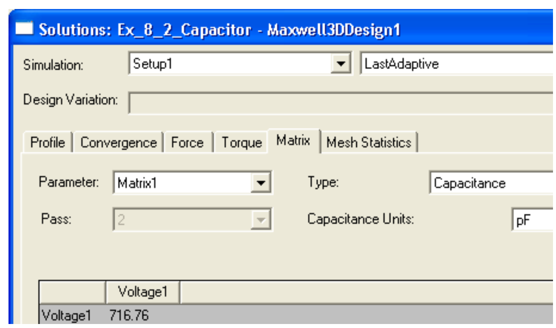
Analyze

- **To start the solution process:**
  - Select the menu item ***Maxwell 3D > Analyze All***

Multiphysics Coupling – Compact Parallel Plate Capacitor

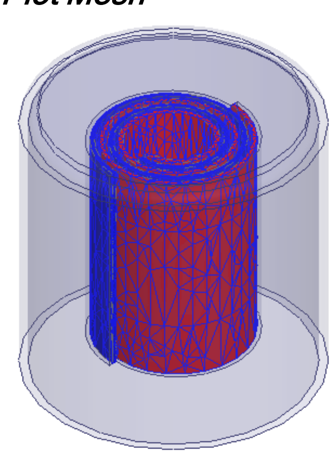
Solution Data

- **To view the Solution Data:**
  - Select the menu item ***Maxwell 3D > Results > Solution Data***
    - To view the Profile:
      1. Click the **Profile** Tab.
    - To view the Convergence:
      1. Click the **Convergence** Tab
         - **Note:** The default view is for convergence is **Table**. Select the **Plot** radio button to view a graphical representations of the convergence data.
    - To view the Mesh information:
      1. Click the **Mesh Statistics** Tab
    - To view capacitance Values
      1. Click the **Matrix** tab



Plot Mesh on Electrodes

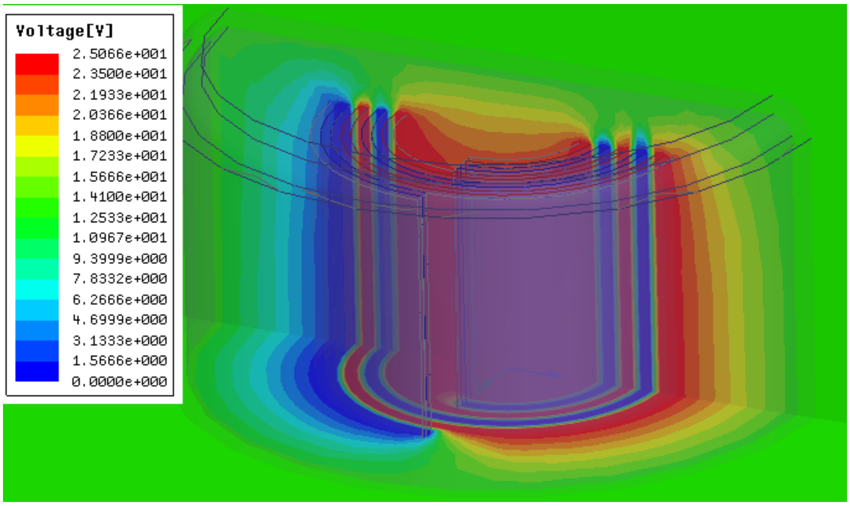
- **Hide Region**
  - Select the object **Region** from the history tree
  - Select the menu item ***View > Visibility > Hide Selection > Active View***
- **To Plot mesh**
  - Press **Ctrl** and select the objects **Electrode_A** and **Electrode_B** from history tree
  - Select the menu item ***Maxwell 3D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    1. Press **Done**



Multiphysics Coupling – Compact Parallel Plate Capacitor

Plot the Voltage field on the Global YZ and XY planes

- **To Plot Voltage on Planes**
  - Expand history tree for Planes and press **Ctrl** and select the planes **Global:XY** and **Global:YZ**
  - Select the menu item ***Maxwell 3D > Fields > Fields > Voltage***
  - In Create Field Plot window,
    1. Press **Done**



Export Geometry

- **Export Geometry for ANSYS Mechanical**
  - Select the menu item ***Modeler > Export***
  - Save the file in Parasolid Text Format with the name **"Ex_8_2_capacitor.x_t"**

Save and Exit Maxwell

- **Save File**
  - Select the menu item ***File > Save***
- **Exit Maxwell**
  - Select the menu item ***File > Exit***

- **Note:** Next section of tutorial shows how forces in Maxwell can be mapped to ANSYS mechanical to calculate Stress

Multiphysics Coupling – Compact Parallel Plate Capacitor

ANSYS Mechanical

- In this section of tutorial, we will map the forces calculated by Maxwell to ANSYS Mechanical Static Structural Solver. Solution in ANSYS Mechanical will give final stress distribution of the objects
- Mapping data from Maxwell to ANSYS Mechanical will be done through Workbench interface

Launch ANSYS Workbench

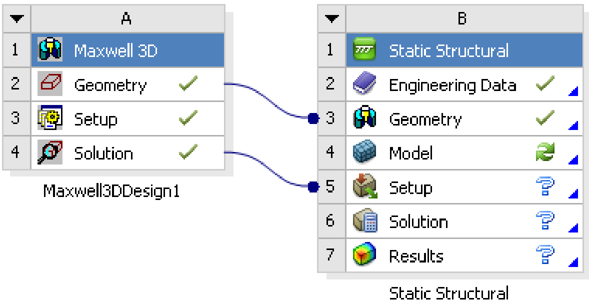
- **To Launch ANSYS Workbench**
  - Select the Microsoft **Start** button and Select **Programs > ANSYS 14.0 > Workbench**

Import Maxwell File

- **To Import Maxwell Project**
  - In Workbench Project window, select the menu item ***File > Import***
  - Set the file type to **Maxwell Project File**
  - Browse to the file **Ex_8_2_Capacitor.mxwl** and **Open** it

Create ANSYS Mechanical Design

- **To Create Static Structural Design**
  - Select a **Static Structural** Analysis System from Analysis Systems list
  - Drag and drop it on the **Solution** tab of Maxwell 3D Design
  - Drag the geometry tab of Maxwell design and drop it on Geometry tab of Static Structural Analysis System
  - Right click on Solution tab of Maxwell 3D Design and Select **Update** to update the solution cell
  - Right click on Geometry tab of Static Structural Analysis System and select **Refresh**



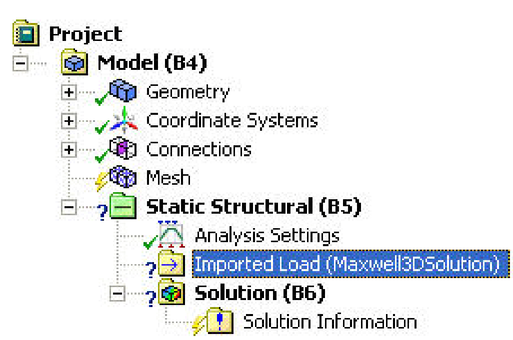
Define Material Database

- **To Define Material Database**
  - Right click on the tab **Engineering Data** and select **Edit**
  - In Engineering Data window,
    1. Select the icon **Engineering Data Sources** 
    2. Select the tab **General Materials** from Data Sources
    3. Locate **Aluminum Alloy** material from the list and select **Add**
- **To Define Material Polyimide**
  - Polyimide material is not available in material database. Hence it needs to be defined by creating new material
  - Add a new material with the properties shown in below image

Property	Value	Unit
Density	1.4	g cm^-3
Isotropic Elasticity		
Derive from	Young's Modulus a...	
Young's Modulus	2500	MPa
Poisson's Ratio	0.34	
Bulk Modulus	2.6042E+09	Pa
Shear Modulus	9.3284E+08	Pa

Launch ANSYS Mechanical

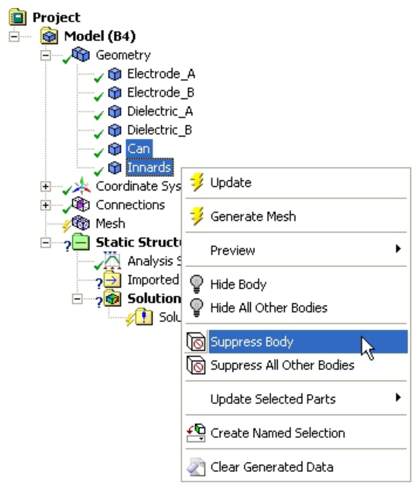
- **To Launch ANSYS Mechanical**
  - Right click on **Model** tab of Static Structural analysis system and select **Edit**
  - A tab corresponding to Maxwell data is automatically added in the tree to enable data mapping



Multiphysics Coupling – Compact Parallel Plate Capacitor

Setup Geometry

- **Suppress Unwanted Objects**
  - Expand the Project tree for **Geometry** under Model
  - Press **Ctrl** and select the bodies **Can** and **Innards,** right click on the Bodies and select **Suppress Body**
  - This will keep only electrodes and dielectric layers for simulation



- **Specify Material**
  - Press **Ctrl** and select the bodies **Electrode_A** and **Electrode_B** from the tree and goto Details View window
  - In Details View window,
    1. Material
       - Assignment: set to **Aluminum Alloy**

Graphics Properties	
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	Aluminum Alloy
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	

  - Similarly assign **Polyimide** material for the bodies **Dielectric_A** and **Dielectric_B**

Multiphysics Coupling – Compact Parallel Plate Capacitor

Map Data from Maxwell

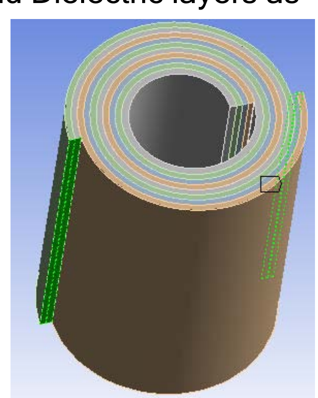
- **To Map Forces from Maxwell**
  - Right click on the **Imported Load (Maxwell3DSolution)** and select ***Insert > Body Force Density***
  - In Details View window,
    1. Geometry: Select all bodies
  - Right click on the tab **Body Force Density** and select **Import Load**
  - Forces calculated in Maxwell be mapped to the mesh in ANSYS Mechanical
  - A Summery of mapped Forces is shown below Imported Heat Generation tab. Scaling factor shown in this summery should be close to **1** to ensure correct data mapping. If not, mesh needs to be refined in important regions. To simplify the problem, we will continue without mesh refinement.

Exporting Volume Force Density...

Object	Total Force	Scaling Factor
Electrode_A	(4.99012E-038N, -4.98144E-037N, -1.13592E-037N)	0.226582
Electrode_B	(0N, 0N, 0N)	0
Dielectric_A	(-1.03533E-006N, 3.43367E-008N, -2.10378E-010N)	1.88786
Dielectric_B	(7.51684E-007N, 3.20343E-006N, -2.05646E-009N)	0.839576

Specify Fixed Supports

- **To Specify Fixed Support**
  - Right click on **Static Structural** tab from the specification tree and select ***Insert > Fixed Support***
  - In Details View window,
    1. Geometry:
       - Change Selection Filter to **Faces** 
       - Select the faces of the Electrode and Dielectric layers as shown in below image
       - Press **Apply** in details view window



Multiphysics Coupling – Compact Parallel Plate Capacitor

Create Stress Plot

- **To Plot Stresses on Electrode_A**
  - Right click on **Solution** tab from specification tree and select ***Insert > Stress > Equivalent (Von-Mises)***
  - In Details View window,
    1. Geometry:
       - Change Selection Filter to **Bodies**
       - Select the body corresponding to **Electrode_A**
       - Press **Apply** in details view window

Solve

- **To Run the Solution**
  - Right click on **Solution** tab from specification tree and select **Solve**

Stress Plot

- **To View Stress Plot**
  - Select the **Equivalent Stress** plot from specification tree under **Solution**
  - Stress distribution on **Electrode_A** will be displayed on the graphic window