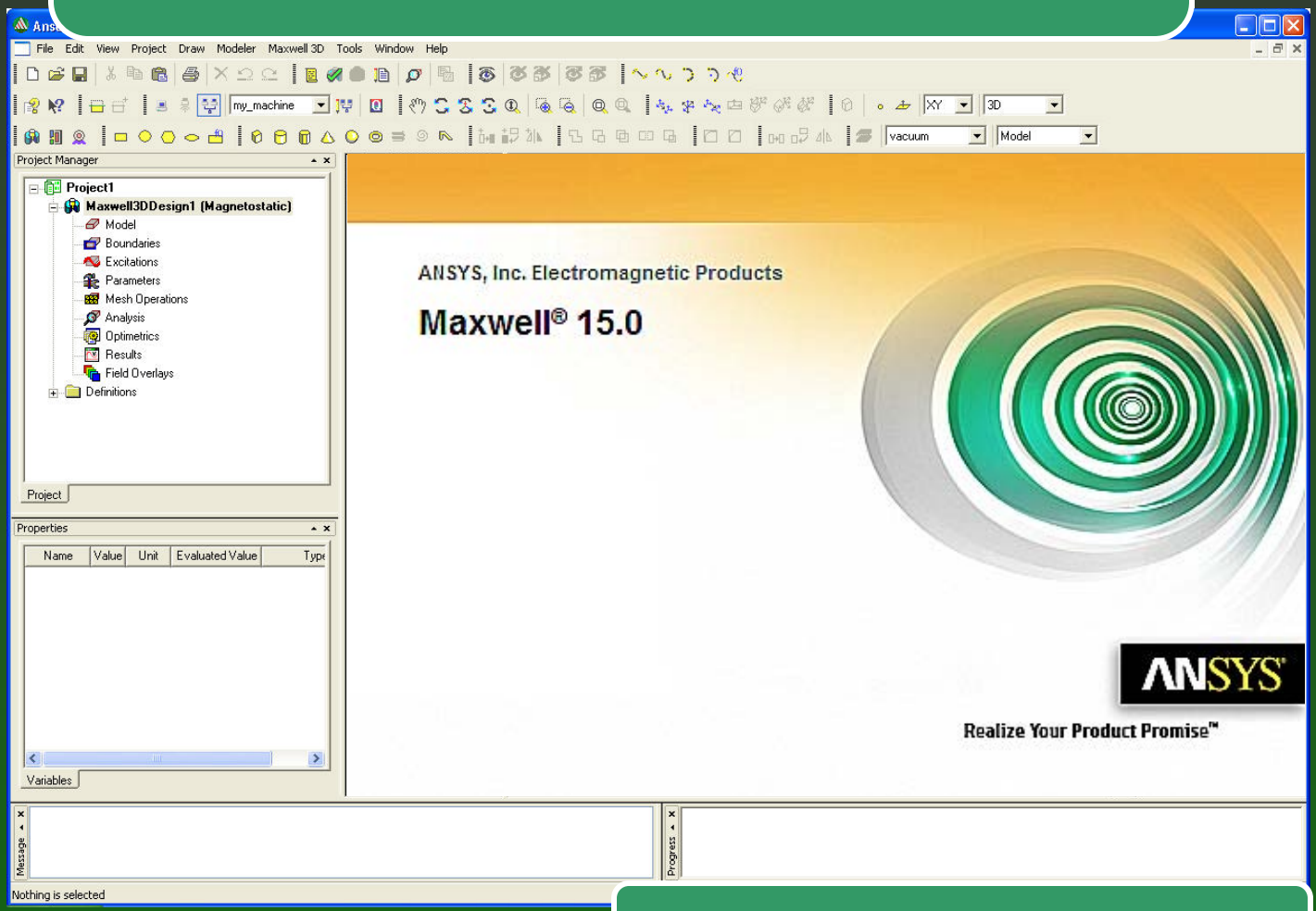
ANSYS®

Maxwell 2D



electronic design automation software

user's guide - Maxwell 2D

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.

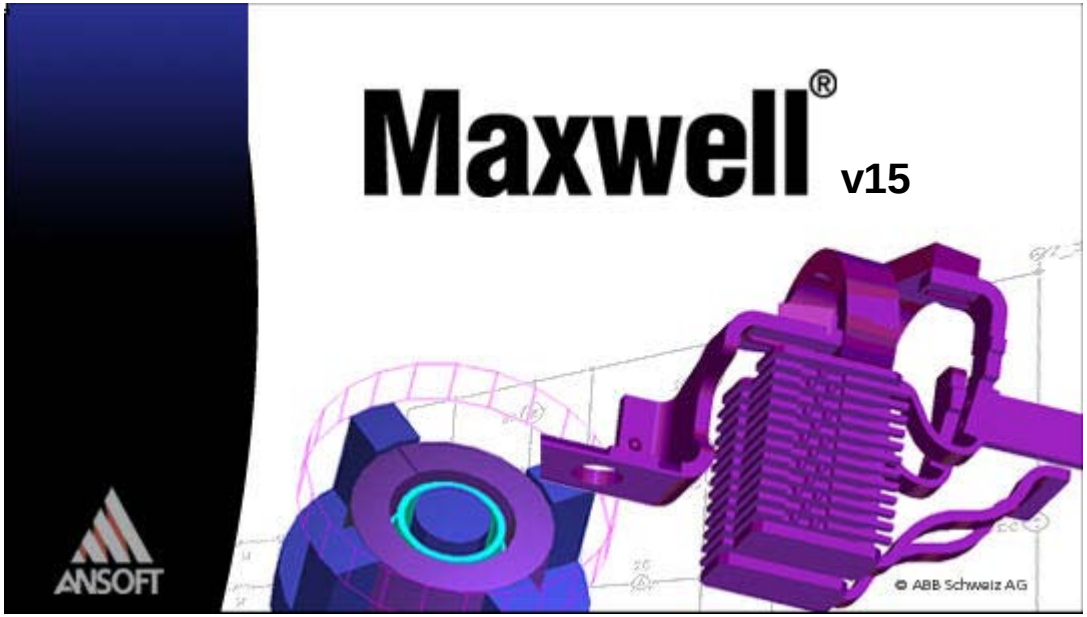
New editions of this manual will incorporate all material updated since the previous edition. The manual printing date, which indicates the manual's current edition, changes when a new edition is printed. Minor corrections and updates which are incorporated at reprint do not cause the date to change. 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:

- Overview
  - 1.0 - Maxwell 2D
- Examples – Eddy Current
  - 6.1 – Jumping Rings Axisymmetric Model
  - 6.3 – Instantaneous Forces on Busbars
- Examples – Transient
  - 7.1 – Gapped Inductor Model
  - 7.2 - Solenoid Problem with an External Circuit
  - 7.4 – Core Loss
- Examples – Basic Exercises
  - 9.1 – Electrostatic
  - 9.2 – DC Conduction
  - 9.3 – Magnetostatic
  - 9.4 – Parametric
  - 9.5 – Transient
  - 9.6 – Transient with Circuit Editor
  - 9.7 – Post-processing
  - 9.8 – Optimetrics
  - 9.10 – Scripting
  - 9.12 – Eddy Current
  - 9.13 – Rotational Transient Motion
  - 9.14 – Boundary Conditions
  - 9.15 – Permanent Magnets Assignment
- Examples – Motors
  - 11.1 - ANSYS Electrical Machine Design Reference
  - 11.2 - Permanent Magnet Synchronous Machine
  - 11.3 - Three-Phase Induction Machine
  - 11.4 - Permanent Magnet Motor (Prius IPM)
  - 11.5 - OptimetricsAnalysis with RMxprt
  - 11.6 – Torque Optimization using Design Explorer



- Maxwell 2D is a high-performance interactive software package that uses finite element analysis (FEA) to solve electric, magnetostatic, eddy current, and transient problems.

Maxwell® v15

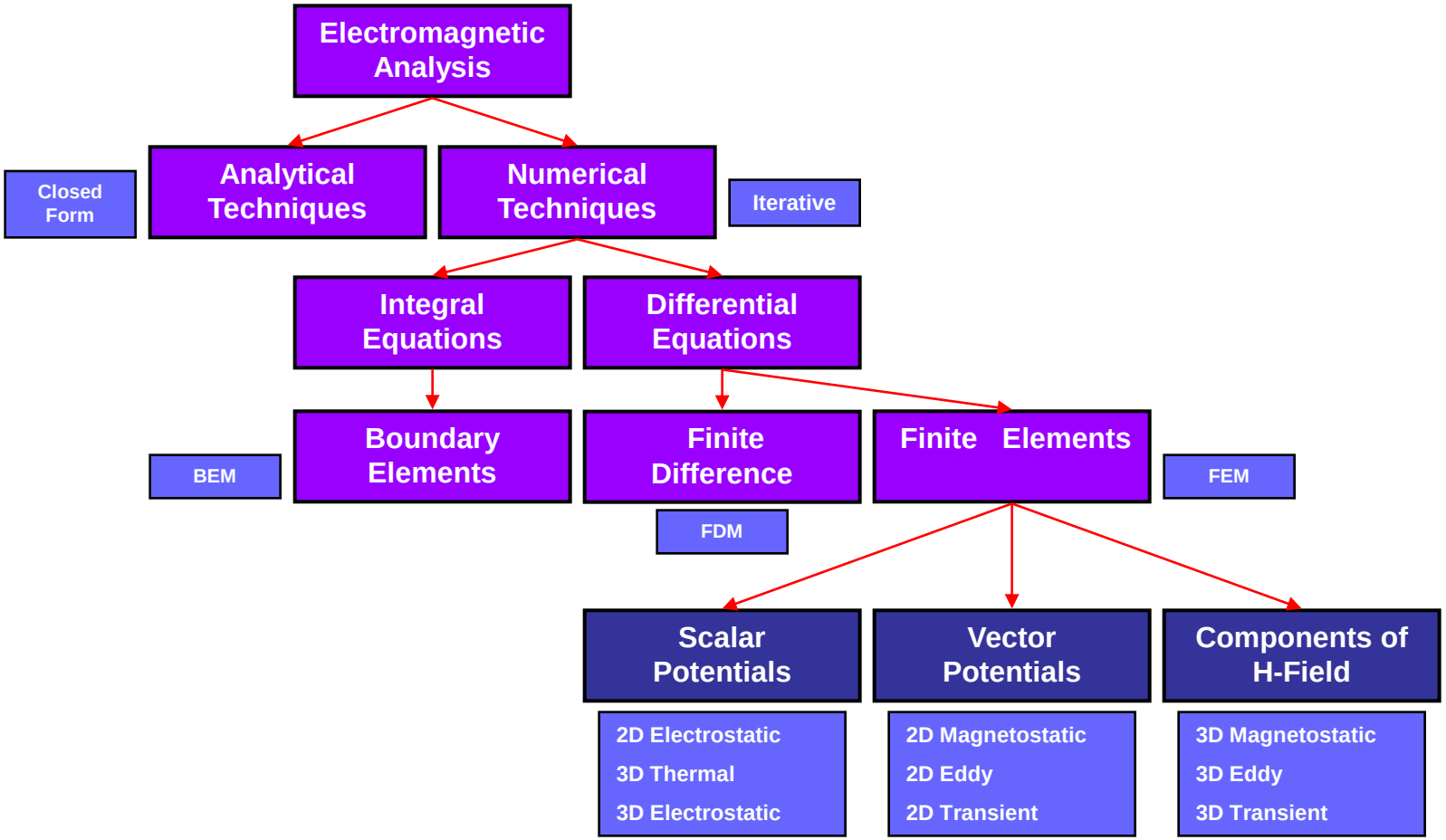
- Maxwell 2D solves the electromagnetic field problems for a given model with appropriate materials, boundaries and source conditions applying Maxwell's equations over a finite region of space.
- There are two geometry modes available in Maxwell 2D:
  - Cartesian (XY) model
  - Axisymmetric (RZ) model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

  } Electric Fields

  - Magnetostatic
  - Eddy Current
  - Transient Magnetic

  } Magnetic Fields

Different Methods of Electromagnetic Analysis



Differential Form of Maxwell's Equations

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

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

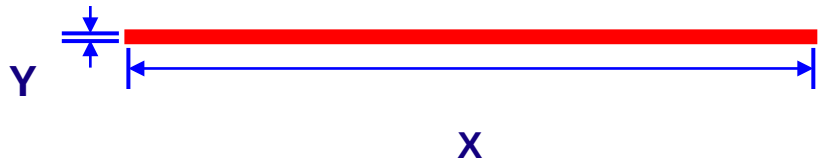
Ampere's Law $\nabla \times H = J + \frac{\partial D}{\partial t}$

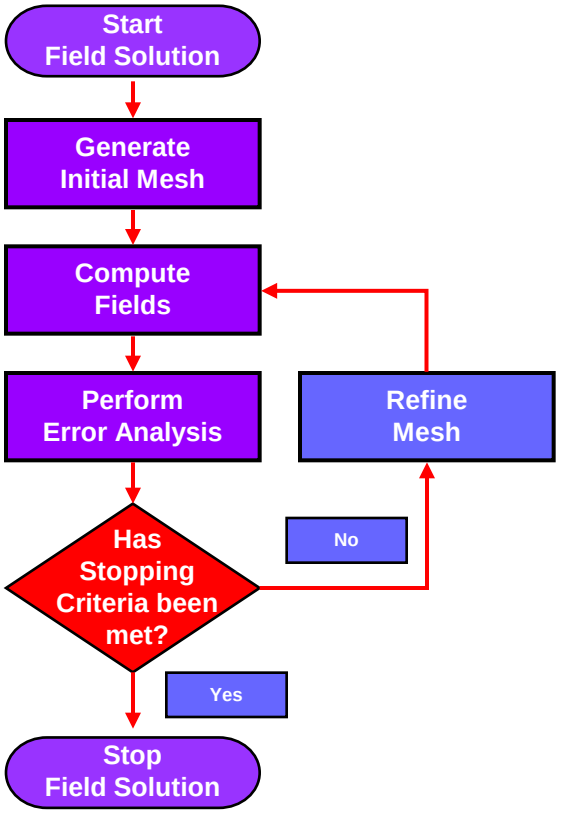
Gauss's Law for Electricity $\nabla \bullet D = \rho$

FEM and adaptive meshing

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

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



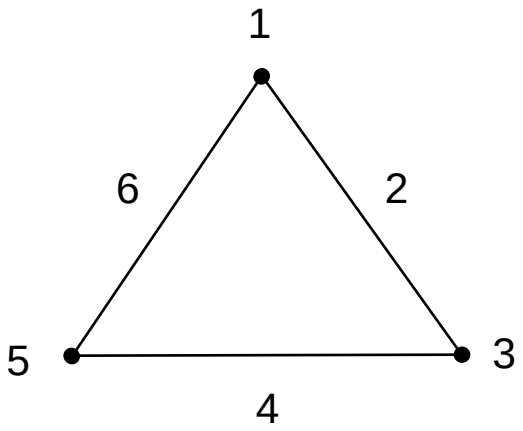


FEM Approximation Functions

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

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

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



FEM Variational Principle

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

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

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

FEM Matrix Equation

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

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

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

FEM Error Evaluation

- Put the approximate solution back into Poisson's equation

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

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

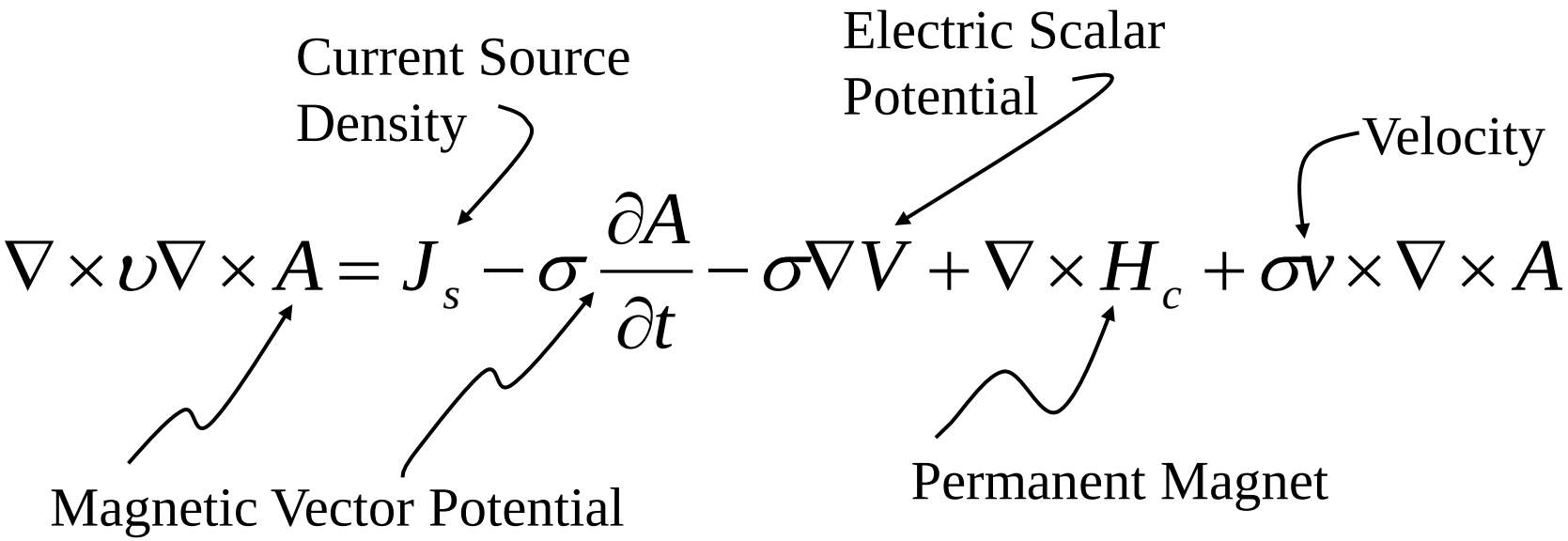
FEM Percent Error Energy

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

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

- Local errors can exceed Percent Error Energy

Transient Solver
Fully Coupled Dynamic Physics Solution



Time-varying Electric and Magnetic Fields

Transient Solver - Magnetic Field Diffusion

- Magnetic fields “diffuse” into materials at different rates depending on:
  - Material properties of the component
  - Physical size of the component

- For a cylindrical conductor, diffusion time is:

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

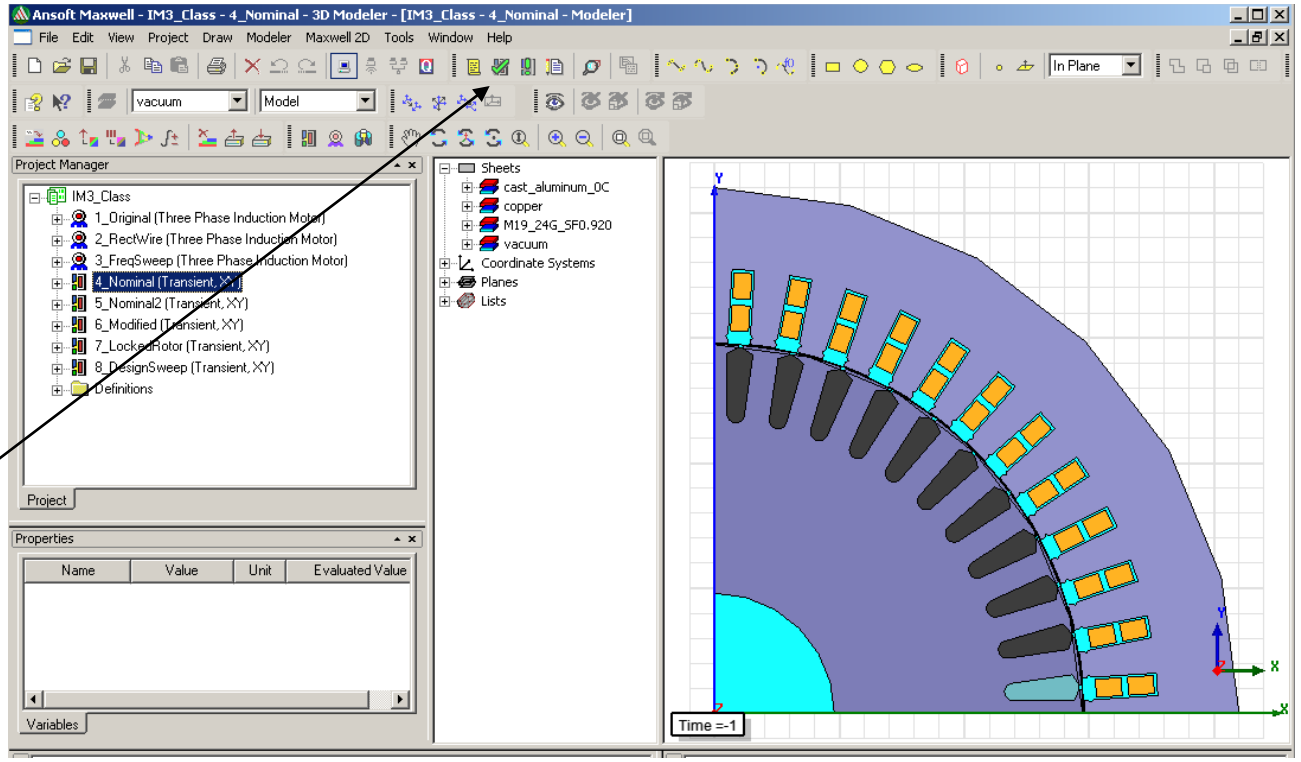
$$where: u = perm, \sigma = conductivity, a = radius\ in\ meters$$

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

GUI - Desktop

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





ACIS solid modeling kernel

- The underlying solid modeling technology used by 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).
- Exports directly .sat, .dxf, .sm3, .sm2, .step, .iges

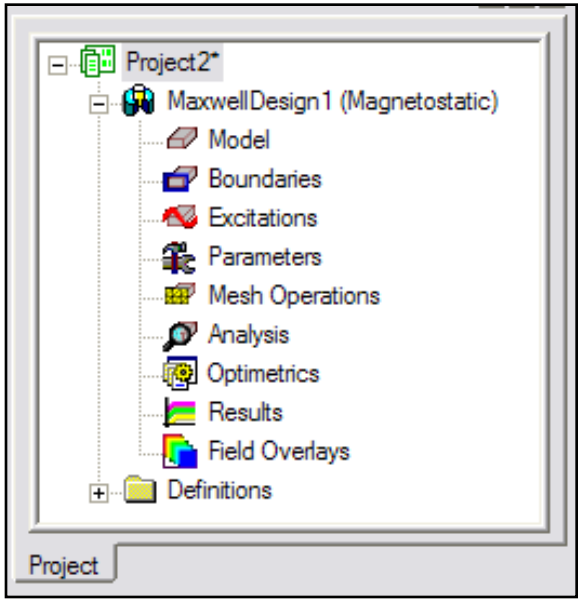
v15 Supported platforms

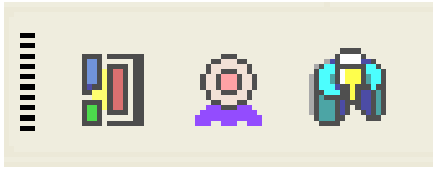
- Windows XP 32-bit Service Pack 2
- Windows Server 2003 32-bit Service Pack 1
- Windows XP 64-bit Service Pack 2
- Windows Server 2003 64-bit Service Pack 1
- Windows HPC Server 2008
- Windows 7 Business Editions (32-bit and 64-bit versions)
- Red Hat (32 and 64 bit) v.4, v.5
- SuSE (32 and 64 bit)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

- ## Starting Maxwell
  - Click the Microsoft **Start** button, select **Programs**, and select the ***Ansoft > Maxwell 15> 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 2D Design***



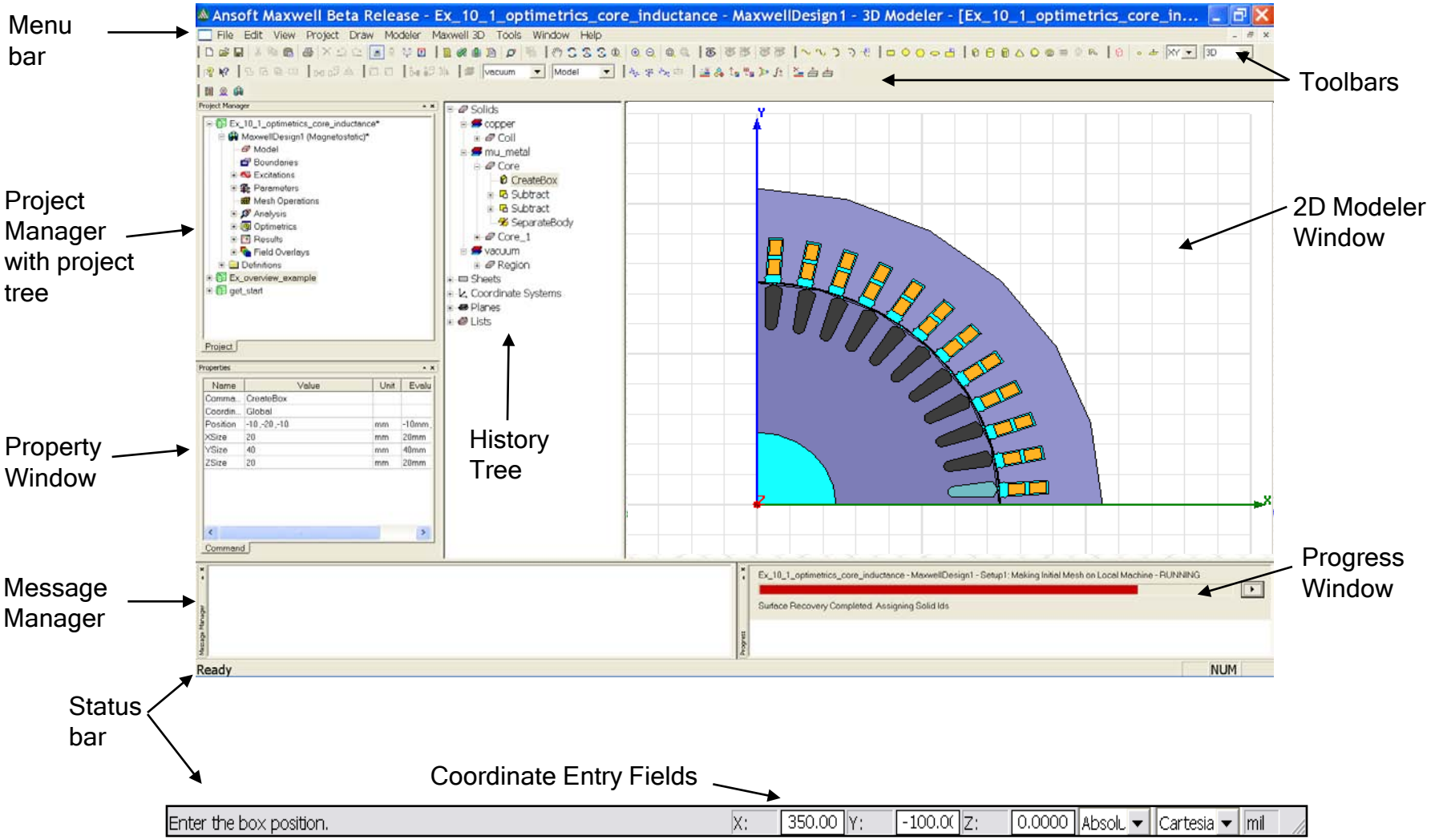


Toolbar:
Insert Maxwell 2D Design

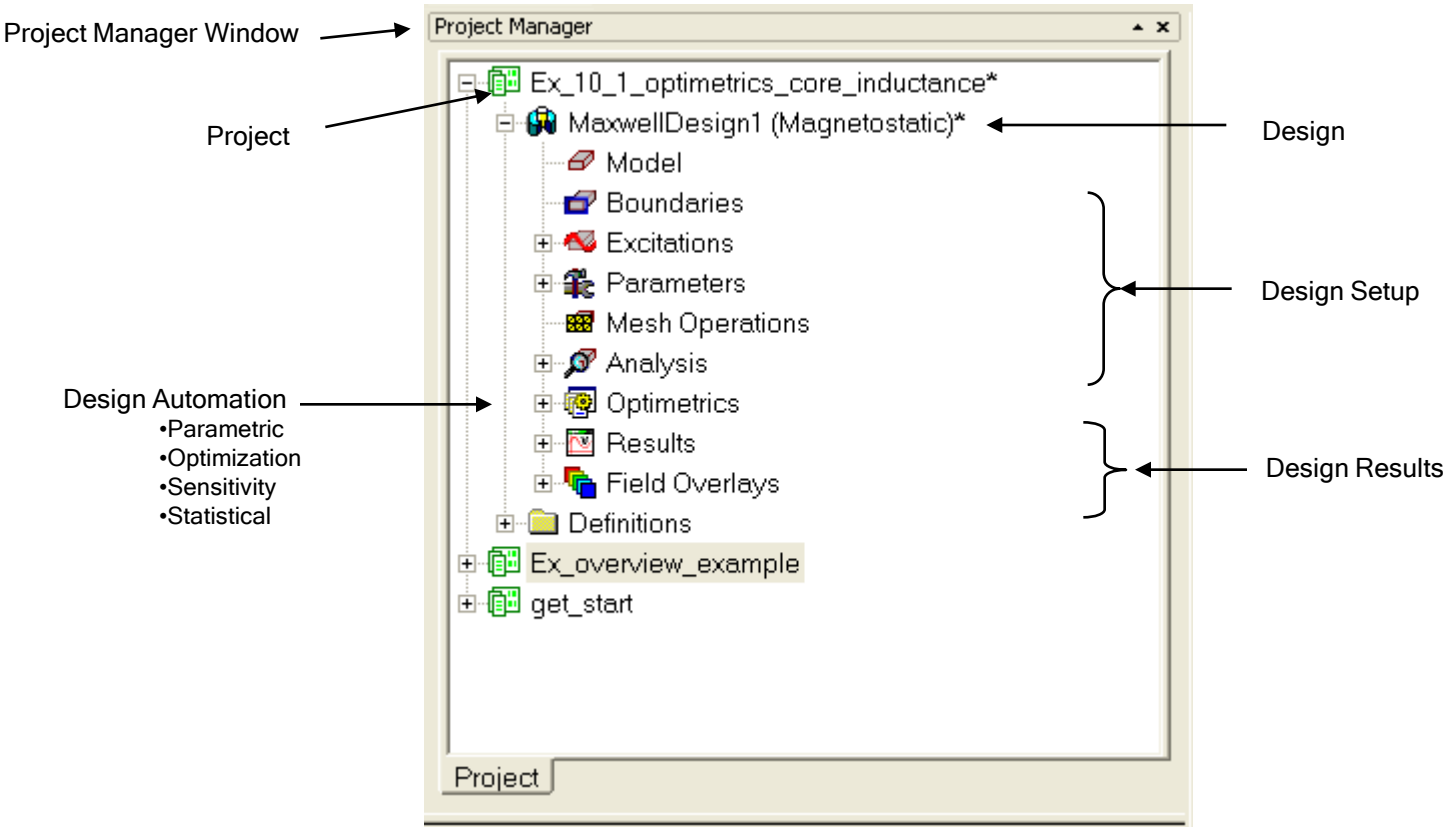
Insert Maxwell 3D Design

Insert RMxprt Design

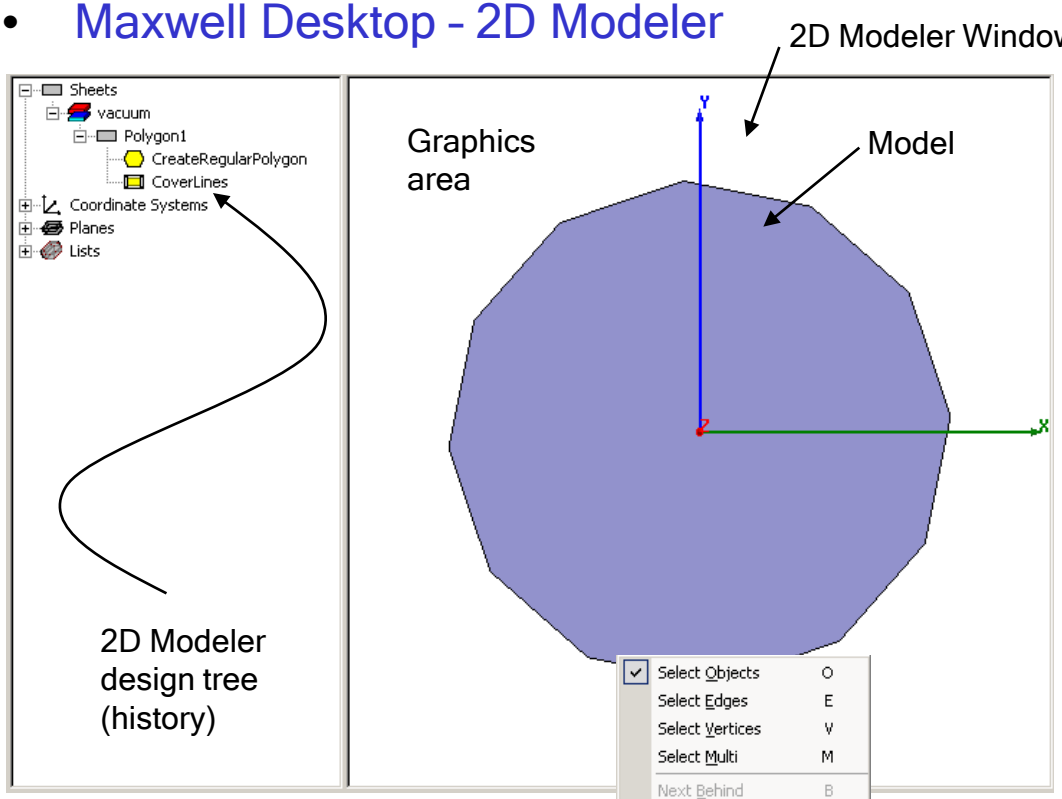
- Maxwell Desktop



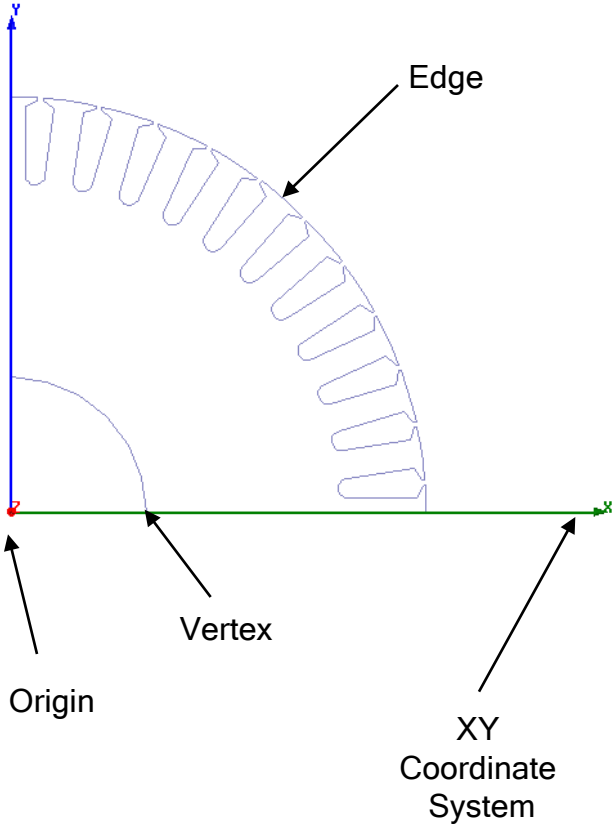
- Maxwell Desktop – Project Manager
  - Multiple Designs per Project
  - Multiple Projects per Desktop
  - Integrated Optimetrics Setup (requires license for analysis)



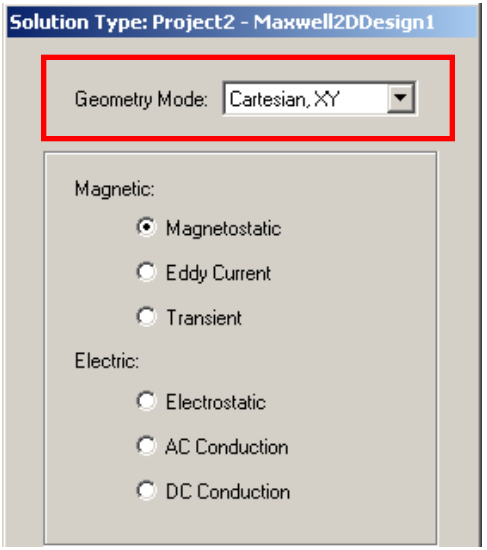
- Maxwell Desktop - 2D Modeler



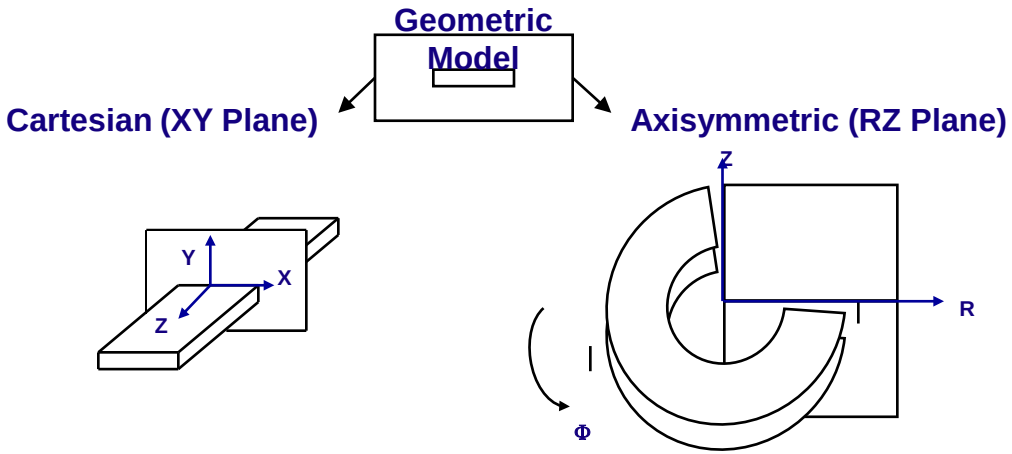
Context menu (right mouse click on 2D modeler window)

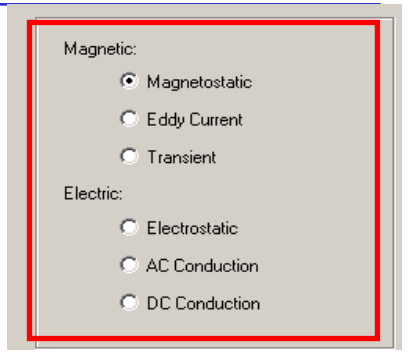


- Geometry Mode
  - **To set the geometry mode:**
    1. Select the menu item *Maxwell 2D > Solution Type*
    2. Solution Type Window:
       - Choose Geometry Mode: **Cartesian XY**



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

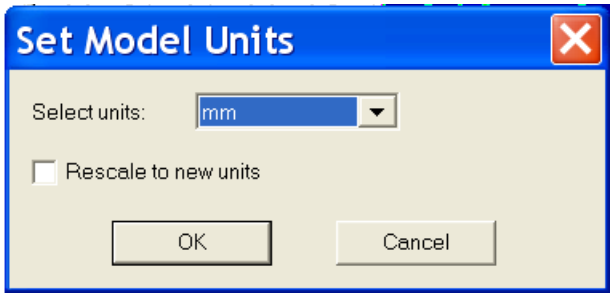




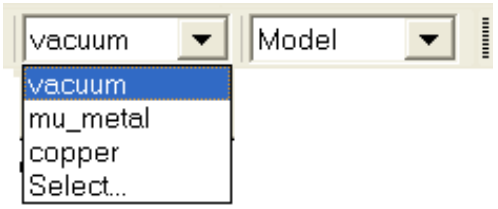
- ## Set Solution Type
  - **To set the solution type:** select the menu item ***Maxwell 2D > Solution Type***

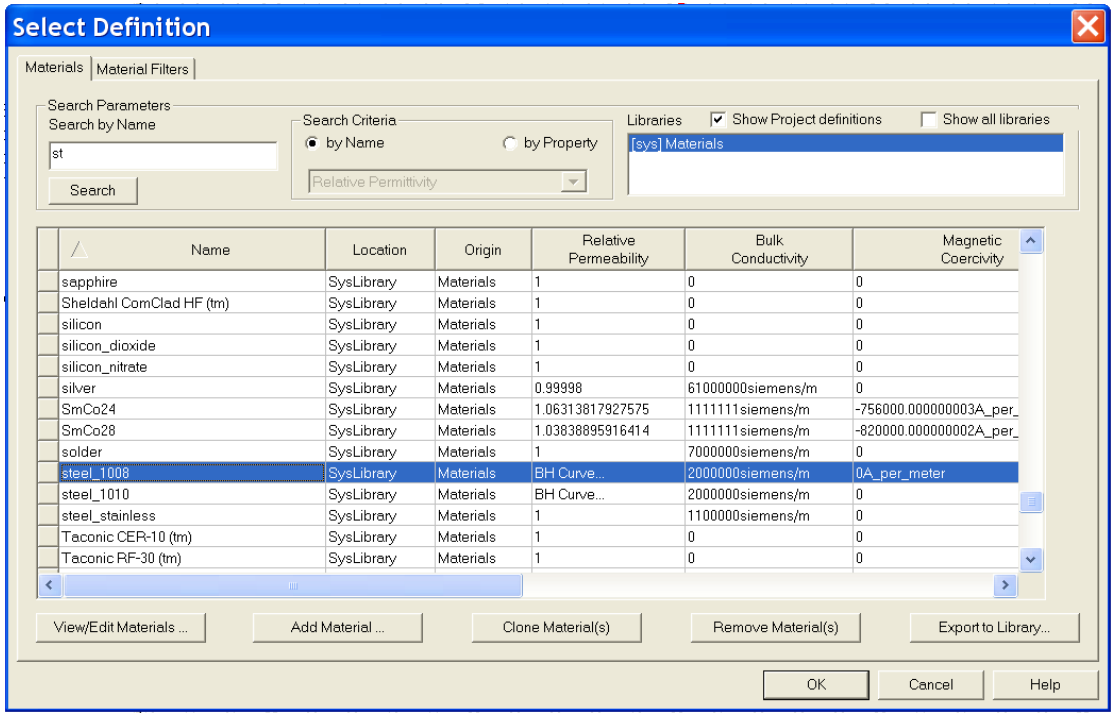
- ## Magnetic Solution Types
  - **Magnetostatic**
    Computes the static magnetic field that exists in a structure given a distribution of DC currents and permanent magnets. The magnetic field may be computed in structures with both nonlinear and linear materials. An inductance matrix, force, torque,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

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

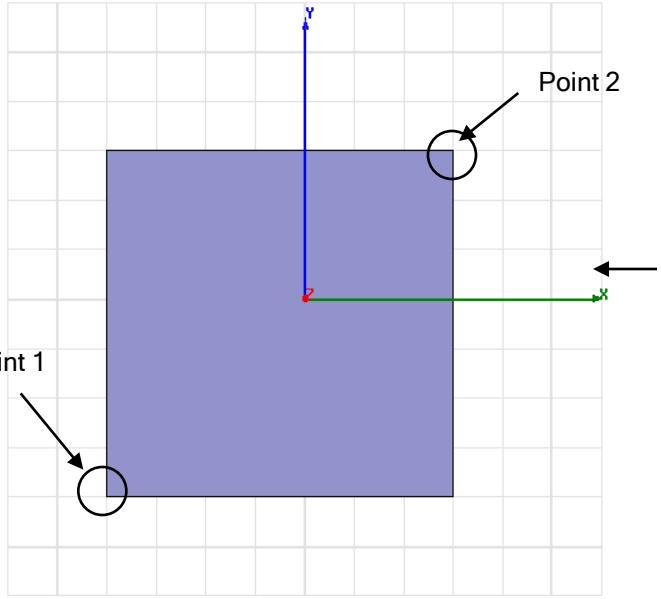


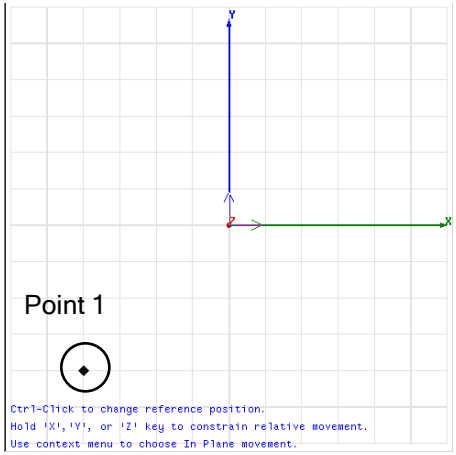
- ## Set Default Material
  - **To set the default material:**
    1. Using the Modeler Materials toolbar, choose **Select**
    2. Select Definition Window:
       1. Type **steel_1008** in the **Search by Name** field
       2. Click the **OK** button

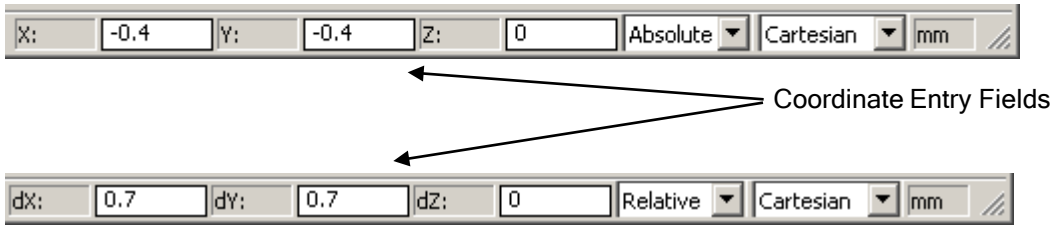


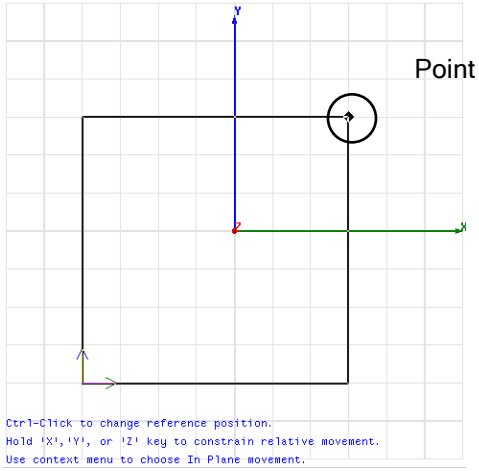


- Modeler - Draw a Rectangle





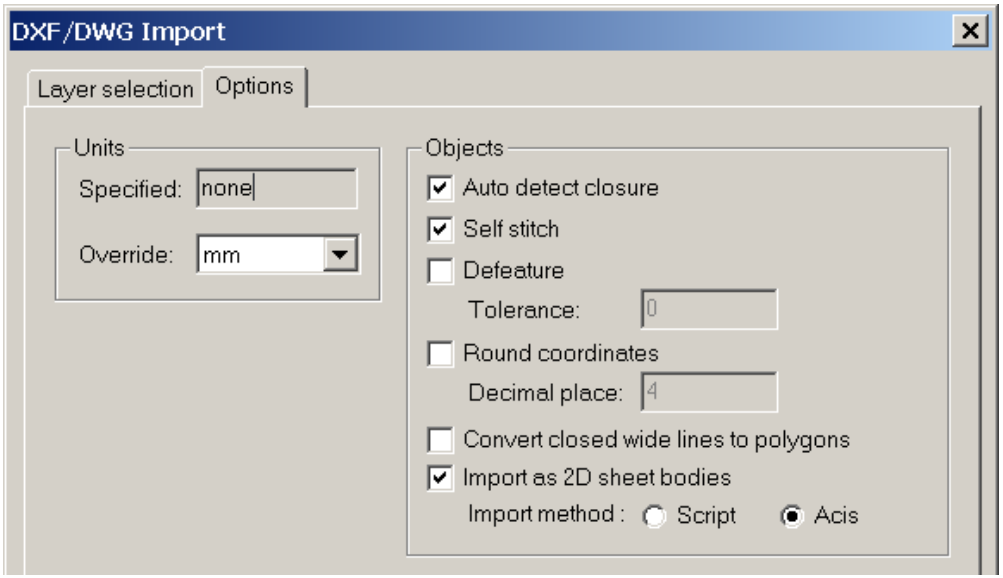




- The Coordinate Entry fields allow equations to be entered for position values.
  - Examples: 2*5, 2+6+8, 2*cos(10*(pi/180)).
- Variables are not allowed in the Coordinate Entry Field
- **Note**: Trig functions are in radians

Modeler – Importing .dxf and .dwg CAD files

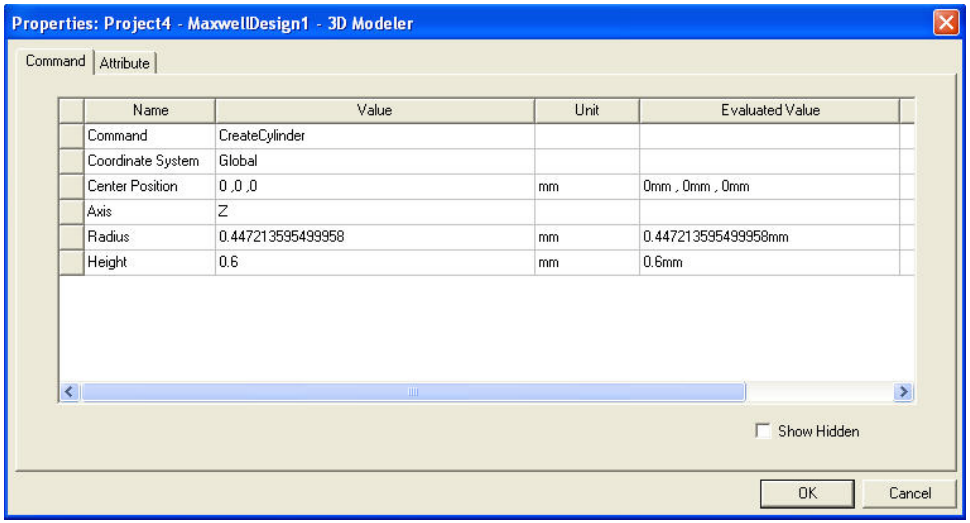
- Default options are shown below
- To change the number of segments on an imported curve:
  - Change to face select mode: Edit > Select > Faces and click on face
  - Modeler > Surface > Uncover Faces
  - Change to object select mode: Edit > Select > Objects and click on open polyline
  - Modeler > Purge History
  - Modeler > Generate History
  - Expand the history tree for that polyline and change number of segments as desired
  - Select the polyline and: Modeler > Surface > Cover Lines



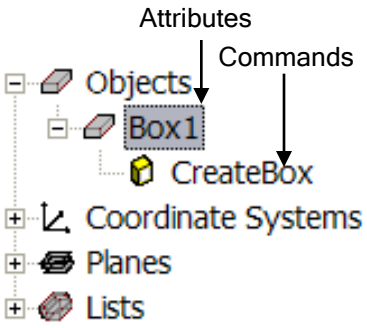
- Modeler – Object Properties

Commands

(dimensions and history)

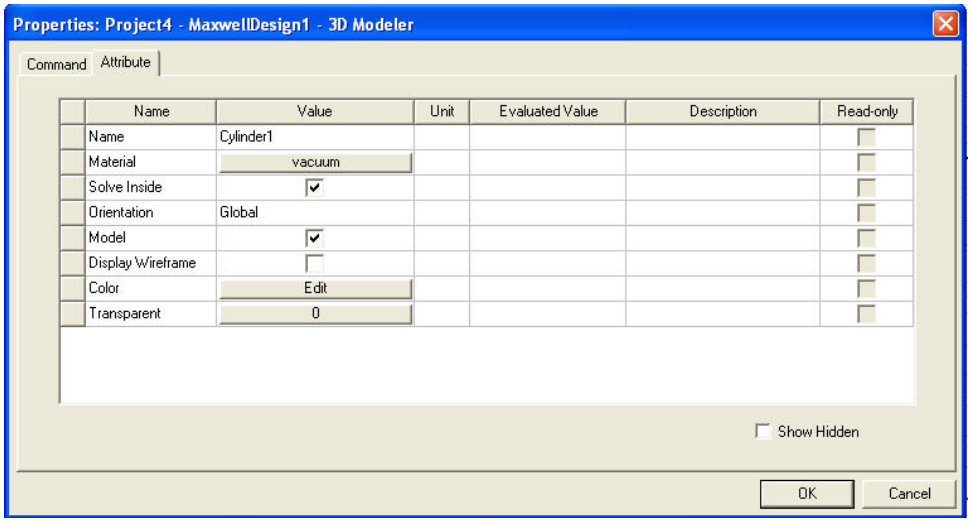


In History Tree:

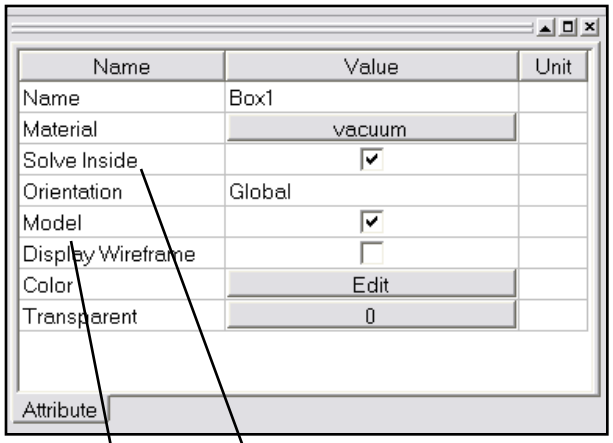


Attributes

(properties of the object)

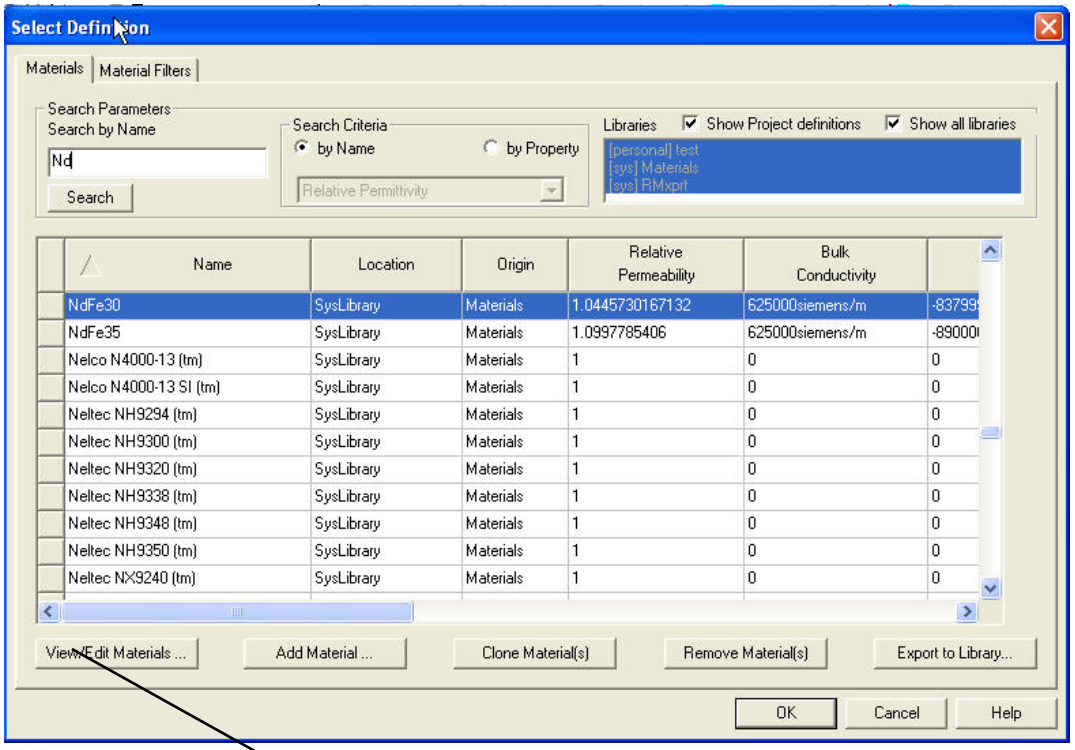


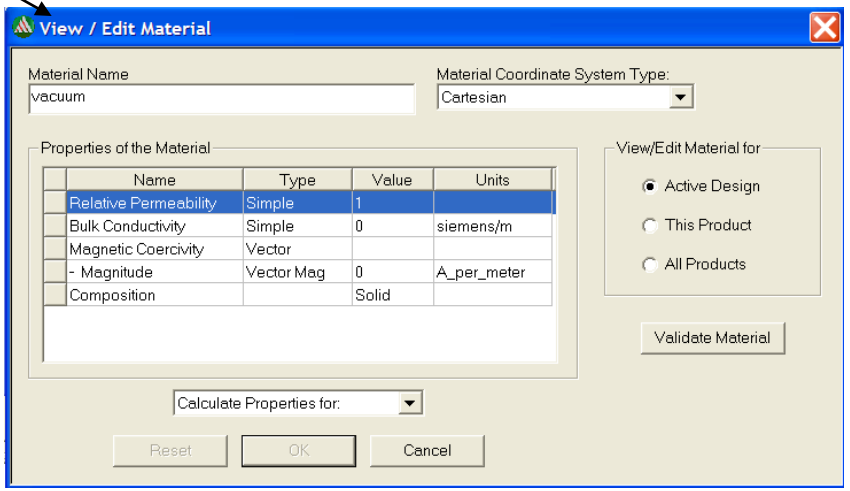
- Modeler – Attributes



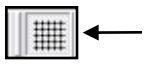
Solve Inside – if unchecked meshes but no solution inside (like the old exclude feature in material manager)

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

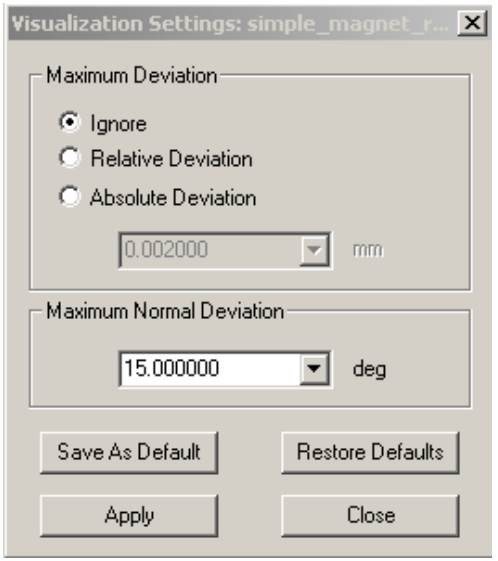


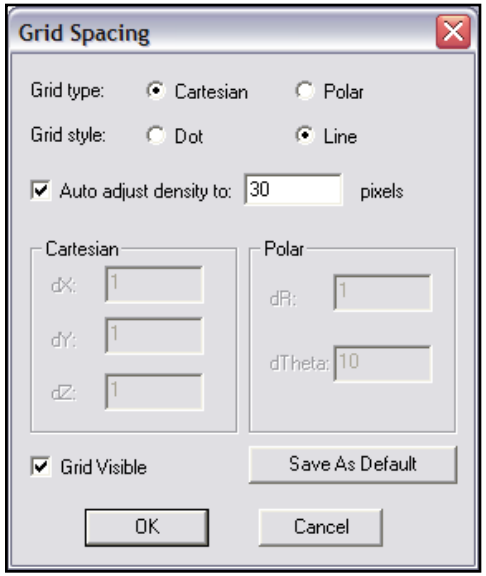


- Modeler - Views
  - *View > Modify Attributes >*
    - *Orientation* – Predefined/Custom View Angles
    - *Lighting* – Control angle, intensity, and color of light
    - *Projection* – Control camera and perspective
    - *Background Color* – Control color of 3D Modeler background
  - *View > Visualization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

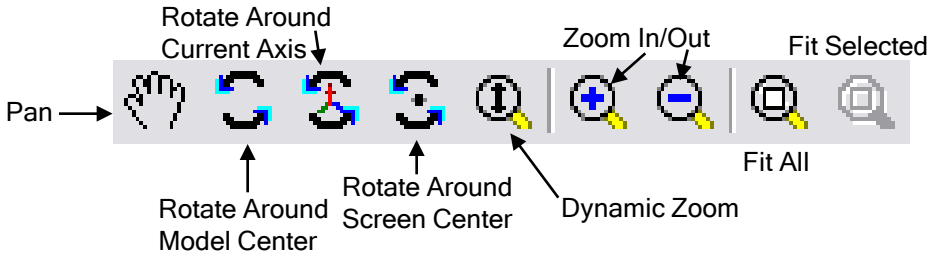


Toolbar: Toggle Grid Visibility

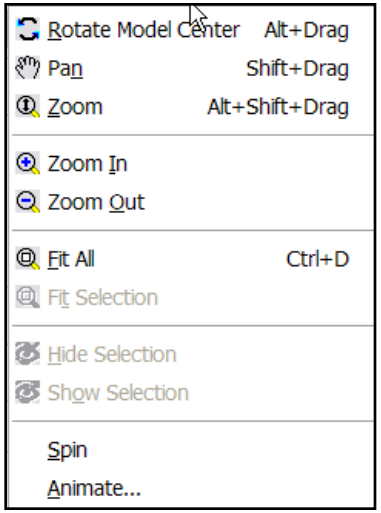




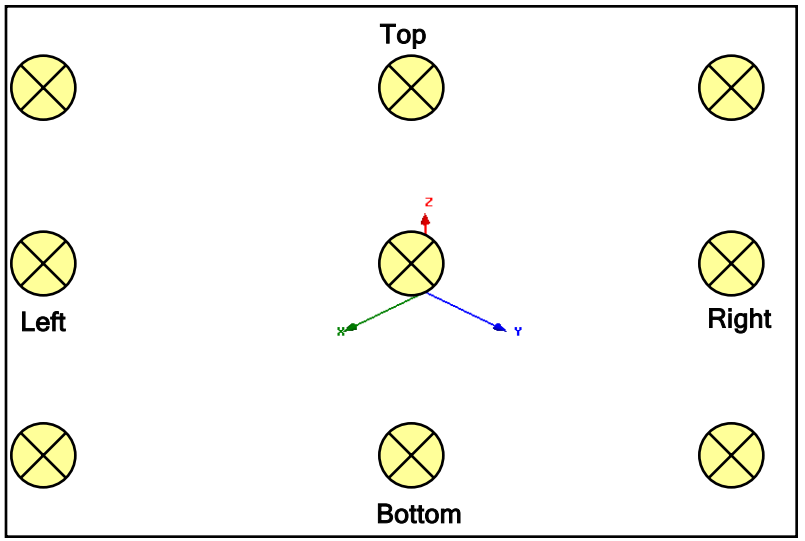
- Changing the View
  - **Toolbar**



  - **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 V12 Keyboard Shortcuts

General Shortcuts

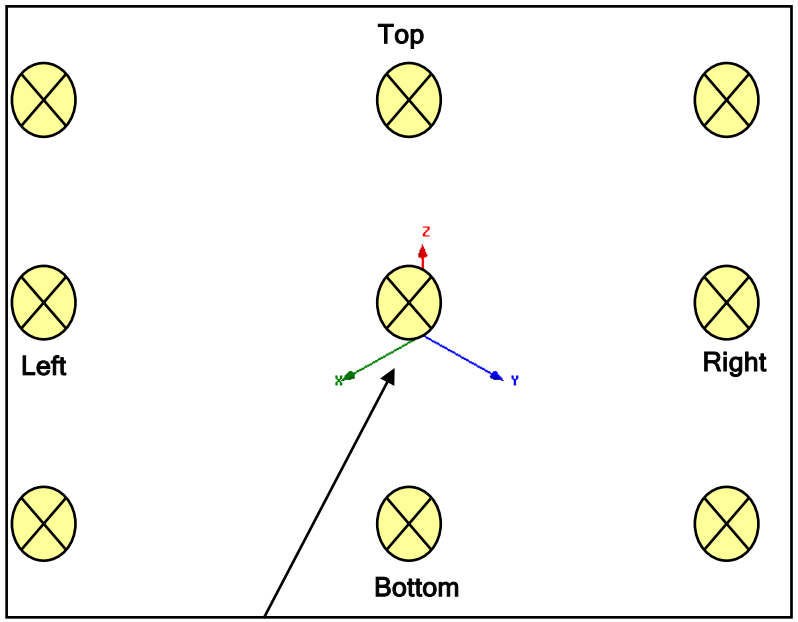
- F1: Help
- Shift + F1: Context help
- CTRL + F4: Close program
- CTRL + C: Copy
- CTRL + N: New project
- CTRL + O: Open...
- CTRL + S: Save
- CTRL + P: Print...
- CTRL + V: Paste
- CTRL + X: Cut
- CTRL + Y: Redo
- CTRL + Z: Undo
- CTRL + 0: Cascade windows
- CTRL + 1: Tile windows horizontally
- CTRL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

Modeller Shortcuts

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

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

Predefined View Angles

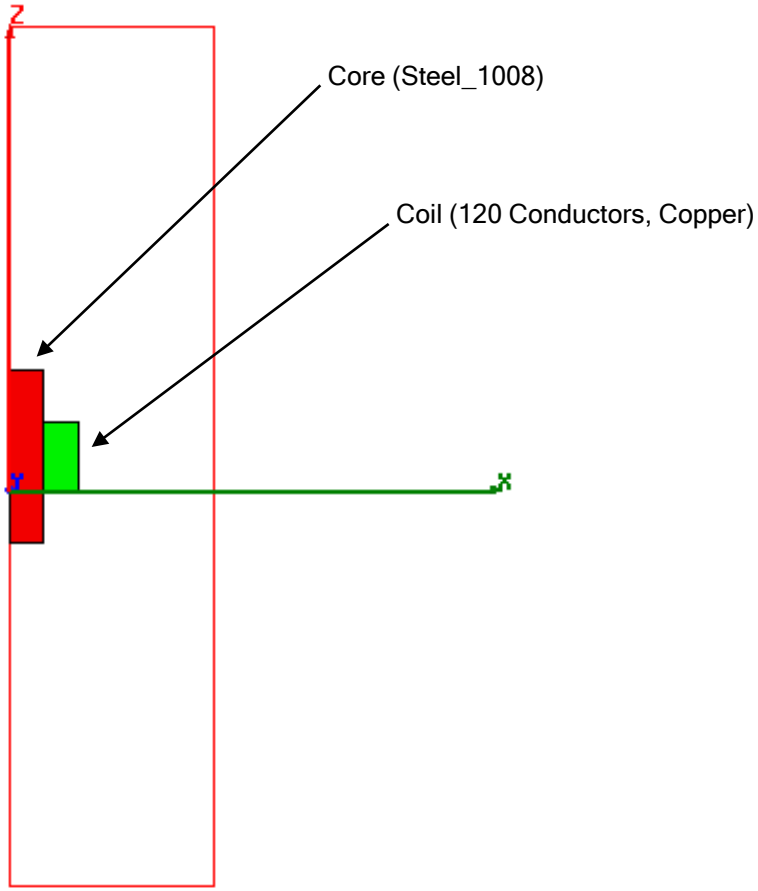


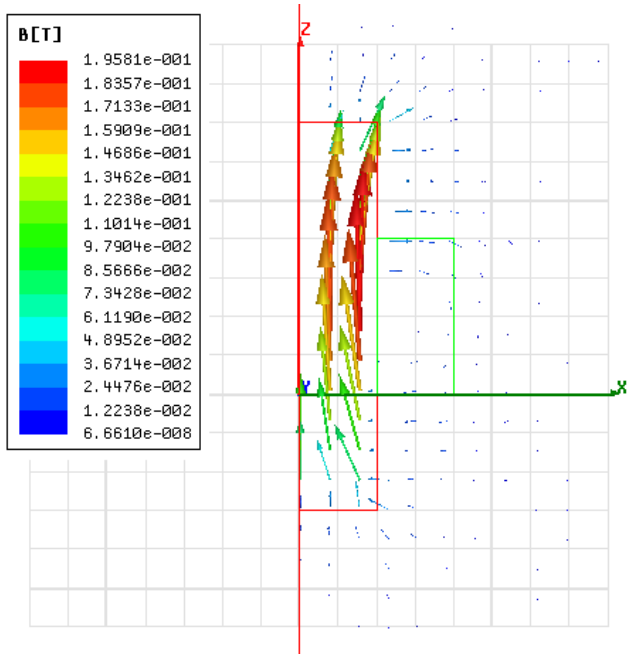
Alt + double left Click here to restore view in an RZ model

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

Simple Example

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





- Setup the geometry mode and solver
  - Choose *Cylindrical about Z* under *Maxwell 2D > Solution Type*
  - Choose **Magnetostatic**
  - Click the **OK** button

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

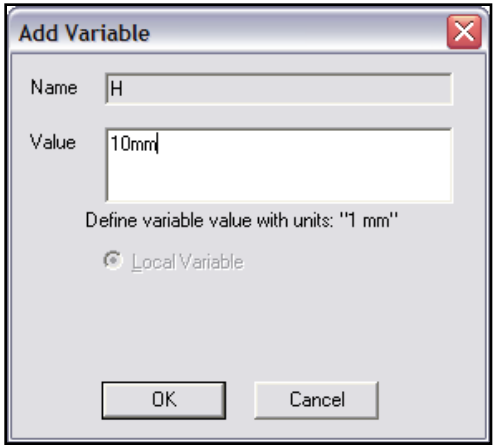


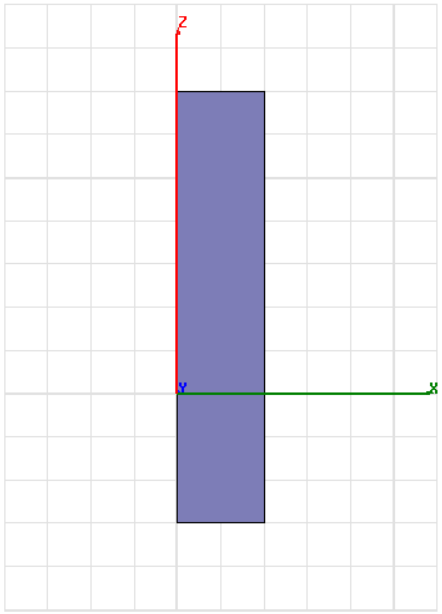
    3. Using the coordinate entry fields, enter the opposite corner of the rectangle
       - dX: **2.0**, dY: **0.0**, dZ: **10.0**, Press the **Enter** key



  - **Continued on Next Page**

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



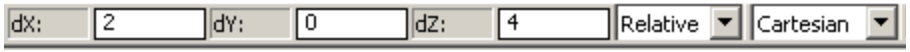


- ## Set Default Material
  - **To set the default material:**
    1. Using the 3D Modeler Materials toolbar, choose **Select**
    2. Select Definition Window:
       1. Type **copper** in the **Search by Name** field
       2. Click the **OK** button

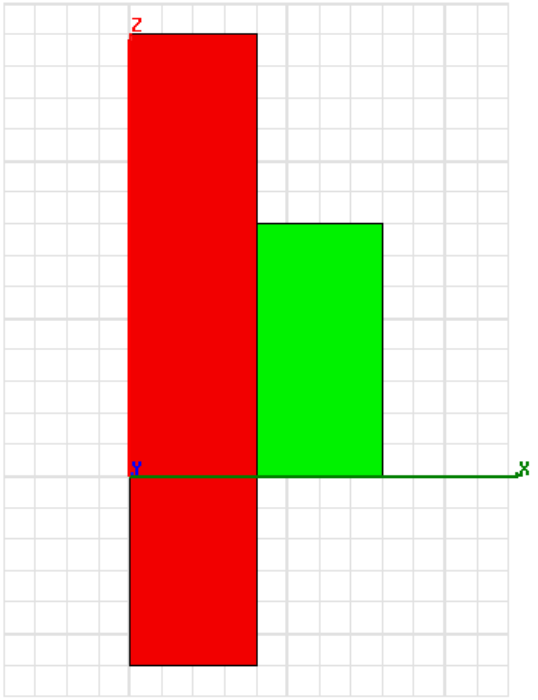
- ## Create Coil
  - **To create the coil for the current to flow:**
    1. Select the menu item *Draw > Rectangle*
    2. Using the coordinate entry fields, enter the center position
       - X: **2.0**, Y: **0.0**, Z: **0.0,** Press the **Enter** key

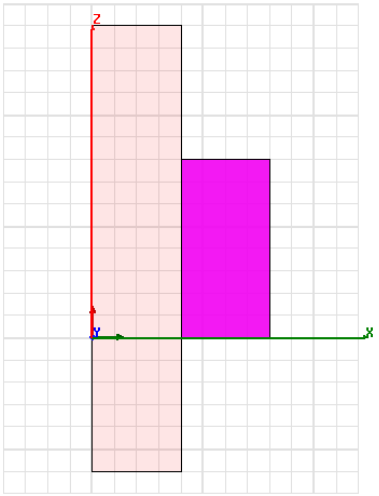
    3. Using the coordinate entry fields, enter the opposite corner of the re
       - dX: **2.0**, dY: **0.0**, dZ: **4.0**, Press the **Enter** key

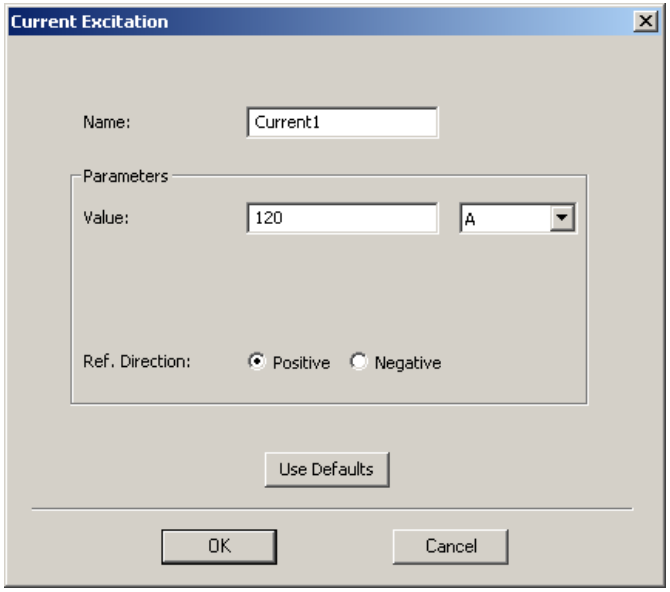
       

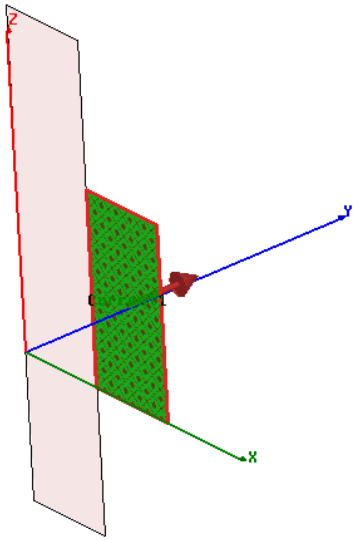
  - **To set the name:**
    1. Select the **Attribute** tab from the **Properties** window.
    2. For the **Value** of **Name** type: **Coil**
    3. Click the **OK** button
  - **To fit the view:**
    1. Select the menu item *View > Fit All > Active View*



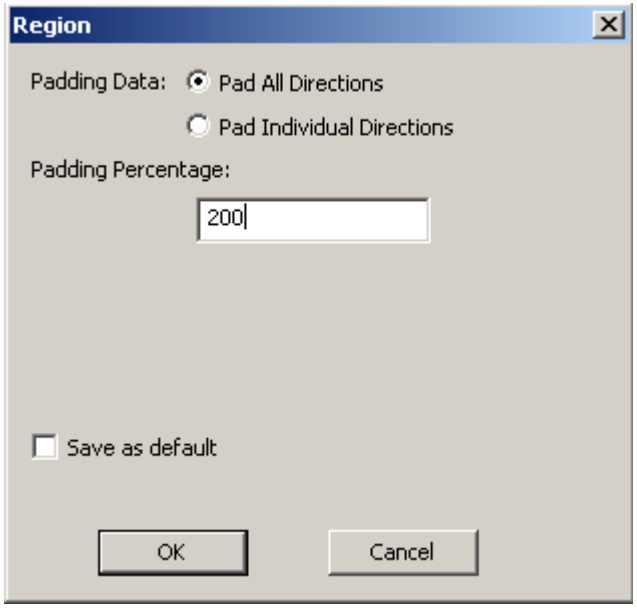
- Create Excitation
  - **Assign Excitation**
    1. Click on the coil.
    2. Select the menu item *Maxwell 2D > Excitations > Assign > Current*
    3. Current Excitation : General
       1. Name: **Current1**
       2. Value: **120 A** (Note: this is 120 Amp-turns)
       3. Ref. Direction: **Positive**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

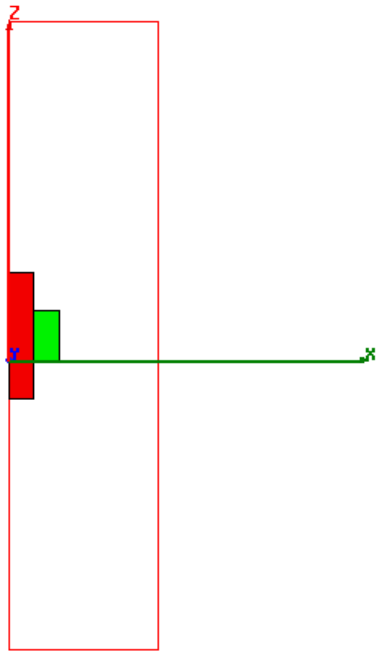






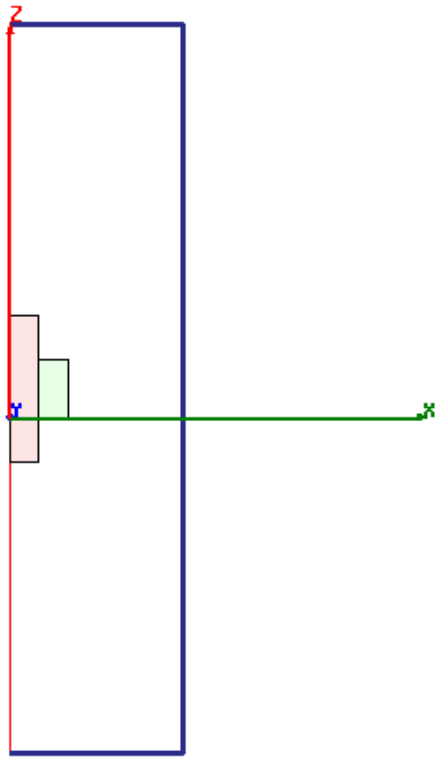
- ## Define a Region
- Before solving a project a region has to be defined. A region is basically an outermost object that contains all other objects. The region can be defined by a special object in ***Draw > Region***. This special region object will be resized automatically if your model changes size.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

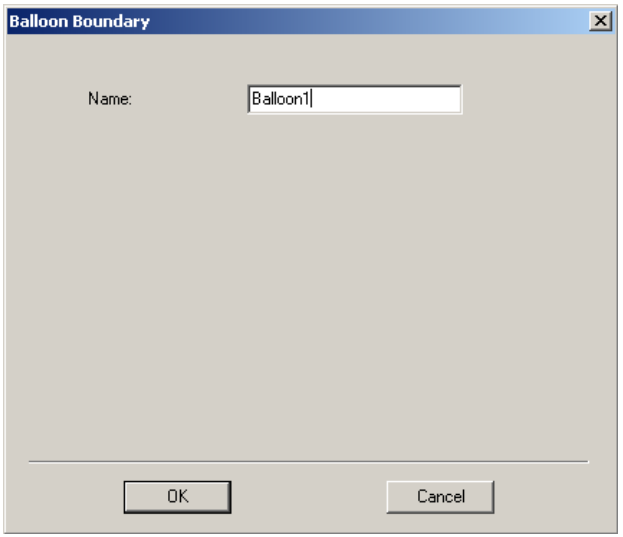




Note: Since there will be considerable fringing in this device, a padding percentage of at least 2 times, or 200% is recommended

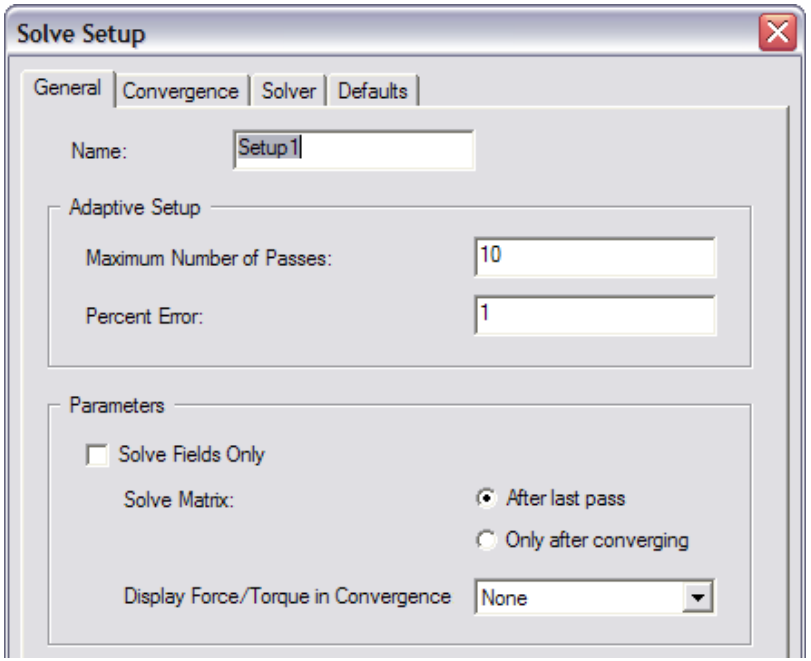
- Setup Boundary
  - **Assign Boundary**
    1. Change to edge selection mode by choosing: ***Edit > Select > Edges***
    2. Using the mouse, click on the top, right and bottom edges while holding down the CTRL key.
    3. Select the menu item ***Maxwell 2D > Boundary > Assign > Balloon***
    4. Click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

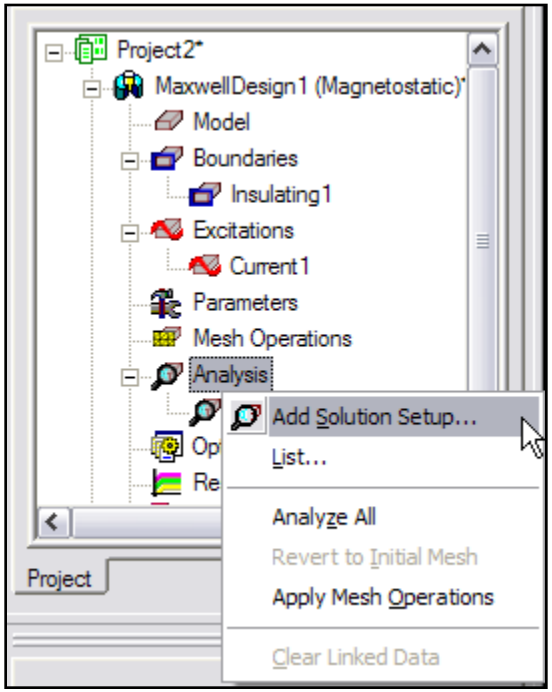




- Solution Setup - Creating an Analysis Setup
  - **To create an analysis setup:**
    1. Select the menu item ***Maxwell 2D> Analysis Setup > Add Solution Setup***
    2. Solution Setup Window:
       1. Click the **General** tab:
          - Maximum Number of Passes: **10**
          - Percent Error: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

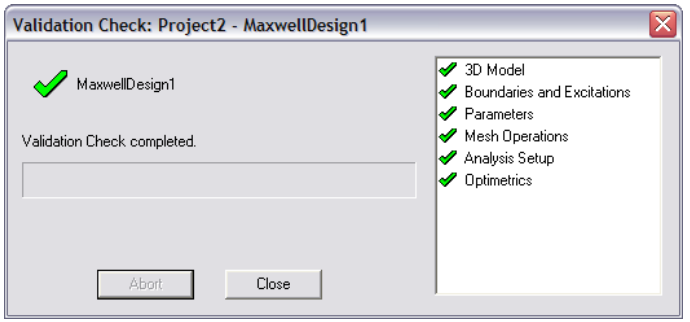




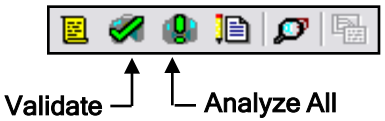


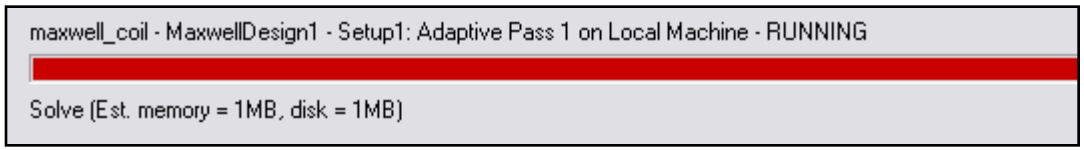
- ## Save Project
  - **To save the project:**
    1. In a Maxwell window, select the menu item ***File > Save As***.
    2. From the **Save As** window, type the Filename: **2D_simple_example**
    3. Click the **Save** button

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



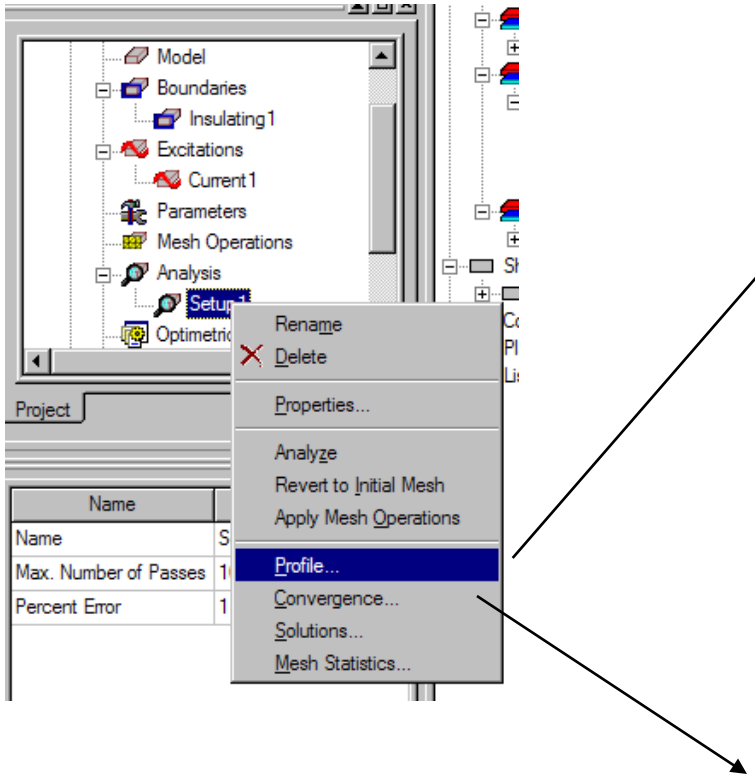
- ## Analyze
  - **To start the solution process:**
    1. Select the menu item ***Maxwell 2D> Analyze All***

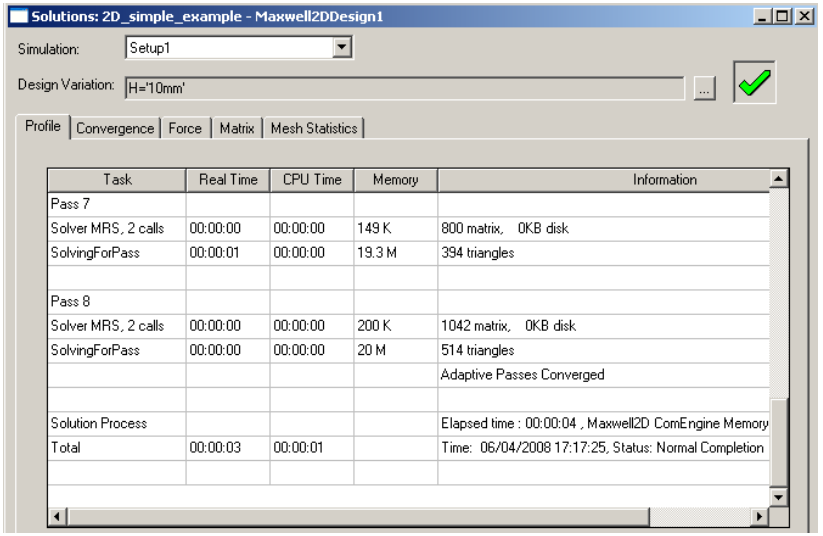


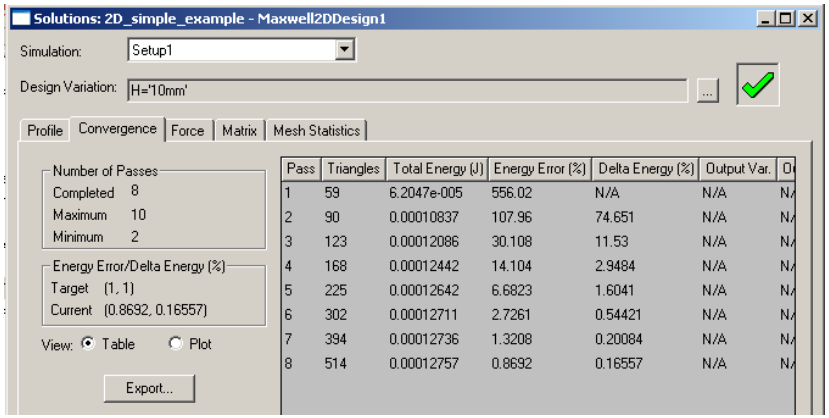


- View detailed information about the progress

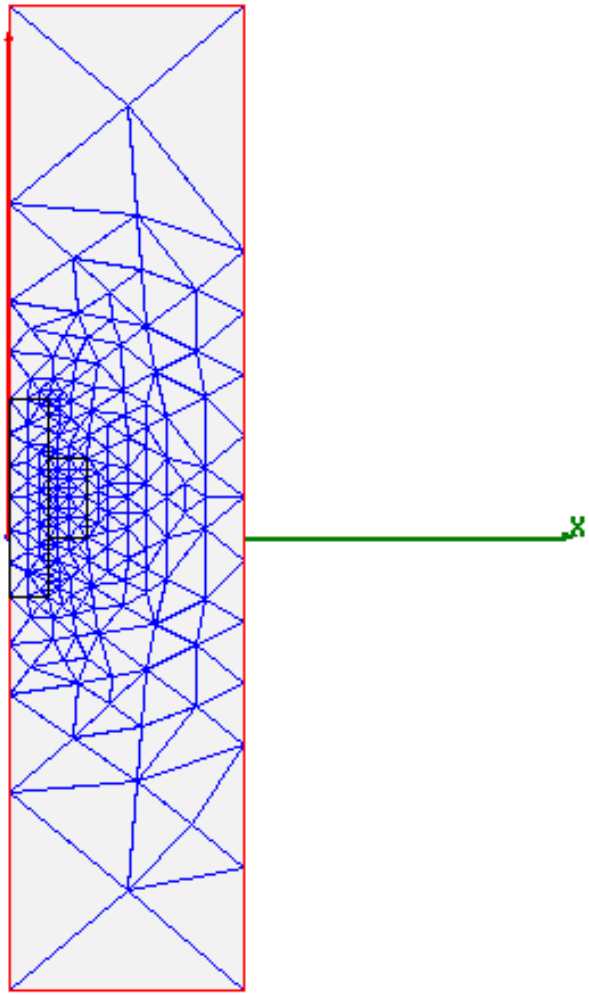
  - In the Project Tree click on **Analysis > Setup1** with the right mouse button und select **Profile**



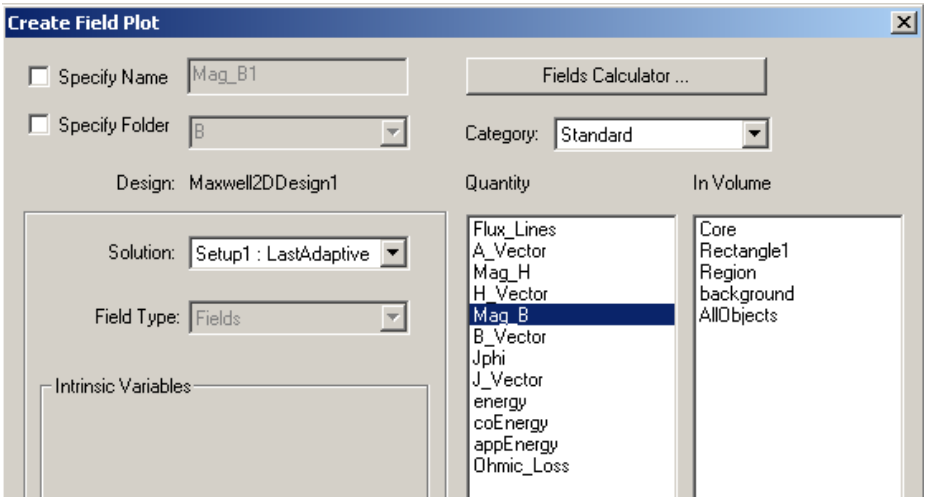


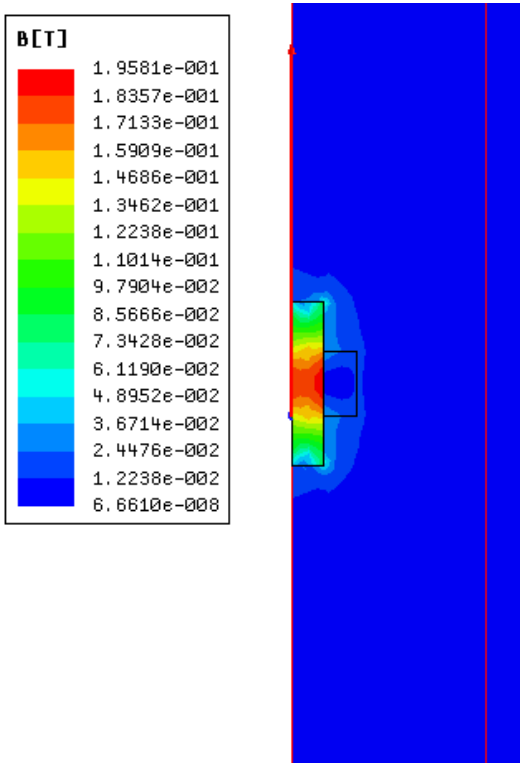


- Mesh Overlay
  - **Create a plot of the mesh**
    1. Select the menu item *Edit > SelectAll*
  - **To create a mesh plot:**
    1. Select the menu item *Maxwell 2D > Fields > Plot Mesh*
    2. Create Mesh Window:
       1. Click the **Done** button

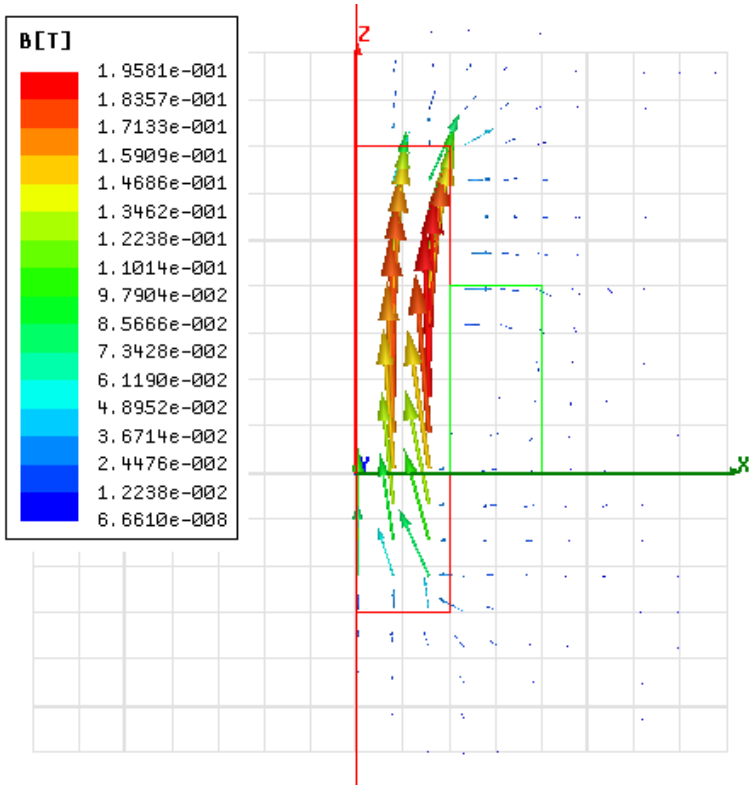


- Field Overlays
  - **To create a field plot:**
    1. In the object tree, select the plane for plotting:
       1. Using the Model Tree, expand **Planes**
       2. Select **Global:XZ**
    2. Select the menu item *Maxwell 2D> Fields > Fields > B > Mag_B*
    3. Create Field Plot Window
       1. Solution: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

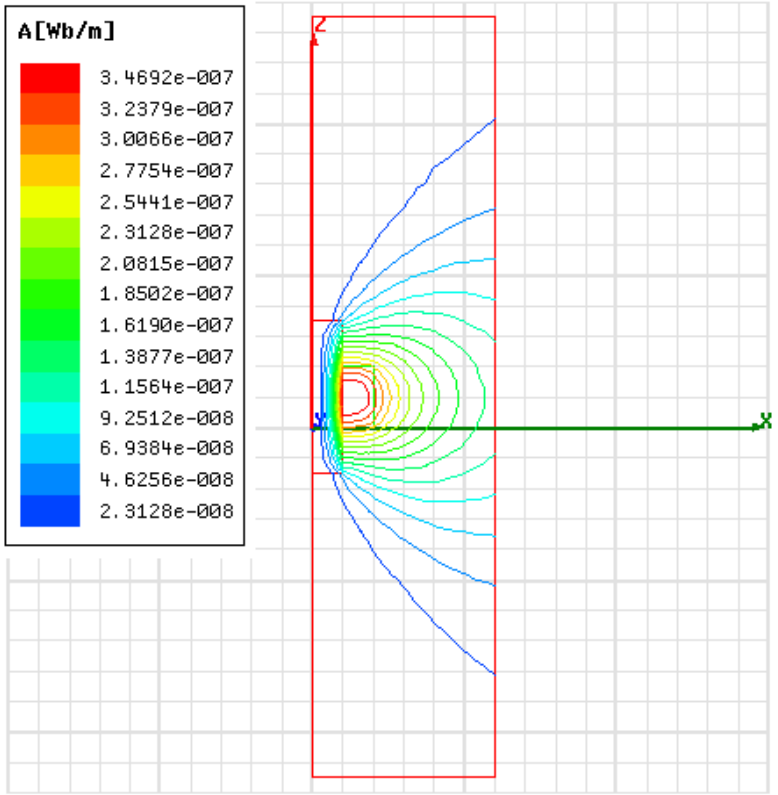




- Field Overlays (cont)
  - **Create another field plot:**
    1. In the object tree, select the plane for plotting:
       1. Using the Model Tree, expand **Planes**
       2. Select **Global:XZ**
    2. Select the menu item *Maxwell 2D> Fields > Fields > B > B_Vector*
    3. Create Field Plot Window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



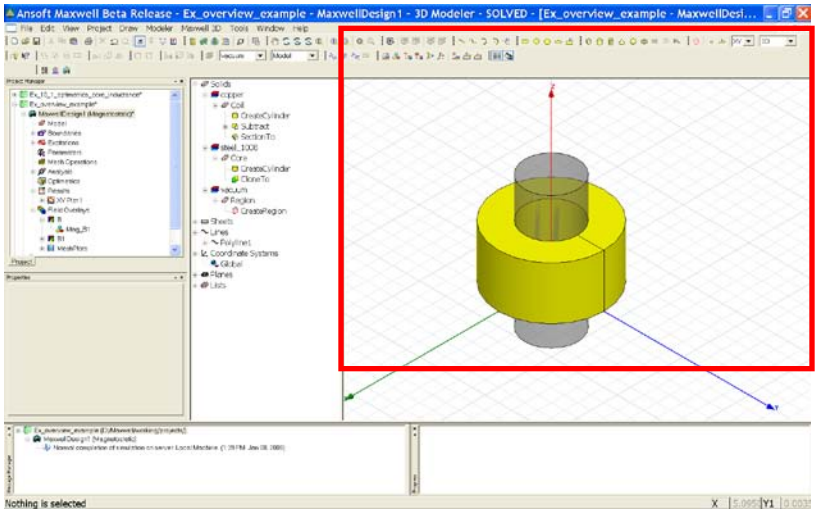
- Field Overlays (cont)
  - **Create another field plot:**
    1. In the object tree, select the plane for plotting:
       1. Using the Model Tree, expand **Planes**
       2. Select **Global:XZ**
    2. Select the menu item *Maxwell 2D> Fields > Fields > A > Flux_Lines*
    3. Create Field Plot Window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

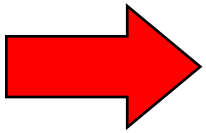


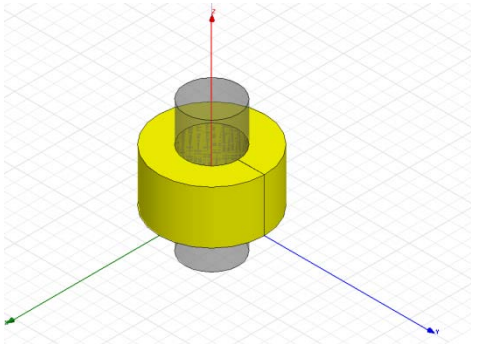
This completes the simple example.

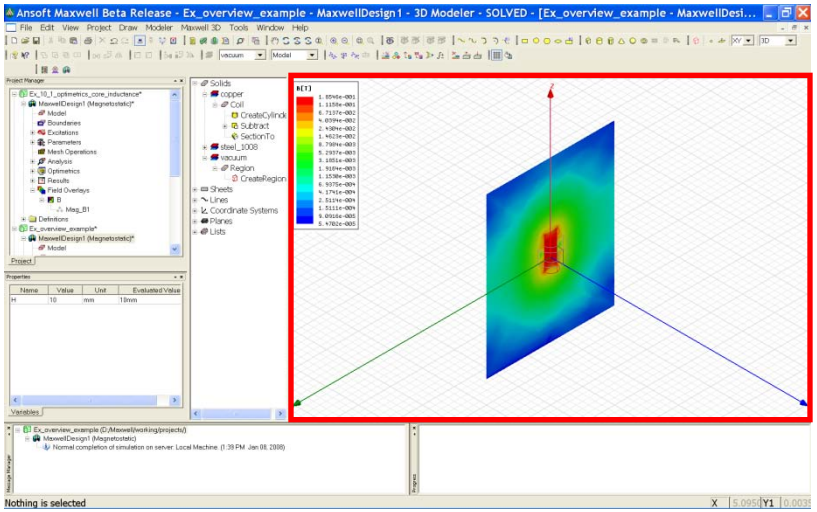
Screen Capturing

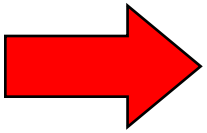
- To save the drawing Window or a plot to the clipboard select the menu item: ***Edit > Copy Image***
- In any Windows application, select: ***Edit > Paste*** to paste the image

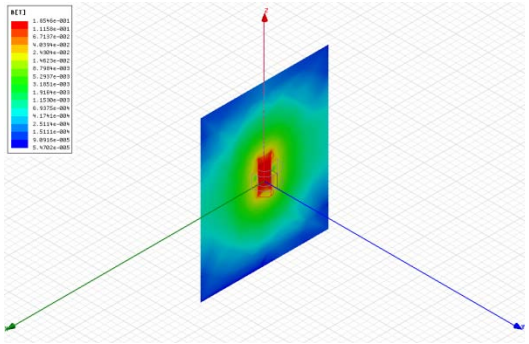




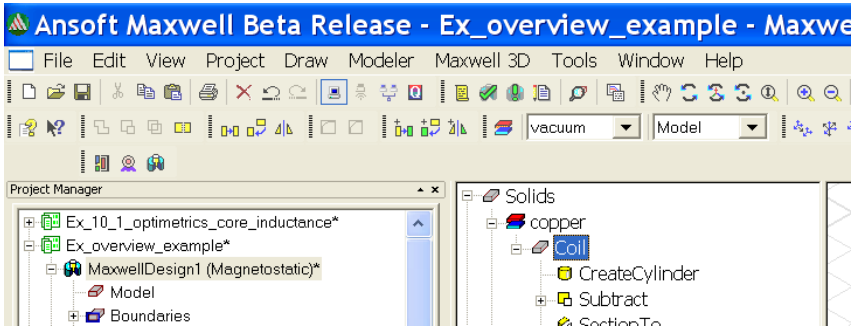


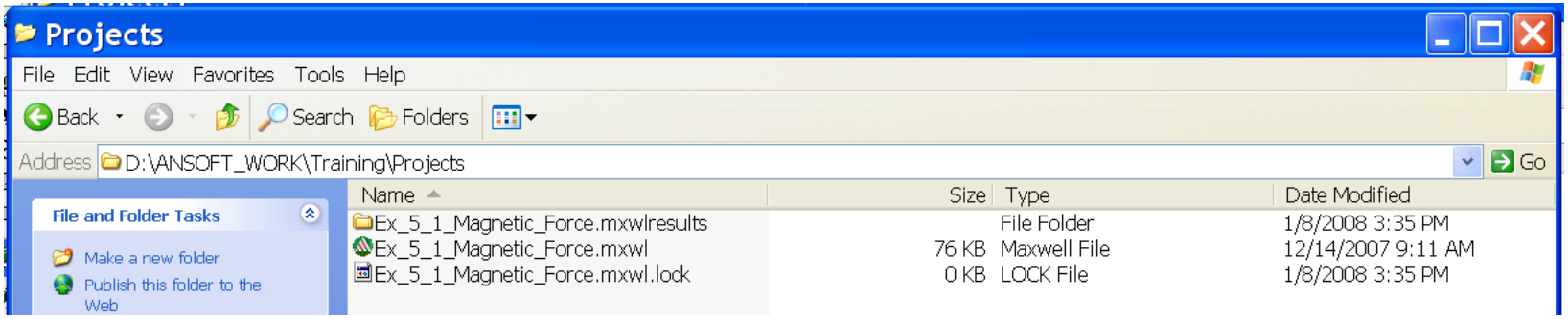




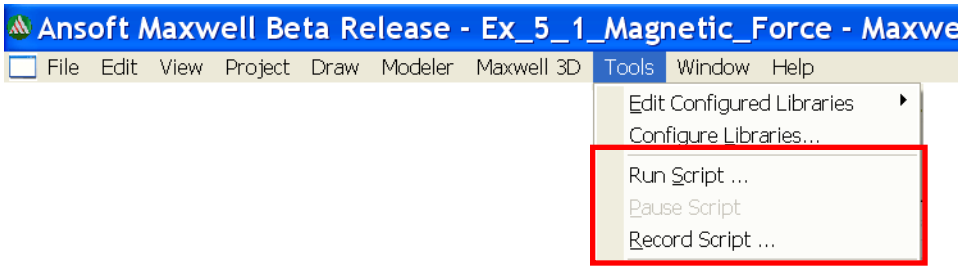


- File Structure
  - Everything regarding the project is stored in an ascii file
    - File: **<project_name>.mxwl**
    - Double click from Windows Explorer will open and launch Maxwell
  - Results and Mesh are stored in a folder named **<project_name>.mxwlresults**
  - Lock file: **<project_name>.lock.mxwl**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

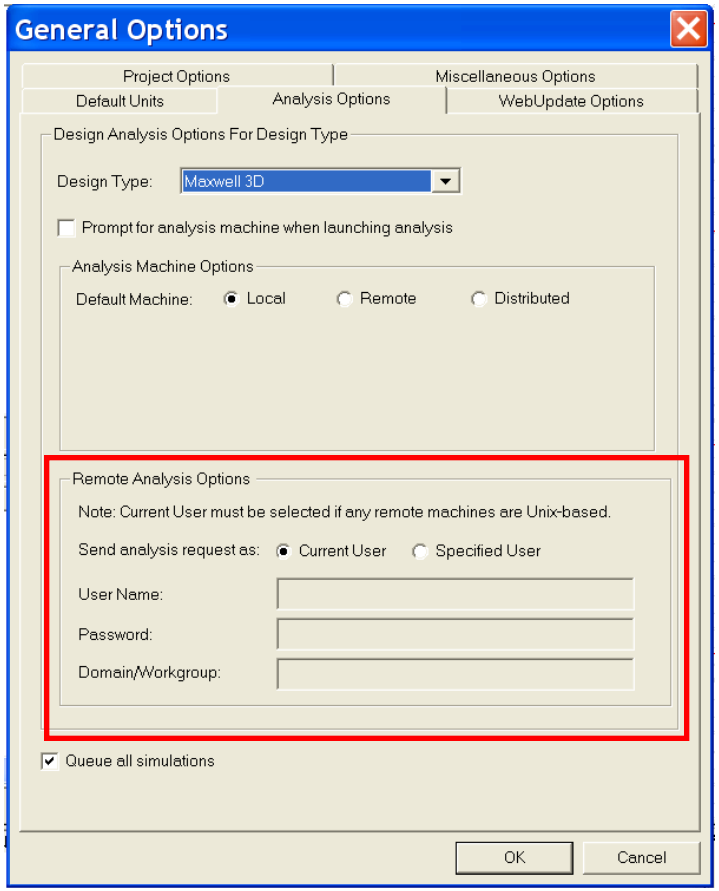




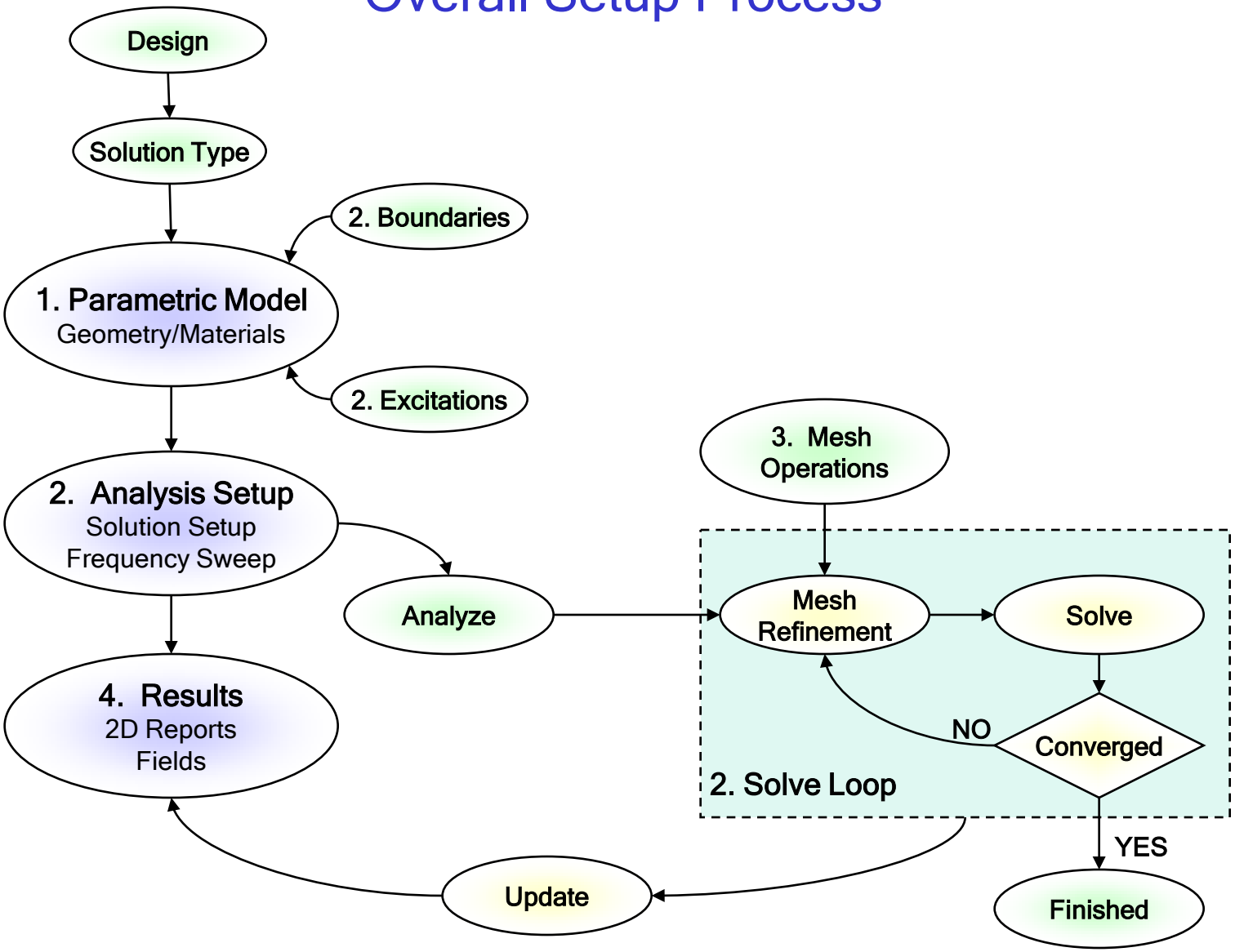
- Scripts
  - Default Script recorded in Maxwell
    - Visual Basic Script



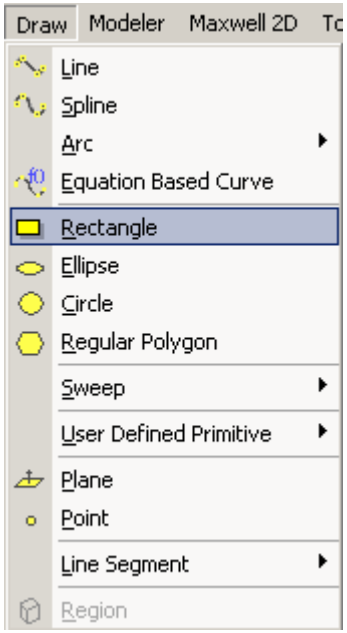
- Remote Solve (Windows Only)
  - *Tools > Options > General Options > Analysis Options*

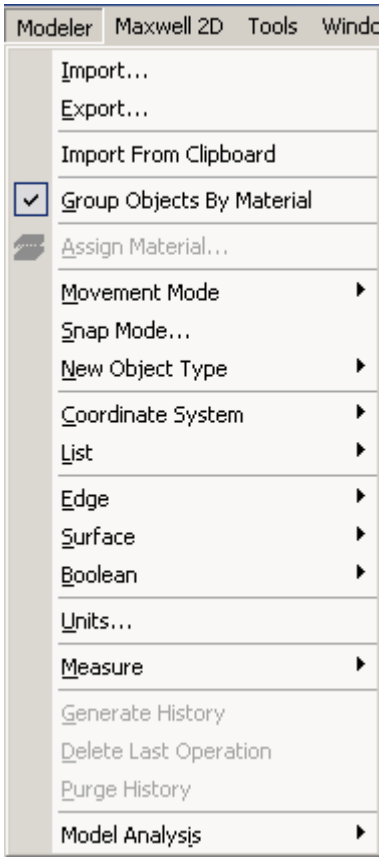


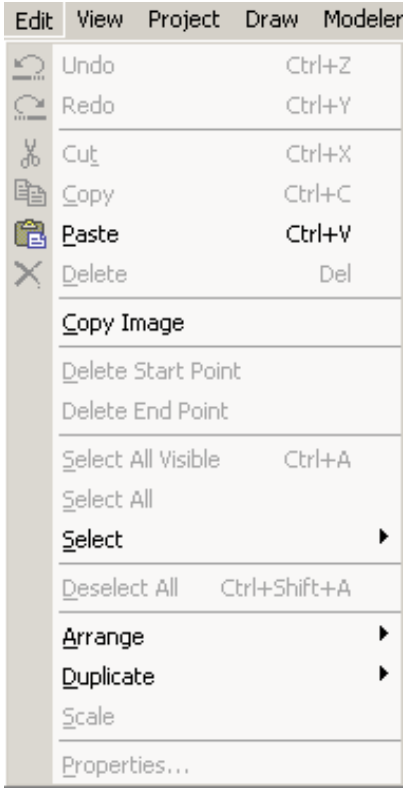
Overall Setup Process

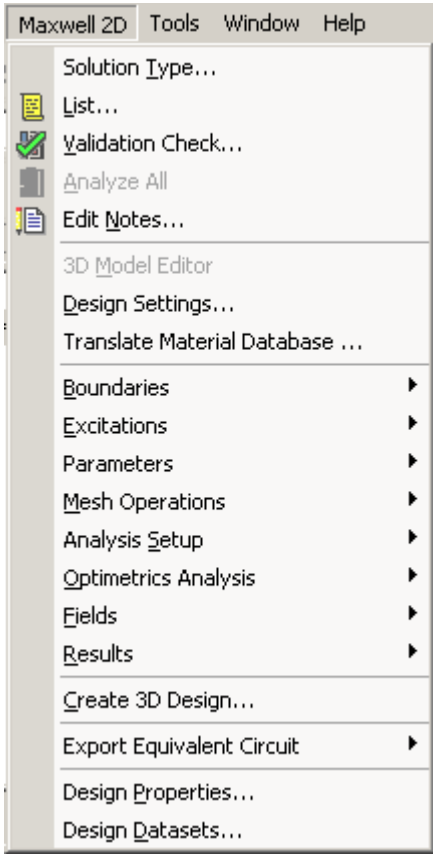


- Menu Structure
  - **Draw** – Primitives
  - **Modeler** – Settings and Boolean Operations
  - **Edit** – Copy/Paste, Arrange, Duplicate
  - **Maxwell 2D** – Boundaries, Excitations, Mesh Operations, Analysis Setup, Results

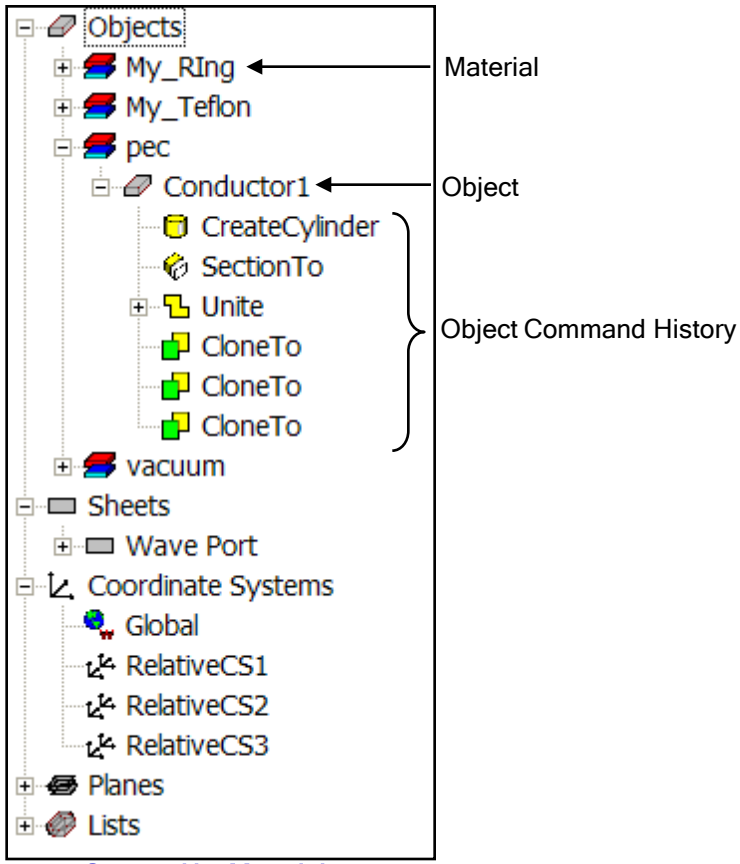




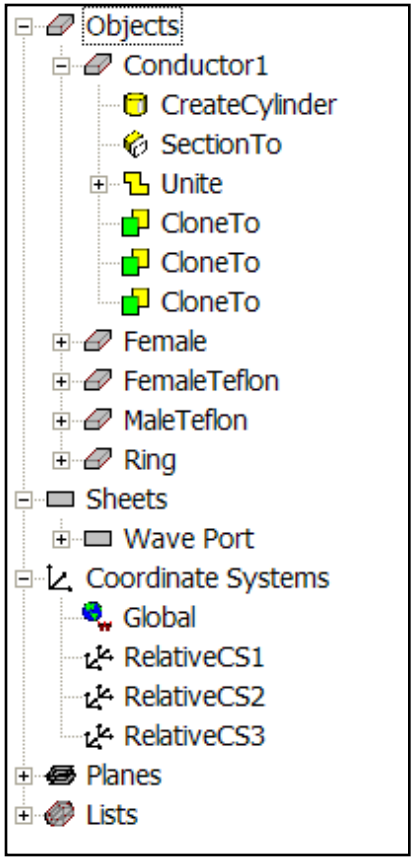




- Modeler - Model Tree
  - Select menu item *Modeler > Group by Material*

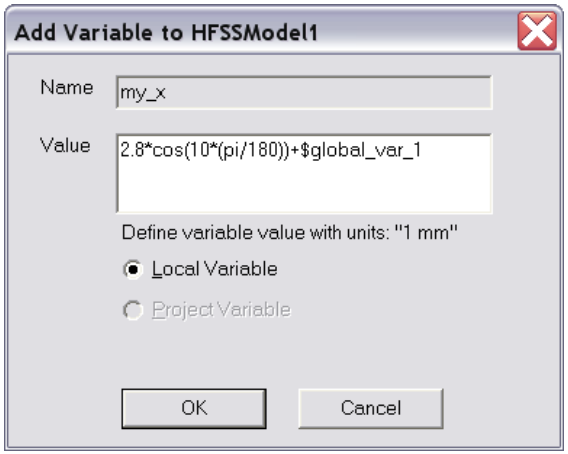


Grouped by Material



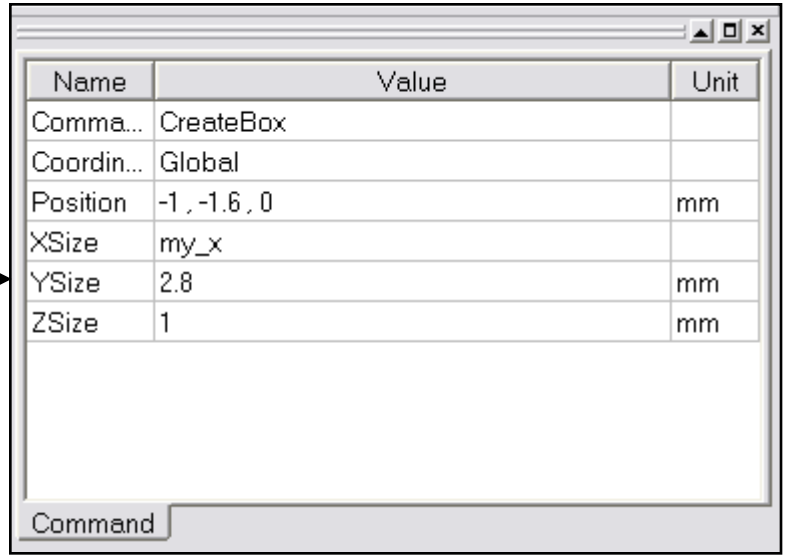
Object View

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

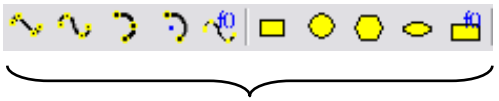


Name	Value	Unit
Command	CreateBox	
Coordinate System	Global	
Position	-1 , -1.6 , 0	mm
XSize	2.6	mm
YSize	2.8	mm
ZSize	1	mm

Command

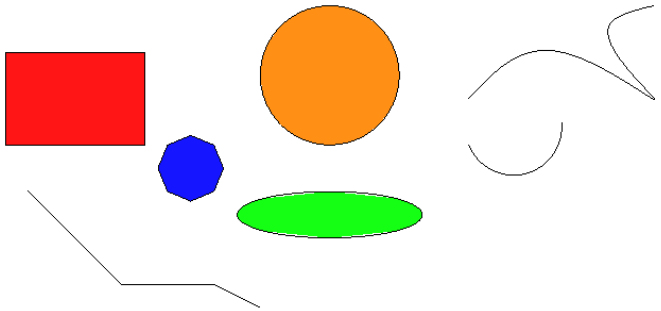


- Modeler – Primitives

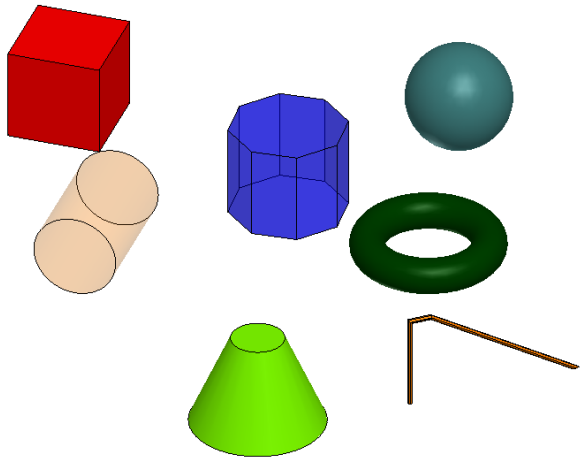


Toolbar: 2D Objects

  - **2D Draw Objects**
    - The following 2D Draw objects are available:
      - Line, Spline, Arc, Equation Based Curve, Rectangle, Ellipse, Circle, Regular Polygon, Equation Based Surface

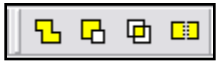


  - **3D Draw Objects**
    - Note that 3D objects can be pasted into the 2D model window, but they are ignored by the solution
    - The following 3D Draw objects are available (in Maxwell 3D):
      - Box, Cylinder, Regular Polyhedron Cone, Sphere, Torus, Helix, Spiral, Bond Wire

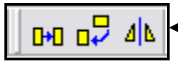


  - **True Surfaces**
    - Circles, Cylinders, Spheres, etc are represented as true surfaces. In versions prior to release 11 these primitives would be represented as faceted objects. If you wish to use the faceted primitives, select the Regular Polyhedron or Regular Polygon.

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


Toolbar: Boolean

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

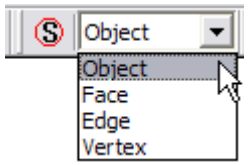

Toolbar: Arrange

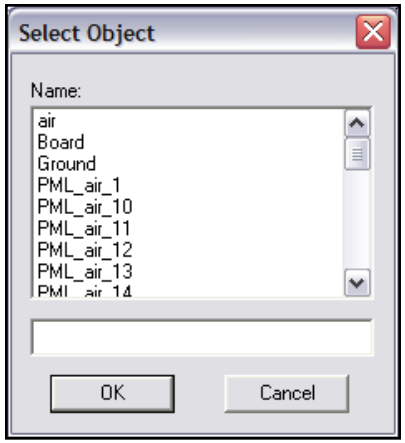
  - *Edit > Duplicate >*
    - **Along Line** – Create multiple copies of an object along a vector
    - **Around Axis** – Create multiple copies of an object rotated by a fixed angle around the x, y, or z axis
    - **Mirror** – Mirrors the shape around a specified plane and creates a duplicate


Toolbar: Duplicate

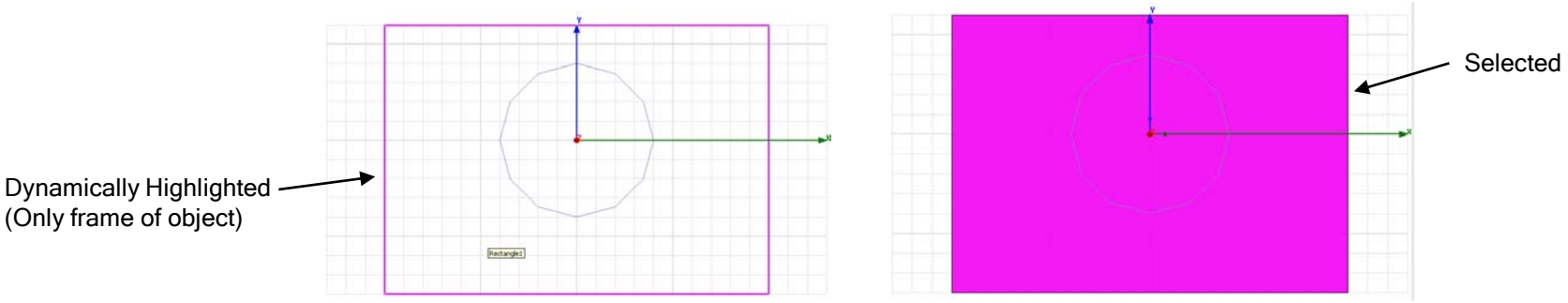
  - *Edit > Scale* – Allows non-uniform scaling in the x, y, or z direction

- Modeler - Selection
  - Selection Types
    - Object (Default)
    - Face
    - Edge
    - Vertex

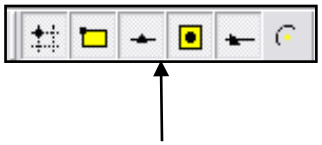




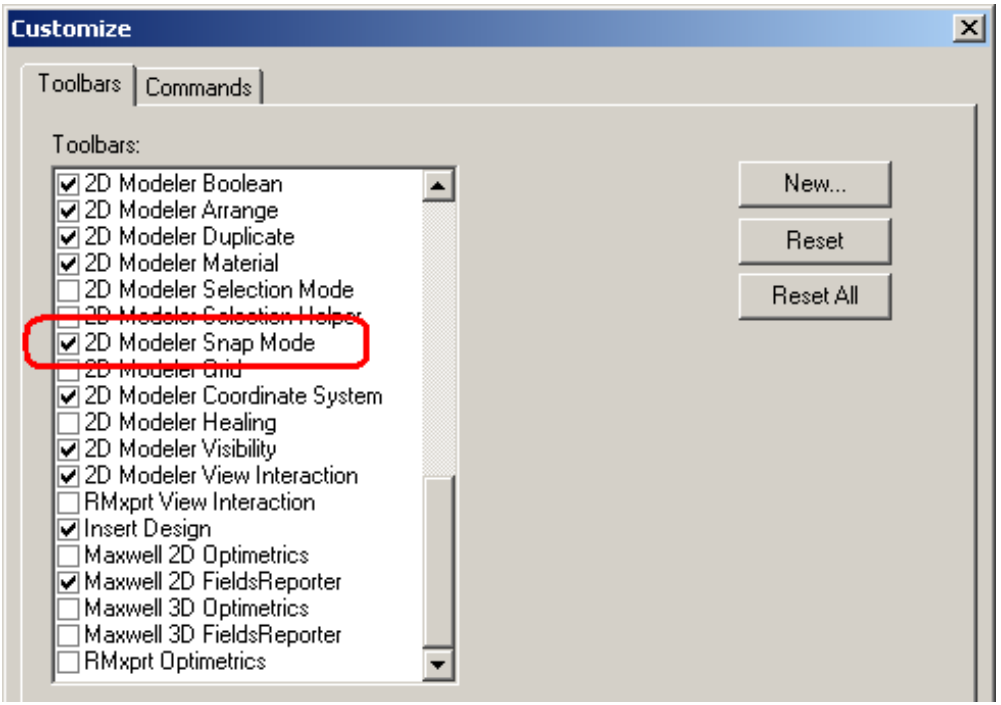
  - Selection Modes
    - All Objects
    - All Visible Object
    - By Name
  - Highlight Selection Dynamically – By default, moving the mouse pointer over an object will dynamically highlight the object for selection. To select the object simply click the left mouse button.
    - **Multiple Object Selection** – Hold the **CTRL** key down to graphically select multiple objects
    - **Next Behind** – To select an object located behind another object, select the front object, press the **b** key to get the next behind. Note: The mouse pointer must be located such that the next behind object is under the mouse pointer.
    - **To Disable:** Select the menu item *Tools > Options > Modeler Options*
      - From the **Display Tab**, uncheck **Highlight selection dynamically**

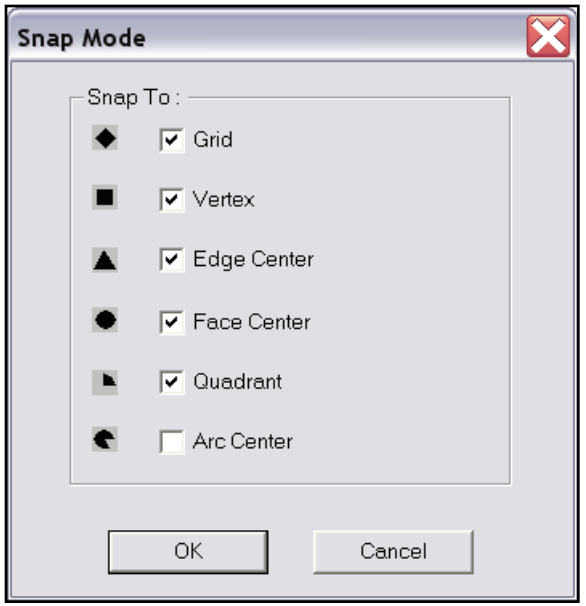


- Modeler – Moving Around
  - Modeler > Snap Mode to set the snaps
  - Tools > Customize…
    Snap Mode to view Snap Mode toolbar

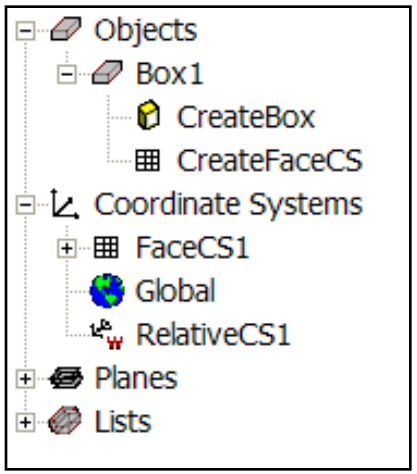


Toolbar: Snap Mode



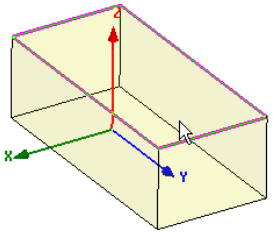


- Modeler – Coordinate Systems
  - Can be Parameterized
  - **Working Coordinate System**
    - Currently selected CS. This can be a local or global CS
  - **Global CS**
    - The default fixed coordinate system
  - **Relative CS**
    - User defined local coordinate system.
      - Offset
      - Rotated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

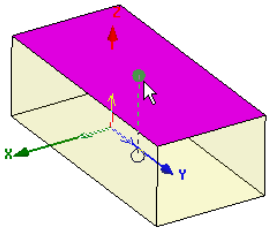




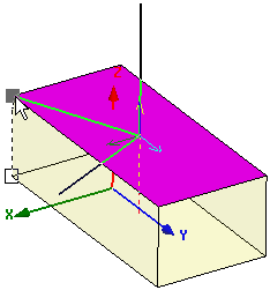
Toolbar: C oordinate System



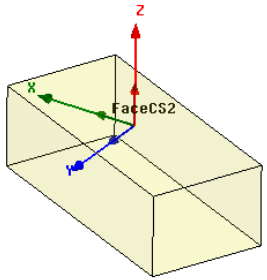
Step 1: Select Face



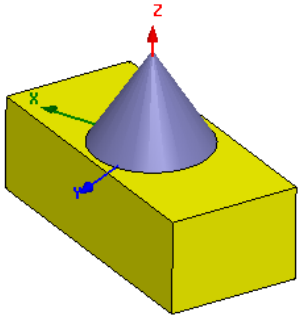
Step 2: Select Origin



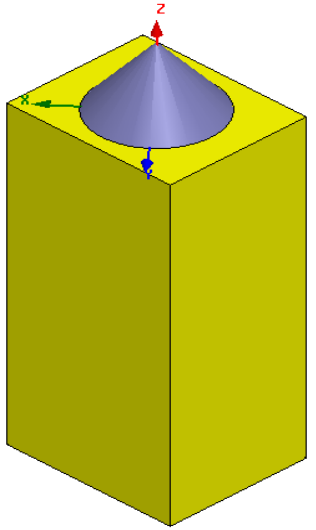
Step 3: Set X-Axis



New Working CS

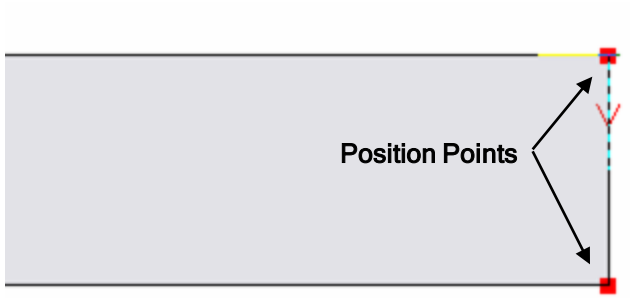


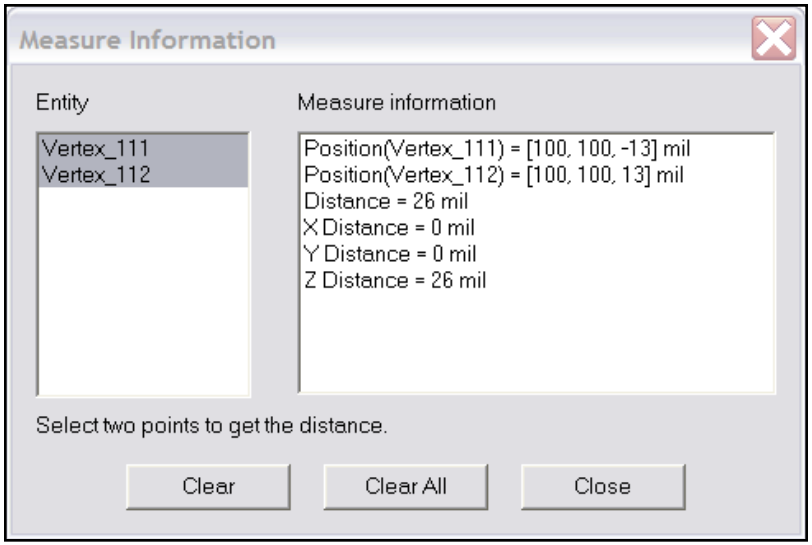
Cone created with Face CS



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

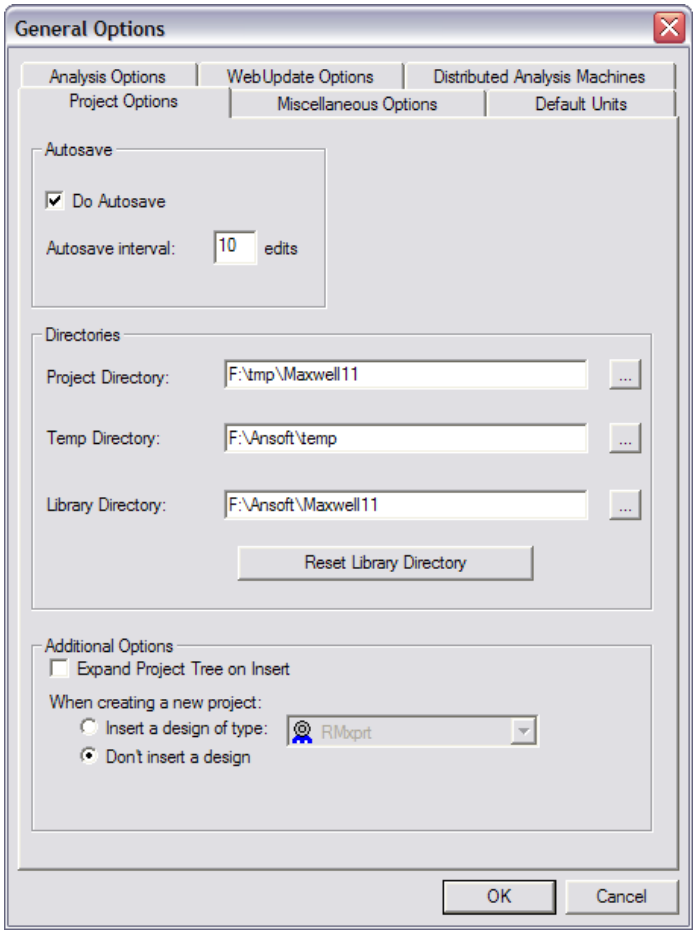
- 2D Measure
  - *Modeler > Measure >*
    - *Position* - Location, Distance, and Area
    - *Edge* - Edge Length
    - *Face* - Surface Area
    - *Object* - Surface Area, Object Volume

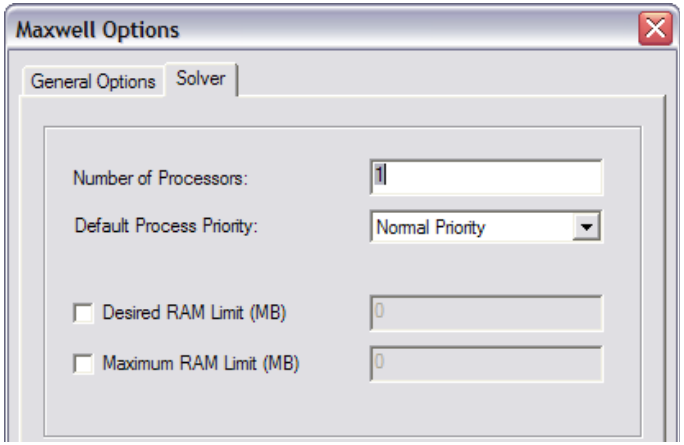




- Options - General
  - *Tools > Options > General Options > Project Options*
    - **Temp Directory** – Location used during solution process
      - Make sure it has at least 512MB free disk.

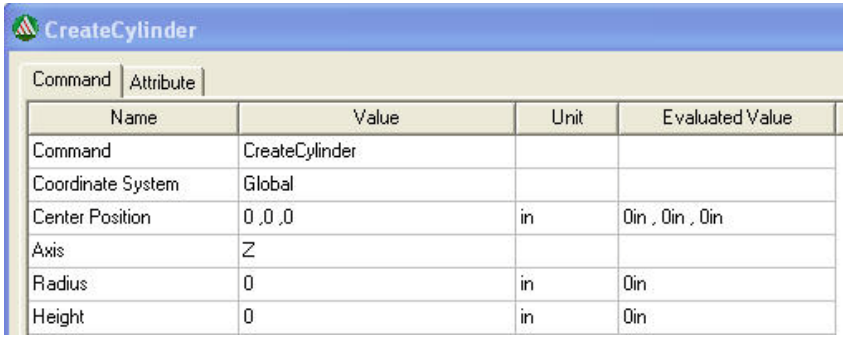
- Options - Maxwell
  - *Tools > Options > Maxwell Options > Solver*
    - Set Number of Processors = 2 for 1 dual-core processor or two single-core processors. Requires additional license
    - Default Process Priority – set the simulation priority from Critical (highest) to Idle (lowest)





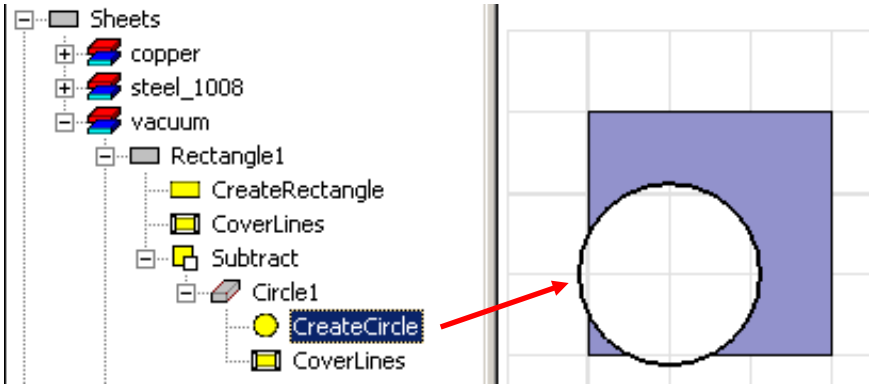
- Options – Modeler Options
  - *Tools > Options > Modeler Options > Drawing* for Point and Dialog Entry Modes
  - Can enter in new dimensions using either Point (mouse) or Dialog entry mode
  - Alternatively use F3 and F4 to switch between Point and Dialog entry modes

Typical "Dialog" entry mode window

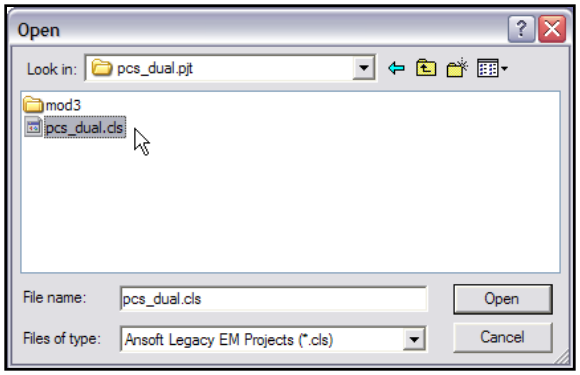


Name	Value	Unit	Evaluated Value
Command	CreateCylinder		
Coordinate System	Global		
Center Position	0 ,0 ,0	in	0in , 0in , 0in
Axis	Z		
Radius	0	in	0in
Height	0	in	0in

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

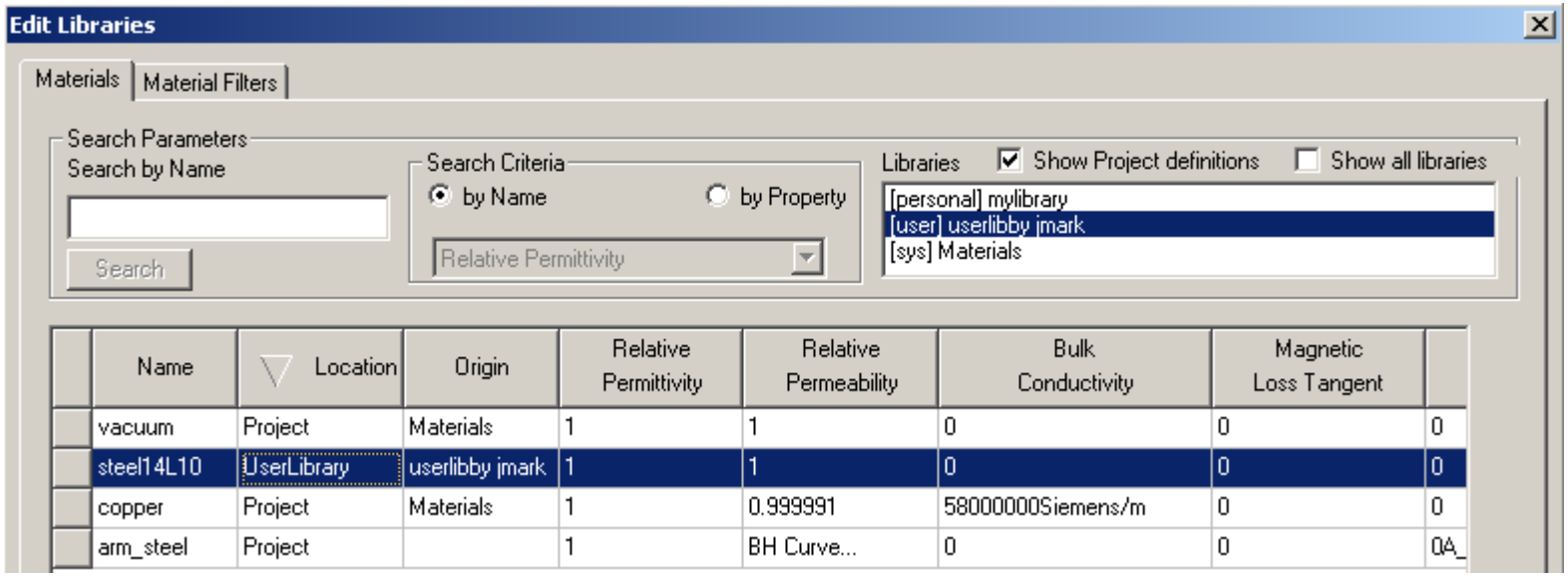


- Converting Older Maxwell Projects
  - **From Maxwell v 11 and older**,
    1. Select the menu item ***File > Open***
    2. Open dialog
       1. Files of Type: **Ansoft Legacy EM Projects (.cls)**
       2. Browse to the existing project and select the .cls file
       3. Click the **Open** button

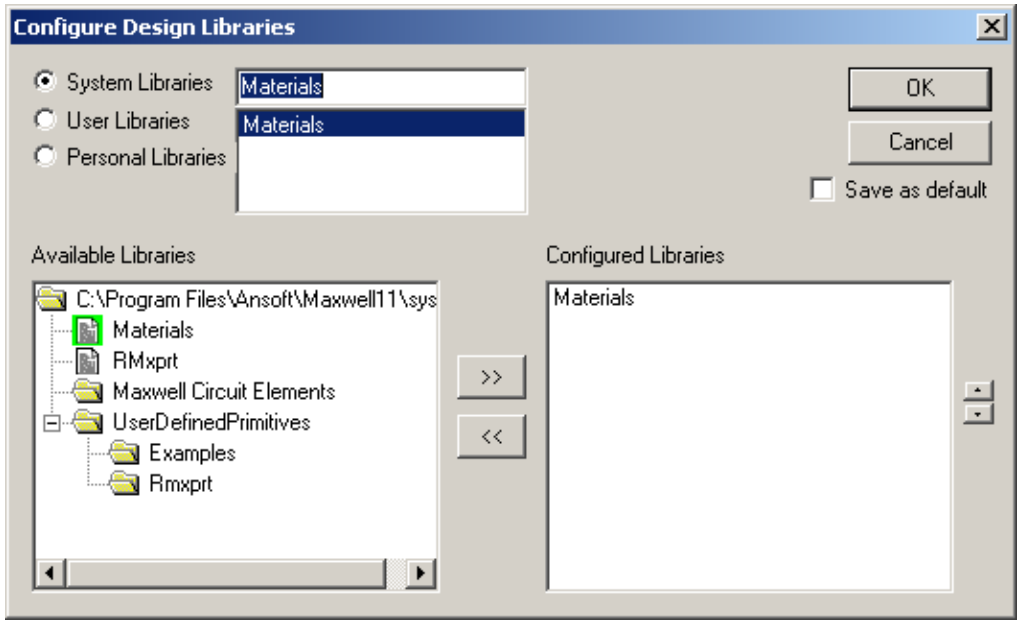


  - **What is Converted?**
    - Converts Entire Model: Geometry, Materials, Boundaries, Sources and Setup
    - *Solutions, Optimetrics projects and Macros are not converted*

- Material Setup - Libraries
  - *3-Tier library structure*
    - *System (global) level – predefined from 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

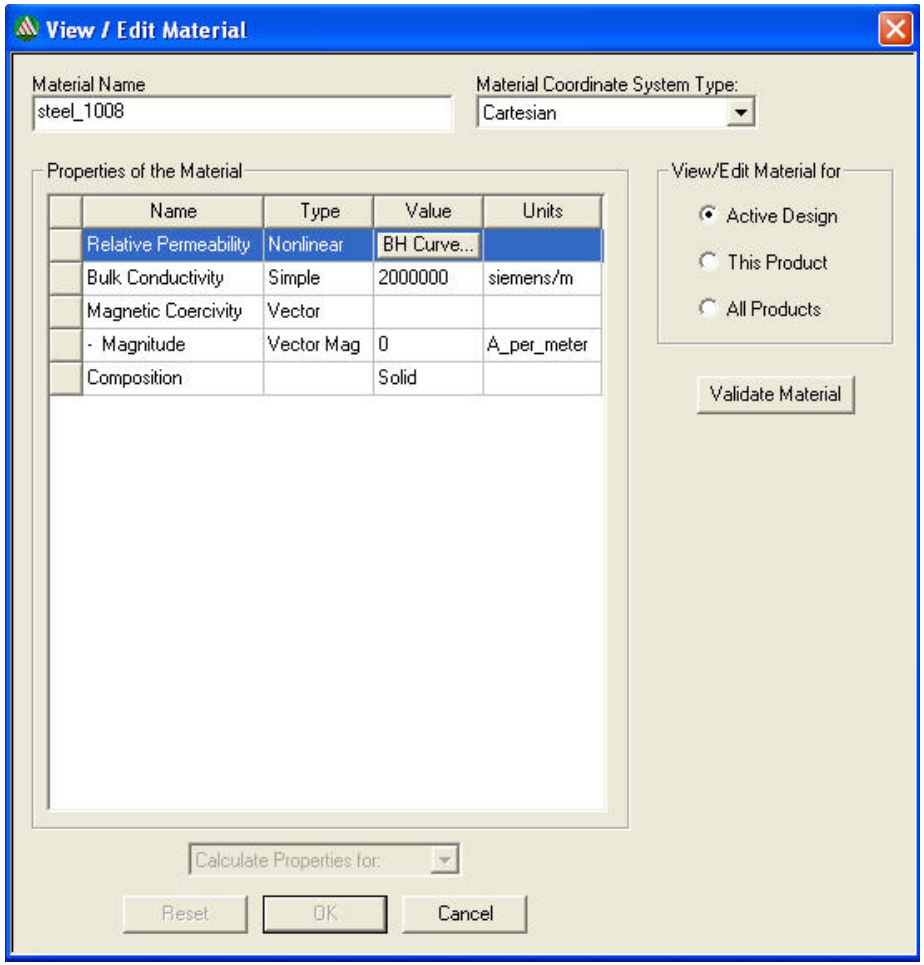


- Click “Add Material …”. The Material is only available in Project
- To add a material in the user or personal library: click on “Export Library” and save it in the desire library.
- In the main project window, click on *Tools > Configured Libraries*. Locate the library to have the material available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



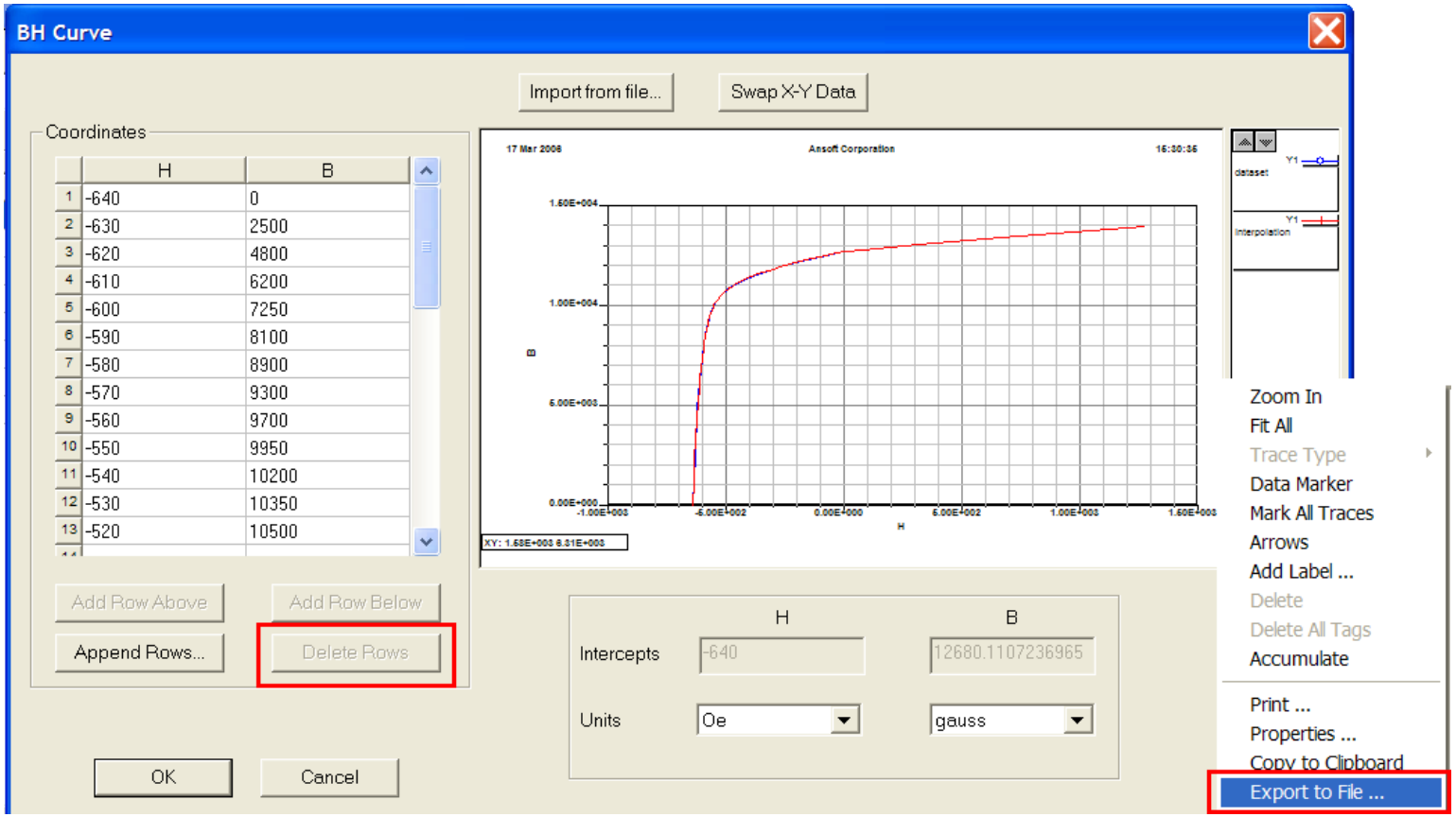
- Materials Setup - Editing



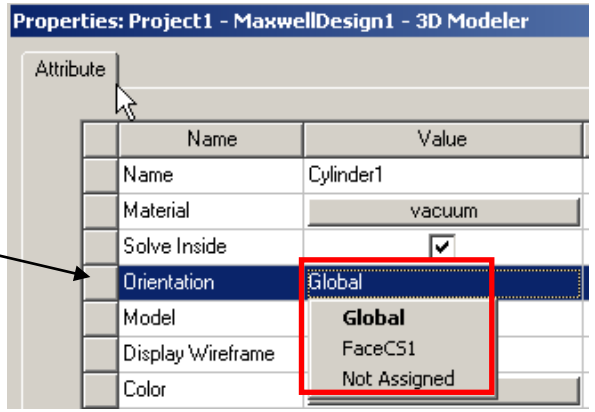


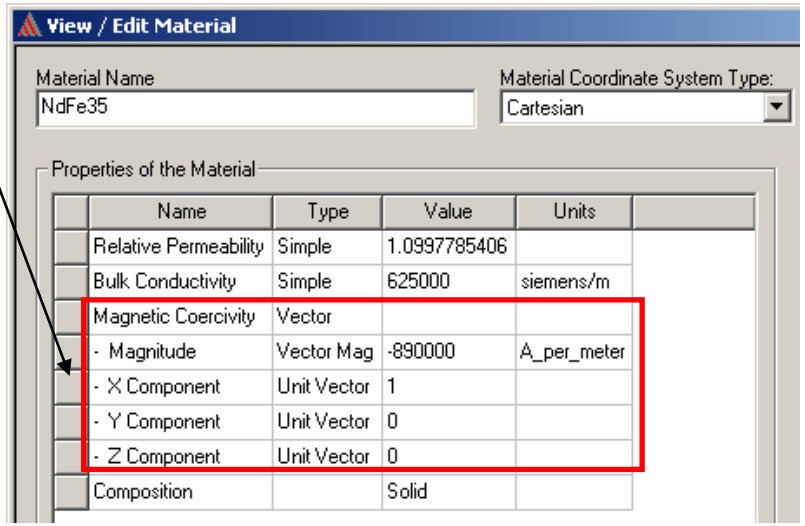
Material Setup – BH curve

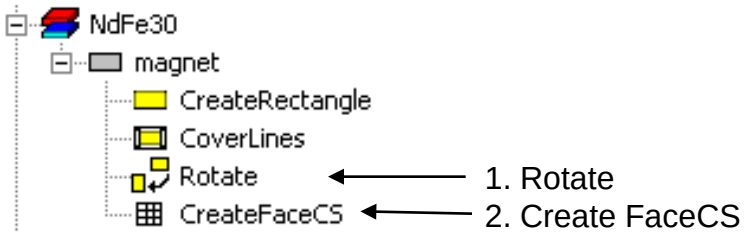
- Robust BH curve entry – can delete points if you make a mistake
- Can import data from a file
- To export BH curve for use in future, right-mouse-click on curve and select Export to File…



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







- Material Setup - Anisotropic Material Properties

  - ε_1, μ_1, and σ_1 are tensors in the X direction.
  - ε_2, μ_2, and σ_2 are tensors in the Y direction.
  - ε_3, μ_3, and $\sigma 3$ are tensors in the Z direction.

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

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

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

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

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

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

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

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

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

Magnetic Field Sources (Magnetostatic)

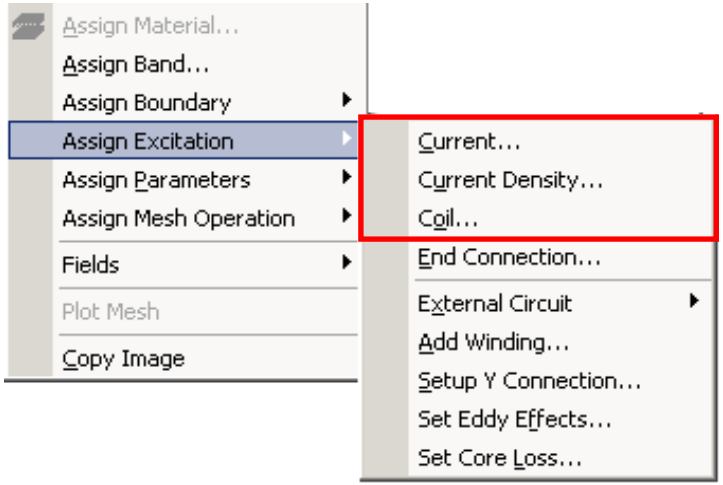
Source	Type of Excitation
Current	The total current in a conductor.
Current Density	The current density in a conductor.
Notes: In the Magnetostatic solver, current is RMS ampturns. Permanent magnets will also act as a source in the Magnetostatic solver.	

Magnetic Field Sources (Eddy Current)

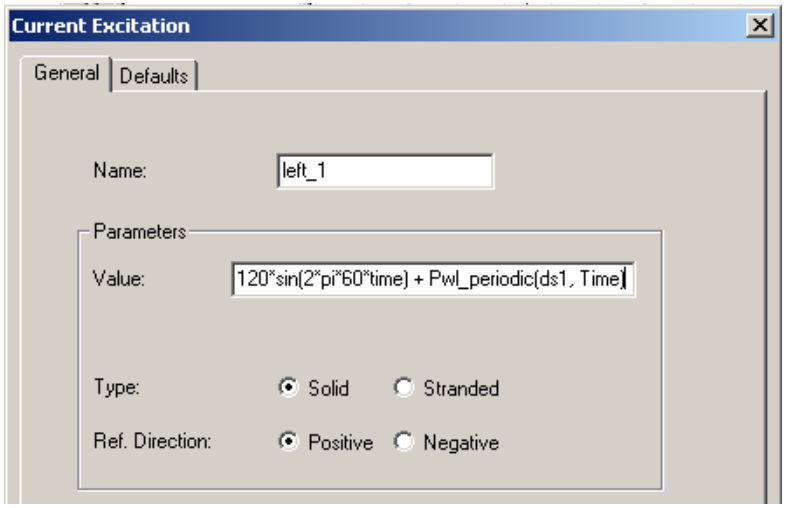
Source	Type of Excitation
Current	The total current in a conductor.
Parallel Current	The total current in a a group of parallel conductors.
Current Density	The current density in a conductor.
Notes: In the Eddy Current solver, current is peak amp-turns. Sources can be solid (with eddy effects) or stranded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

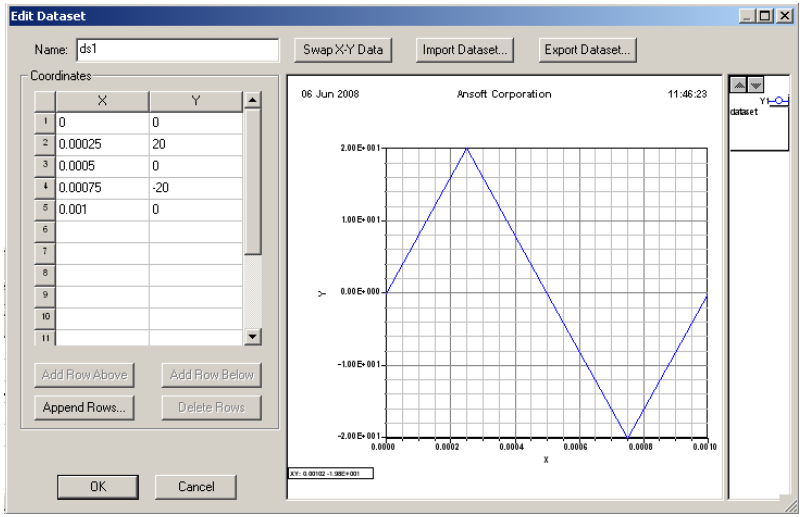
Magnetic Field Sources (Transient)

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



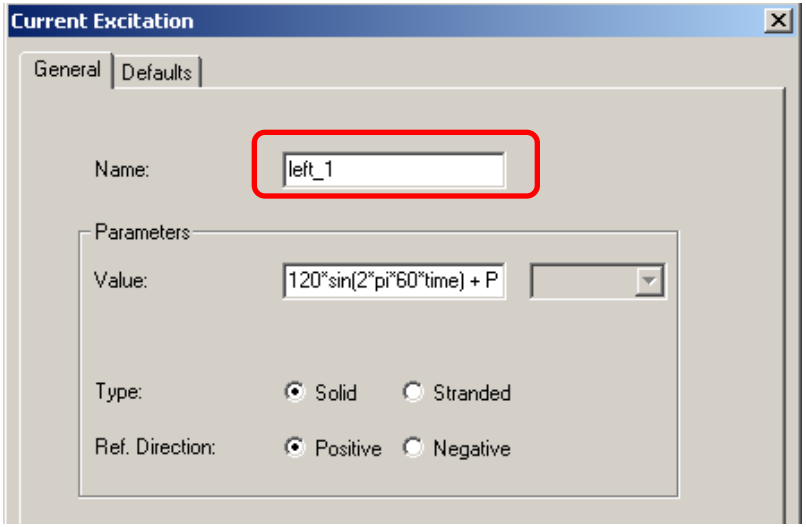
- Current and voltage sources (solid or stranded) can be constant or functions of intrinsic variables: speed (rpm or deg/sec), position (degrees), or time (seconds)
- Dataset function can be used for piecewise linear functions: Pwl_periodic (ds1, Time)





Magnetic Field Sources (Transient)

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



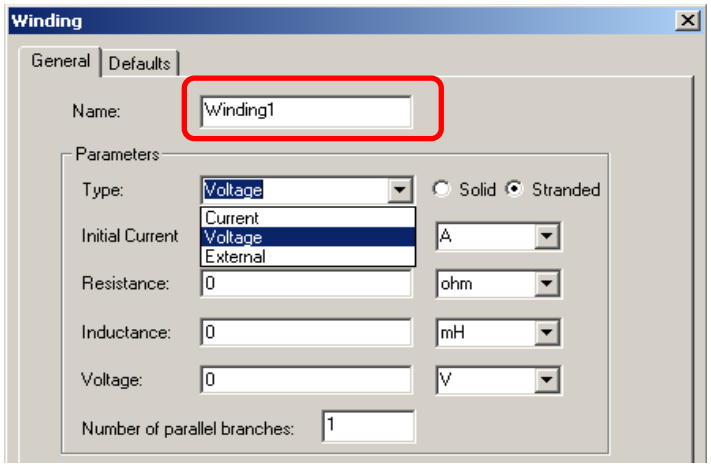


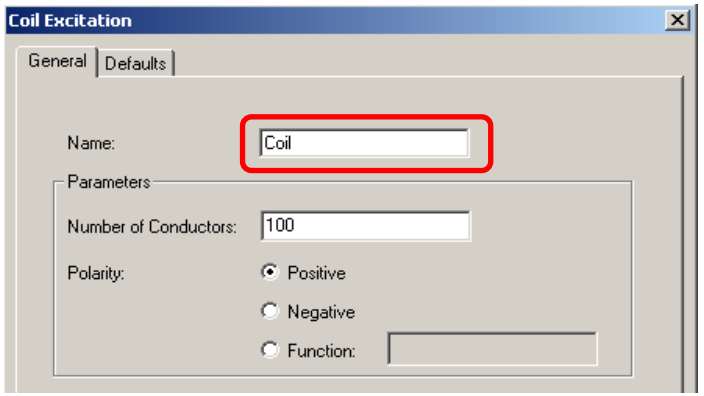
Magnetic Field Sources (Transient)

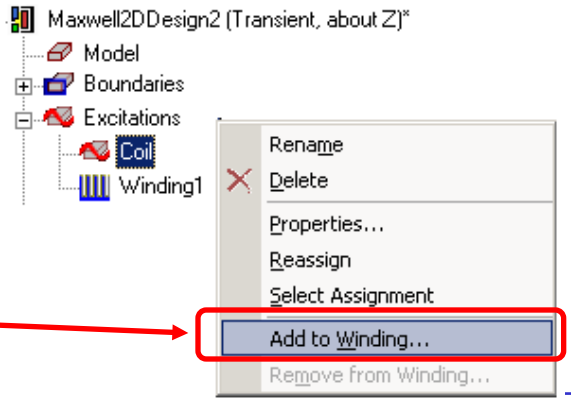
- Maxwell 2D > Excitation > Add Winding
  - Current – applies current in amps
    - Solid or Stranded
    - Input current and number of parallel branches as seen from terminal
  - Voltage – applies voltage (total voltage drop over the length of a solid conductor or the entire winding)
    - Solid or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

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

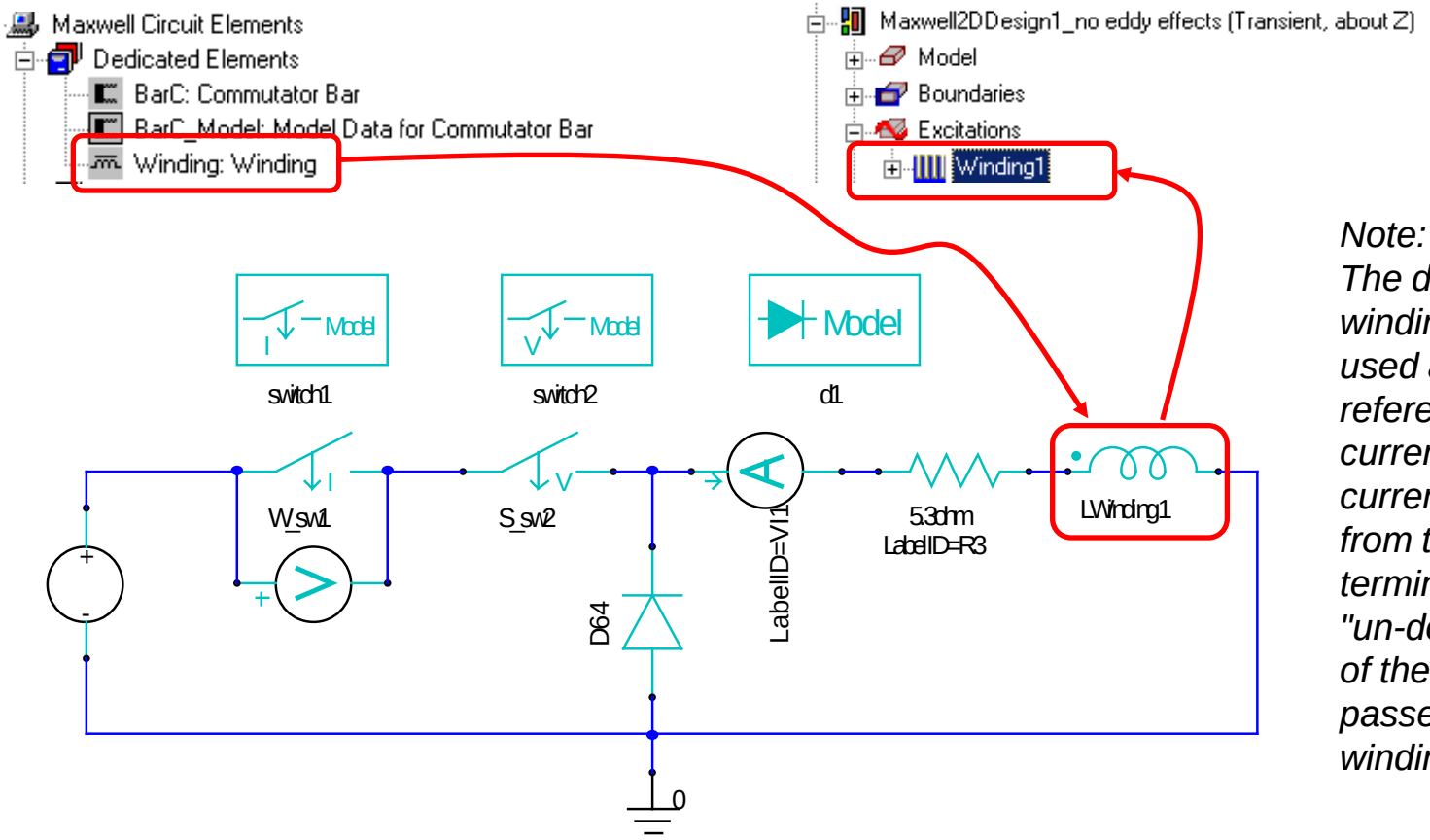
  Note: Windings in the XY solver will usually have 2 coils: one positive and one negative polarity. Both coils will be added to the appropriate winding by right-mouse clicking on **Coil** in the project tree and choosing **Add to Winding**





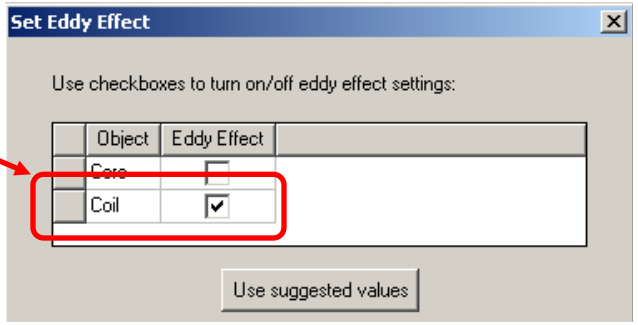


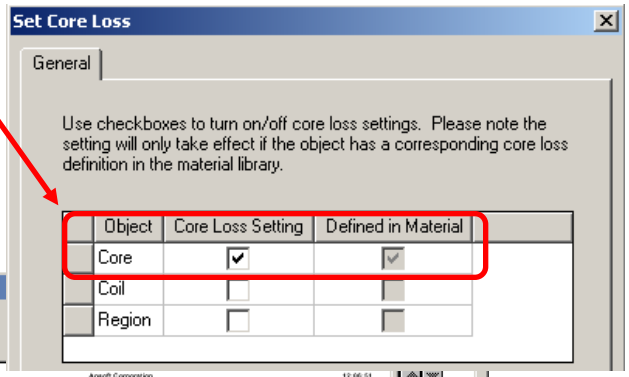
- To Create an External Circuit
  1. Select: *Maxwell2D > Excitations > External Circuit > Edit External Circuit > Import Circuit*
  2. After circuit editor opens, add elements to construct the circuit. Note that the name of the Winding in the circuit (Winding1) must match the name of the Winding in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

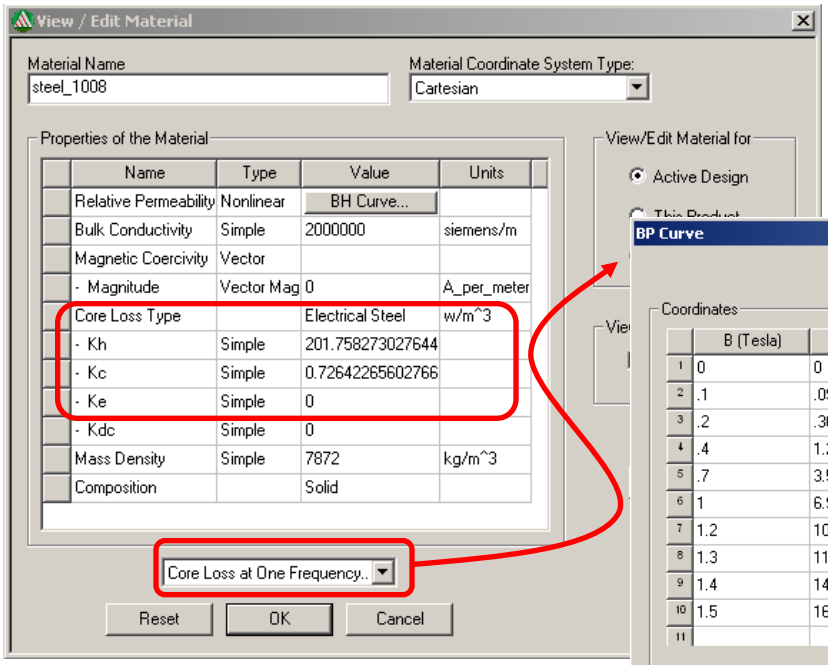


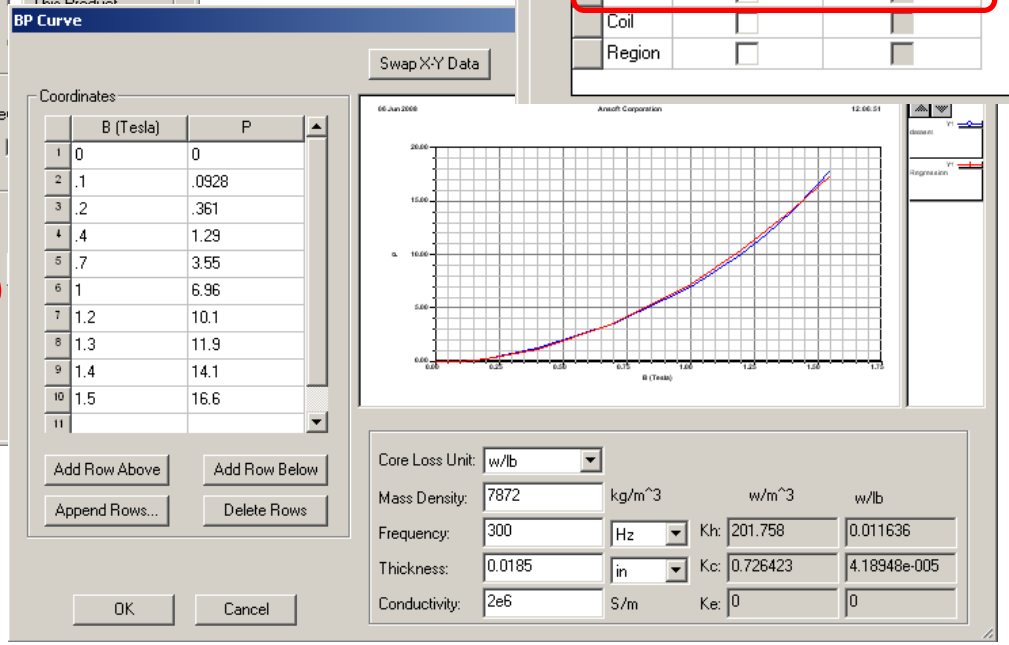
Note:
The dot on the winding symbol is used as the positive reference for the current (positive current is oriented from the "dotted" terminal towards to "un-dotted" terminal of the winding as it passes through the winding).

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









Core Loss Calculation Method

- The core loss for electrical steel is based on:

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

where:

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

- The power ferrite core loss is based on:

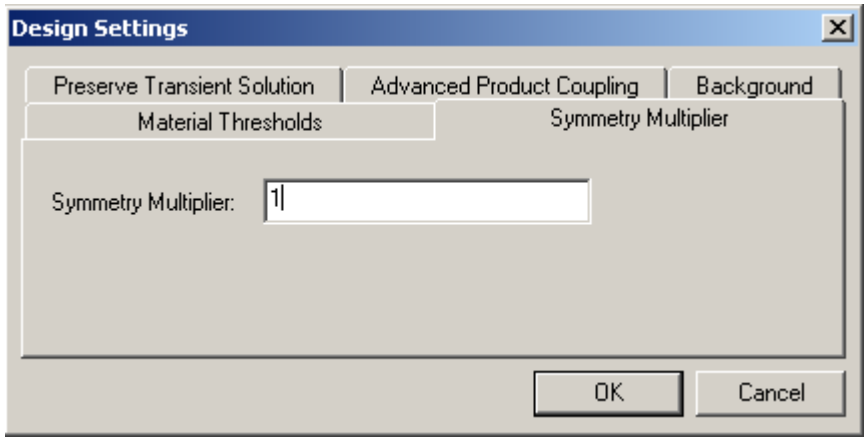
$$p = C_m f^x B_{\max}^y$$

where:

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

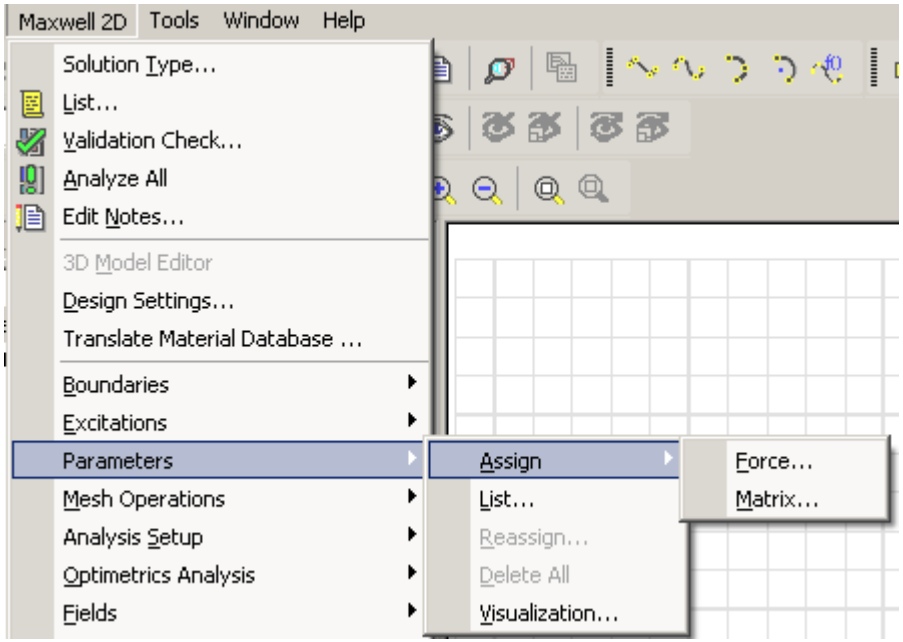
Maxwell 2D > Design Settings

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

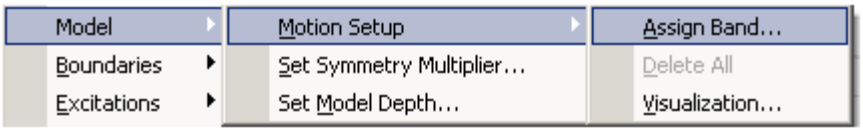


- Set the Material Threshold for treating materials as conductors vs. insulators.
- Set Preserve Transient Solution options (For Transient Solutions Only).
- Set transient coupling with Simplorer on the Advanced Product Coupling tab (For Transient Solutions Only)
- Set the Model Depth (Maxwell2D XY Transient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

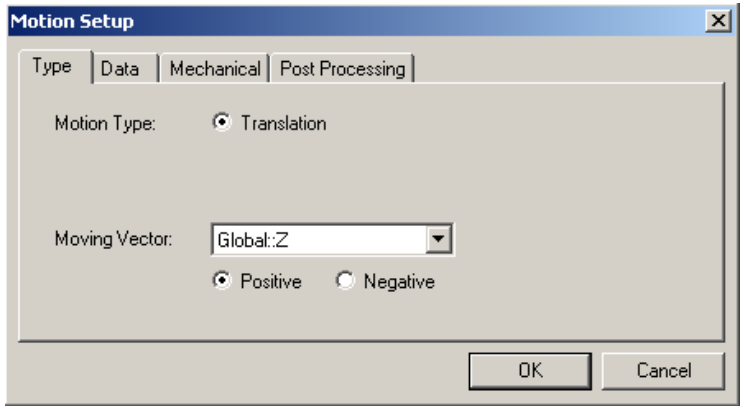
- Maxwell 2D > Parameters
  - Allows the automatic calculation of parameters following the field solution
  - Includes: Force, Torque, Flux linkage, Core loss, and Matrix

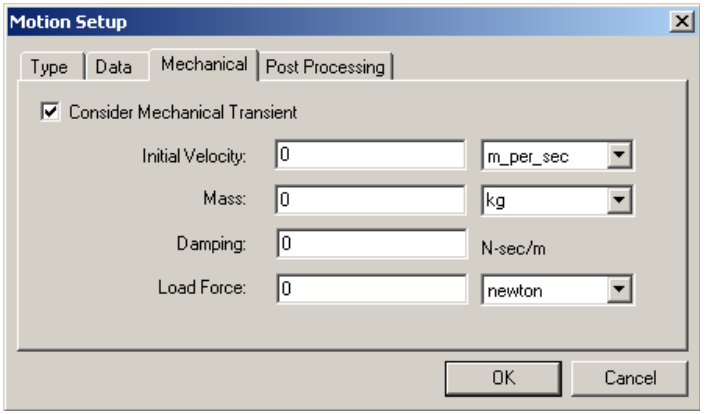


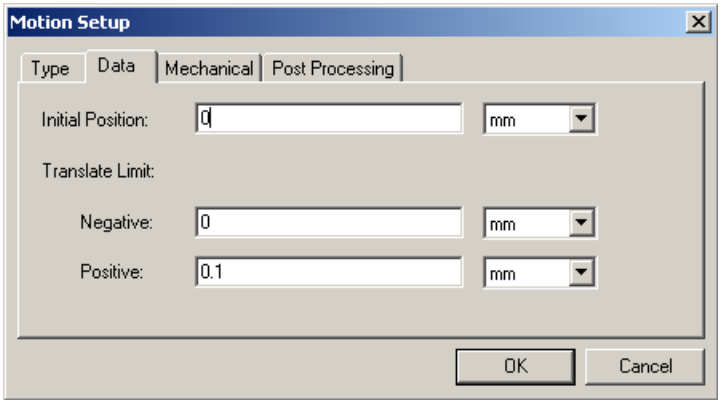
- Maxwell 2D > Model > Motion Setup > Assign Band



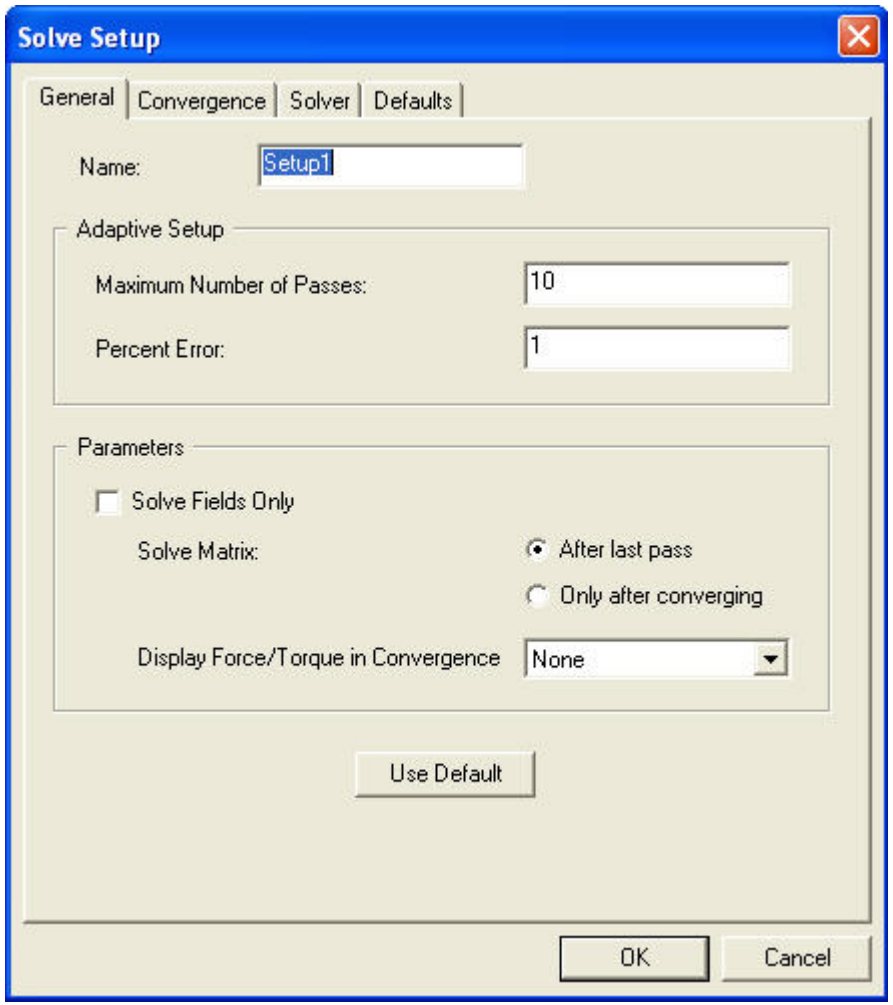
1. Defines the direction and type of motion (translation or rotation)
2. Defines the mechanical parameters such as mass, damping, and load force
3. Defines limits of motion

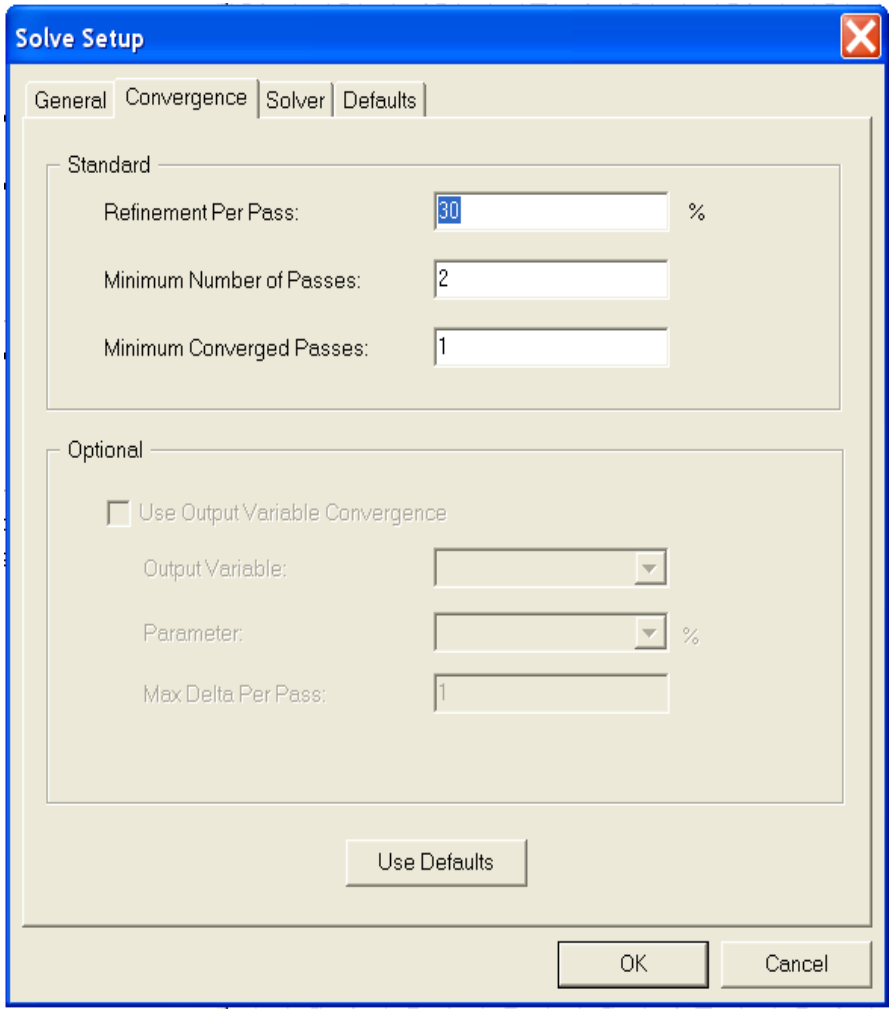




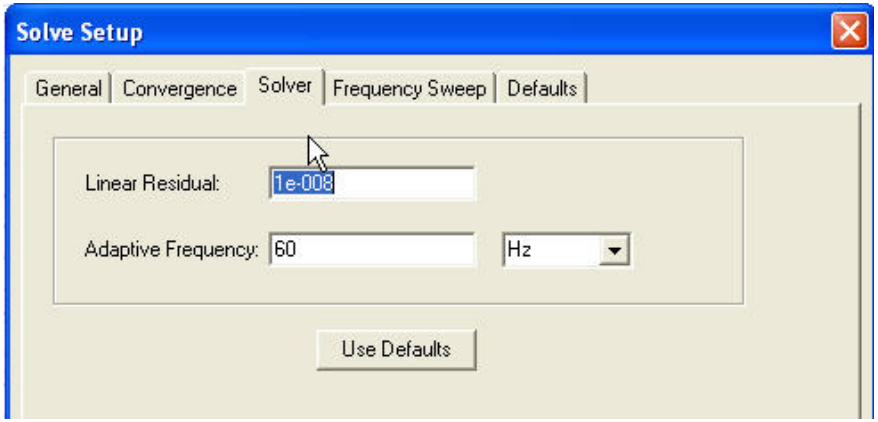


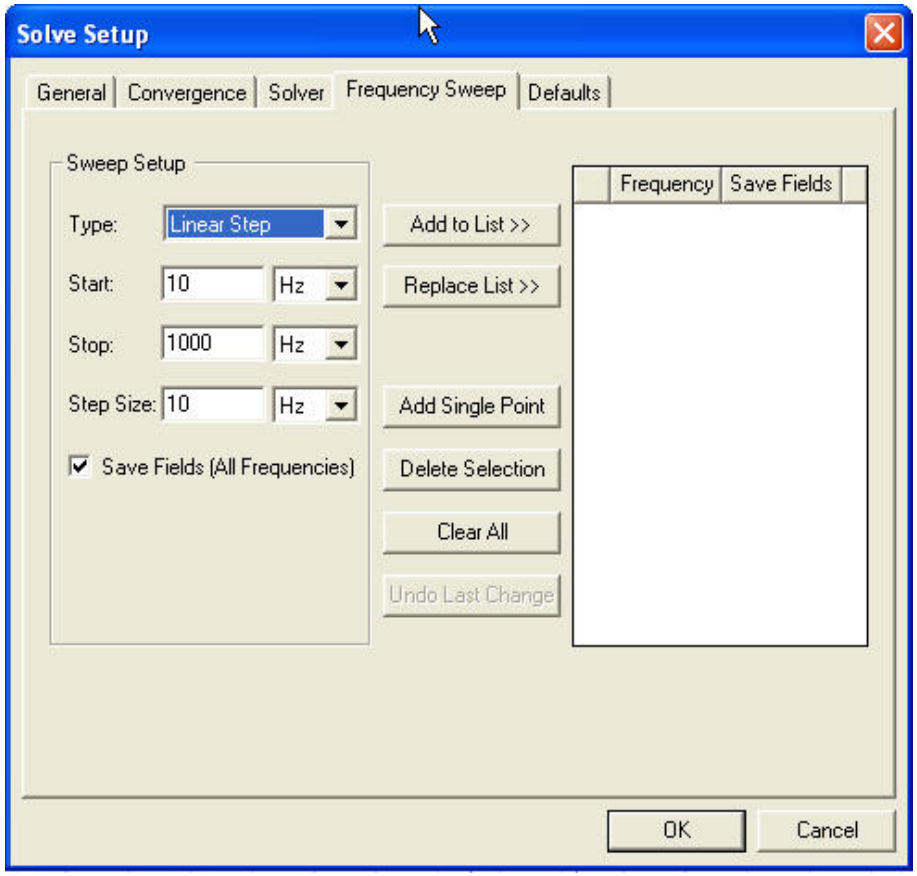
- Magnetostatic and Electric Solution Setup
  - Start the menu of solution setup by: ***Maxwell > Analysis Setup > Add Solution Setup …***
  - For Magnetostatic solver on Solver tab, suggest setting nonlinear residual = 0.001. On default tab choose Save Defaults to set this value for all future projects.



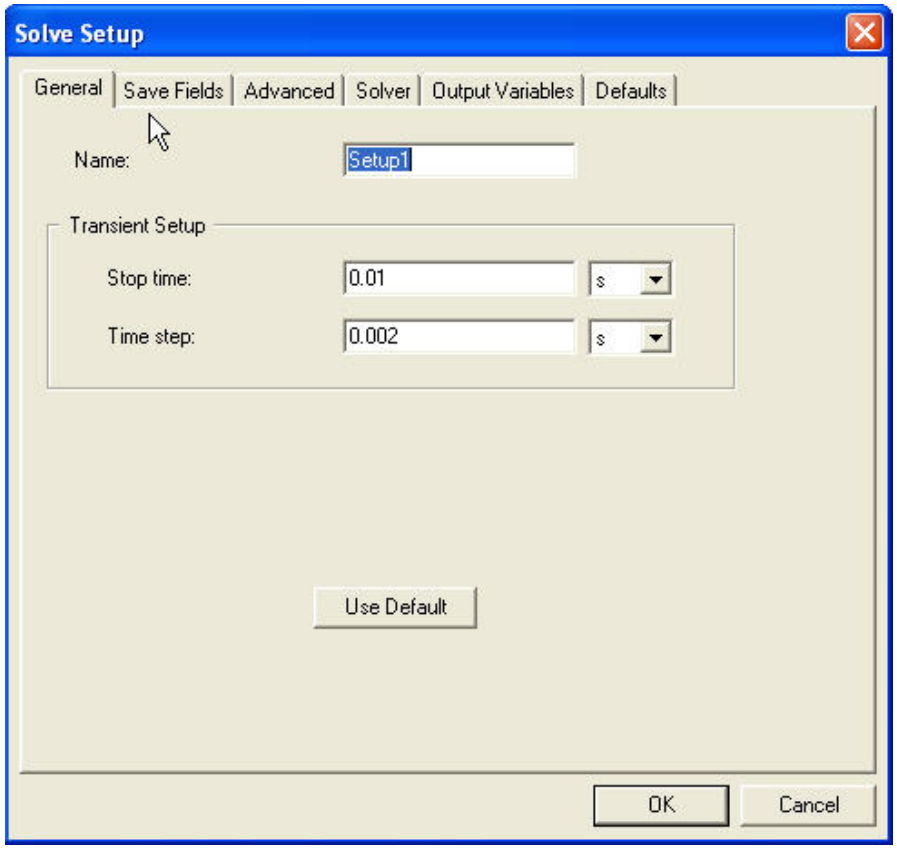


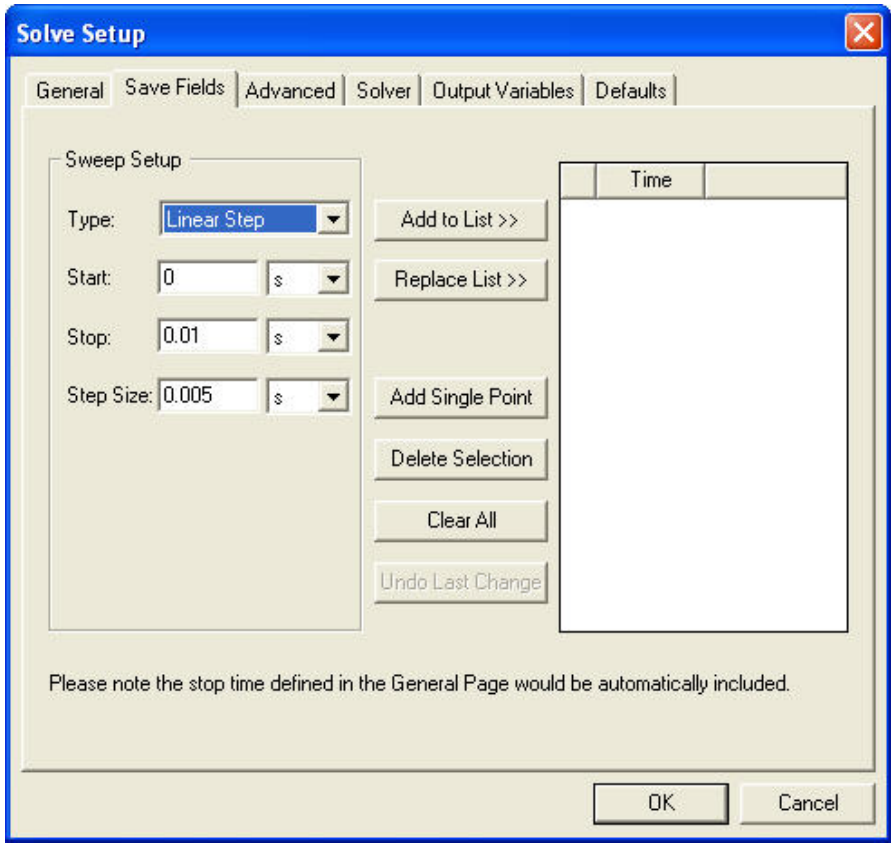
- Eddy Current Solution Setup



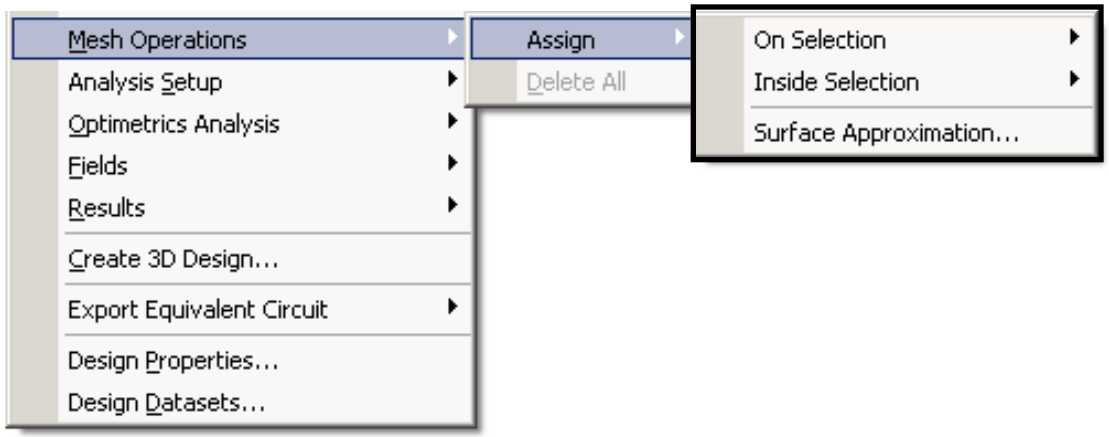


- Transient Solution Setup





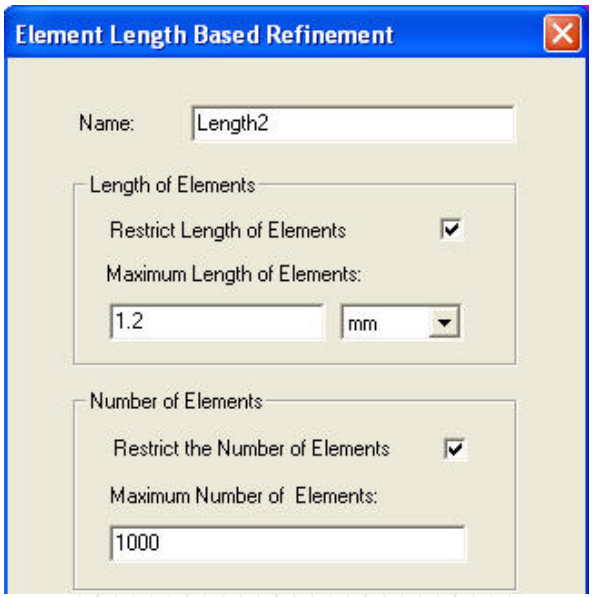
- Mesh Operations
  - To assign Mesh operations to Objects, select the Menu item: ***Maxwell 2D > Assign Mesh Operations***
    1. ***On Selection*** is applied on the surface of the object
    2. ***Inside Selection*** is applied through the volume of the object
    3. ***Surface approximation*** is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

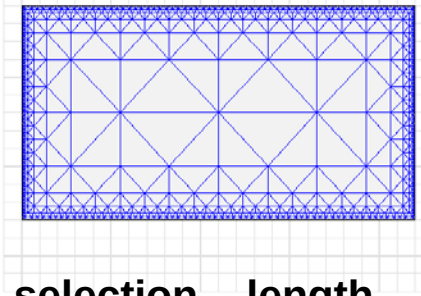


1. Mesh Operations "On selection" applied on the perimeter of the object

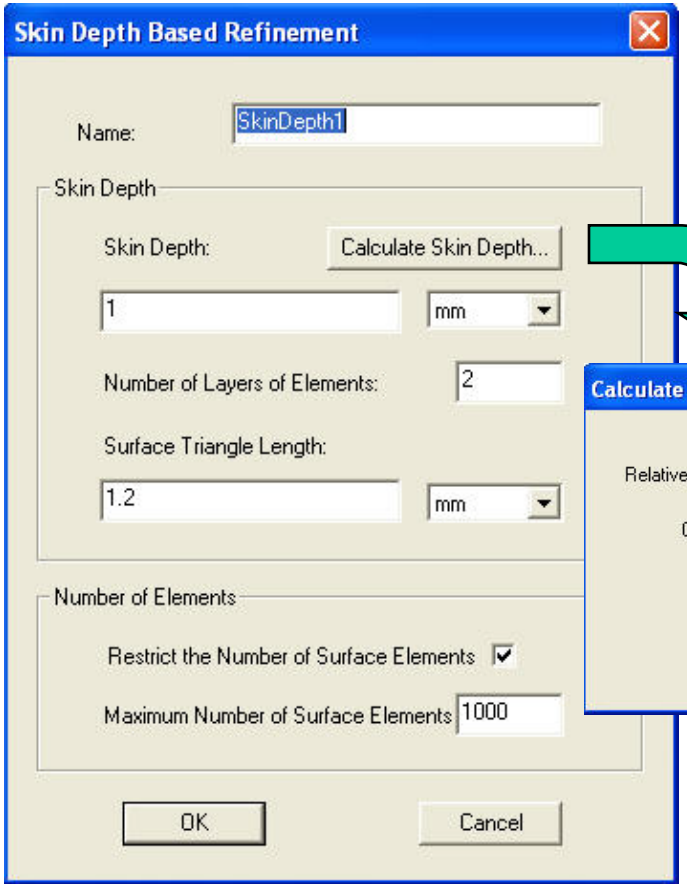
- Element length based refinement: ***Length Based***
- Skin Depth based refinement: ***Skin Depth Based***

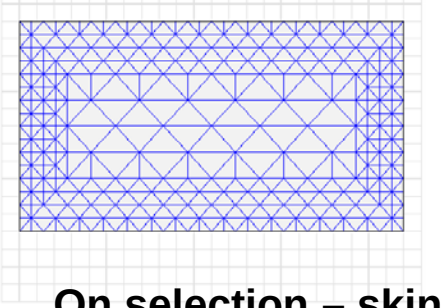




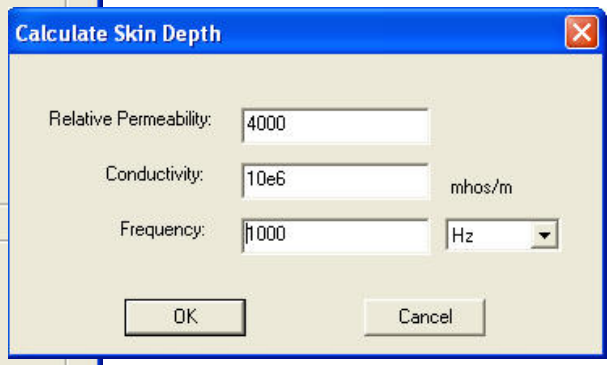


On selection – length based



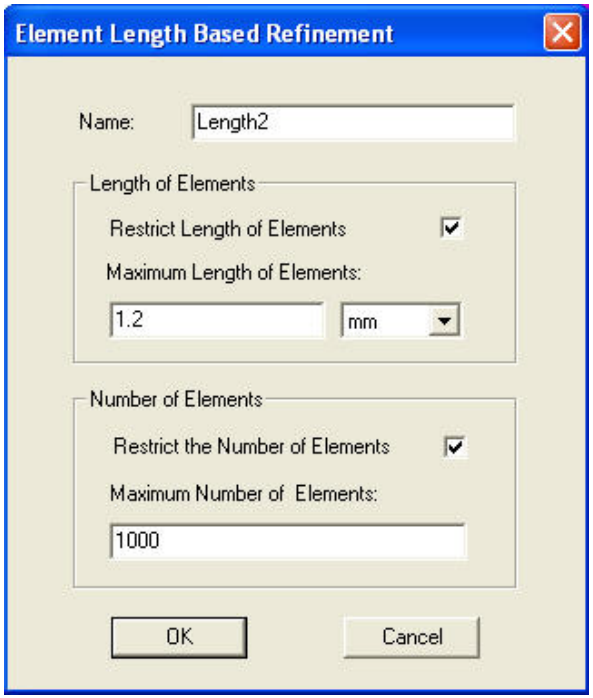


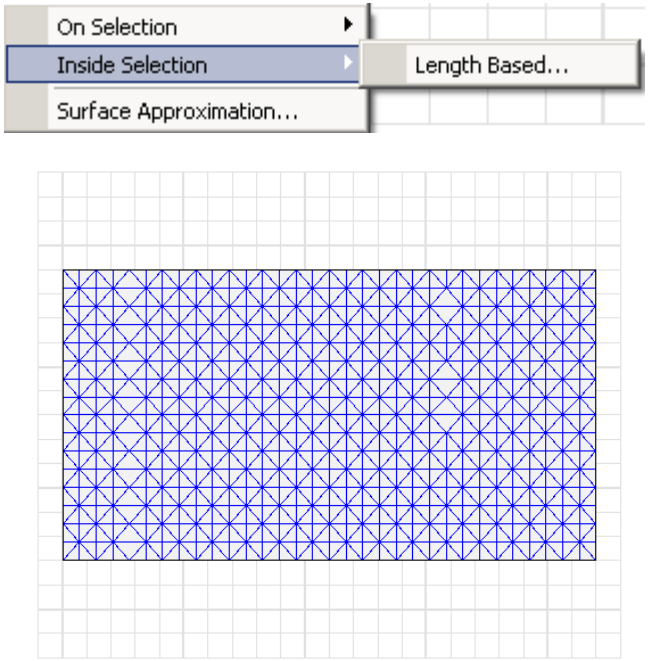
On selection – skin depth based (2 layers)



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

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

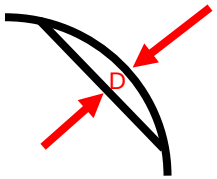




Inside selection – length based

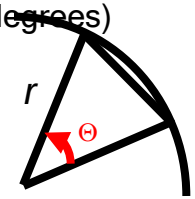
3. Mesh Operations "Surface Approximation"

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



D = Maximum Surface Deviation

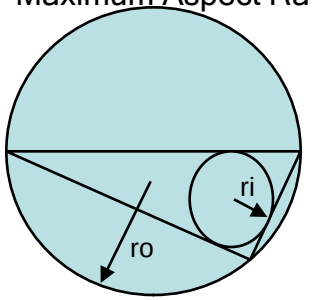
  - Maximum Surface Normal Deviation (degrees)



Θ = Maximum Surface Normal Deviation

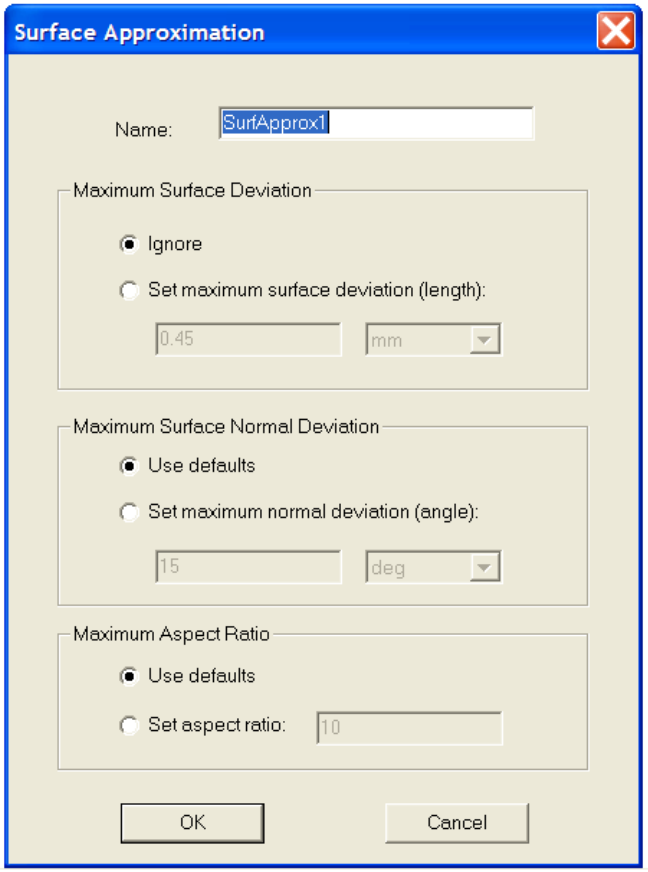
$$D = r(1 - \cos(\Theta / 2))$$

  - Maximum Aspect Ratio

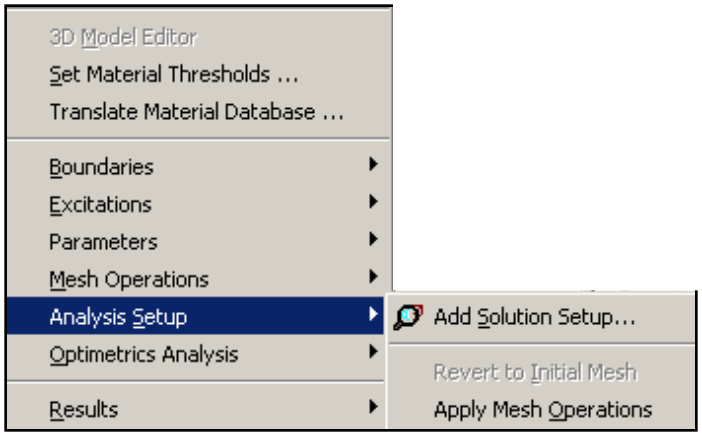


$$AspectRatio = \frac{ro}{2 * ri}$$



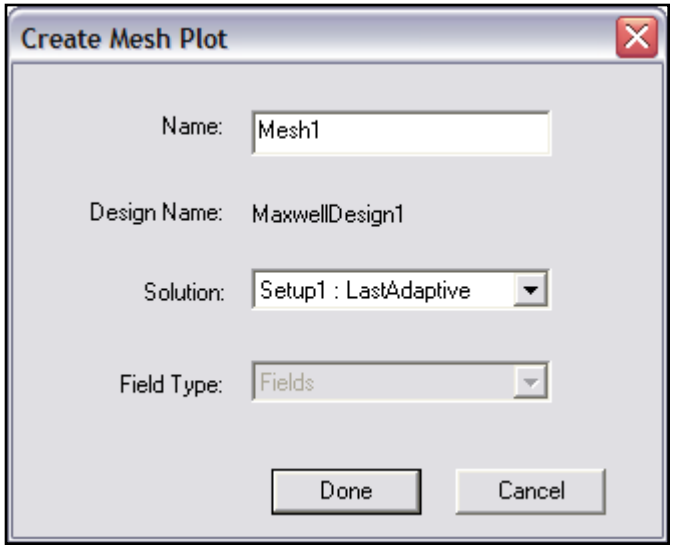


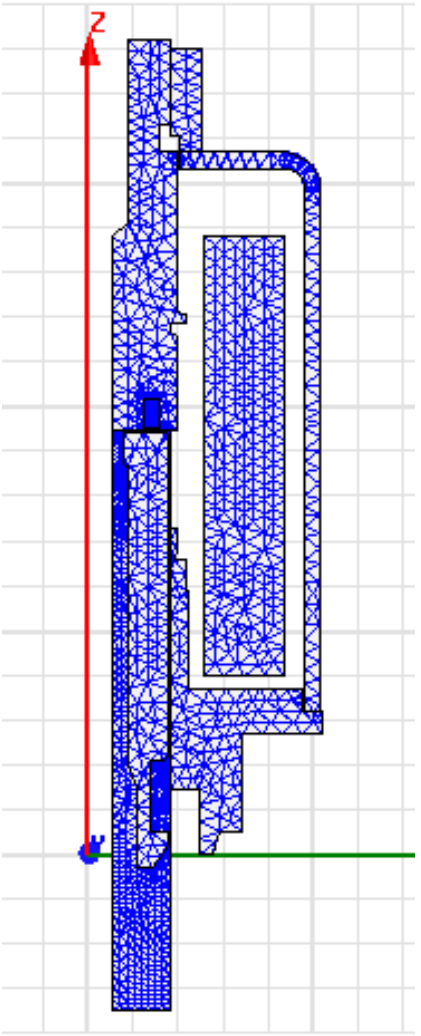
- Manual mesh creation
  - To create the initial mesh: Click *Maxwell > Analysis Setup > Apply Mesh Operations*
  - To refine the mesh without solving
    1. Define mesh operations as previously discussed
    2. Click *Maxwell > Analysis Setup > Apply Mesh Operations*
    3. Click *Maxwell > Analysis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



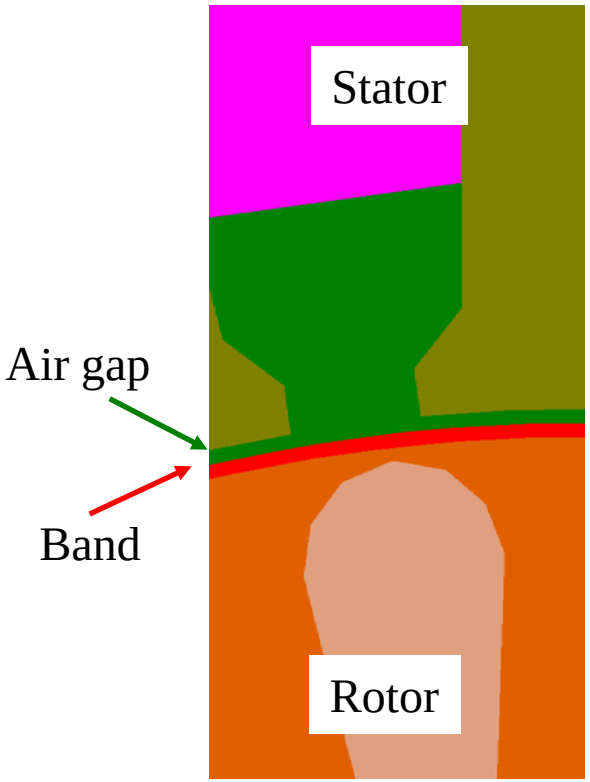
  - To view mesh information: Click *Maxwell > Results > Solution Data* and click on the tab *Mesh Statistics*

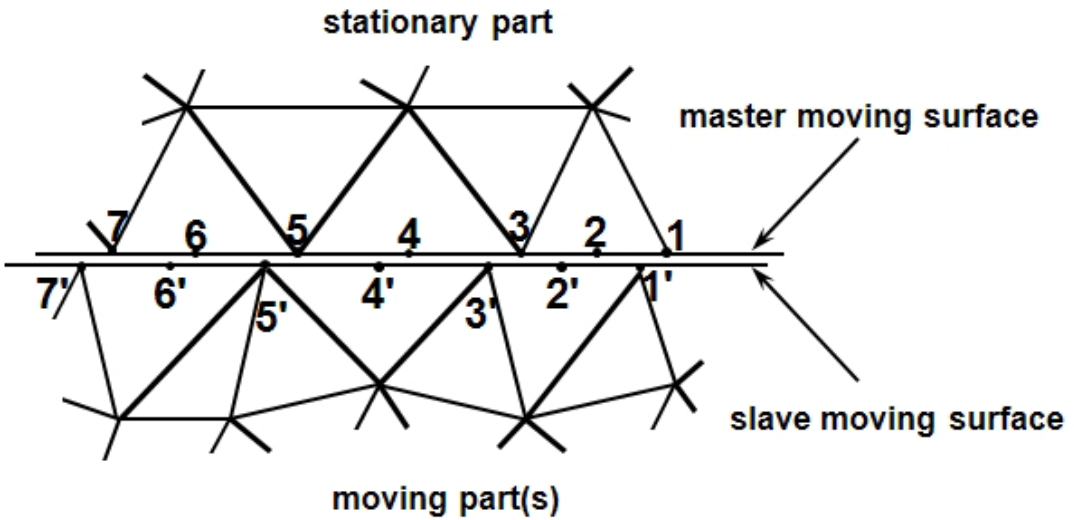
- Mesh Display
  1. Select an object
  2. Select the menu item *Maxwell 2D > Fields > Plot Mesh*



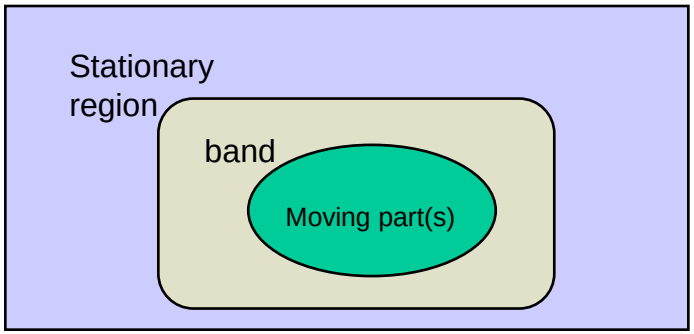


- 2D transient meshing for rotational models
- "Moving Surface" method used





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

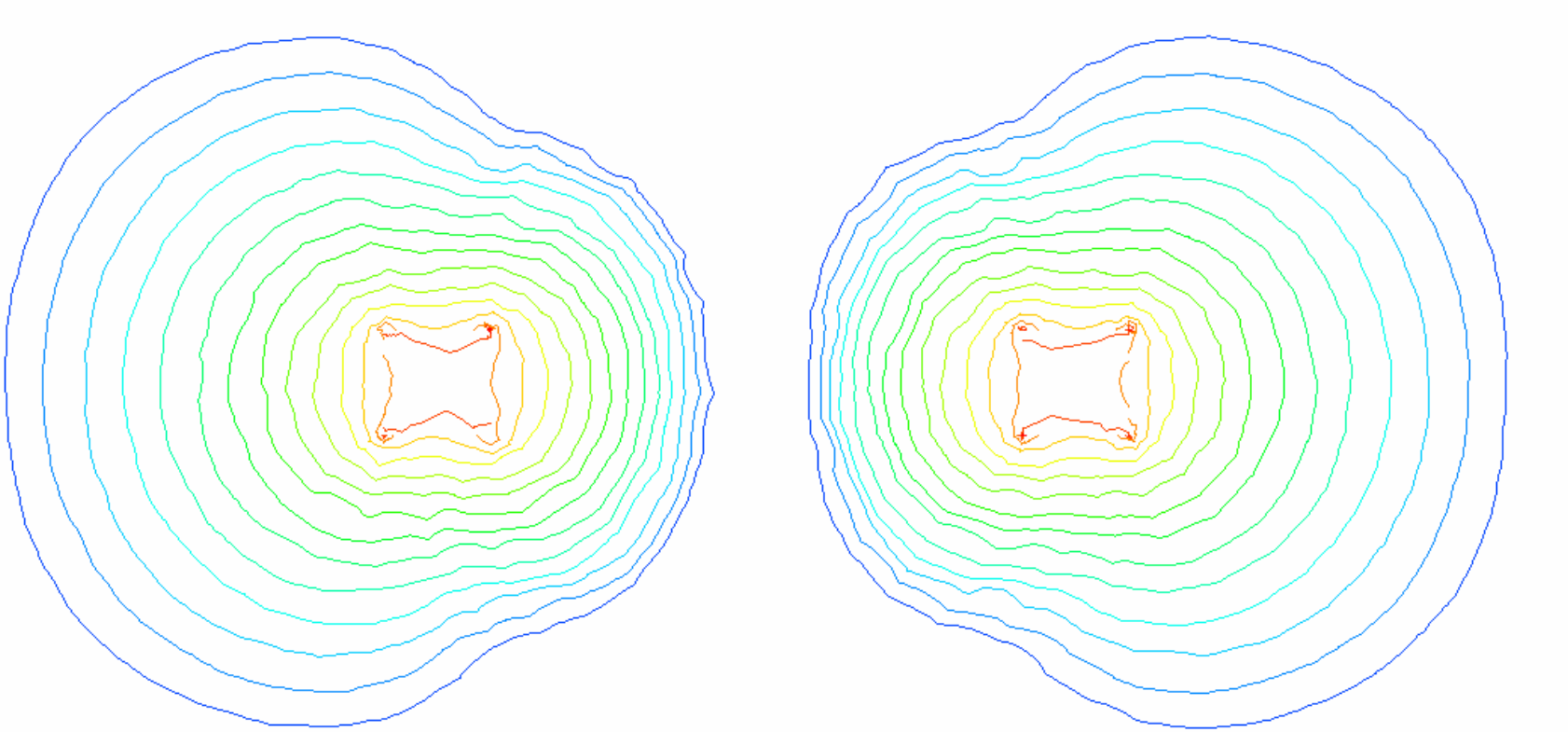


Post Processing

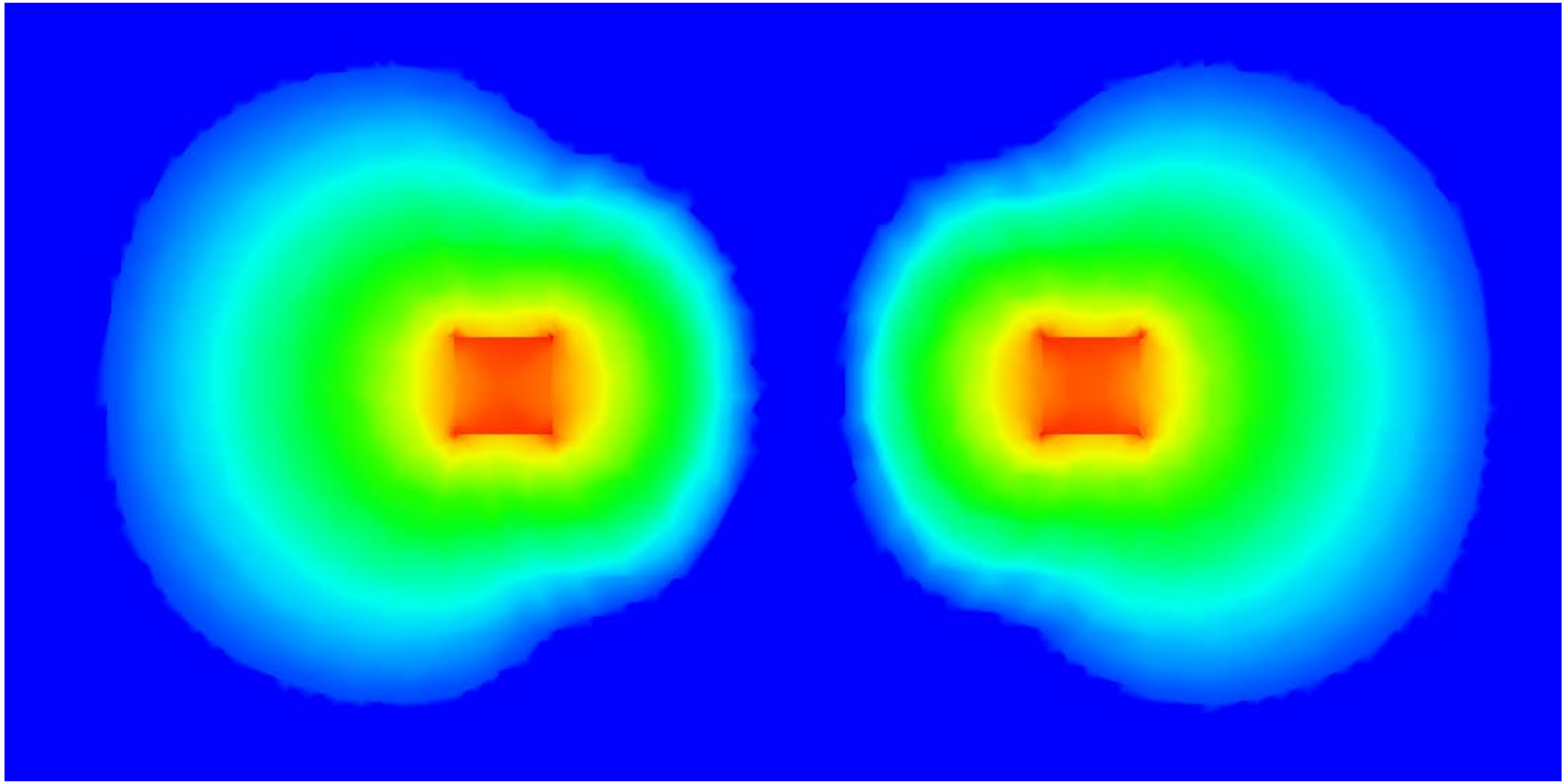
- Two Methods of Post Processing Solutions:
  - Viewing Plots
  - Manipulating Field Quantities in Calculator

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

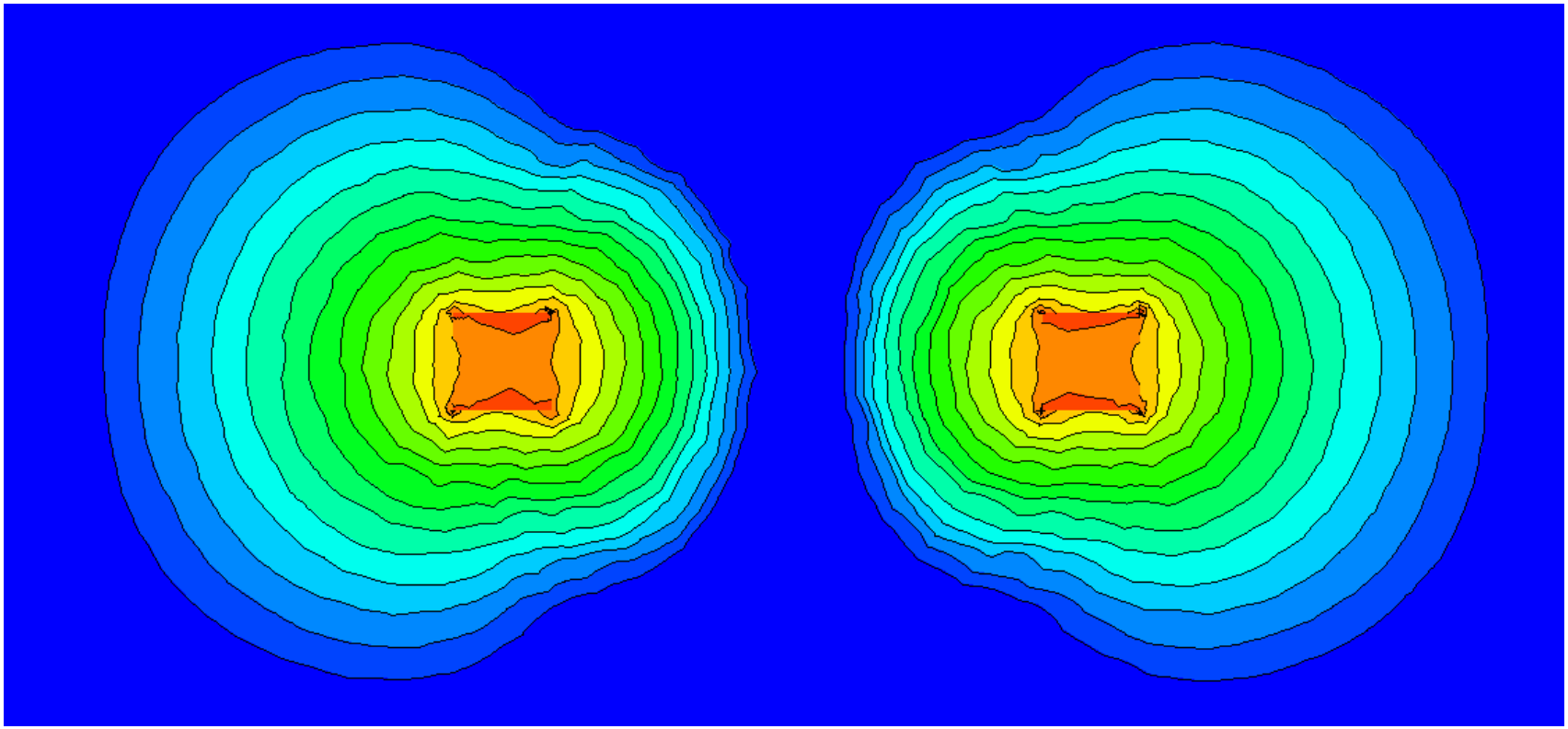
- Contour plot



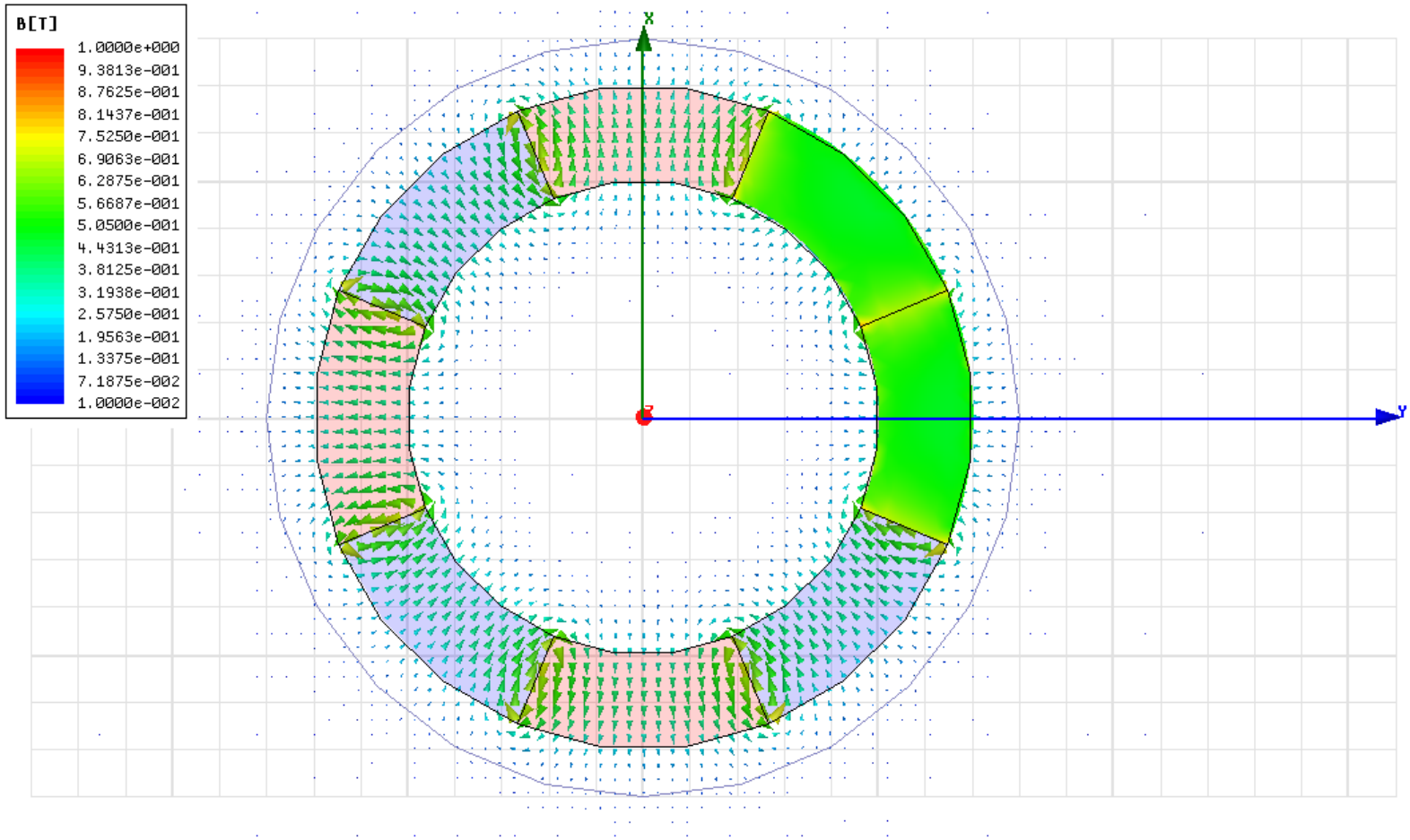
- Shade plot (tone)



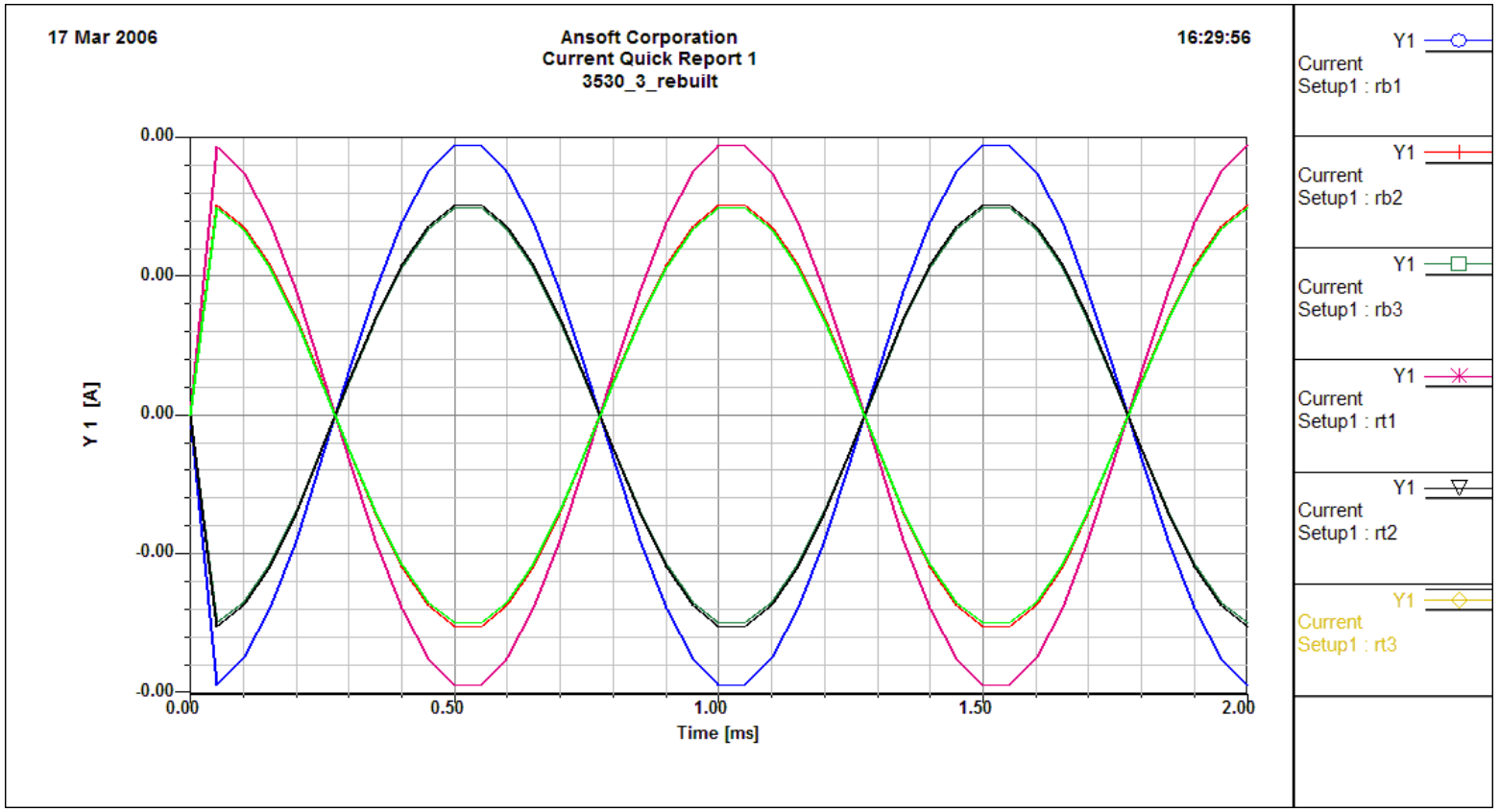
- Shade plot (fringe with outline)



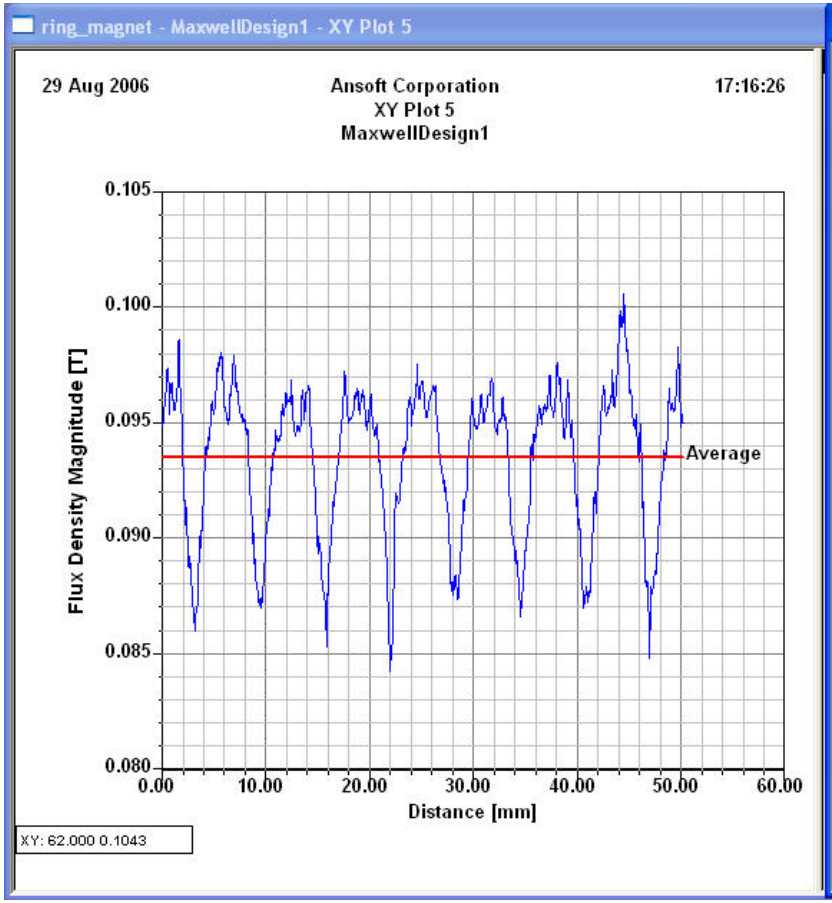
Arrow plot

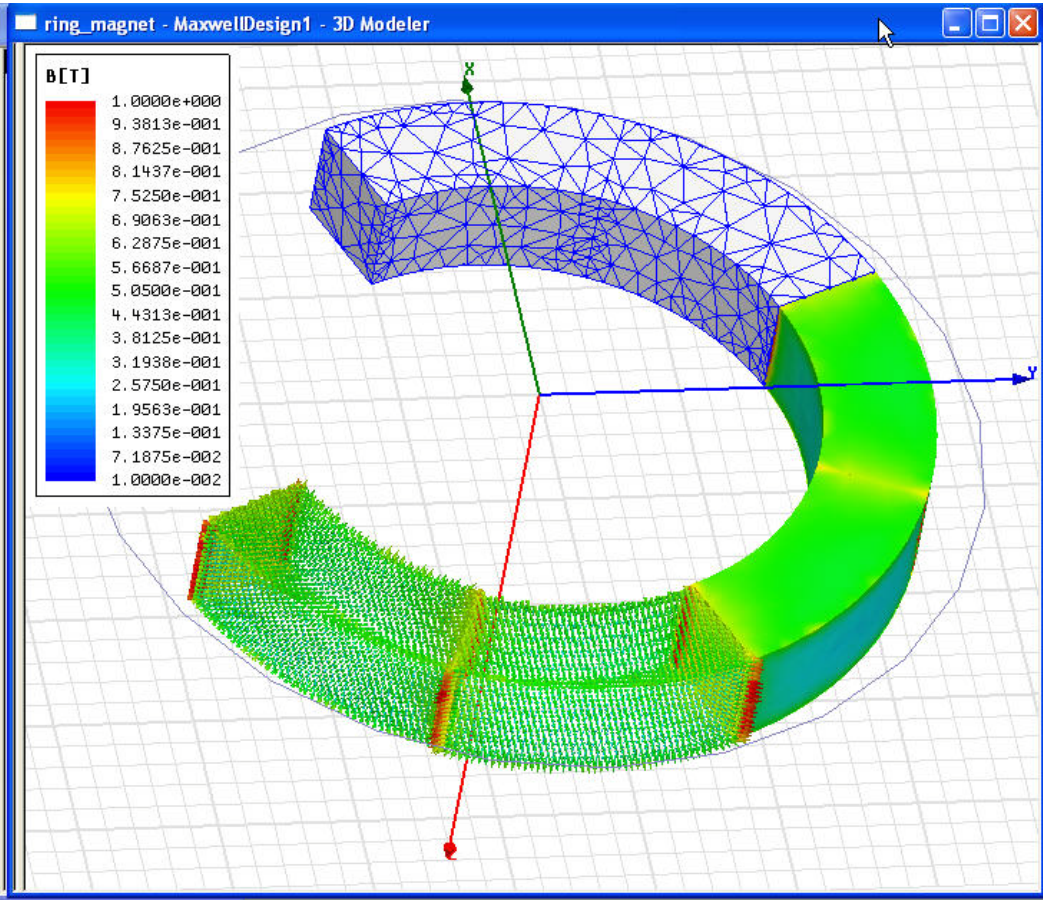

B[T]
1.0000e+000
9.3813e-001
8.7625e-001
8.1437e-001
7.5250e-001
6.9063e-001
6.2875e-001
5.6687e-001
5.0500e-001
4.4313e-001
3.8125e-001
3.1938e-001
2.5750e-001
1.9563e-001
1.3375e-001
7.1875e-002
1.0000e-002
X
Z
Y

Line plot


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

- Multiple windows and multiple plots

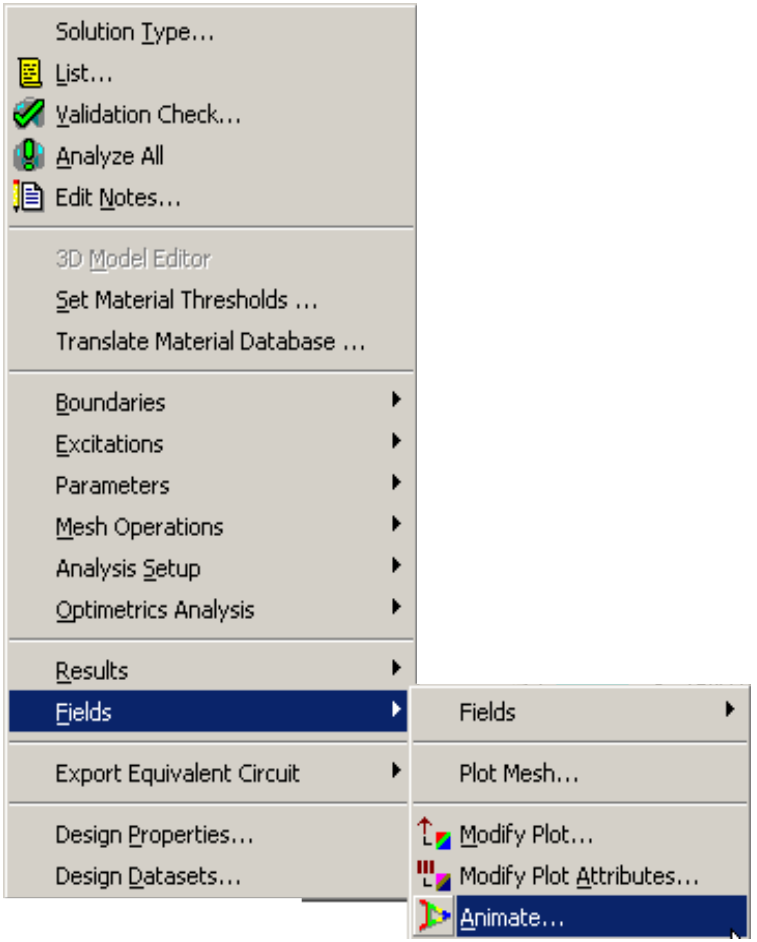




- Animation plot

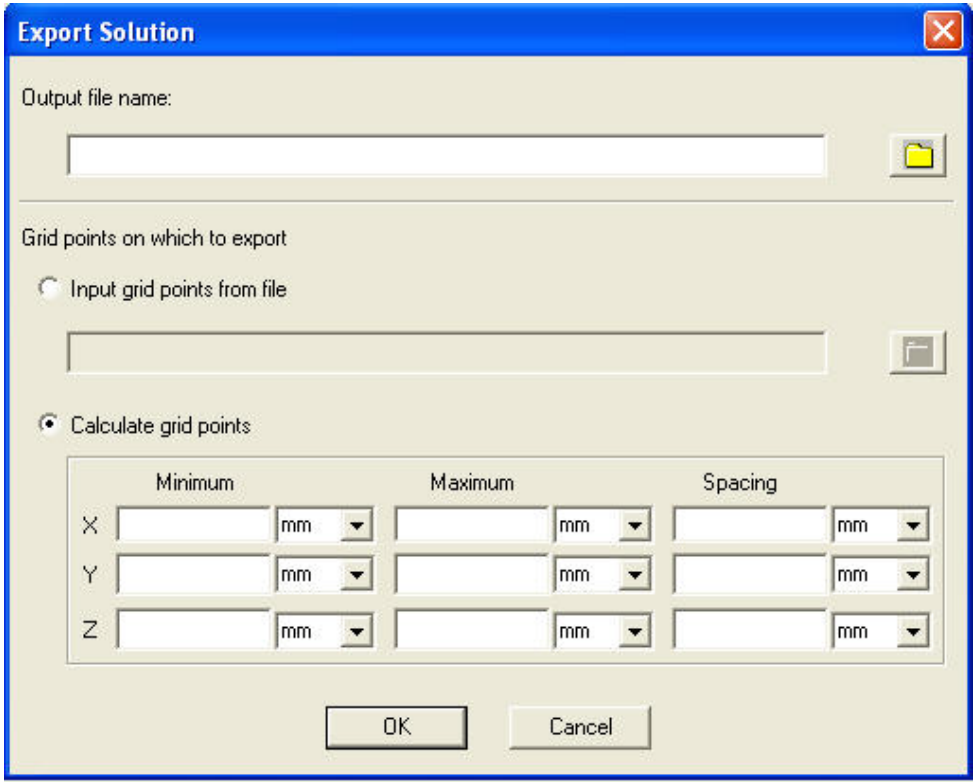
  - Various types of animated plots are possible:
    - Animate with respect to phase angle (eddy solver)
    - Animate with respect to time (transient solver)
    - Animate with respect to position (for parametric analysis)
    - Animate with respect to shape change (for parametric analysis)

  - Export to .gif or .avi format



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

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



- 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

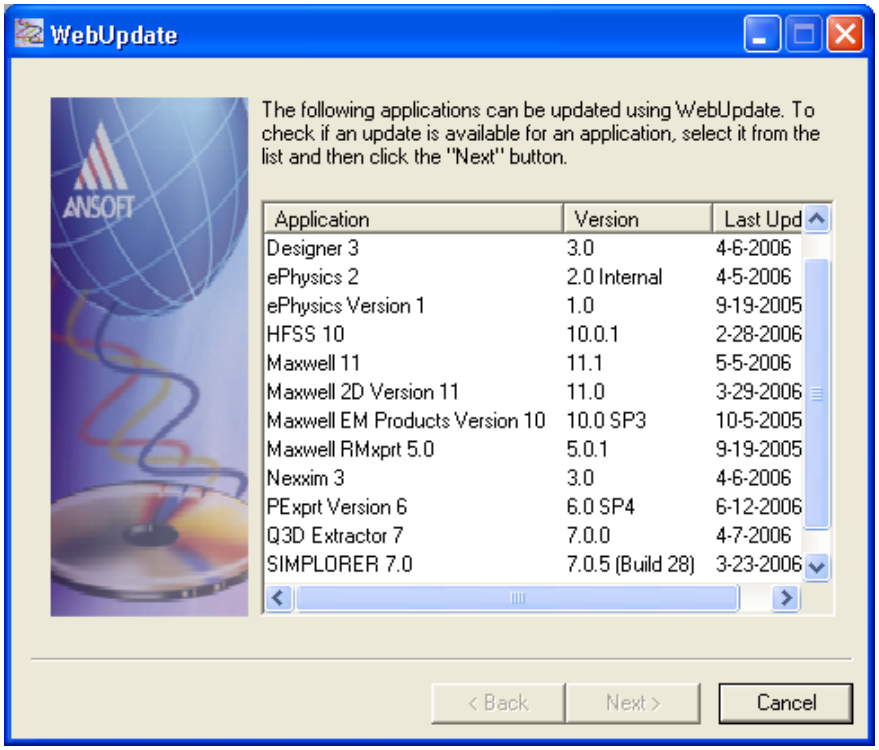
Vector data "<Ex,Ey,Ez>"
Grid Output Min: [0 0 0] Max: [2 2 2] Grid Size: [1 1 1

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.
      - **Tooltip**s are available to provide information about tools on the toolbars or dialog boxes. When you hold the pointer over a tool for a brief time, a tooltip appears to display the name of the tool.
      - As you move the pointer over a tool or click a menu item, the **Status Bar** at the bottom of the Maxwell window provides a brief description of the function of the tool or menu item.
      - The 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.

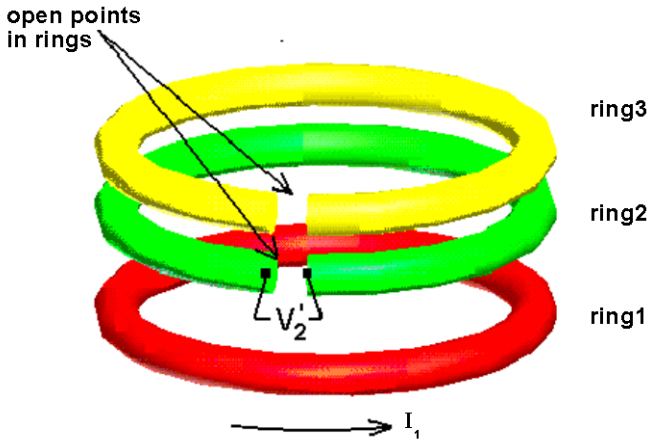


- Application Support for North America
  - The names and numbers in this list may change without notice
    - Technical Support:
      - 9-4 EST:
        - **Pittsburgh, PA**
        - (412) 261-3200 x199
        - **Burlington, MA**
        - (781) 229-8900 x199
      - 9-4 PST:
        - **San Jose, CA**
        - (408) 261-9095 199
        - **Portland, OR**
        - (503) 906-7946 x199
        - **Irvine, CA**
        - (714) 417-9311 x199

- Chapter 6.0 - Eddy Current Examples
  - 6.1 - Jumping Rings Axisymmetric Model
  - 6.3 - Instantaneous Forces on Busbars

Introduction

- This example investigates the classical “jumping rings” experiment using a 2D axi-symmetric eddy current model. Three rings are stacked on top of each other around a common axis. The bottom ring provides a 10 kHz excitation that induces eddy currents and losses in the other two rings. These rings are repelled from ring1 and can be suspended by the magnetic field as the current in ring1 is increased.
- The model consists of three solid copper rings. The bottom ring1 has a peak current of 1A, while ring2 and ring3 have no excitation and are open-circuited. The open-circuit condition is simulated by constraining the total current to zero. A physical layout of the actual device is shown in:



- After the problem is solved, you can do the following:
  - View the impedance matrix.
  - Calculate the power loss using two independent methods, and compare it to the loss in the convergence table.
  - Calculate the induced voltage (V2’) across the open ends of ring2.
- The analysis includes all skin and proximity effects in the calculation of the impedance matrix, power losses, and voltage.

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

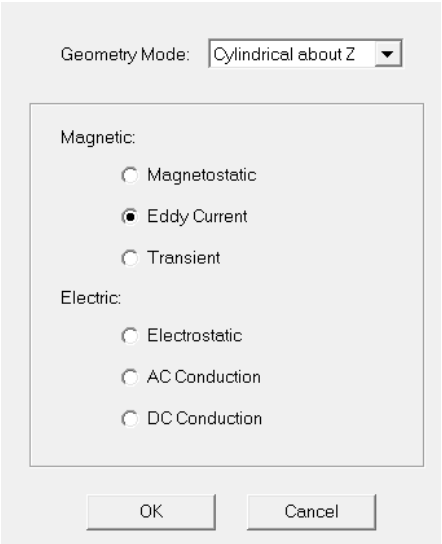
- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  - Select the menu item *Tools > Options > Maxwell 2D Options*
    - Maxwell Options Window:
      1. Click the **General Options** tab
         - Use Wizards for data input when creating new boundaries: ☑ **Checked**
         - Duplicate boundaries/mesh operations with geometry: ☑ **Checked**
      2. Click the **OK** button
  - 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 2D Design***, or click on the icon

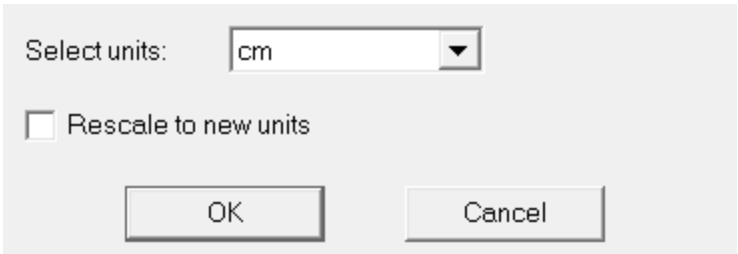
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetostatic > Eddy Current**
    3. Click the **OK** button



Set Default Units

- **To Set Default Units**
  - Select the menu item ***Modeler > Units***
  - In Units window,
    - Select units: **cm (centimeters)**
    - Press **OK**

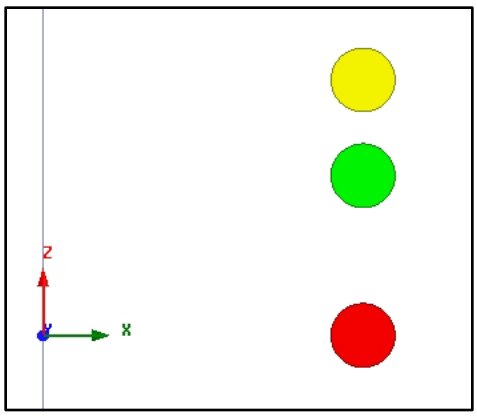


Define Solution Region

- **To Create Simulation Region**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **-10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **20**, dY: **0**, dZ: **20**, Press the **Enter** key
  - Select the resulting sheet and goto the properties window
    - Change the name of resulting sheet to **Region**
    - Check the option **Display Wireframe**

Create Model

- The model consists of three donut-shaped rings. A cross-section of the model is shown below. This is a 2-dimensional axi-symmetric drawing; an axi-symmetric model is rotated 360° around the z-axis (displayed as the v-axis in the drawing).

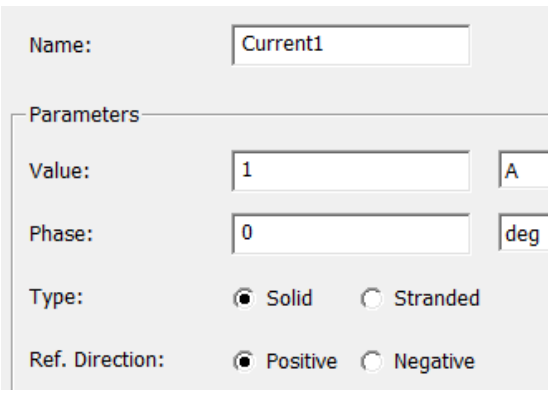


- **Create Ring1**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **1**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.1**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and go to the Properties window
    - Change the name of the sheet to **Ring1** and color to **red**
    - Change the material of the sheet to **Copper**

- **Create Ring2**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **1**, Y: **0**, Z: **0.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.1**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and go to the Properties window
    - Change the name of the sheet to **Ring2** and color to **Green**
    - Change the material of the sheet to **Copper**
- **Create Ring3**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **1**, Y: **0**, Z: **0.8**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.1**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and goto the Properties window
    - Change the name of the sheet to **Ring3** and color to **Yellow**
    - Change the material of the sheet to **Copper**

Assign Excitations

- A current of 1A will be assigned to the Ring1 while 0A will be assigned to both Ring2 and Ring3. This forces the total current flow around these rings to be zero in order to model the “open-circuit” condition.
- **To Assign Excitations for Ring1**
  - Select the sheet **Ring1** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window
    - Name: **Current1**
    - Value: **1 A**
    - Type: **Solid**
    - Ref. Direction: **Positive**
    - Press **OK**



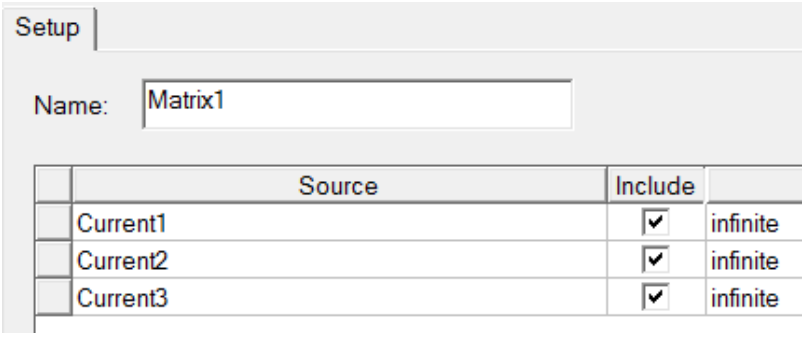
- **To Assign Excitations for Ring2**
  - Select the sheet **Ring2** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window
    - Name: **Current2**
    - Value: **0 A**
    - Type: **Solid**
    - Ref. Direction: **Positive**
    - Press **OK**
- **To Assign Excitations for Ring3**
  - Select the sheet **Ring3** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window
    - Name: **Current3**
    - Value: **0 A**
    - Type: **Solid**
    - Ref. Direction: **Positive**
    - Press **OK**
- Note: Choosing **Solid** specifies that the eddy effects in the coil will be considered. On the other hand, if **Stranded** had been chosen, only the DC resistance would have been calculated and no AC effects in the coil would have been considered.

Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - In Balloon Boundary window,
    - Press **OK**

Assign Matrix Parameters

- In this example, the compete [3x3] impedance matrix will be calculated. This is done by setting a parameter.
- **To Calculate Impedance Matrix**
  - Select the menu item *Maxwell 2D > Parameters > Assign > Matrix*
  - In Matrix window,
    - For all current Sources
      - Include: ☑ **Checked**
    - Press **OK**



Compute the Skin Depth

- Skin depth is a measure of how current density concentrates at the surface of a conductor carrying an alternating current. It is a function of the permeability, conductivity and frequency
- Skin depth in meters is defined as follows:

$$\delta = \sqrt{\frac{2}{\omega \mu_o \mu_r \sigma}}$$

where:

- ω is the angular frequency, which is equal to $2\pi f$. (f is the source frequency which in this case is 10000Hz).
- σ is the conductor's conductivity; for copper its 5.8e7 S/m
- μ_r is the conductor's relative permeability; for copper its 1
- μ_o is the permeability of free space, which is equal to $4\pi \times 10^{-7}$ A/m.

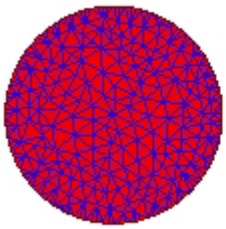
- For the copper coils, the skin depth is approximately 0.066 cm which less than the diameter of 0.200cm for the conductors.

Analysis Setup

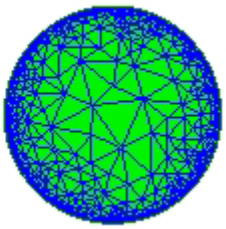
- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.01**
    2. **Solver** Tab
       - Adaptive Frequency: **10 KHz**
    3. Click the **OK** button

Specify Mesh Operations

- In order to accurately compute the mutual resistance terms in the impedance matrix, a uniform mesh is needed in all conductors.
- **To Assign Mesh Operation for Coils**
  - Press **Ctrl** and select all coils from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based…*
  - In Element Length Based Refinement window,
    - Name: **Coils_Inside**
    - Restrict Length Of Elements: ☐ **Unchecked**
    - Restrict Number of Elements: ☑ **Checked**
    - Maximum Number of Elements: **1000**
    - Press **OK**
  - Note that by choosing “Inside Selection” instead of “On Selection”, the mesh operation is applied evenly through the area of the conductors as opposed to being applied only on the outer perimeter of the conductor.



Mesh Operation
“Inside Selection”



Mesh Operation
“On Selection”

Analyze

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

Solution Data

- **To View Solution Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
    - To View Convergence
      - Select the **Convergence** tab
      - Note that the total loss is around **1.99e-004 Watts**

Pass	Triangles	Total Energy (J)	Energy Error (%)	Delta Energy (%)	Loss (W)	Delta Loss (%
1	1840	7.48e-009	182.28	N/A	0.00019922	N/A
2	2512	8.1046e-009	14.782	8.3505	0.0001994	0.092509
3	3432	8.1553e-009	1.3329	0.62543	0.0001996	0.096457
4	4606	8.1579e-009	0.25051	0.031529	0.00019961	0.0050957
5	5988	8.1584e-009	0.059327	0.006351	0.00019961	0.00071978
6	7785	8.1585e-009	0.016854	0.0011279	0.00019961	0.00016556
7	10121	8.1585e-009	0.0056084	0.00029916	0.00019961	1.8798e-005

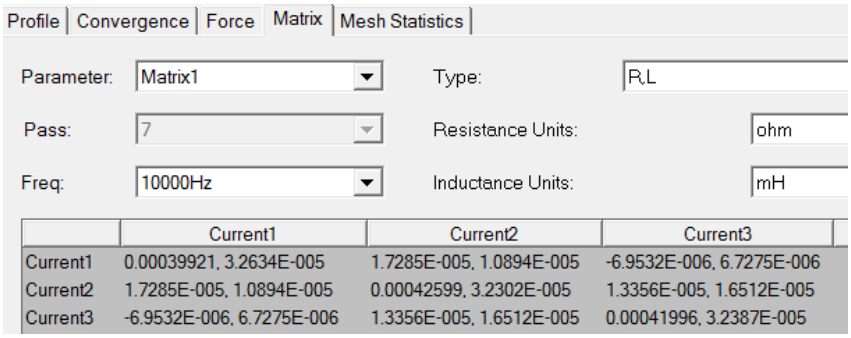
    - To View Mesh information
      - Select the **Mesh Statistics** tab

	Num Elements	Min edge length	Max edge length	RMS edge length	Min elem area	Max elem area	Mean elem area
Region	7933	3.9748e-005	0.05	0.0034398	9.76094e-010	0.000503872	5.04104e-006
Ring1	860	2.98471e-005	0.000226717	9.38052e-005	5.1926e-010	1.9098e-008	3.65135e-009
Ring2	514	4.87817e-005	0.000245152	0.000124404	1.23262e-009	1.79498e-008	6.10926e-009
Ring3	462	4.93591e-005	0.000241493	0.000130502	1.52626e-009	2.05693e-008	6.79688e-009

    - To View Impedance matrix
      - Select **Matrix** tab
      - By default, the results are displayed as [R, Z] but can be also shown as [R, L] or as coupling coefficients.

$$\begin{bmatrix} R_{11}, L_{11} & R_{12}, L_{12} & R_{13}, L_{13} \\ R_{21}, L_{21} & R_{22}, L_{22} & R_{23}, L_{23} \\ R_{31}, L_{31} & R_{32}, L_{32} & R_{33}, L_{33} \end{bmatrix}$$

- **About Impedance Matrix**
  - The diagonal resistance terms represent the self-resistance of each coil due to DC component and skin effects, as well as the proximity effects in all other conductors. The off-diagonal resistance terms result from proximity effect currents induced in one coil due to excitation in the other coil.
  - The diagonal inductance terms represent the self-inductance of each coil, while off-diagonal terms represent the mutual inductance due to coupling.
  - The matrix results should closely resemble the results shown in the following matrix. Negative resistance *R*13 means that current in **Ring1** induces a current in **Ring3**, which actually reduces AC resistance of **Ring3**:



	Current1	Current2	Current3
Current1	0.00039921, 3.2634E-005	1.7285E-005, 1.0894E-005	-6.9532E-006, 6.7275E-006
Current2	1.7285E-005, 1.0894E-005	0.00042599, 3.2302E-005	1.3356E-005, 1.6512E-005
Current3	-6.9532E-006, 6.7275E-006	1.3356E-005, 1.6512E-005	0.00041996, 3.2387E-005

  - The diagonal term *R*11 is made up of the following resistive components due to **Ring1**, **Ring2**, and **Ring3**. (The ring1 DC resistance is obtained by running a separate simulation a 0.1Hz. The R11 term as well as ring2 and ring3 proximity terms are taken from the matrix above. Finally, The ring1 skin effect term is calculated as the difference between R11 and the sum of all of other terms.)

    ring1 DC resistance = 3.441e-004
    ring1 skin effect = 4.447e-005
    ring2 proximity effect from I1 = 1.728e-005
    ring3 proximity effect from I1 = -6.953e-006
    R11 = 3.992e-004 ohms

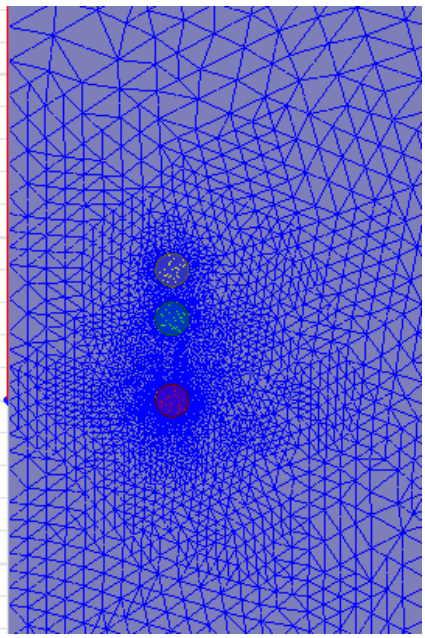
  - In this example, with a 1 A peak current in **ring1**, and with both **ring2** and **ring3** open-circuited, the total power loss can be calculated by hand from the impedance matrix using the following formula:

    $P = \frac{1}{2} * Ipeak^2 * R11 = \frac{1}{2} * 1^2 * 3.9921e\text{-}004 = 1.9961\ e\text{-}004$

  - This value also corresponds to the **Total Power Loss** in the convergence table.

Plot Mesh

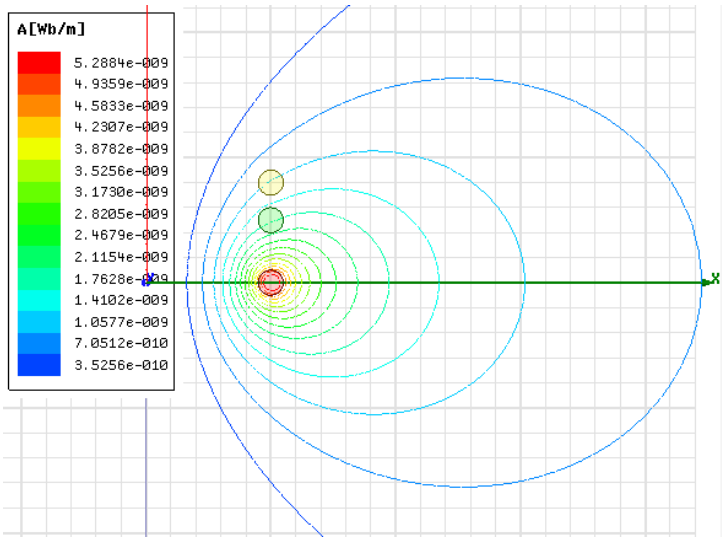
- **To Plot Mesh**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  - In Create Mesh Plot window,
    - Press **Done**



View the Results

- **Calculate Total Power Loss in Coils**
  - Press **Ctrl** and select all Rings
  - Select the menu item *Modeler > List > Create > Object List*
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > OhmicLoss**
    - Select **Input > Geometry**
      - Select the radio button **Volume**
      - Select **Objectlist1** from the list
      - Press **OK**
    - Select **Scalar > Integral > RZ**
    - Select **Output > Eval**
  - The evaluated loss in the **Coils** should be about: **1.996 e-004 (W).** This value is equal to the power calculated by hand from R11 in the impedance matrix.
    - Press **Done**

- **Plot Flux Lines**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window
    - Press **Done**



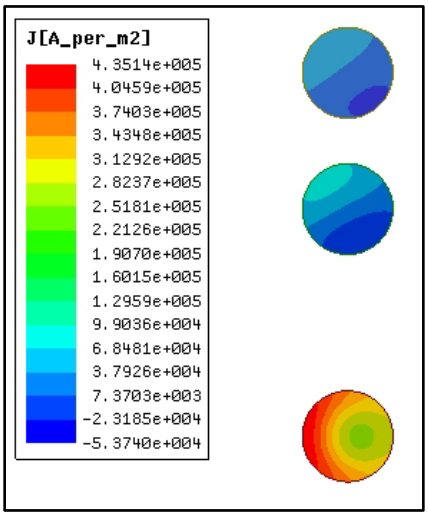
- **Calculate Total Current flowing around each of the Rings**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > J**
    - Select **Vector > Scal? > ScalarPhi**
    - Select **General > Complex > Real**
    - Select **Input > Geometry**
      - Select the radio button **Surface**
      - Select **Ring1** from the list
      - Press **OK**
    - Select **Scalar > Integral > XY** (Note this is a surface integral of J dot dA)
    - Select **Output > Eval**
  - Note that the current in ring1 is close to 1 A. Repeating these steps for Ring2 and Ring3 yields a net current ~0 A, which represents an open-circuited ring.
  - Click **Done.**

- **Calculate Rotating Current in open rings**
  - Although the net current flow in ring2 and ring3 is zero, there is a small rotating current flowing down one side and back on the opposite of each open ring. Taking the absolute value of J will return two times the current flowing in the open rings.
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > J**
    - Select **Vector > Scal? > ScalarPhi**
    - Select **General > Complex > Real**
    - Select **General > Abs**
    - Select **Input > Geometry**
      - Select the radio button **Surface**
      - Select **Ring2** from the list
      - Press **OK**
    - Select **Scalar > Integral > XY**
    - Select **Output > Eval**
  - The magnitude of the total current in Ring2 is displayed. Note that the current in Ring2 is close to **0.087A.** Repeat the test for Ring3. Evaluated value should be close to **0.047A.**
  - The current flowing along each side of Ring2 is a “rotational” eddy current equal to ½ * 0.087 = **0.044A**. For Ring3, the current flowing along each side of is ½ * 0.048 = **0.024A**. This current flows in opposite directions on either side of Ring2 and Ring3 unlike the current flow in Ring1, which is only in one direction.
    - Click **Done.**

Plot Current Density Magnitude

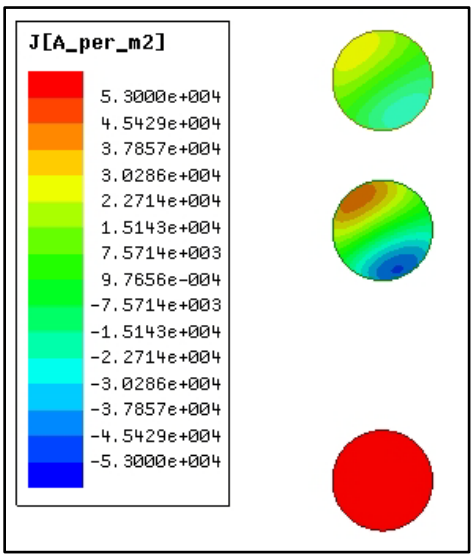
- **Plot Current Density**
  - Select the sheet **Region** from history tree
  - Select the menu item *View > Hide Selection > Active View*
  - Select the menu item *Edit > Select All Visible*
  - Select the menu item *Maxwell 2D > Fields > Fields > J > JAtPhase*
  - In Create Field Plot window
    - Press **Done**

Note: On Ring1, the skin effect causes higher current density on the surface. Current density is higher towards the axis of symmetry due to the DC spirality effect.



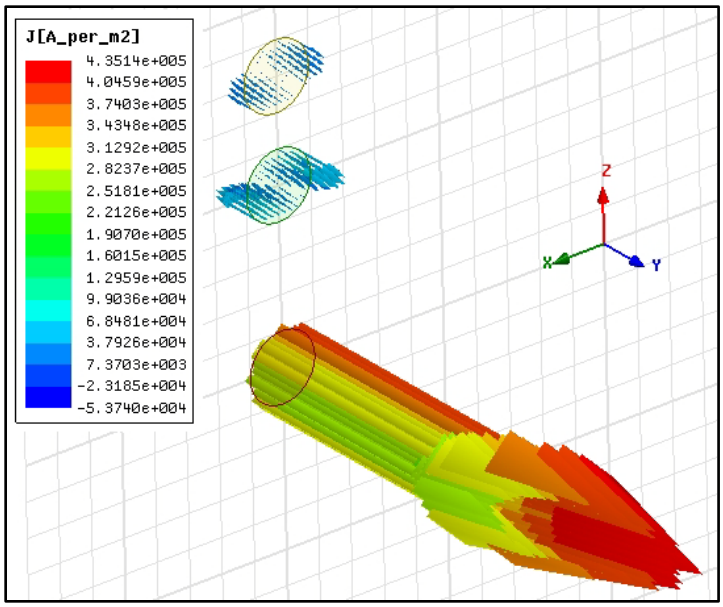
- **Modify Plot Attributes**
  - Double click on the legend to modify its attributes
  - In the window, select **Scale** tab
    - Select **User Limits**
      - Min: **-53000**
      - Max: **53000**
    - Press **Apply** and **Close**

Note: On Ring2 and Ring3, the rotational eddy currents cause positive and negative current density.



Plot Current Density Vectors

- **To Plot Current Density Vectors**
  - Select the sheet **Region** from history tree
  - Select the menu item *View > Hide Selection > Active View*
  - Select the menu item *Edit > Select All Visible*
  - Select the menu item *Maxwell 2D > Fields > Fields > J > J_Vector*
  - In Create Field Plot window
    - Press **Done**
- **Modify Plot**
  - Double click on the legend to modify plot attributes
  - In the window,
    - **Scale** tab
      - Change limits to **Auto**
    - **Marker/Arrow** tab
      - Change arrow size as desired
    - **Plots** tab
      - Plot: Select **J_Vector**
      - Spacing:
        - Min: **0.02**
        - Max: **0.02**
    - Press **Apply** and **Close**



- **Animate Vector Plot**
  - Expand the Project Manager tree to view **Field Overlays > J** plots
  - Right click on the plot **J_Vector 1** and select **Animate**
  - In Setup Animation window, press **OK** to accept default settings
  - Select **Export** button in Animation window to export animation in GIF format

Calculate Open Circuit Voltage on Ring2 and Ring3

- **To Calculate Open Circuit Voltage for Ring2**
  - Calculate the voltage (V2') induced across the open-circuit point in Ring2. This voltage is the negative of the voltage that is required to ensure that the total current flow around Ring2 is zero. It can be calculated by hand from the impedance matrix using the following formula:

$$
\begin{aligned}
V_2^{'} &= -I_1 * Z_{12} \\
&= -I_1 * (1.728\text{e}^{-5} + \text{j}\omega 1.089\text{e}^{-8}) \\
&= -1 * (1.728\text{e}^{-5} + \text{j}2\pi * 10000 * 1.089\text{e}^{-8}) \\
&= -1.728\text{e}^{-5} - \text{j}6.842\text{e}^{-4} \\
&= 6.844\text{e}^{-4} \angle 88.5^{\circ} \quad (\text{V peak})
\end{aligned}
$$

- **Alternative Method to calculate Open Circuit voltage**
  - The open circuit voltage (V2') can also be calculated by integrating the average electric field in Ring2 around its circumference using the following formula, where E = - jωA, ω = 2 pi (10000), and area = 3.1257e-6:

$$
\begin{aligned}
V_2^{'} &= \int \overline{E} \bullet d\overline{L} \\
&= \frac{1}{area} \int_{RZ} \overline{E} \bullet d\overline{V} \\
&= \frac{1}{area} \int_{RZ} -j\omega \overline{A} \bullet d\overline{V} \\
&= \frac{2 * \pi * 10000}{3.1257e^{-6}} \int_{RZ} -j\overline{A} \bullet d\overline{V} \\
&= 6.85e^{-4} (Vpeak)
\end{aligned}
$$

Calculate Complex Magnitude of Voltage

- **To calculate the complex magnitude of the voltage**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > A**
    - Select **Vector > Scal? > ScalarPhi**
    - Select **General > Complex > CmplxMag**
      - Since A_vector is a complex number, the CmplxMag includes both real and imaginary components. Note that the complex magnitude is equal to:

$$A_{CmplxMag} = \sqrt{A^2_{\phi_real} + A^2_{\phi_imag}}$$

  - To multiply by ω; select:
    - Select **Input > Number**
      - Type: **Scalar**
      - Value: **2**
      - Press **OK**
    - Select **Input > Function**
      - Select **Scalar**
      - Select **Freq** from list
      - Press **OK**
    - Select **Input > Constant > Pi**
    - Select **General > ***
    - Select **General > ***
    - Select **General > ***
  - To divide by area; select:
    - Select **Input > Number**
      - Type: **Scalar**
      - Value: **1**
      - Press **OK**
    - Select **Input > Geometry**
      - Select **Surface**
      - Select **Ring2**
      - Press **OK**

- Select **Scalar > Integral > XY**
- Select **General > /**

- Finally, do an RZ integration
  - Select **Input > Geometry**
    - Select **Volume**
    - Select **Ring2**
    - Press **OK**
  - Select **Scalar > Integral > RZ**
  - Select **Output > Eval**

- The open circuit voltage induced across the open point in ring2 is 6.86e-004 V. This equals the voltage calculated by hand from Z12 in the impedance matrix, as well as that calculated by integrating the average electric field. This is the complex magnitude of the voltage. The real and imaginary components can be individually determined by substituting Complex/Imag and Complex/Real in the steps above. These voltages are: V2'(real) = -1.80e-005 and V2'(imaginary) = -6.85e-004 which are nearly the same as the voltage calculated by hand on the previous page.

- Reference: “Prediction and Use of Impedance Matrices for Eddy-Current Problems,” IEEE Transactions on Magnetics, Kent R. Davey and Dalian Zheng, vol. 33 pp. 2478-2485, 1997.

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.

$$F_{DC} = \frac{1}{2}\int \mathrm{Re}\left| \bar{J} \times \bar{B}^{*} \right| dV$$

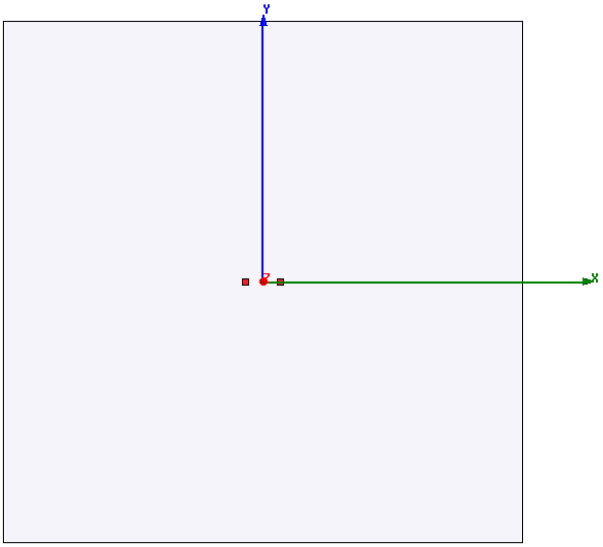
$$F_{AC} = \frac{1}{2}\int \left| \bar{J} \times \bar{B} \right| dV \qquad evaluated\ at\ phase\ (\omega t = degrees)$$

$$F_{INST} = F_{DC} + F_{AC}$$

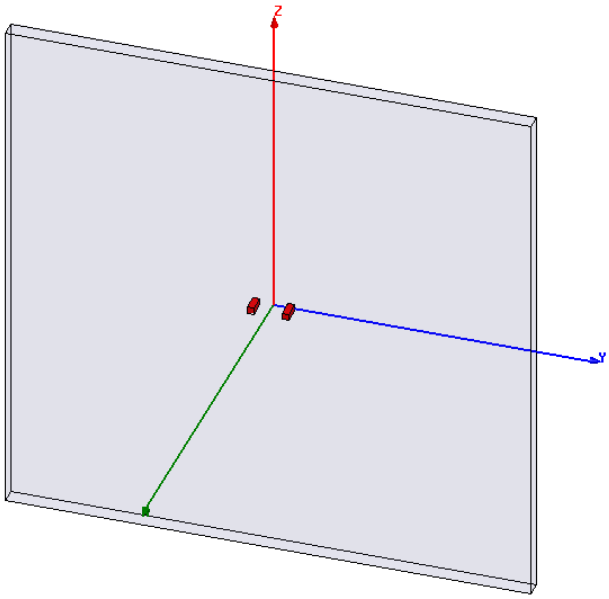
Description

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

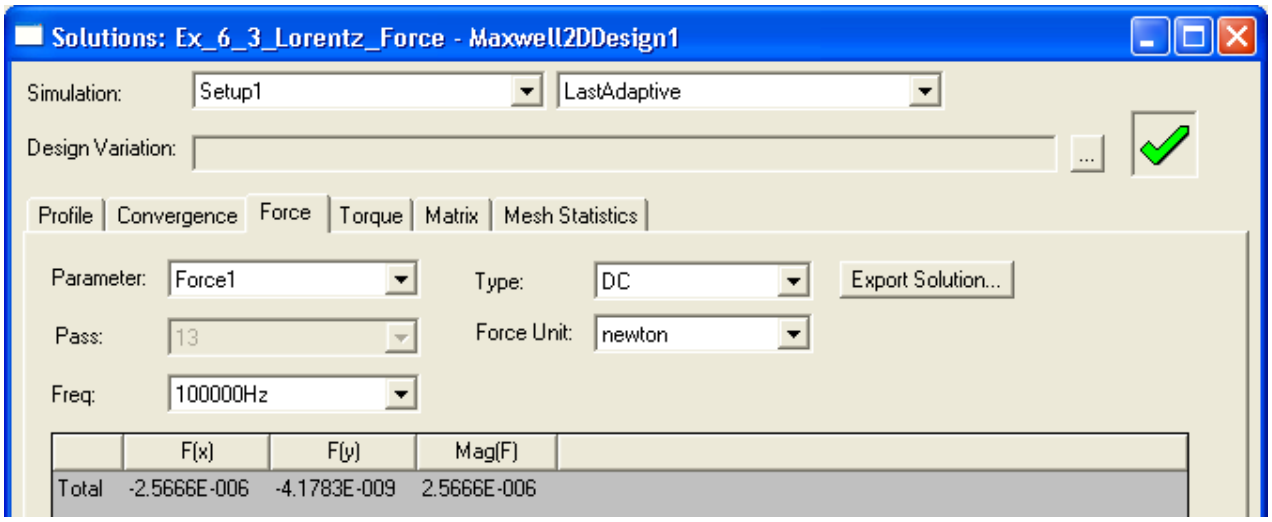
1. Click Right on the **Analysis** folder in the Model Tree and select **Add Solution Setup…**
2. On the General tab, re-set the Number of passes to **15.**
3. Percent Error to **0.01**
4. On the Solver tab, re-set the Adaptive Frequency to **100kHz.**

Solve the Problem

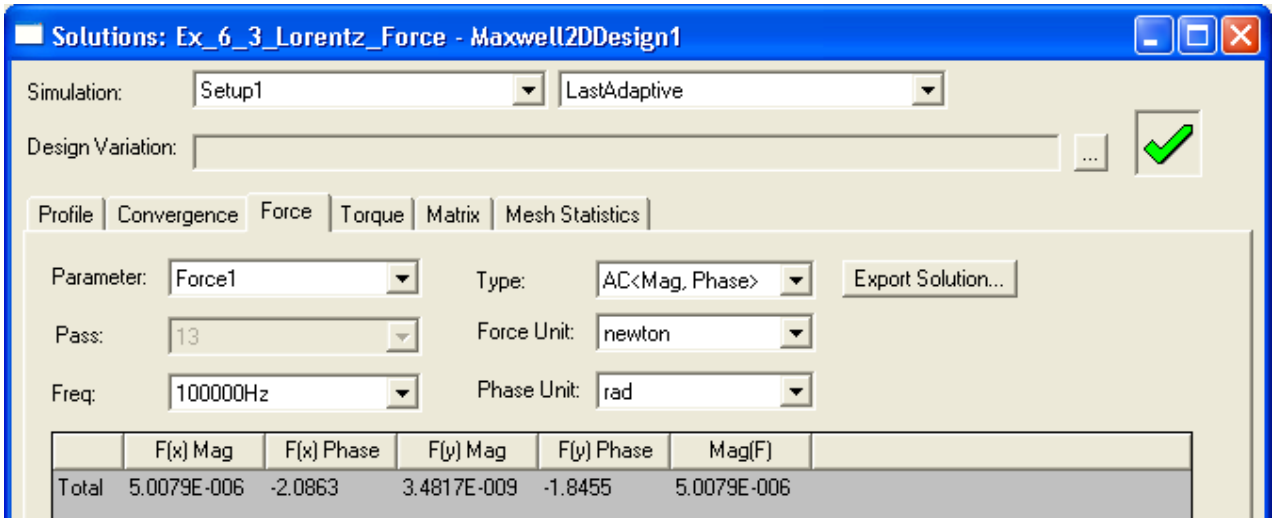
1. Save the project by clicking on menu item **File > Save As**
2. Select the menu item **Maxwell 2D > Validation Check** to verify problem setup
3. Click on **Maxwell 2D > Analyze All**.

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**

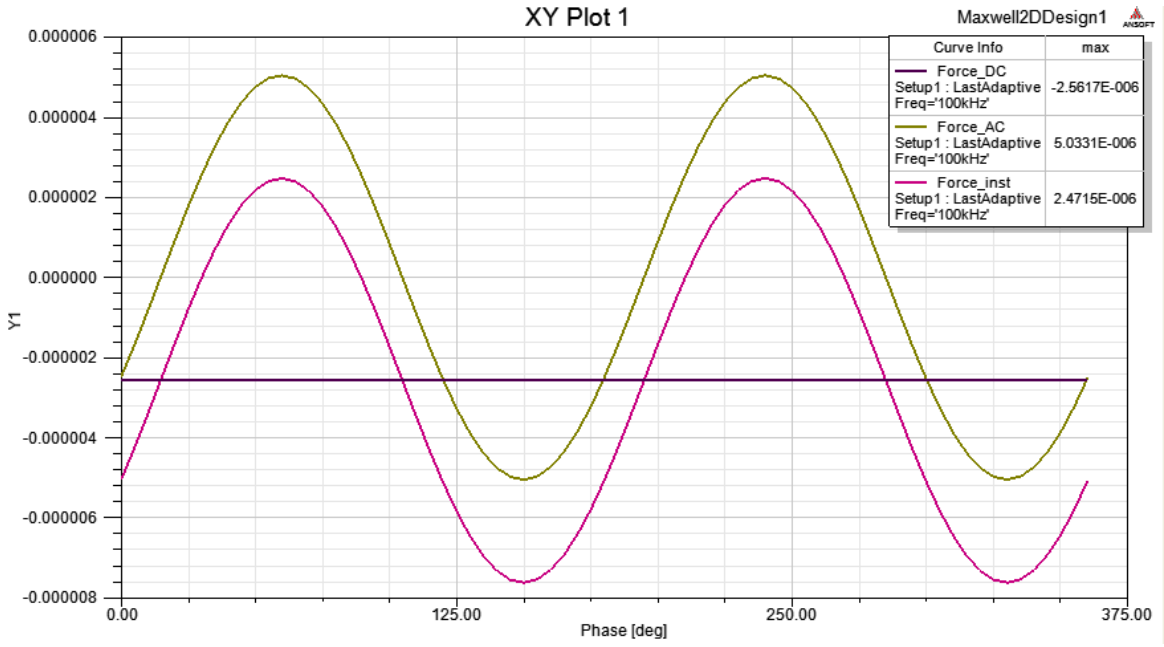
2. Determine the AC component of Lorentz force:
   - **Quantity > J**
   - **Quantity > B > Cross**
   - **Scalar X**
   - **Function > Phase > OK**
   - **Complex > AtPhase**
   - **Number > Scalar > 0.5 > OK**
   - **Multiply**
   - **Geometry > Volume > left > OK**
   - **Integrate**
   - **Add…** 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:

$$F_{MST} = \int \left(\overline{B} \cdot \overline{n}\right) \overline{H} - 0.5\,(\overline{B} \cdot \overline{H})\overline{n}\; dV \quad evaluated\ at\ phase\ (\omega t = degrees)$$

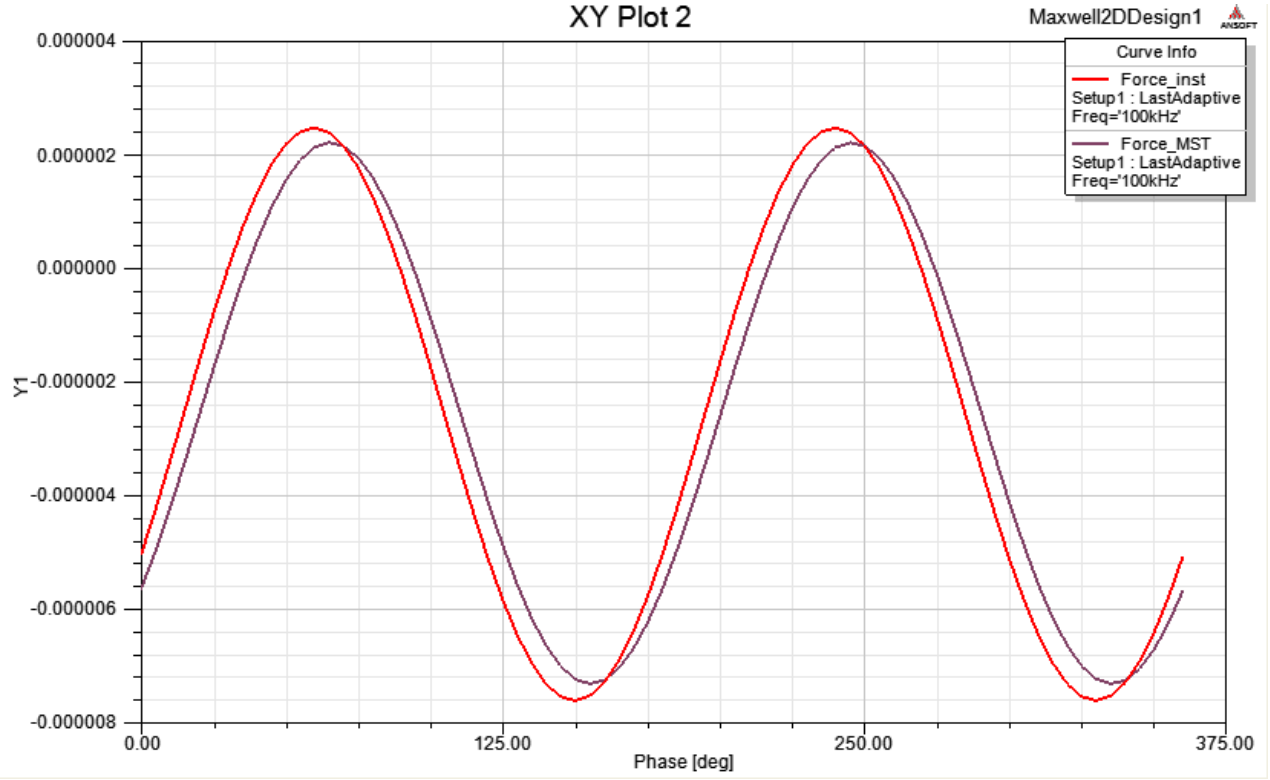
Determine the instantaneous component of force at time wt=0 using the Maxwell Stress Tensor method in the calculator:

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

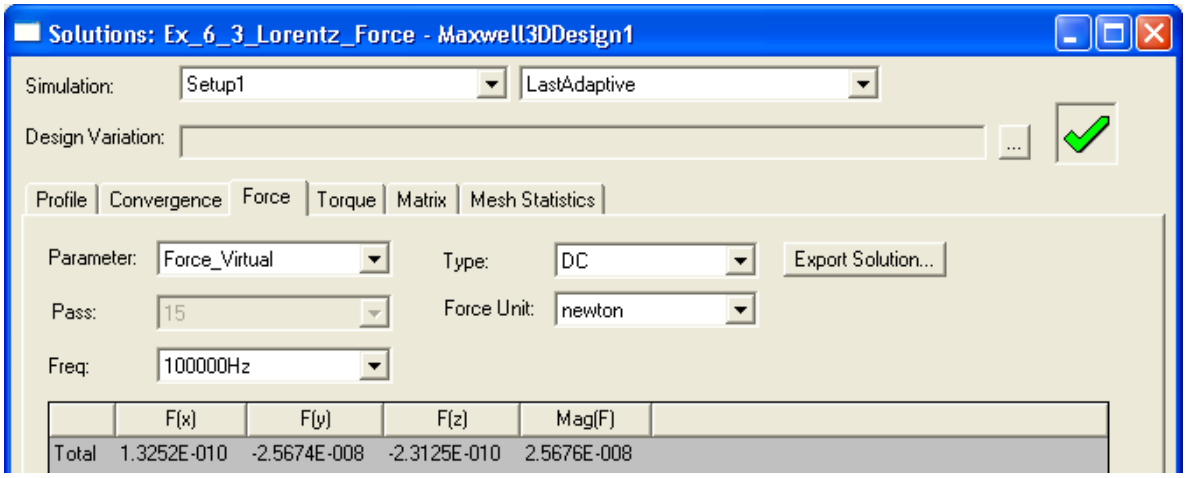
1. Click Right on the **Analysis** folder in the Model Tree and select **Add Solution Setup…**
2. On the General tab, re-set the Number of passes to **15.**
3. Percent Error to **0.01**
4. On the Solver tab, re-set the Adaptive Frequency to **100kHz.**
5. Click **OK** to save the setup.

Solve the Problem

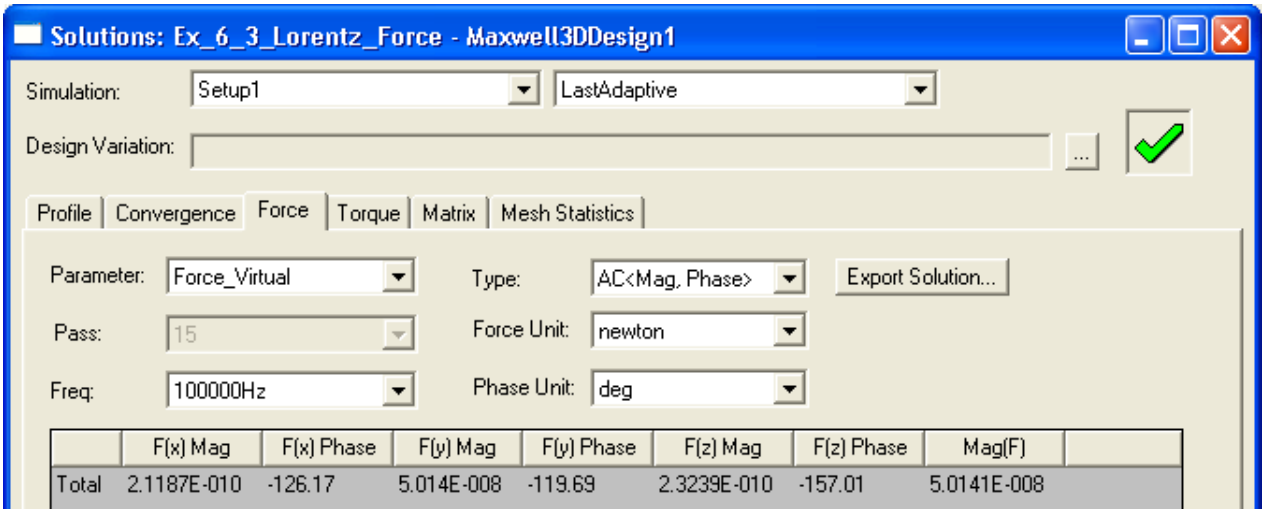
1. Save the project by clicking on menu item **File > Save**
2. Select the menu item **Maxwell 3D > Validation Check** to verify problem setup
3. Click on **Maxwell 3D > Analyze All**.

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

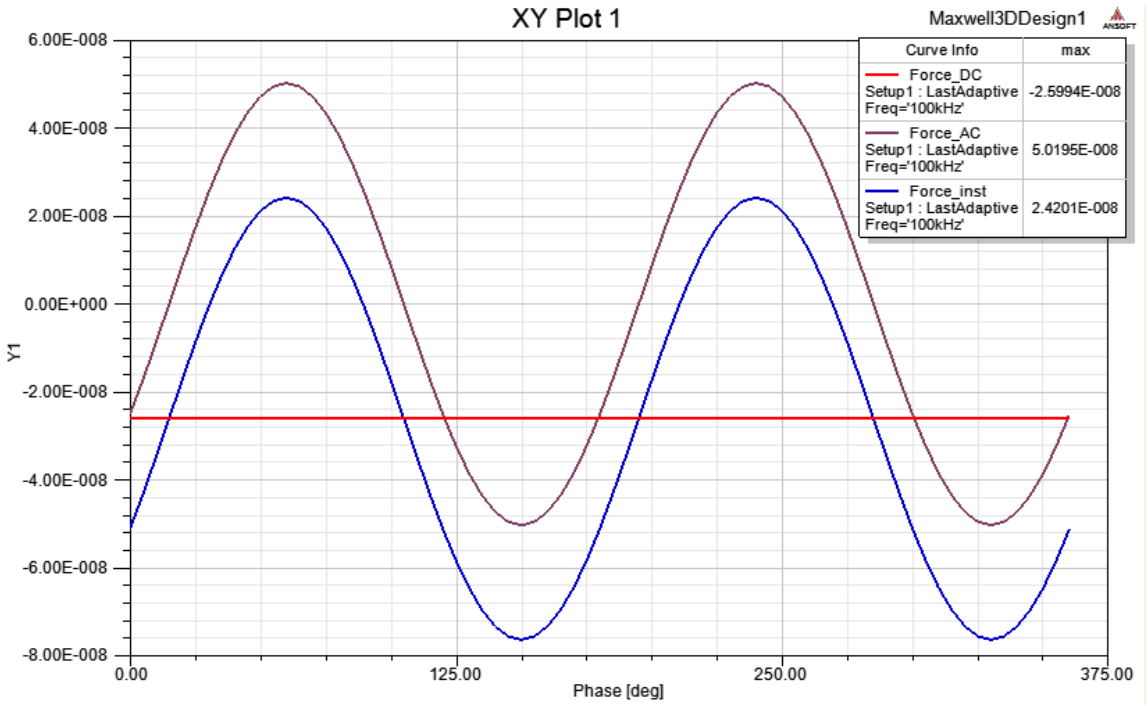
The time-averaged, AC, and instantaneous components Lorentz force can be plotted vs. time by creating named expressions in the calculator using the formulas at the beginning of the application note.

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

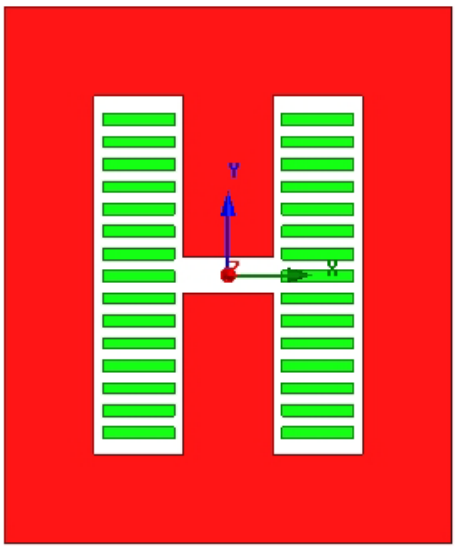
Chapter 7.0 - Transient Examples

- 7.1 - Gapped Inductor Model
- 7.2 - Solenoid Problem with an External Circuit
- 7.4- Core Loss

Transient – Gapped Inductor in XY Plane

Introduction

- The Maxwell 2D Field Simulator's XY transient solver can be used to demonstrate the difference between sinusoidal and non-sinusoidal excitation in a gapped inductor. In addition, the fringing flux effect on AC losses can be considered in this device.
- The inductor consists of a ferrite core with a gap in the center leg. The winding has 15 copper turns which are connected in series. The inductor is excited by a 120A-60Hz sinusoidal current and a 20A-1kHz triangular current superimposed on it.
- Although no motion occurs in this problem, the transient time-stepping solver is needed because of the complex waveform of the current.



- After the problem is solved, the user can do the following:
  - View the flux lines and power loss density in the winding.
  - Plot the instantaneous power loss in the winding vs. time.
  - Calculate the average power loss over time.
- A second simulation will be done using only a sinusoidal excitation in order to compare the losses.

Transient - Gapped Inductor in XY Plane

Launching Maxwell

- **To access Maxwell:**
  - 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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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

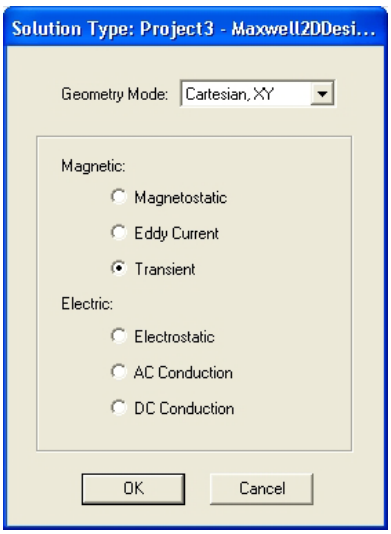
Transient - Gapped Inductor in XY Plane

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 2D Design***, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - Solution Type Window:
    1. Geometry Mode: **Cartesian, XY**
    2. Choose **Magnetic > Transient**
    3. Click the **OK** button



Set Default Units

- **To Set Default Units**
  - Select the menu item ***Modeler > Units***
  - In Units window,
    1. Select units: **in (inches)**
    2. Press **OK**



Set Model Depth

- For all transient XY models, the depth must be specified. Then all losses and force results reported are for that particular depth.
- **To Set Model Depth**
  - Select the menu item *Maxwell 2D > Design Settings*
  - In Design Settings window,
    1. **Model Depth** tab
       - Set Model Depth to **1 in**
    2. Press **OK**

Create Model

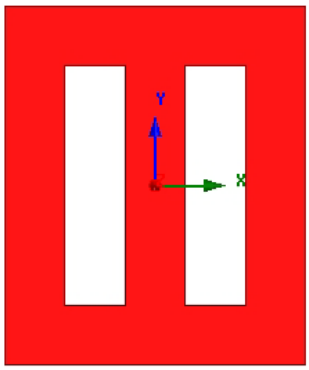
- The model consists of a core and a winding. Note that each turn of the winding is exactly modeled and is "solid" in order to accurately determine the AC losses.

Create Core

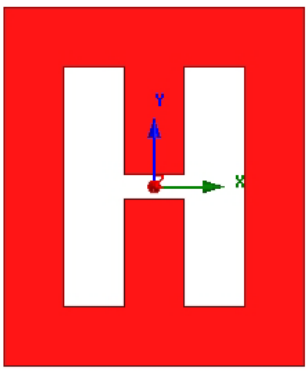
- **Draw Core**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **-2.5**, Y: **-3**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **5**, dY: **6**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and goto the Properties window
    1. Change the name of the object to **Core** and color to **Red**
    2. Change the material of the object to **ferrite**
- **Draw Core windows**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **-1.5**, Y: **-2**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **1**, dY: **4**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and goto the Properties window
    1. Change the name of the object to **window**

Transient - Gapped Inductor in XY Plane

- **Duplicate Core window**
  - Select the sheet **window** from history tree
  - Select the menu item *Edit > Duplicate > Along Line*
    1. Using the coordinate entry fields, enter 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: **2**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Total Number: **2**
    4. Attach to original: ☑ **Checked**
    5. Press **OK**
- **Subtract Sheets**
  - Press **Ctrl** and select the sheets **Core** and **window**
  - Select the menu item *Modeler > Boolean > Subtract*
  - In Subtract window,
    1. Blank Part: **Core**
    2. Tool Part: **window**
    3. Press **OK**



- **Create Core Gap**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **-0.5**, Y: **-0.2**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **1**, dY: **0.4**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and goto the Properties window
    1. Change the name of the object to **gap**
- **Subtract Sheets**
  - Press **Ctrl** and select the sheets **Core** and **gap**
  - Select the menu item *Modeler > Boolean > Subtract*
  - In Subtract window,
    1. Blank Part: **Core**
    2. Tool Part: **gap**
    3. Press **OK**

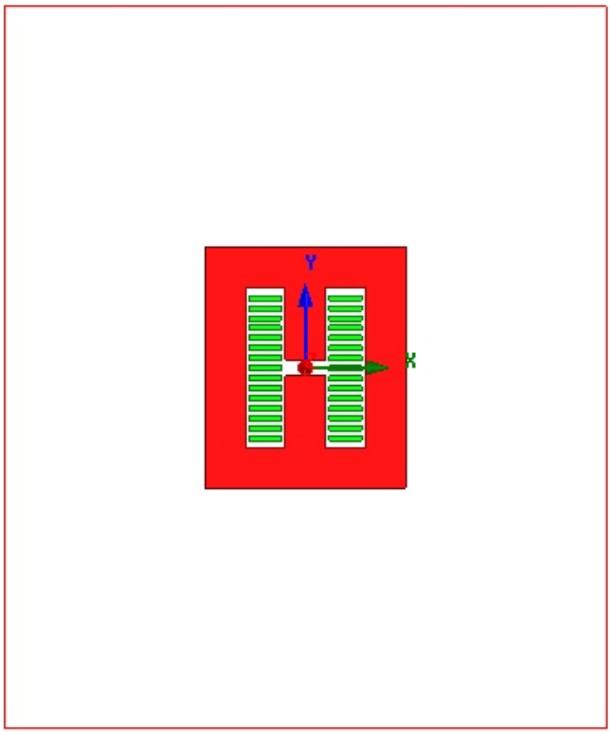


Create Windings

- **Draw Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **-1.4**, Y: **-1.825**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.8**, dY: **0.125**, dZ: **0**, Press the **Enter** key
  - Select the resulting sheet and goto the Properties window
    1. Change the name of the object to **Coil** and color to **Green**
    2. Change the material of the object to **copper**
- **Create Return Coil**
  - Select the sheet **coil** from history tree
  - Select the menu item *Edit > Duplicate > Along Line*
    1. Using the coordinate entry fields, enter 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: **2**, dY: **0**, dZ: **0**, Press the **Enter** key
    3. Total Number: **2**
    4. Attach to original: ☐ **Unchecked**
    5. Press **OK**
  - Select the resulting sheet and goto the Properties window
    1. Change the name of the object to **Coil_return**
- **Duplicate Coils**
  - Press **Ctrl** and select the sheets **Coil** and **Coil_return**
  - Select the menu item *Edit > Duplicate > Along Line*
    1. Using the coordinate entry fields, enter 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: **0.25**, dZ: **0**, Press the **Enter** key
    3. Total Number: **15**
    4. Attach to original: ☐ **Unchecked**
    5. Press **OK**

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: **100**
    4. Press **OK**

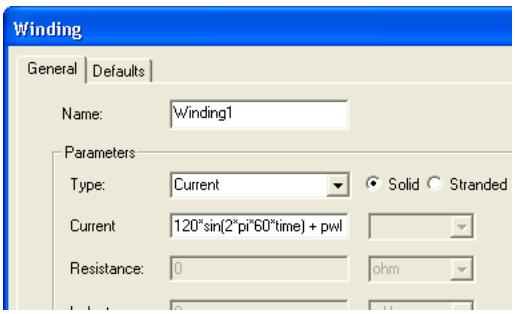


Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - In Balloon Boundary window,
    1. Press **OK**

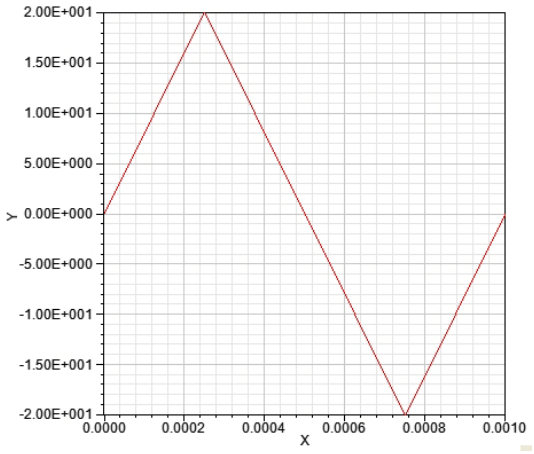
Assign Excitation

- A 120A 60Hz sinusoidal current will be assigned to the 15 series turns in the inductor. In addition, a 20A 1kHz triangular current source will be added on top of the sinusoidal current. The winding consists of a go and a return for the left and right sides of the winding. A simple sinusoidal function with be used to create the 60Hz component while a dataset **"ds1"** will be used to create the triangular component of current.
- **To Assign Coils**
  - Press **Ctrl** and select the sheets **Coil, Coil1, Coil2,….., Coil14**
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
  - In Coil Excitation window,
    1. Base Name: **Left**
    2. Number of Conductors: **1**
    3. Polarity: **Positive**
    4. Press **OK**
  - Press **Ctrl** and select the sheets **Coil_return, Coil_return1,…..,Coil_return14**
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
  - In Coil Excitation window,
    1. Base Name: **Right**
    2. Number of Conductors: **1**
    3. Polarity: **Negative**
    4. Press **OK**
- **To Add a Winding**
  - Select the menu item *Maxwell 2D > Excitations > Add Winding*
  - In Winding window,
    1. Name: **Winding1**
    2. Type: **Current**
    3. Value: **120*sin(2*pi*60*time) + pwl_periodic(ds1, Time)**
    4. Select **Soild**
    5. Press **OK**

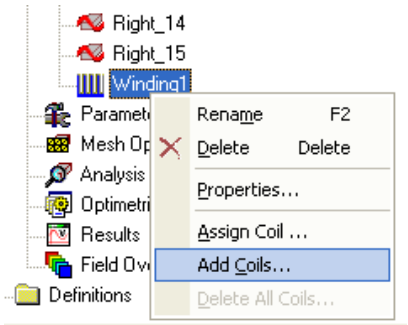


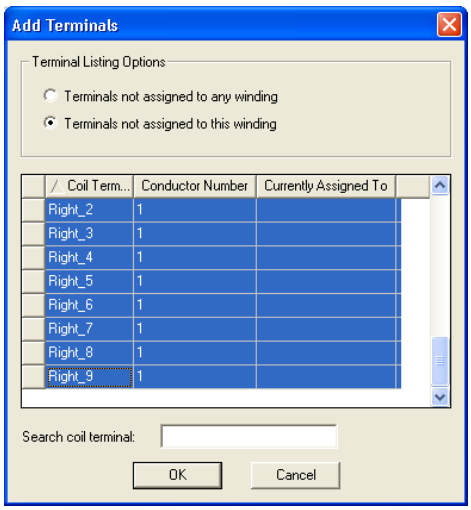
- Add Dataset window will pop up
- In the window,
  1. Specify X-Y Coordinates as give in below table
  2. Press **OK**

	X	Y
1	0	0
2	0.00025	20
3	0.00050	0
4	0.00075	-20
5	0.001	0



- **To Add Coils to the winding**
  - Expand the Project Manager tree to view Excitations
  - Right click on the **Winidng1** from the tree and select **Add Coils**
  - In Add Terminals window,
    - Select the first terminal from the list
    - Press **Shift** and select the last terminal from the list
    - All inbetween terminal will be automatically selected.
    - Press **OK**



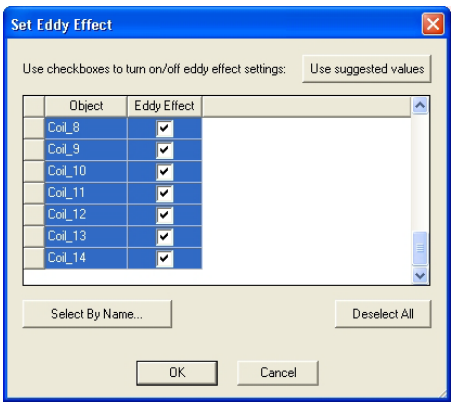


Set Eddy Effects

To Set Eddy Effects

- Select the menu item *Maxwell 2D > Excitations > Set Eddy Effects*
- In Set Eddy Effects window,
  1. Select the button **Select By Name**
  2. In Select By Name window,
     - Type **Coil***
     - Press **OK**
     - All Coils will be selected
  3. While all coils are selected, Check the Eddy Effects box in any of the coils.
  4. Press **OK**

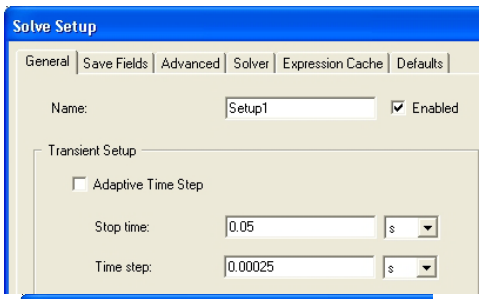


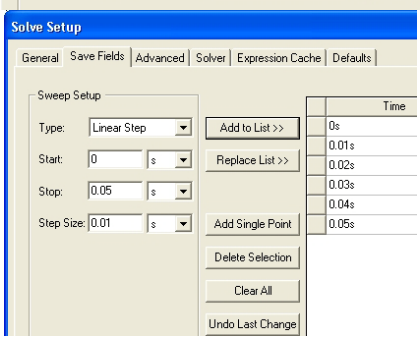


Analysis Setup

To Create Analysis Setup

- Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
- In Solve Setup window,
  1. **General** tab
     - Stop Time: **0.05 sec**
     - Time Step: **0.00025 sec**
  2. **Save Fields** tab
     - Type: **Linear**
     - Start: **0 sec**
     - Stop: **0.05 sec**
     - Step Size: **0.01 sec**
     - Select the button **Add to List**
  3. Press **OK**





Assign Mesh Operation

- In the transient solvers, the mesh is not automatically created. It must either be linked to a magnetostatic or eddy current design, or you can manually create it. In this example, the mesh will be manually created.
- **To Assign Mesh Operation to Coils**
  - Press **Ctrl** and select all coils (total 30) from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Coils_Inside**
    2. Restrict Length of Elements: ☐ **Uncheck**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **500**
    5. Press **OK**
  - Note that by choosing "Inside Selection" instead of "On Selection", the mesh operation is applied evenly through the area of the conductors as opposed to being applied only on the outer perimeter of the conductor.
- **To Assign Mesh Operation to Core**
  - Select the sheet **Core** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Core_Inside**
    2. Restrict Length of Elements: ☐ **Uncheck**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **500**
    5. Press **OK**

Transient - Gapped Inductor in XY Plane

Analyze

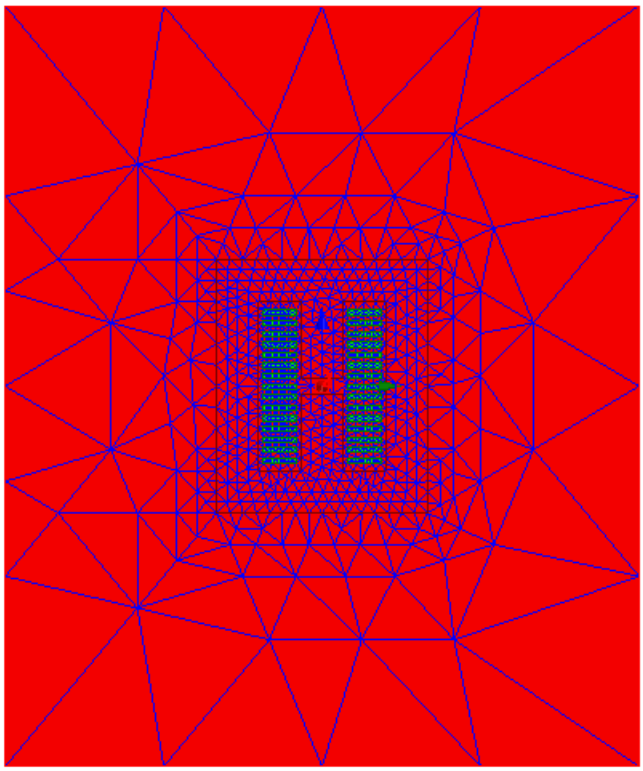
- **To start the solution process:**
  - Select the menu item *Maxwell 2D > Analyze All*

Set Solution Context

- **To Set Solution Context**
  - Select the menu item *View > Set Solution Context*
  - In Set View Context window,
    1. Set Time to **0.02s**
    2. Press **OK**

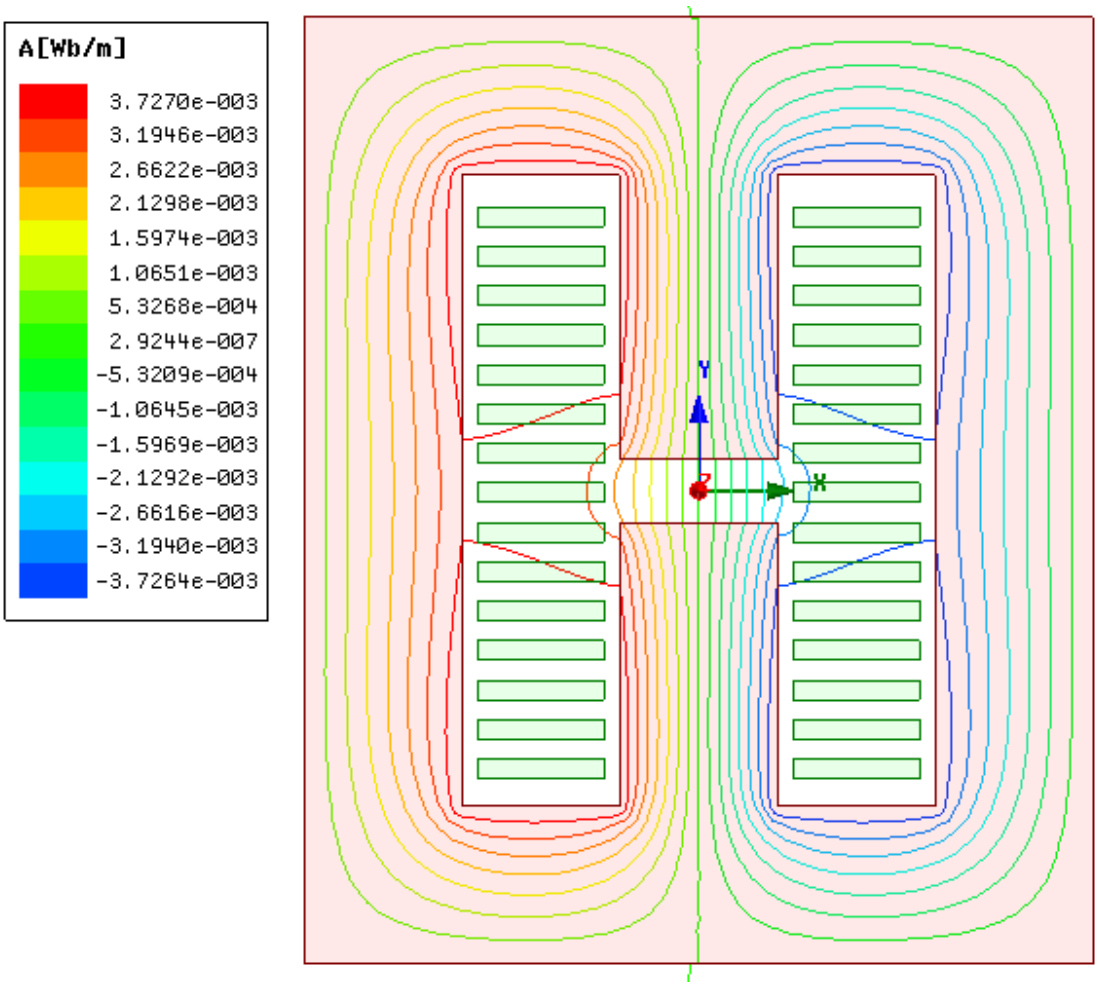
Plot Mesh

- **To Plot Mesh**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  - In Create Mesh Plot window,
    1. Press **Done**



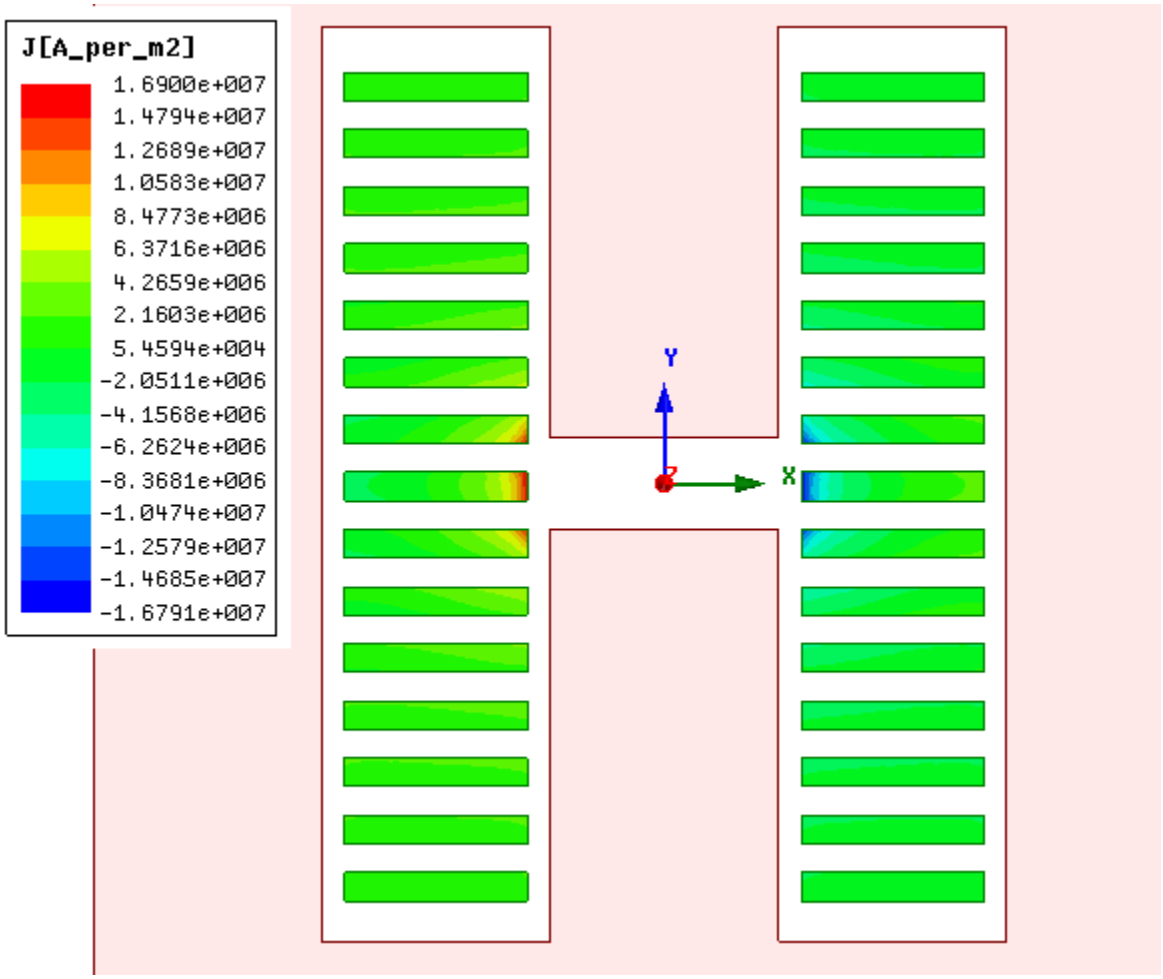
Plot Flux Lines

- **To Plot Flux Lines**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window,
    1. Press **Done**



Plot Current Density

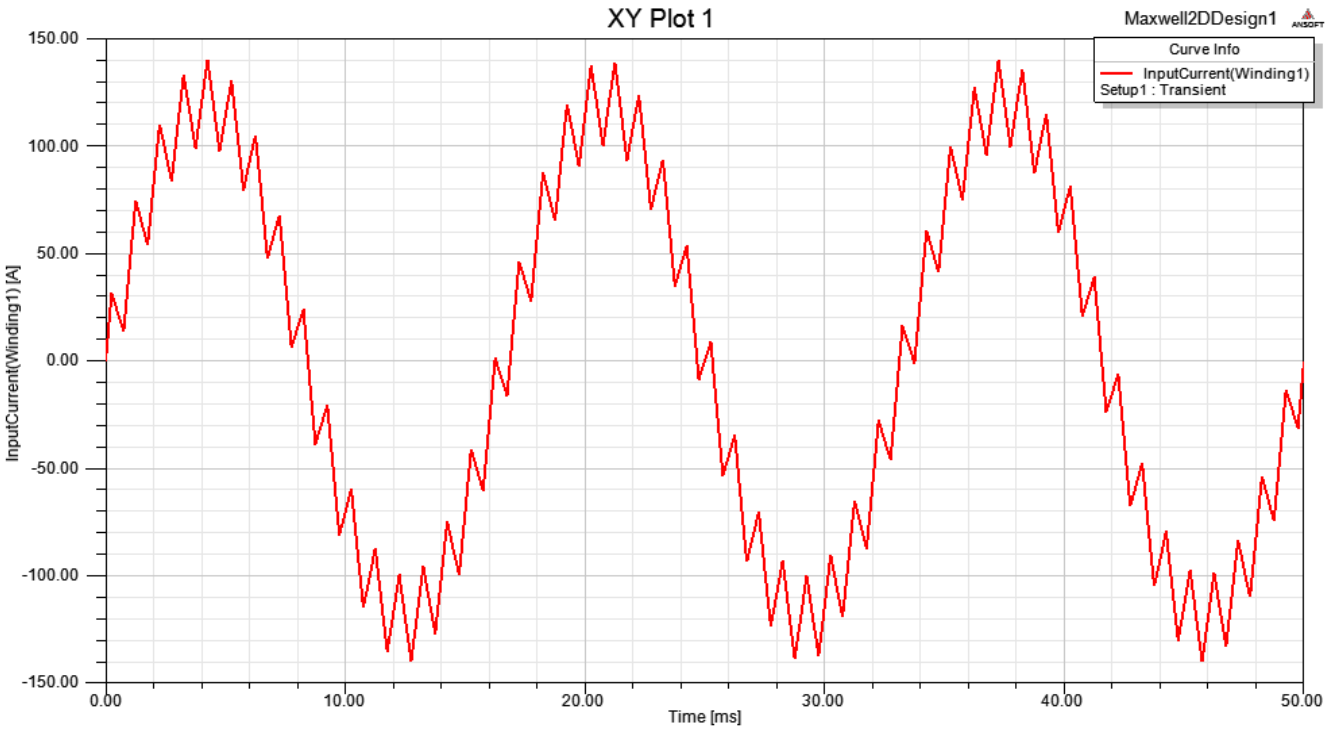
- The current density in the coils will be greater near to the gap in the core because fringing flux caused induced proximity losses in the copper.
- **To Plot Current Density on Coils**
  - Press **Ctrl** and select all coils from history tree
  - Select the menu item *Maxwell 2D > Fields > Fields > J > Jz*
  - In Create Field Plot window,
    1. Press **Done**



Plot Input Current

To Plot Current

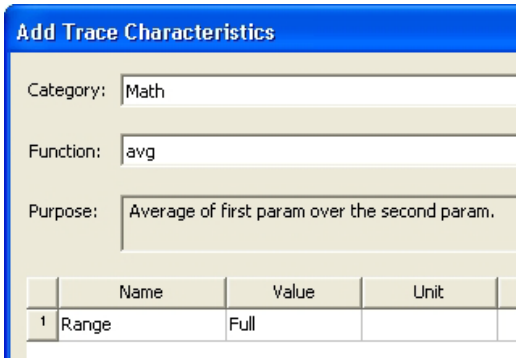
- Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
- In Report window,
  1. Category: **Winding**
  2. Quantity: **InputCurrent(Winding1)**
  3. Select **New Report**

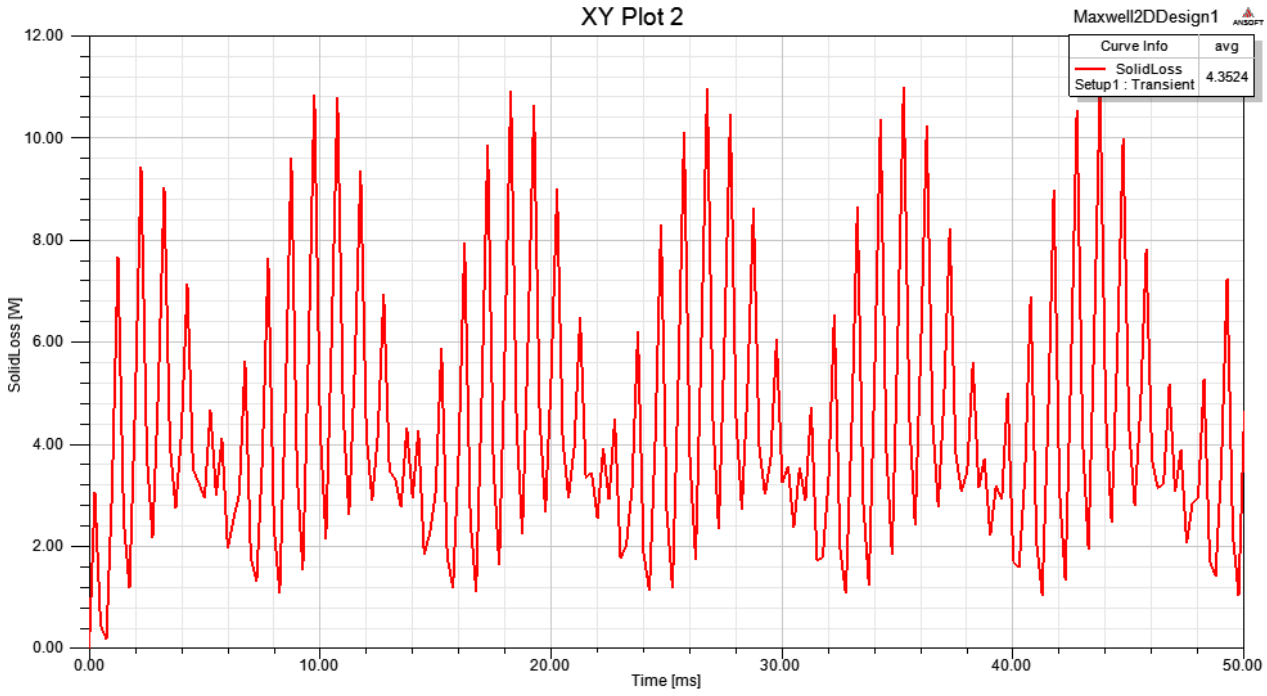


Transient - Gapped Inductor in XY Plane

Plot Losses in Windings

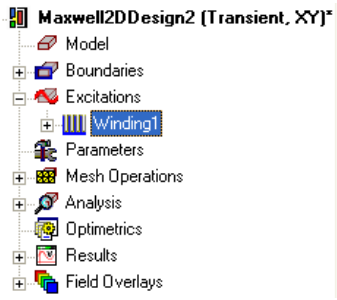
- **To Plot Losses**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In Report window,
    1. Category: **Loss**
    2. Quantity: **SolidLoss**
    3. Select **New Report**
- **Calculate Average Losses**
  - Right click on the plot and select *Trace Characteristics > Add*
  - In Add Trace Characteristics window,
    1. Category: **Math**
    2. Function: **avg**
    3. Select **Add** and **Done**
  - Reported losses are around **4.35 watts**

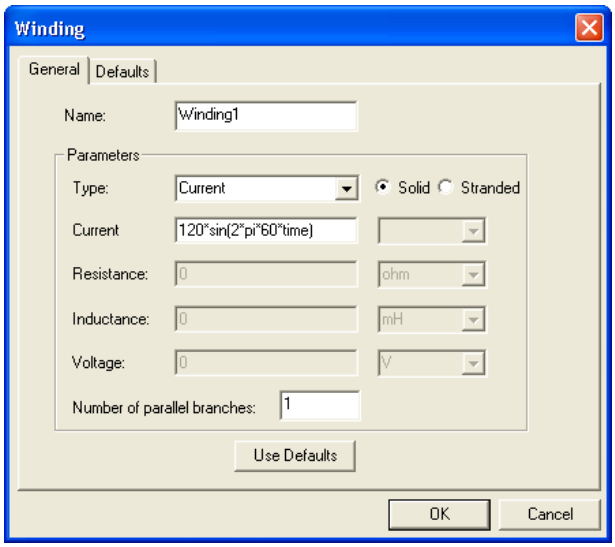




Transient - Gapped Inductor in XY Plane

- Solve with Sinusoidal Excitation
- Copy Design
  - **To Create a Copy of the Design**
    - Select the design **"Maxwell2D Design1"** from Project Manager tree, right click and select **Copy**
    - Right click on the Project name in Project Manager tree and select **Paste**
- Modify Excitations
  - **To Modify Excitation**
    - Expand the Project Manager tree to view **Excitations**
    - Double click on the tab **Winding1** to modify its properties
    - In Winding window,
      1. Change the Value to: **120*sin(2*pi*60*time)**
      2. Press **OK**



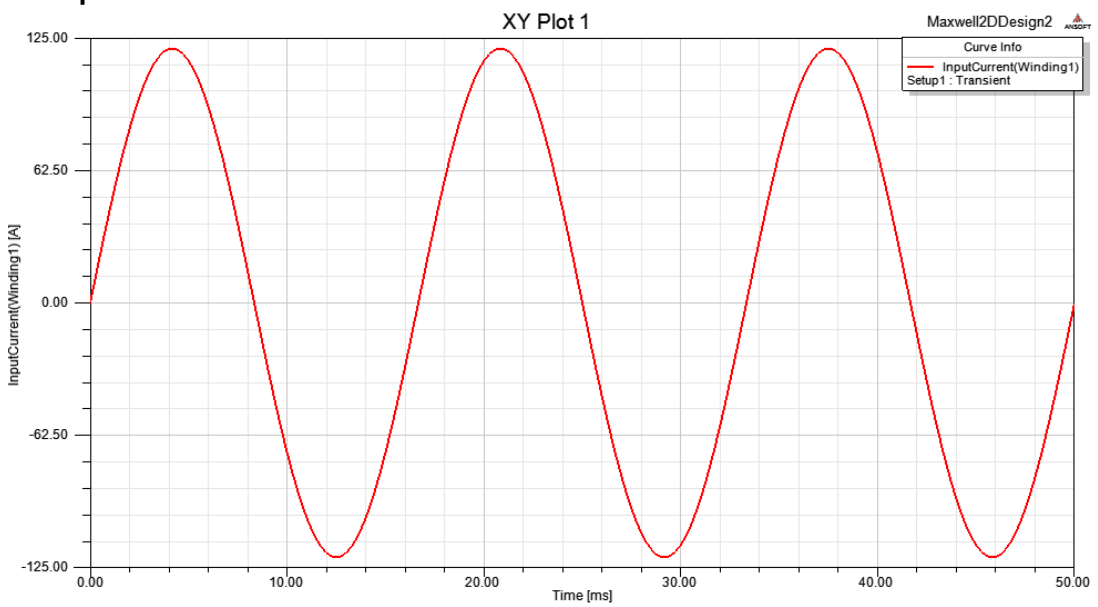


- Analyze
  - **To start the solution process:**
    - Select the menu item *Maxwell 2D > Analyze All*

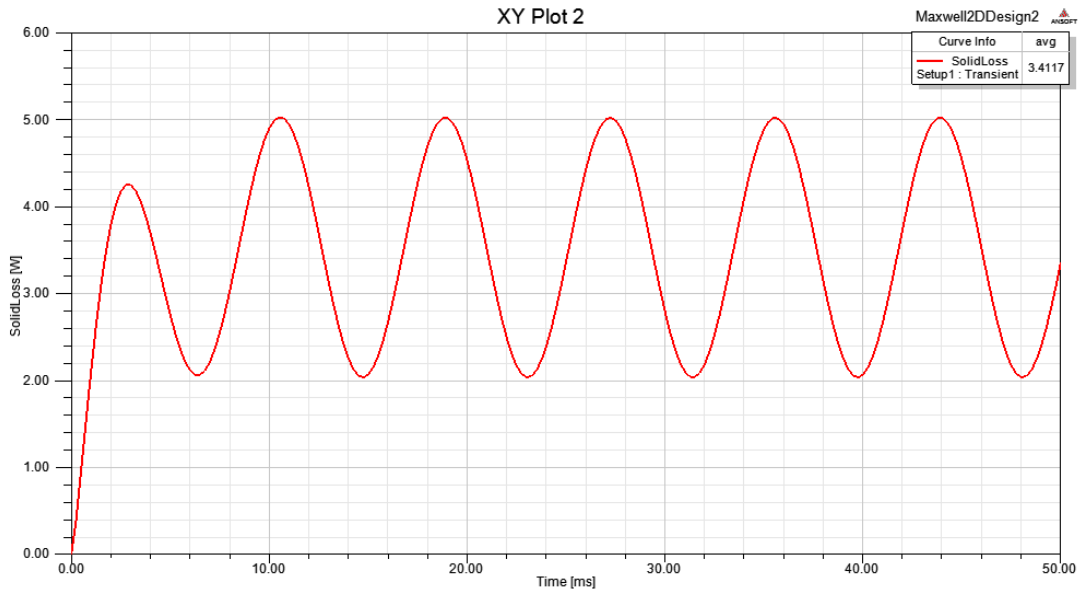
Transient - Gapped Inductor in XY Plane

View Results

Input Current Plot



Solid Losses in windings

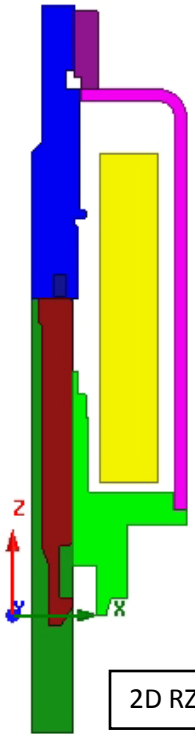


- The average power loss (approx. 3.41W) is smaller than the previous simulation (approx. 4.35W) which included the triangular current component. Also, you can see that the power loss is sinusoidal at twice the excitation frequency.

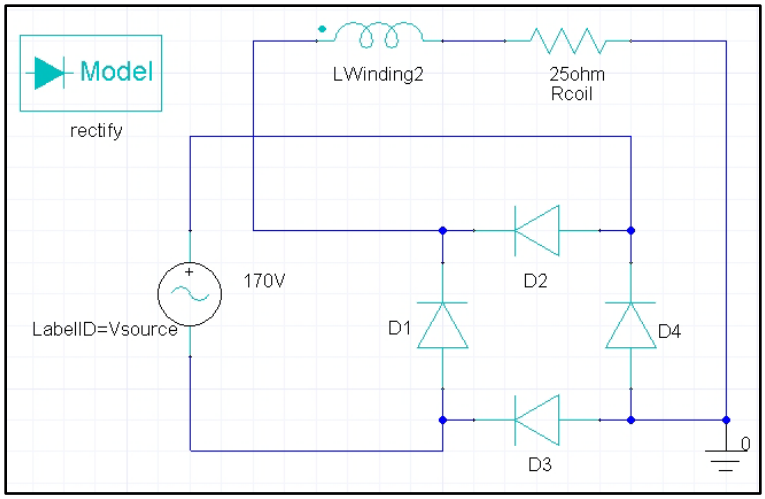
Transient - Solenoid with External Circuit

A Solenoid Problem with an External Circuit

- This example models an AC solenoid using Maxwell 2D. A full wave bridge rectifying drive circuit will be setup to drive the solenoid.



2D RZ Model



Description

- A model of an AC solenoid using an external circuit will be simulated using the 2D RZ transient solver. The source is a 170V 60Hz sinusoidal voltage which is rectified using a full-wave bridge. The mechanical force for a spring and gravity are modeled using an equation. The force, loss, position, speed and winding current, flux, and voltage will be determined.

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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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

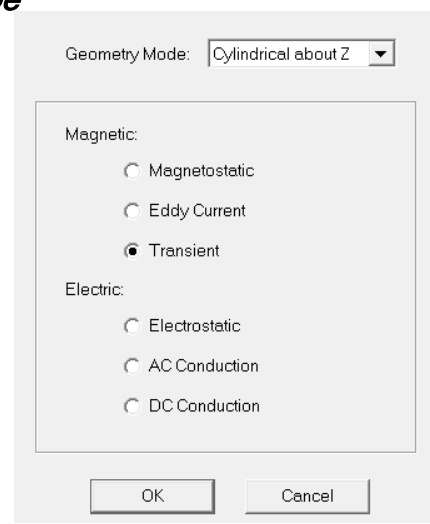
Transient - Solenoid with External Circuit

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 2D Design***, or click on the icon

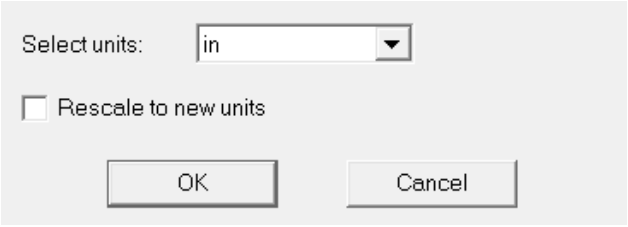
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Transient**
    3. Click the **OK** button



Set Default Units

- **To Set Default Units**
  - Select the menu item ***Modeler > Units***
  - In Units window,
    1. Select units: **in (inches)**
    2. Press **OK**



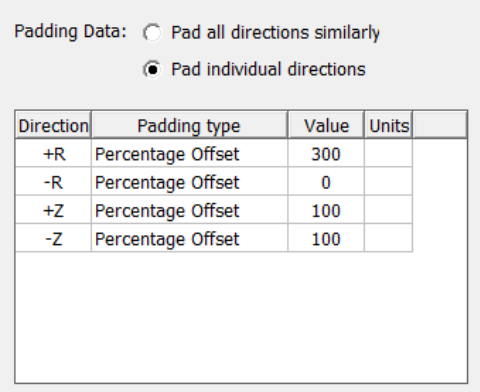
Import Geometry

- **To Import Geometry file**
  - Select the menu item ***Modeler > Import***
  - Locate the file **"Ex_7_02_Solenoid.sm3"** and **Open** it.
  - Check the **"Model Analysis"** and say **"Close"** if the status of objects are **"Good".**

Transient - Solenoid with External Circuit

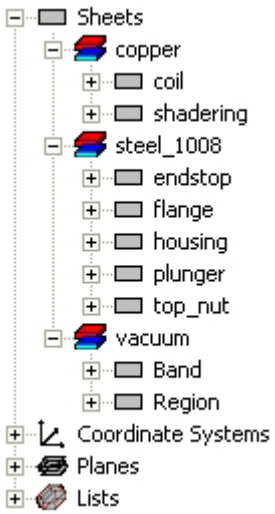
Define Region

- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad individual directions: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value
       - +R = **300**
       - +/- Z = **100**
  - **Note:** Region will not be created in Negative X direction due to RZ-symmetry about the Z-axis.



Assign Materials

- **To Assign Material Copper**
  - Press **Ctrl** and select the sheets **coil** and **shadering**
  - Right click on the objects and select **Assign Material**
  - In Select Definition window,
    1. Type **Copper** in search by name field
    2. Ensure that material **copper** is selected
    3. Press **OK**
- **To Assign Material Vacuum**
  - Select the sheets **Band**
  - Right click on the object and select **Assign Material**
  - Assign the material **vacuum**
- **To Assign Material Steel**
  - Press **Ctrl** and select the sheets **endstop, flange, housing, plunger** and **top_nut**
  - Right click on the objects and select **Assign Material**
  - Assign the material **steel_1008**

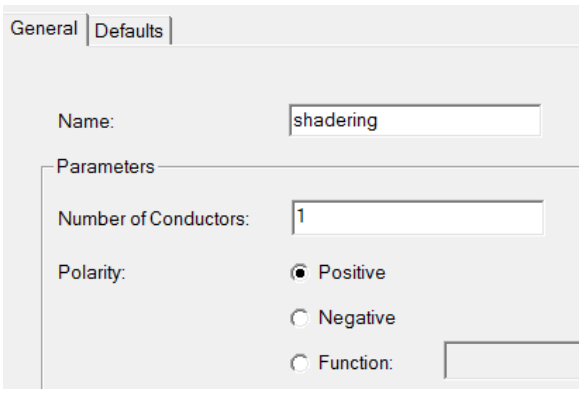


Assign Boundaries

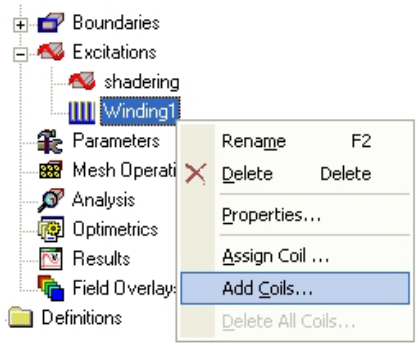
- A no-fringing vector potential boundary will be assigned to outside of the 2D problem region. This forces all flux to stay in the solution region.
- **To Assign Boundary**
  - Select the sheet **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Vector Potential*
  - In Vector Potential window,
    1. Value: **0 weber/m**
    2. Press **OK**
  - Note that even if we have assign vector potential boundary to all edges of the region, the edge that coincides with Z axis will not be considered while simulation since axisymmetric geometry is defined.

Assign Excitations for shadering

- Because the solenoid is a converted "AC" solenoid, it contains a copper "shading ring" which may have eddy currents induced in it. A zero voltage source must be set on the shade ring in order to properly represent a shorted single turn winding and to see if the eddy currents are significant or not.
- **Assign Coil to shadering**
  - Select the sheet **shadering** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
  - In Coil Excitation window,
    1. Name: **shadering**
    2. Number of Conductors: **1**
    3. Polarity: **Positive** (into the screen)
    4. Press **OK**



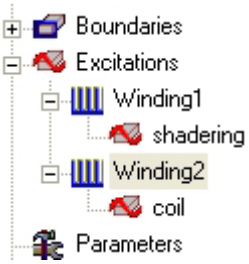
- **Create Winding for shadering**
  - Select the menu item *Maxwell 2D > Excitations > Add Winding*
  - In Winding window,
    1. Name: **Winding1**
    2. Type: **Voltage**
    3. Select **Solid**
    4. Initial Current: **0 A**
    5. Resistance: **0 ohm** (for solid windings, resistance calculated by the solver)
    6. Inductance: **0 mH** (coil inductance always calculated by the solver)
    7. Voltage: **0 V** (zero voltage represents a shorted turn, with no source)
    8. Number of parallel branches: **1**
    9. Press **OK**
- **Add Coil to Winding**
  - Expand the Project Manager tree to view **Excitations**
  - Right click on the tab **Winding1** and select **Add Coils**
  - In Add Terminals window,
    1. Select the terminal **shadering**
    2. Press **OK**



Assign Excitation to Coil

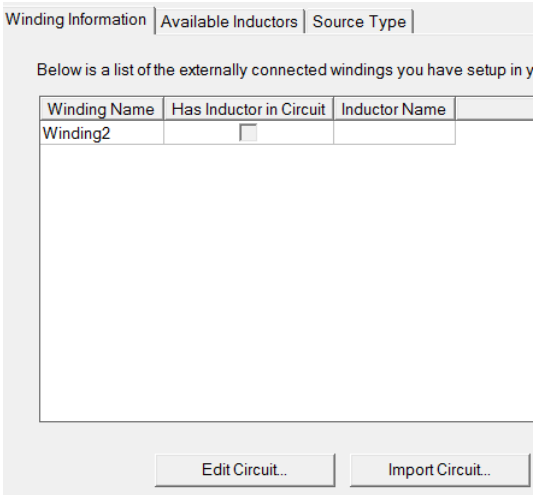
- **Assign Coil terminal**
  - Select the sheet **coil** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
  - In Coil Excitation window,
    1. Name: **coil**
    2. Number of Conductors: **2250**
    3. Polarity: **Positive** (into the screen)
    4. Press **OK**

- **Add Winding for Coil**
  - Select the menu item *Maxwell 2D > Excitations > Add Winding*
  - In Winding window,
    1. Name: **Winding2**
    2. Type: **External**
    3. Select **Starnded**
    4. Initial Current: **0 A**
    5. Number of parallel branches: **1**
    6. Press **OK**
- **Add Coil to Winding**
  - Expand the Project Manager tree to view **Excitations**
  - Right click on the tab **Winding2** and select **Add Coils**
  - In Add Terminals window,
    1. Select the terminal **coil**
    2. Press **OK**

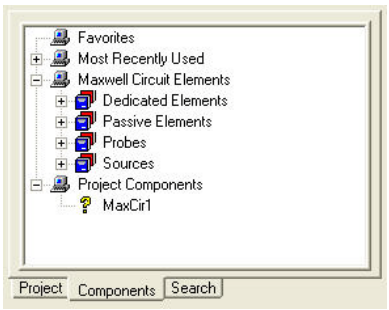


Create External Circuit

- **To Launch Maxwell Circuit Editor**
  - Select the menu item *Maxwell 2D > Excitations > External Circuit > Edit External Circuit*
  - In Edit External Circuit window,
    1. Select **Edit Circuit**
  - Maxwell Circuit Editor will launch in separate window



- **To Create Maxwell Circuit Editor Project**
  - Select the menu item ***File > New***
- **To Add Components to the Circuit**
  - In Maxwell Circuit Editor, select the tab **Components** in Project Manager window to get access to components

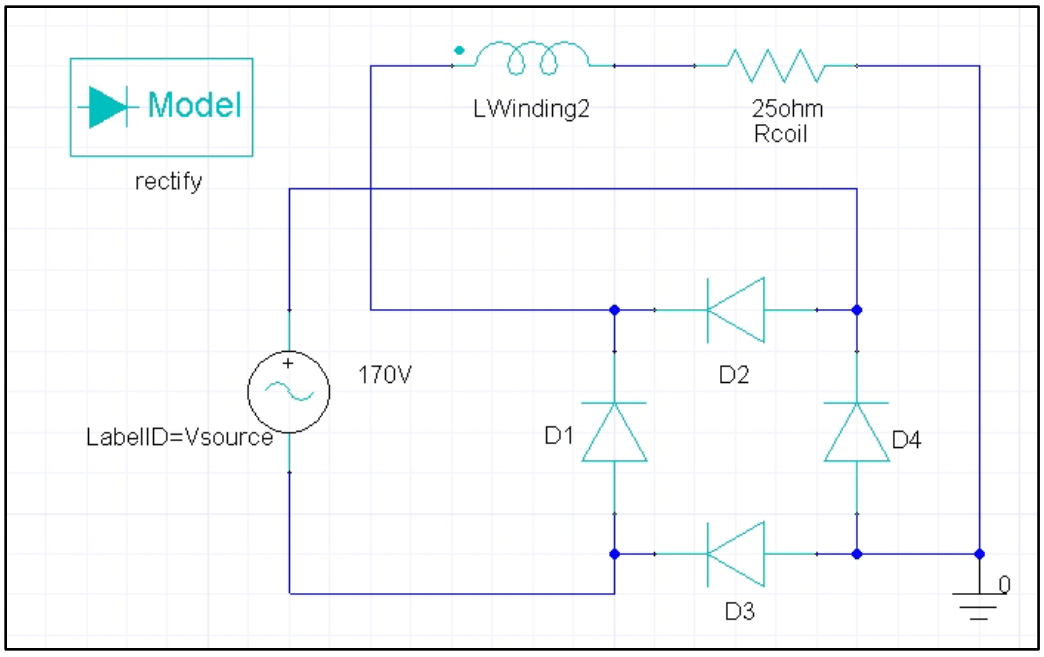


  - Add Diodes
    1. Select the tab **Maxwell Circuit Elements > Passive Elements > Diode** from the tree, drag and drop it on the Project Page
    2. Press **Esc** to exit the component insertion
    3. Double click on the added component to edit its properties
    4. Change Properties as below
       - Name: **1**
       - MOD: **rectify**
       - Press **OK**
    5. Copy the added diode and paste it to create three more diodes named **D2, D3** and **D4**
  - Add Diode Model
    1. Select the tab **Maxwell Circuit Elements > Passive Elements > Diode_Model** from the tree, drag and drop it on the Project Page
    2. Double click on the added component to edit its properties
    3. Change Properties as below
       - Name: **rectify**
       - Press **OK**

- Similarly add below components
  1. Add winding from **Dedicated Elements > Winding**
     - Name: **Winding2**
       - **Note**: This name has to be **exactly** the same name as used in the Winding definition described previously in *Maxwell 2D > Excitations > Add Winding*
  2. Add Resistor from **Passive Elements > Resistor**
     - Name: **coil**
     - R = **25 ohm**
  3. Add Voltage Source from **Sources > VSin**
     - Name: **source**
     - Va = **170 V**
     - VFreq = **60 Hz**
- Add Ground from menu item *Draw > Ground (or Ctrl+G)*

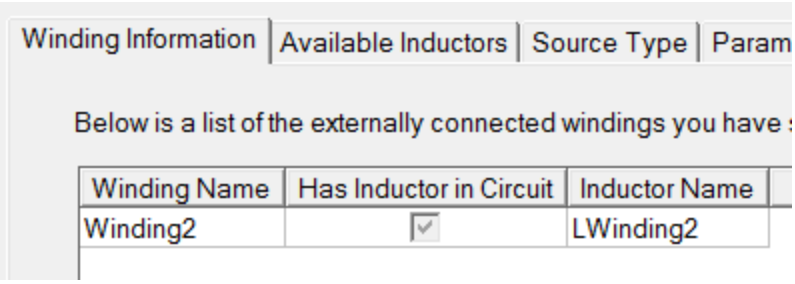
Build the Circuit

- Position the added components and connect them using wires to build a circuit as shown below



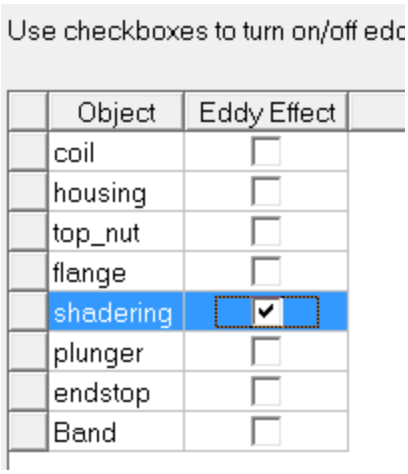
Transient - Solenoid with External Circuit

- **Save File**
  - Select the menu item ***File > Save***
  - Save the file with the name **"Ex_7_02_Solenoid"**
- **Export Circuit to Maxwell**
  - Select the menu item ***Maxwell Circuit > Export Netlist***
  - Save the file with the name **"Ex_7_02_Solenoid.sph"**
- **Import External Circuit**
  - In Maxwell 2D, select the tab **Import Circuit** from **Edit External Circuit** window
  - Select the file **"Ex_7_02_Solenoid.sph"** that was written in last step and select **Open**
  - Edit External circuit window will look as below



Set Eddy Effects

- **To Set Eddy Calculations**
  - Select the menu item ***Maxwell 2D > Excitations > Set Eddy Effects***
  - In Set Eddy Effects window,
    1. For the object **shadering**
       - Eddy Effects: ☑ **Checked**
    2. Press **OK**

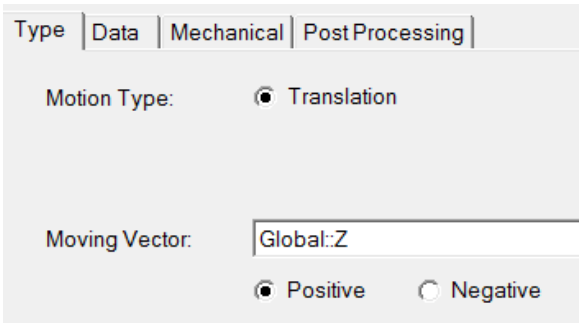


Assign Mesh Operations

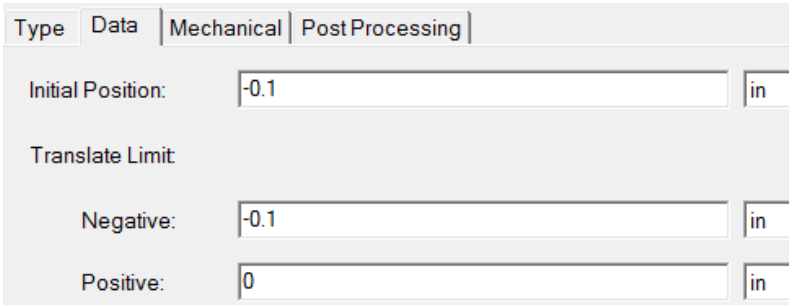
- The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created.
- **Assign Mesh Operations for Band**
  - Select the sheet **Band** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Band_Inside**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **1000**
    5. Press **OK**
- **Assign Mesh Operations for shadering**
  - Select the sheet **shadering** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Shadering_Inside**
    2. Restrict Length of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **50**
    5. Press **OK**
- **Assign Mesh Operations for the Rest**
  - Press **Ctrl** and select the sheets **coil, endstop, flange, housing, plunger,** and **top_nut**
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Others_Inside**
    2. Restrict Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **0.05 in** (Note that units are in inches)
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**

Assign Motion

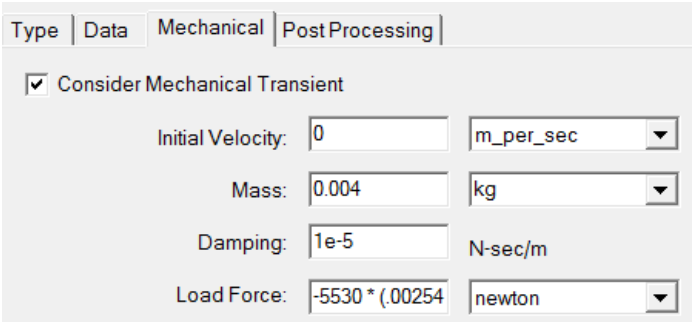
- The plunger is the moving object and is surrounded by the band. (Note: moving objects are never allowed to touch the band. The minimum air gap is 0.002 inches when the solenoid is "fully" closed.) Positive motion is defined as upwards or in the positive Z direction. The starting position is -0.100 inch (or open) so the plunger will move upwards (and close) when the solenoid is energized. The load force acts downward against the direction of motion and consists of: gravity (-0.04N), a spring preload force (-50N), and a variable compression spring force (-5530 * position) which is zero at the starting position and increases as the plunger closes. The units for the intrinsic variable "position" is meters.
- **To Setup Motion**
  - Select the sheet **Band** from history tree
  - Select the menu item ***Maxwell 2D > Model > Motion Setup > Assign Band***
  - In Motion Setup window,
    1. **Type** tab
       - Motion Type: **Translational** (for RZ Model Motion sis always translational)
       - Moving Vector: **Global:Z**
       - Select **Positive**



    2. **Data** tab
       - Initial Position: **-0.1 in**
       - Translate Limit
         - Negative: **-0.1 in**
         - Positive: **0 in**



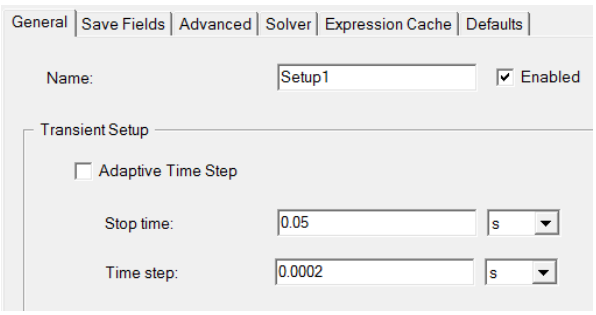
3. **Mechanical** tab
   - Consider Mechanical Transient: ☑ **Checked**
   - Initial Velocity: **0 m_per_sec**
   - Mass: **0.004 kg**
   - Damping: **1e-5 N-sec/m**
   - Load Force: **-5530 * (.00254 + position) -0.04 -50** (units are in Newtons)
   - Press **OK**

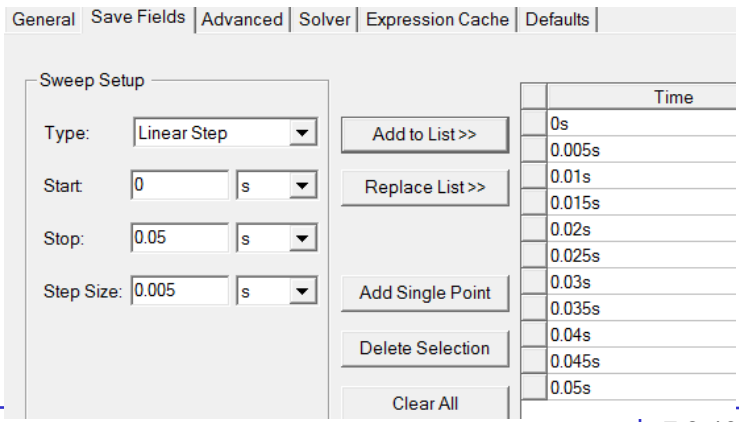


Analysis Setup

- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,

1. **General** tab
   - Stop Time: **0.05 s**
   - Time Step: **0.0002 s**
2. **Save Fields** tab
   - Type: **Linear Step**
   - Start: **0 s**
   - Stop: **0.05 s**
   - Step Size: **0.005 s**
   - Select **Add to List >>**
3. Press **OK**



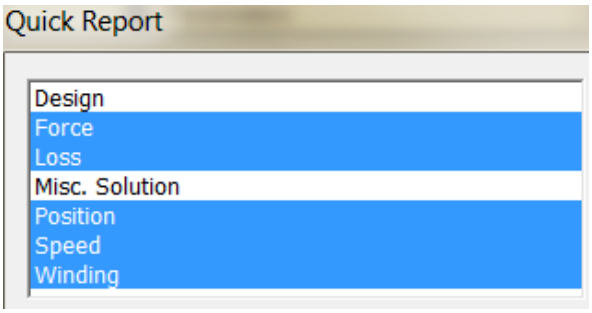


Analyze

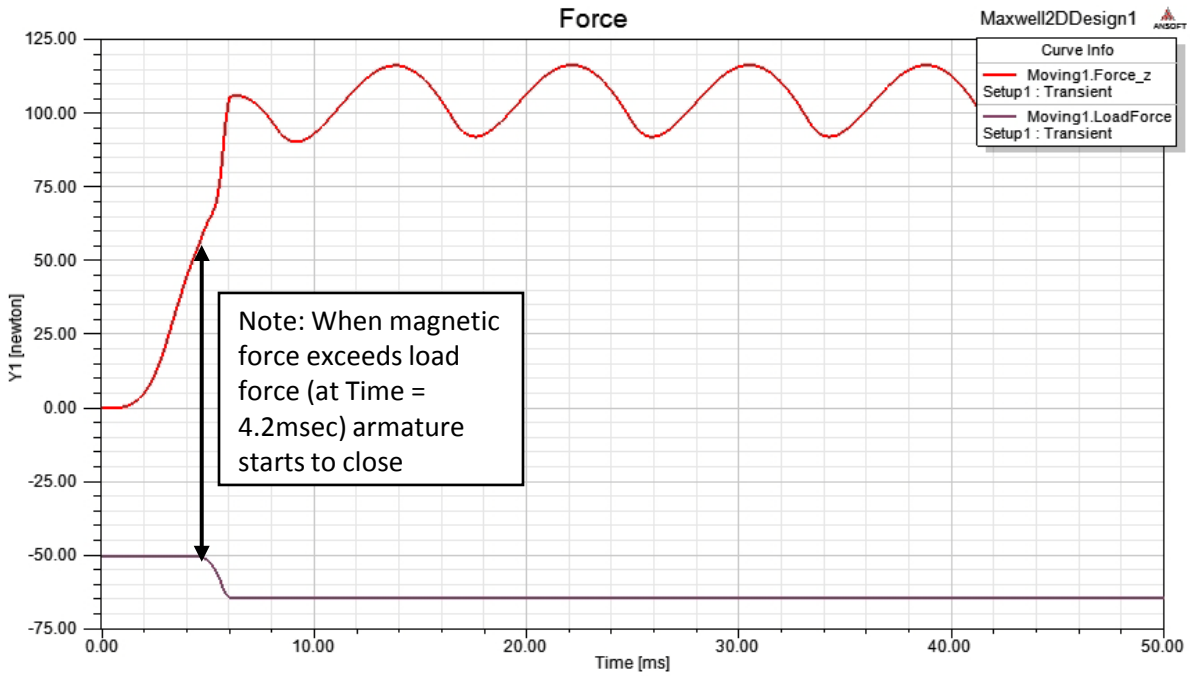
- **To start the solution process:**
  - Select the menu item *Maxwell 2D > Analyze All*

Plot Results

- **To Plot Results Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Quick Report*
  - In Quick Report window,
    1. Press **Ctrl** and select the quantities **Force, Loss, Position, Speed** and **Winding**

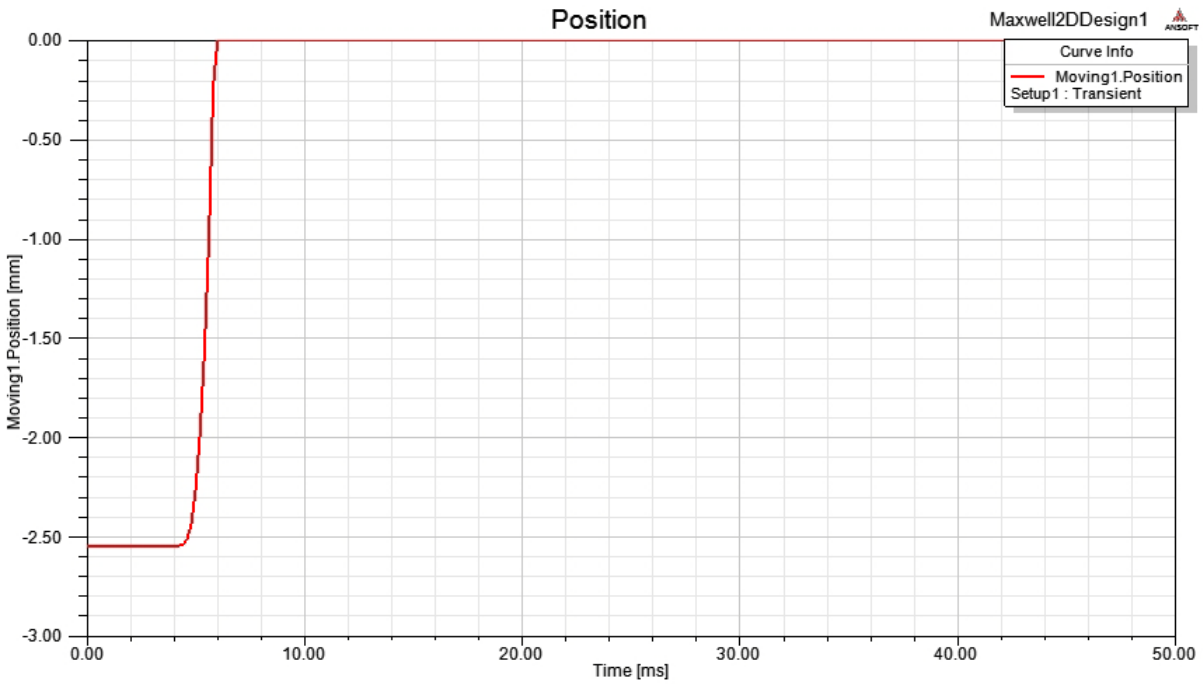


- **Force Vs Time**

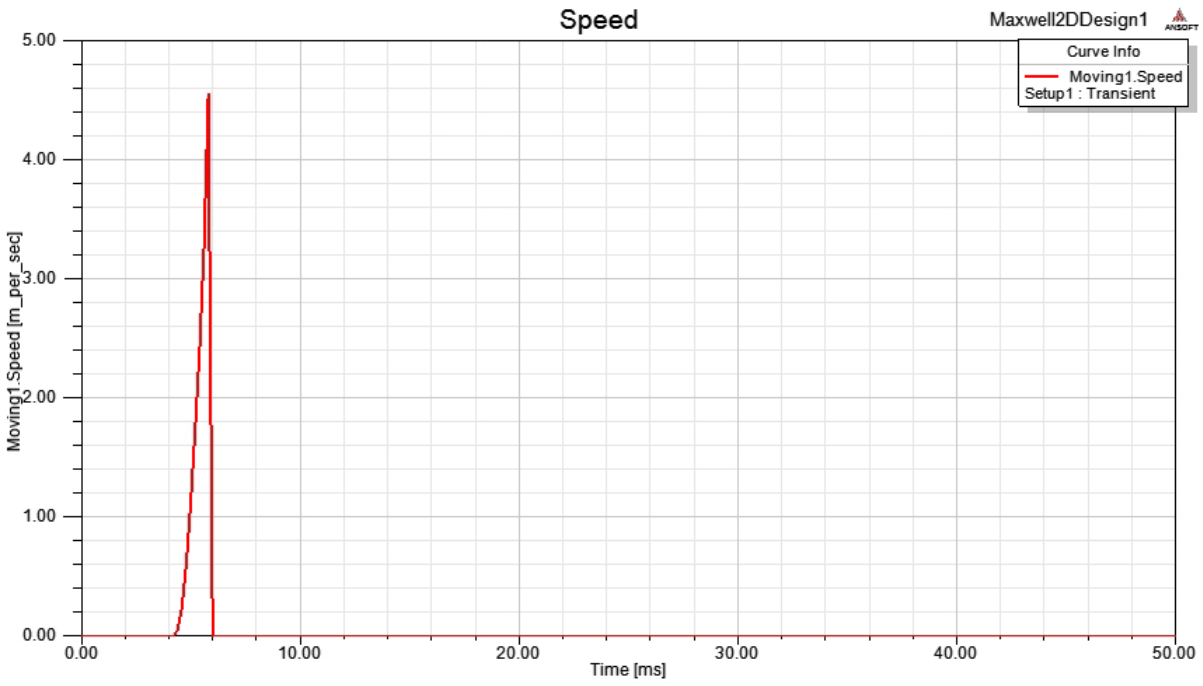


Transient - Solenoid with External Circuit

Position Vs Time

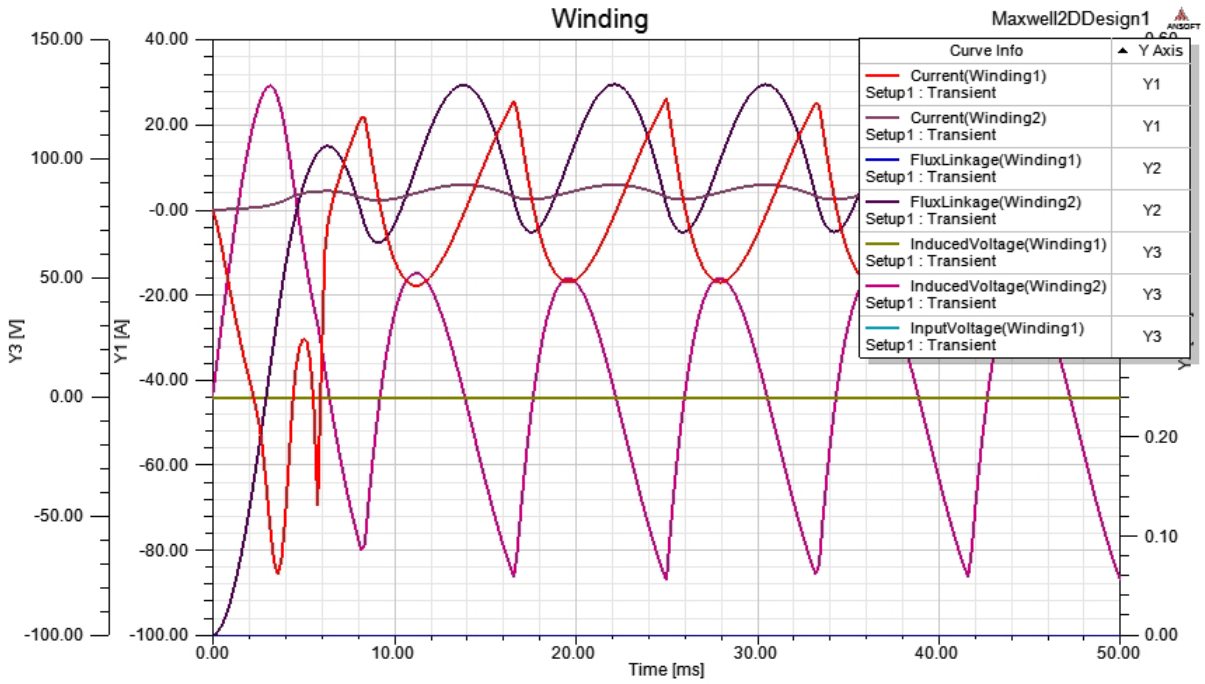


Speed Vs Time

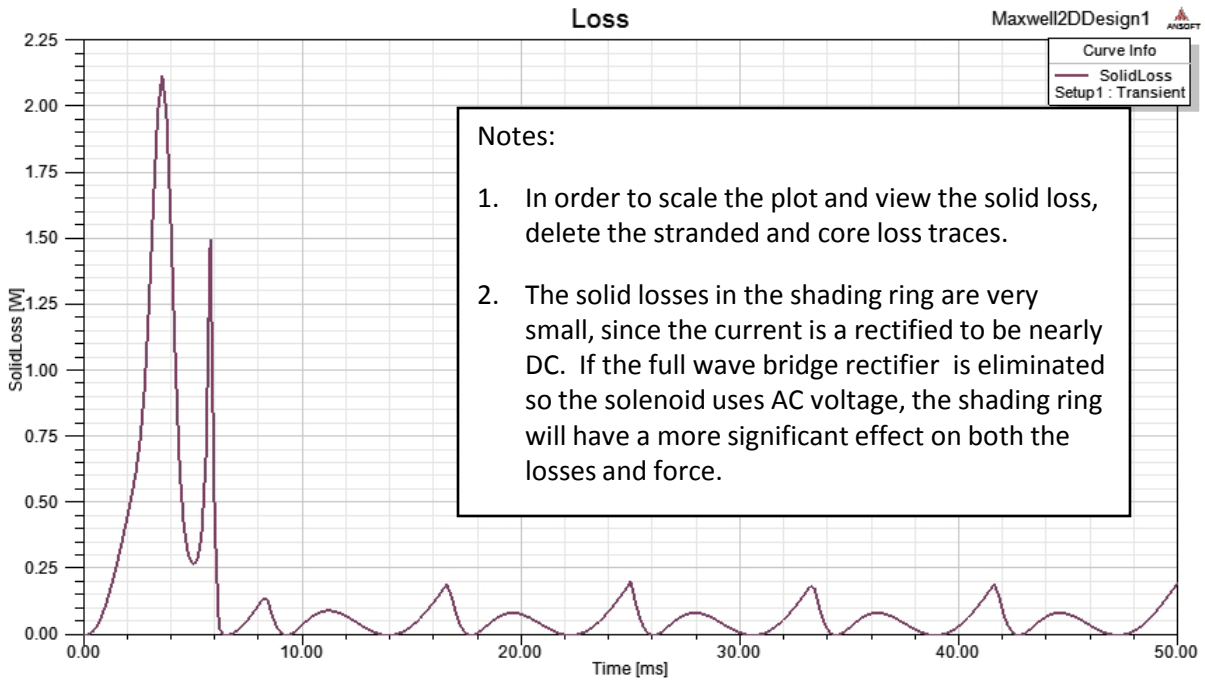


Transient - Solenoid with External Circuit

Winding Vs Time



Loss Vs Time



Notes:

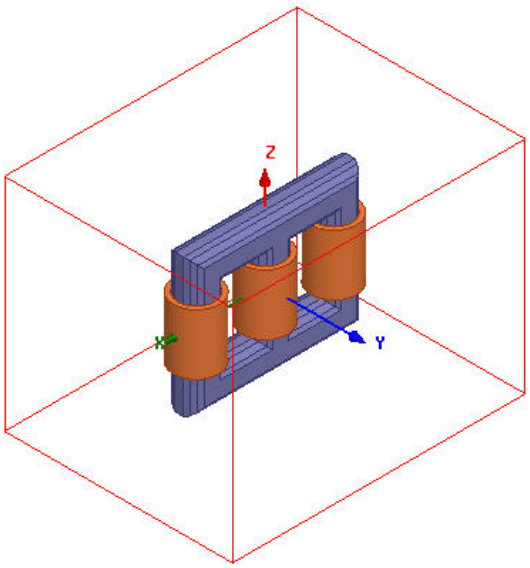
1. In order to scale the plot and view the solid loss, delete the stranded and core loss traces.
2. The solid losses in the shading ring are very small, since the current is a rectified to be nearly DC. If the full wave bridge rectifier is eliminated so the solenoid uses AC voltage, the shading ring will have a more significant effect on both the losses and force.

Example (2D/3D Transient) - Core Loss

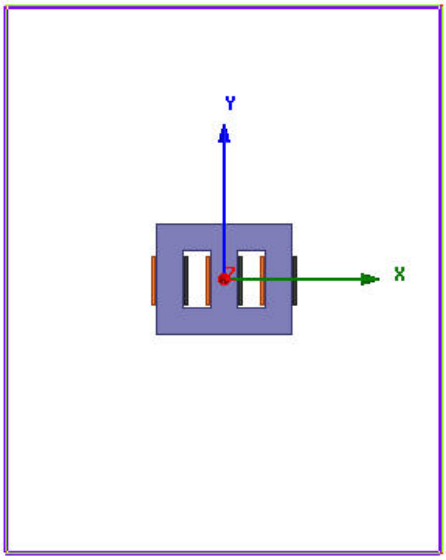
Transformer Core Loss Calculation in Maxwell 2D and 3D

- 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.

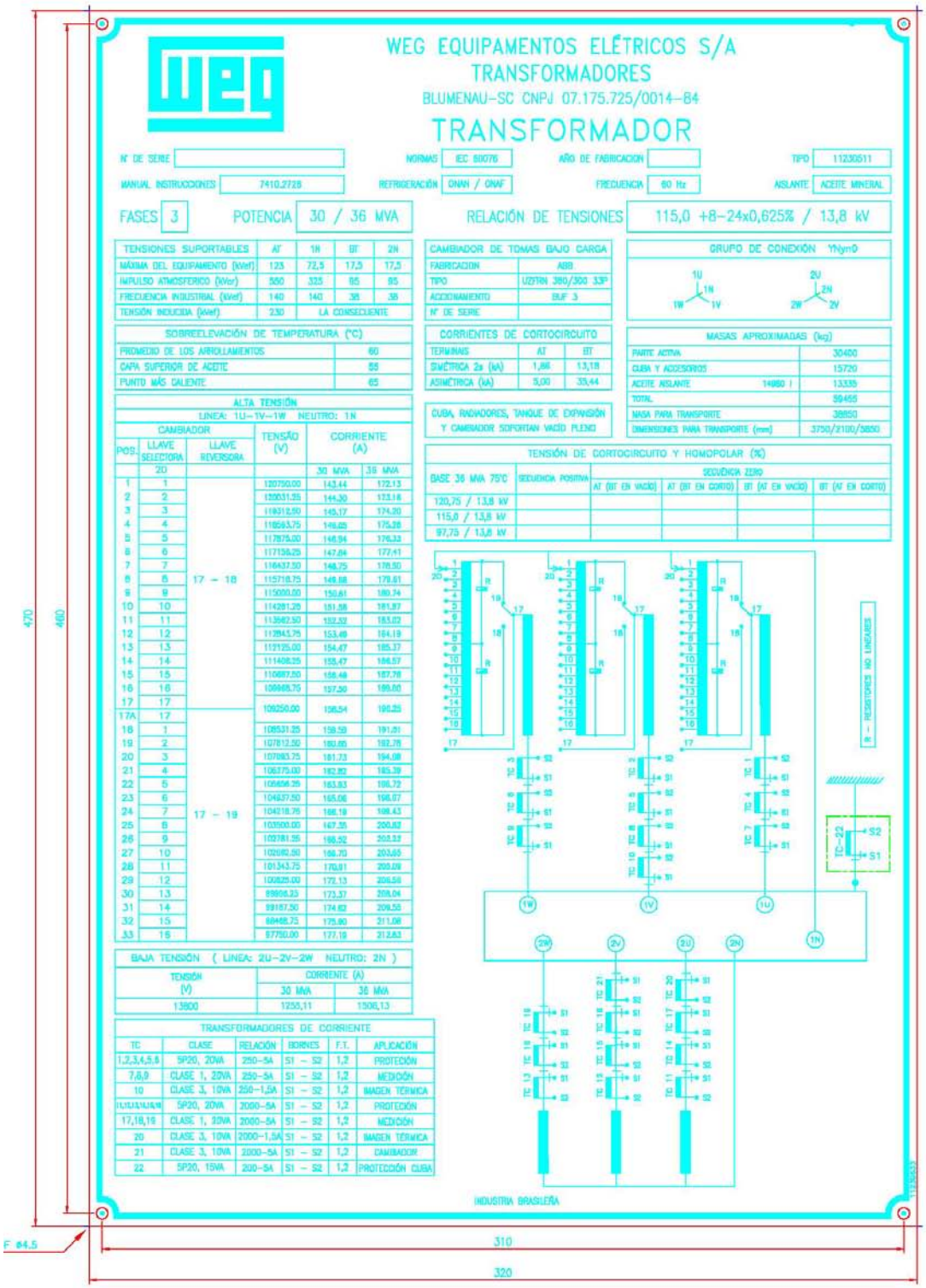
3D Model



2D Model



Example (2D/3D Transient) - Core Loss

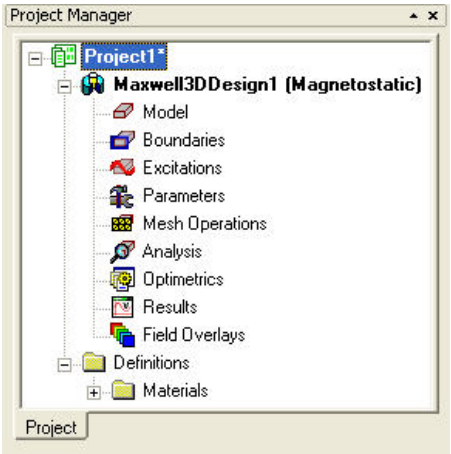


Launch Maxwell

- To access Maxwell
  - Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and then **Maxwell 15.**

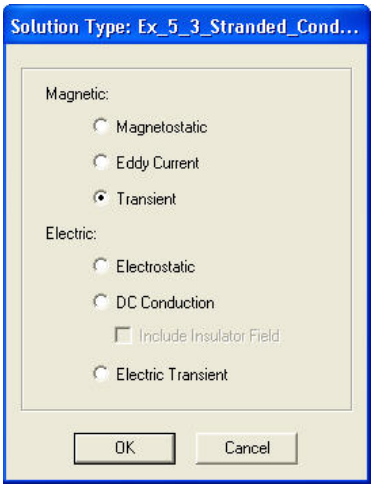
Opening a New Project

- 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

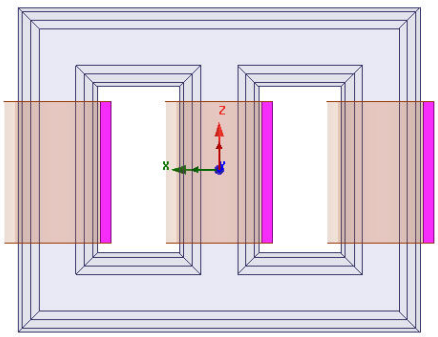


Prepare Geometry

- **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**

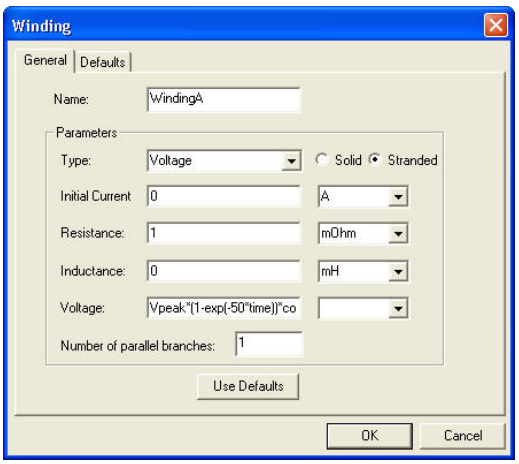
Specify Excitations

- **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**

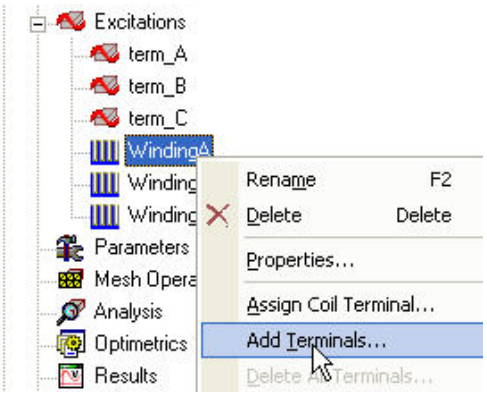


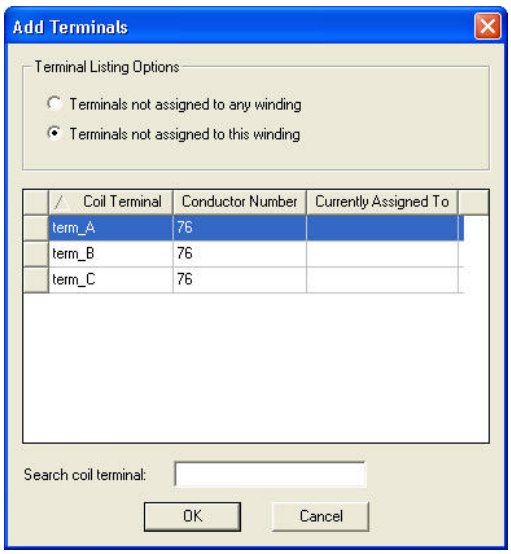
- **Note:** This is an exponentially increasing (in several cycles) sinusoidal 60Hz waveform with peak magnitude of 11,268V..

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))**

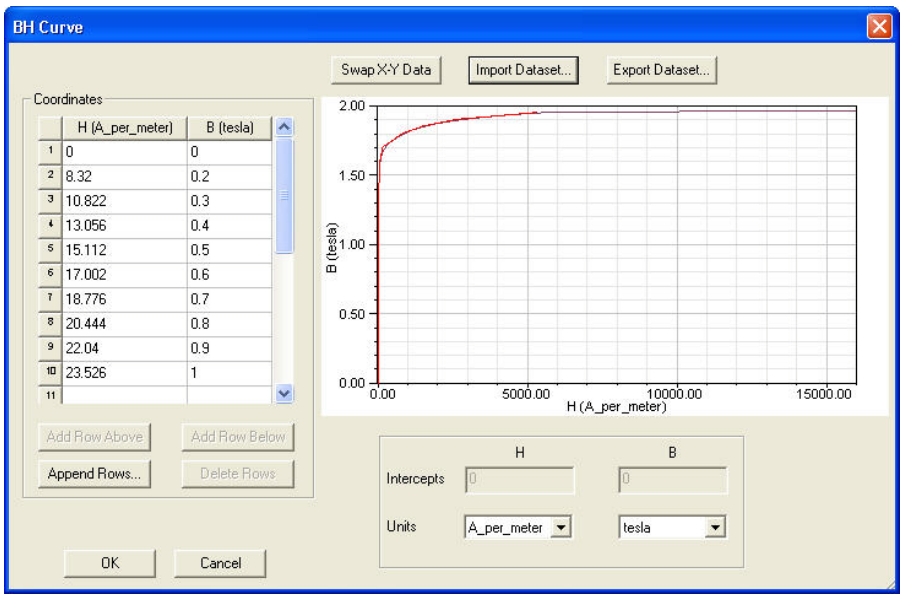
- **Add Terminals to Windings**
  - Expand the Project Manager tree to view Excitations
  - Right click on **WindingA** and select **Add Terminals**
  - In Add Terminals window,
    - Select **term_A**
    - Press **OK**
  - In Similar way add **term_B** to **WindingB**
  - Add **term_C** to **WindingC**





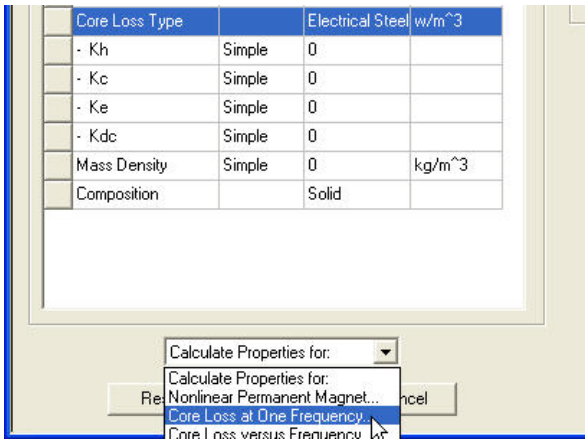
Assign Materials

- **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

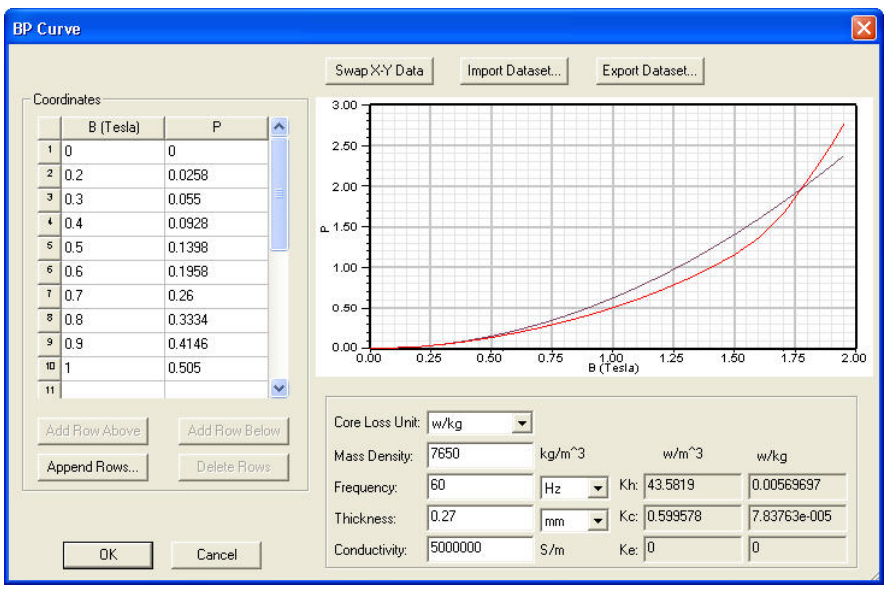


Example (2D/3D Transient) - Core Loss

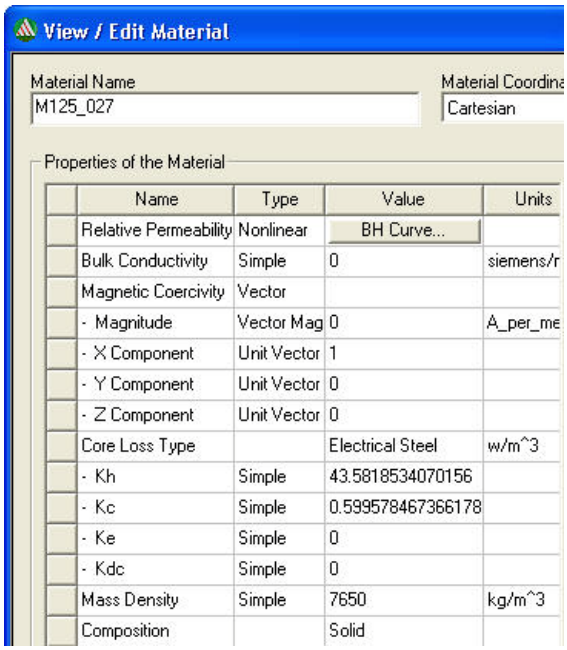
- Return to View/Edit Material window,
  - Core Loss Type: Set to **Electrical Steel**
  - Set the tab at the bottom of window Calculate Properties for to **Core Loss at one Frequency**



  - 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



- 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

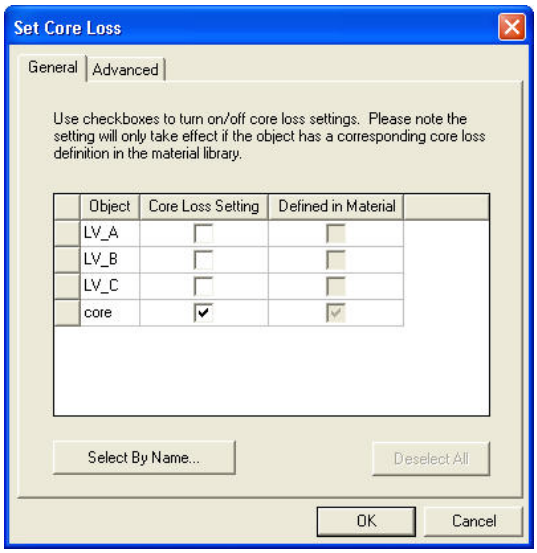
Assign Mesh Operations

- 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

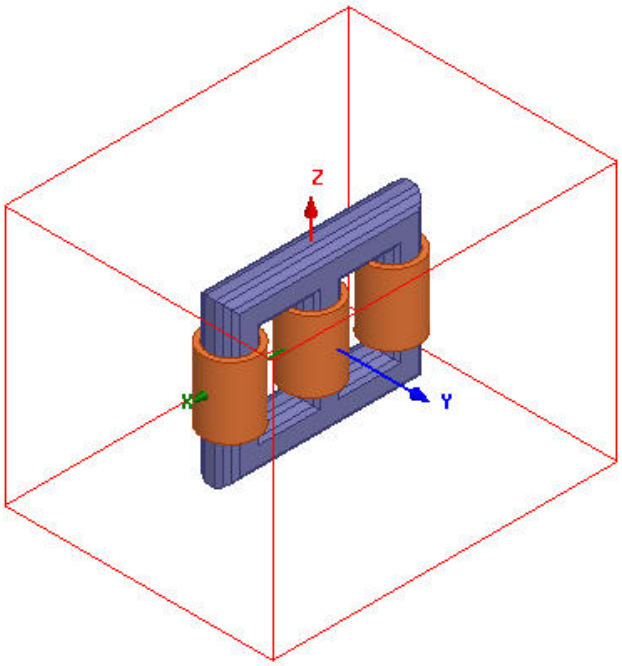


Set Eddy Effects

- 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

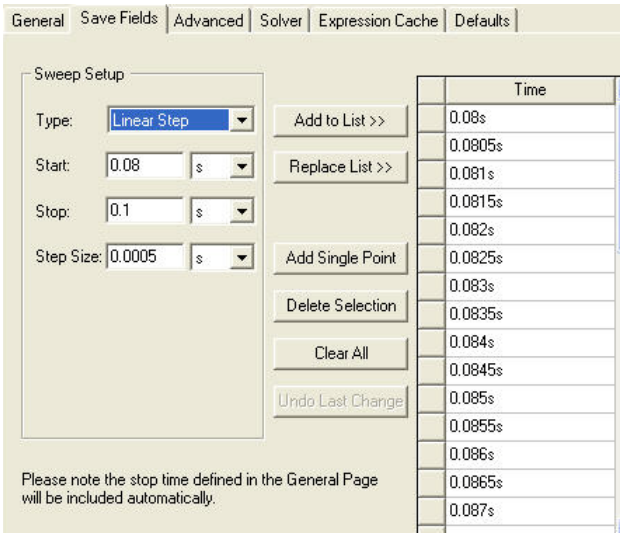
- **To Create Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    - Padding Data: **Pad individual directions**
      - +/- X = **30**
      - +/- Y = **200**
      - +/- Z = **30**
  - **Note:** This small padding % is acceptable as fields are completely concentrated inside the magnetic core and there is little or no fringing



Example (2D/3D Transient) - Core Loss

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

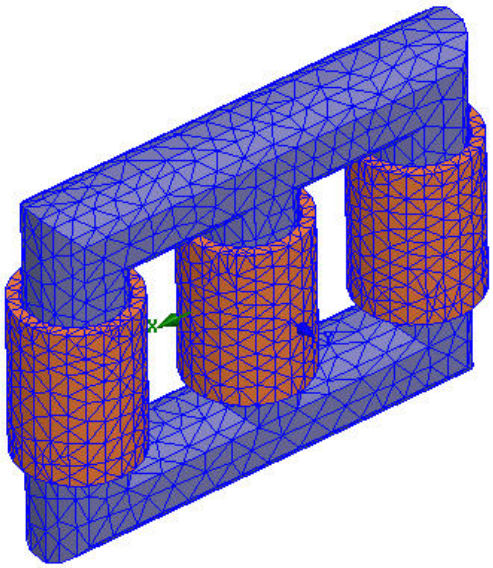
- **To Save File**
  - Select the menu item *File > Save*
  - Save the file with the name **"Ex_7_4_Core_Loss.mxwl"**

Analyze

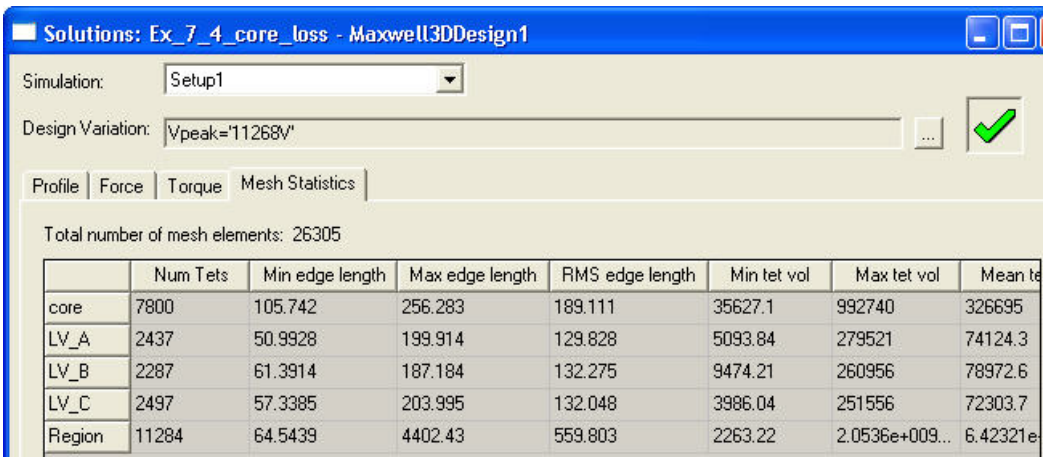
- **To Run Solution**
  - Select the menu item *Maxwell 3D > Analyze All*

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

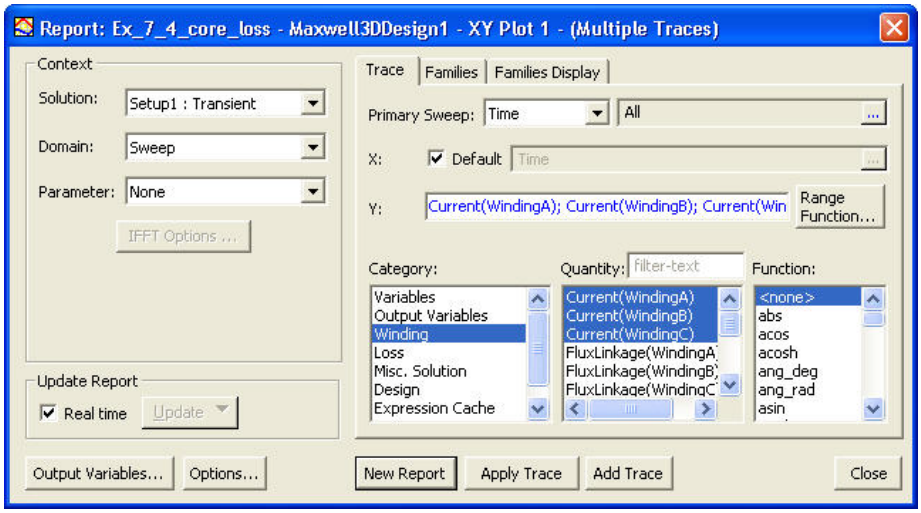


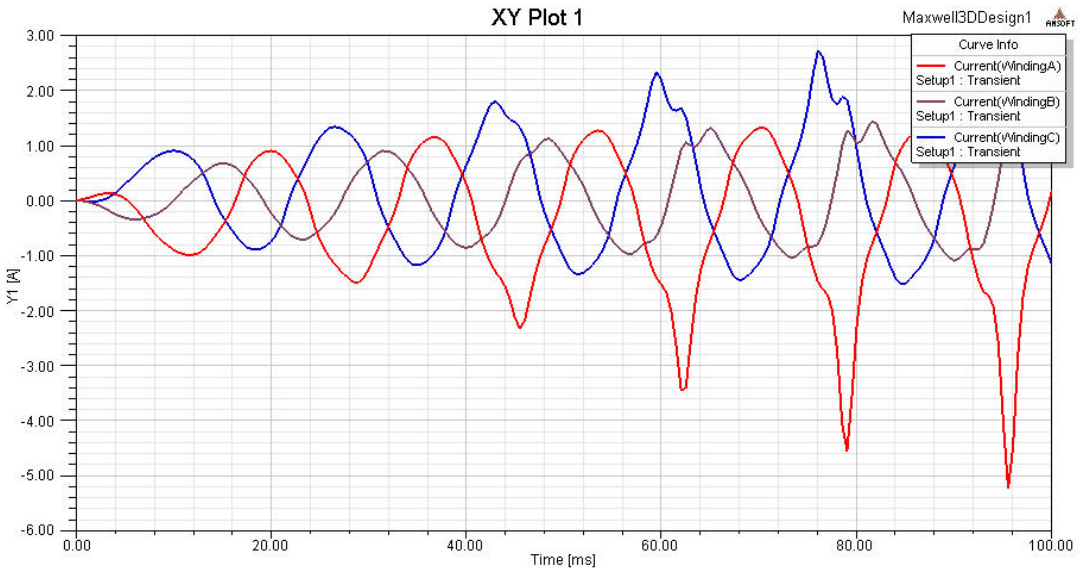
	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean te
core	7800	105.742	256.283	189.111	35627.1	992740	326695
LV_A	2437	50.9928	199.914	129.828	5093.84	279521	74124.3
LV_B	2287	61.3914	187.184	132.275	9474.21	260956	78972.6
LV_C	2497	57.3385	203.995	132.048	3986.04	251556	72303.7
Region	11284	64.5439	4402.43	559.803	2263.22	2.0536e+009...	6.42321e-

Example (2D/3D Transient) - Core Loss

Create Reports

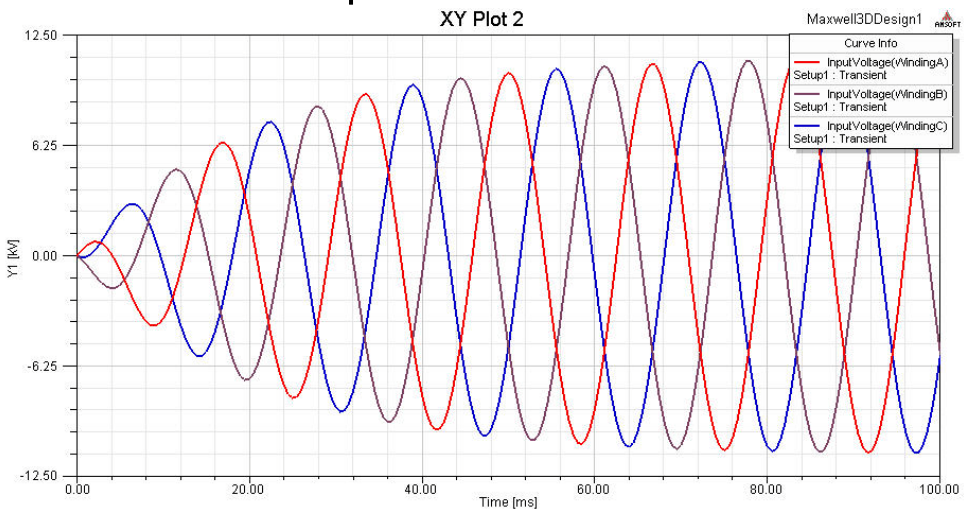
- **Plot Winding Currents Vs Time**
  - Select the menu item *Maxwell 3D > Results > Create Transient Report > Rectangular Plot*
  - In Report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **Current(WindingA)**, **Current(WindingB)** and **Current(WindingC)**
    - Select **New Report**
  - **Note:** Do not close Report window as we will create more plots using same window



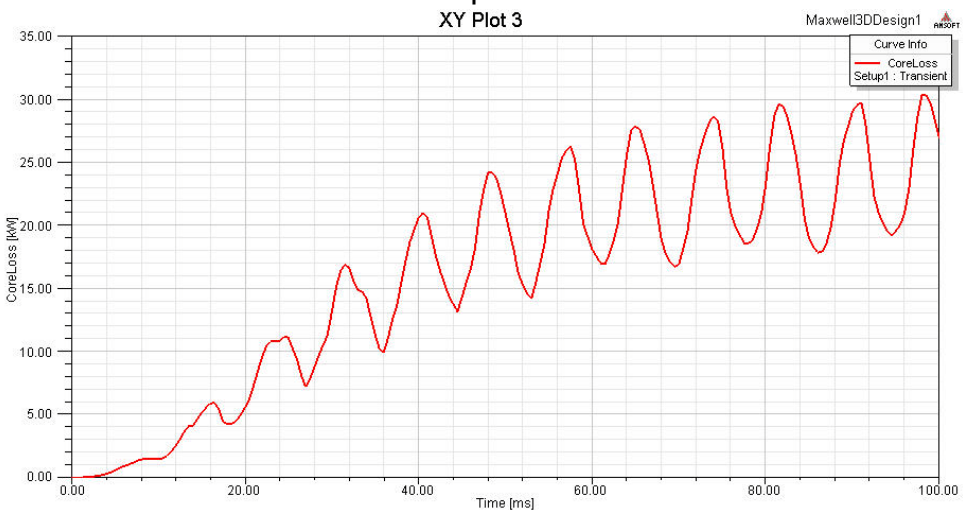


Example (2D/3D Transient) - Core Loss

- **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**

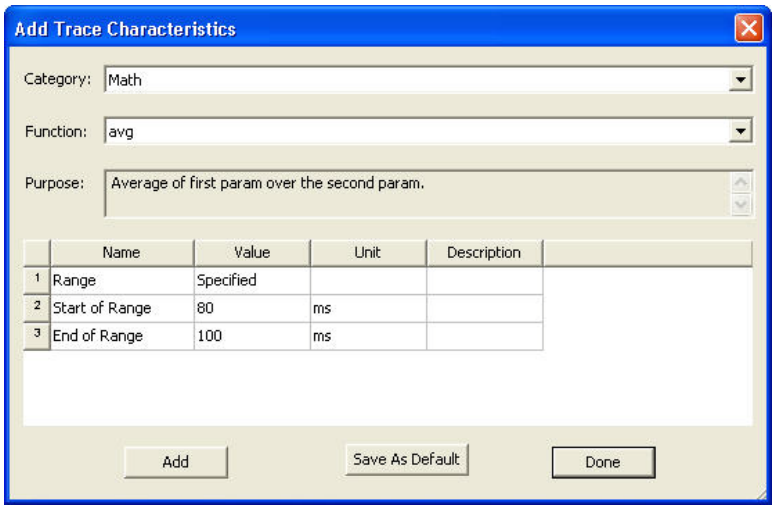


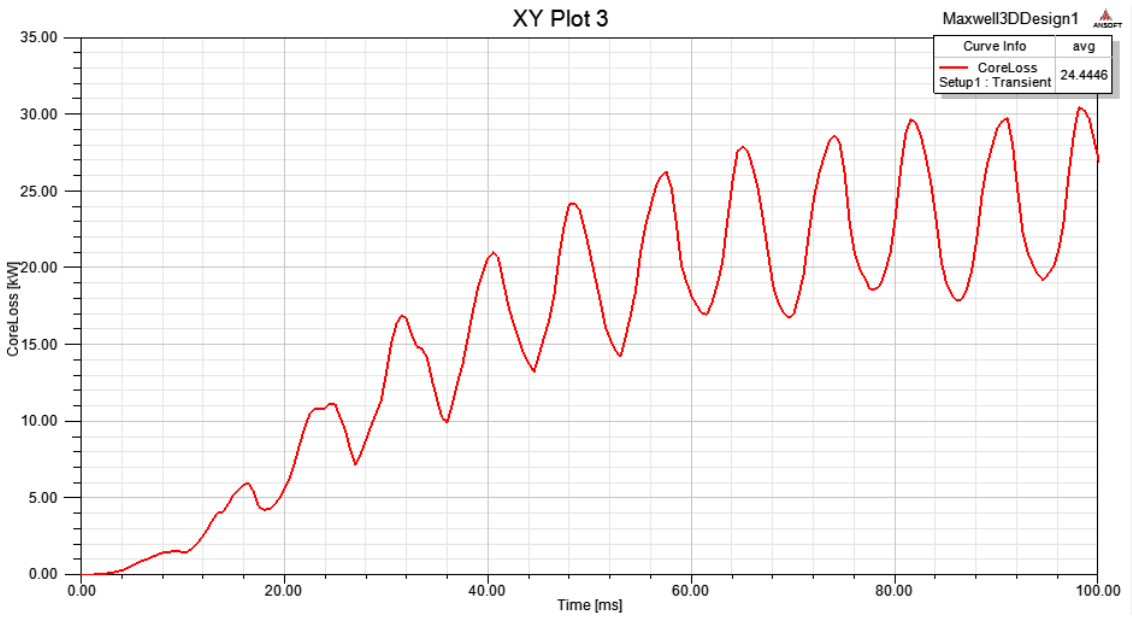
- **Plot Cores Loss vs Time**
  - In Report window,
    - Category: Change to **Loss**
    - Quantity: Select **CoreLoss**
    - Select **New Report**
    - Press **Close** to close report window



Example (2D/3D Transient) - Core Loss

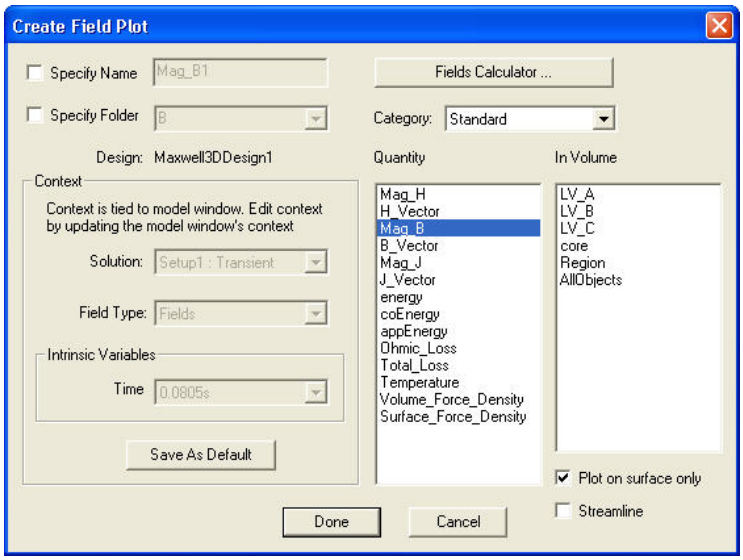
- **Calculate Avg Losses over a Time Range**
  - In XY Plot Corresponding to CoreLoss, right click on the Legend and select *Trace Characteristics > Add*
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **Avg**
    - Change the Range from Full to **Specified**
    - Start of Range: **80 ms**
    - End of Range: **100 ms**
    - Select **Add** and **Done**

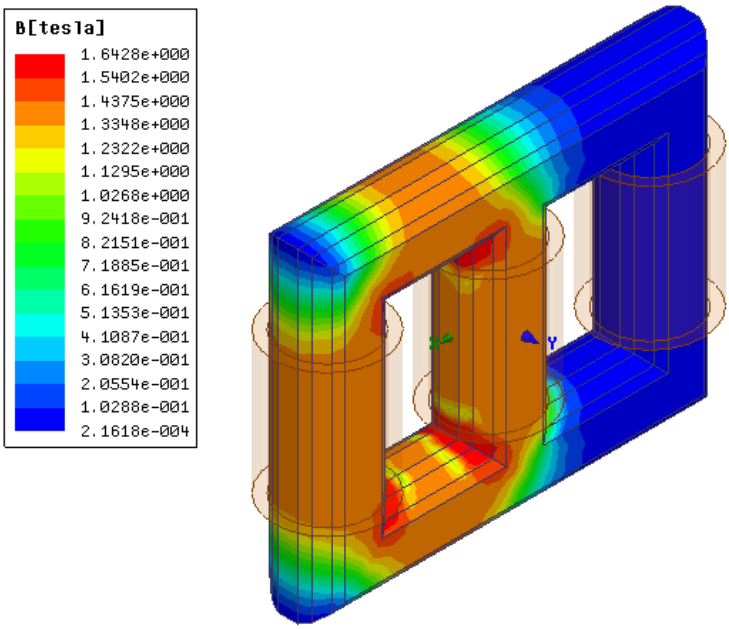




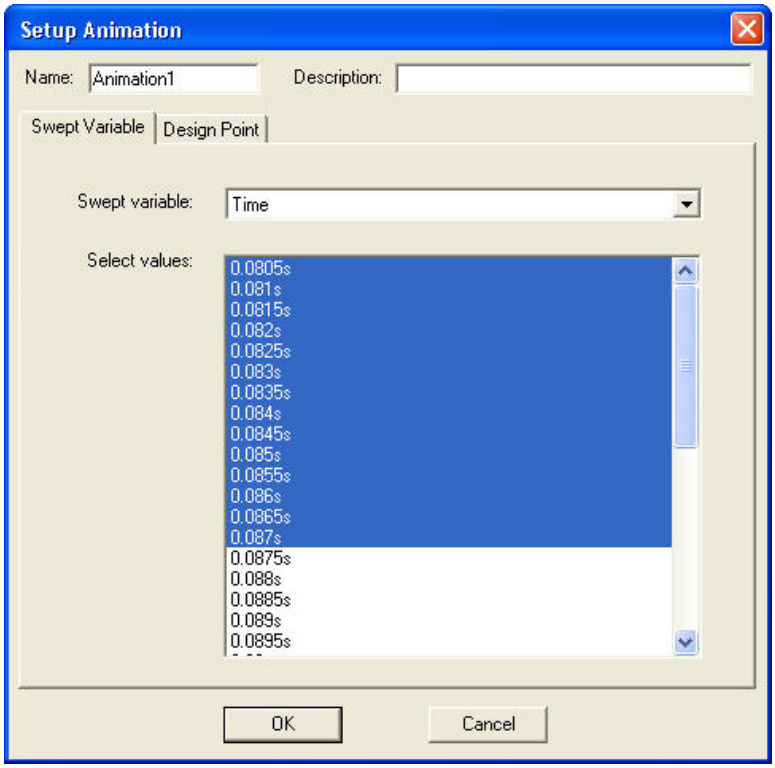
Create Flux Density Plot

- **To Plot Flux Density on Core**
  - Double click on **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**

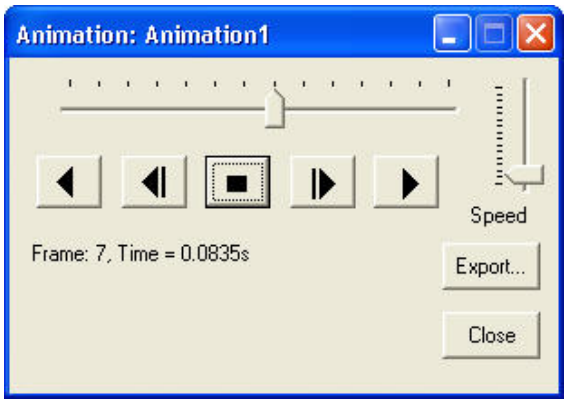




- **To Animate the Plot**
  - Select the menu item *Maxwell 3D > Fields > Animate*
  - In Setup Animation window,
    - Sweep Variable: **Time**
    - Select values: Select the time range from **0.0805s** to **0.087s**
    - Press **OK**



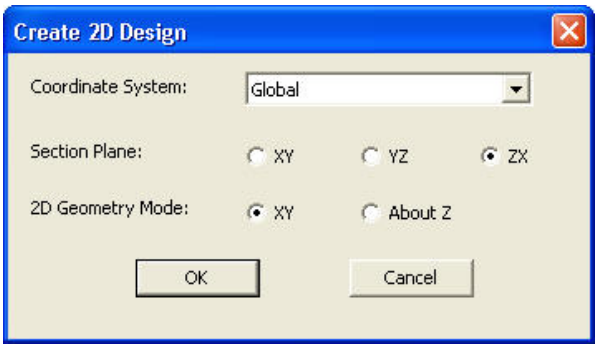
  - An Animation window will pop up which will enable to start, stop, pause the animation. Animation speed can also be varied using same window. The animation can be also exported in GIF or AVI format using Export button



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

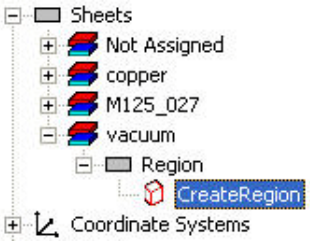
- Set the depth of the 2D XY model to give the same area as in Maxwell 3D = 264704 mm^2. Since the width of the core leg = 580mm, set the depth = 456mm.
- **To Set Model Depth**
  - Select the menu item *Maxwell 2D > Model > Set Model Depth*
  - In Design Settings window,
    - Set Model Depth to **456 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**



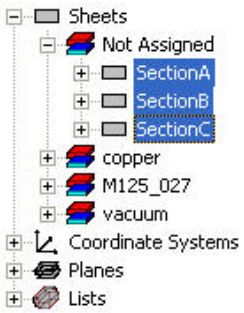
Properties: Ex_7_4_core_loss - Maxwell2DDesign1 - Modeler

Command

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√2 / √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**

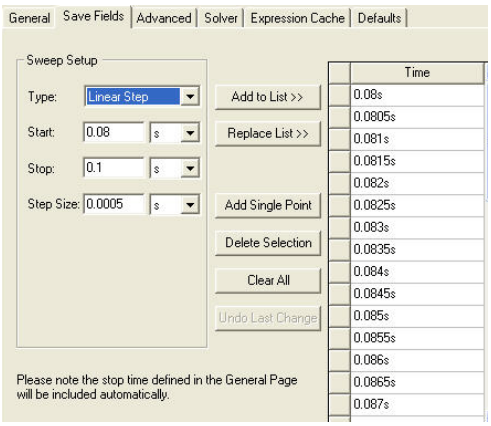
Example (2D/3D Transient) - Core Loss

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

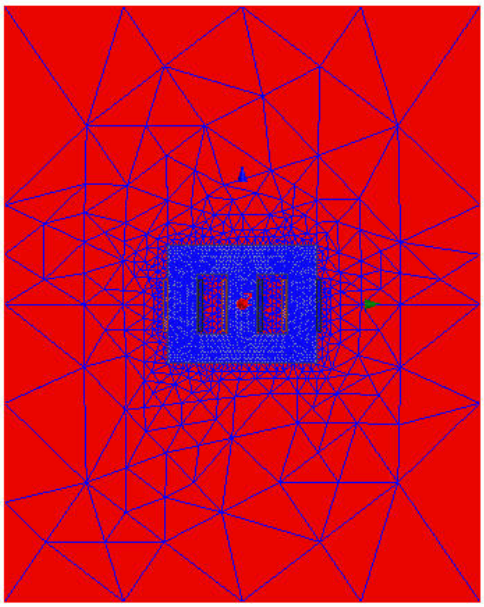
- **To Save File**
  - Select the menu item *File > Save*

Analyze

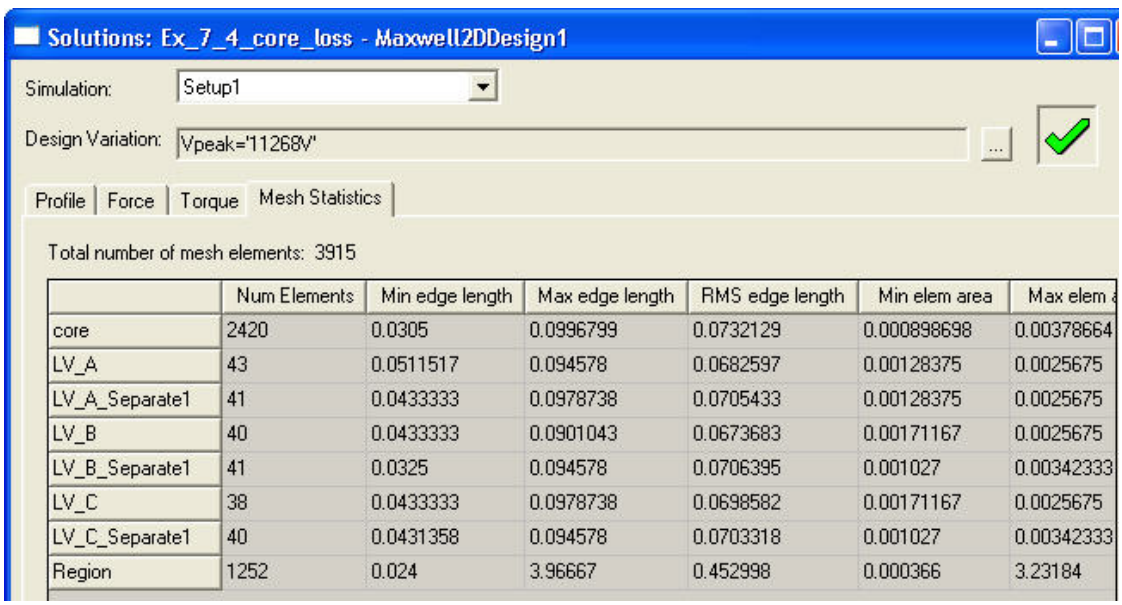
- **To Run Solution**
  - Select the menu item *Maxwell 3D > Analyze All*

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**



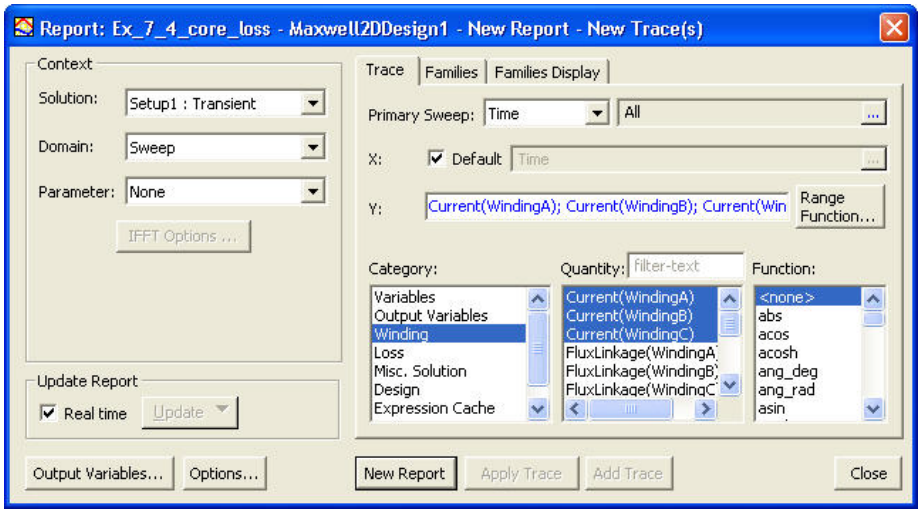
- **To View Mesh Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
  - In Solutions window,
    - Select the tab **Mesh Statistics** to view mesh information

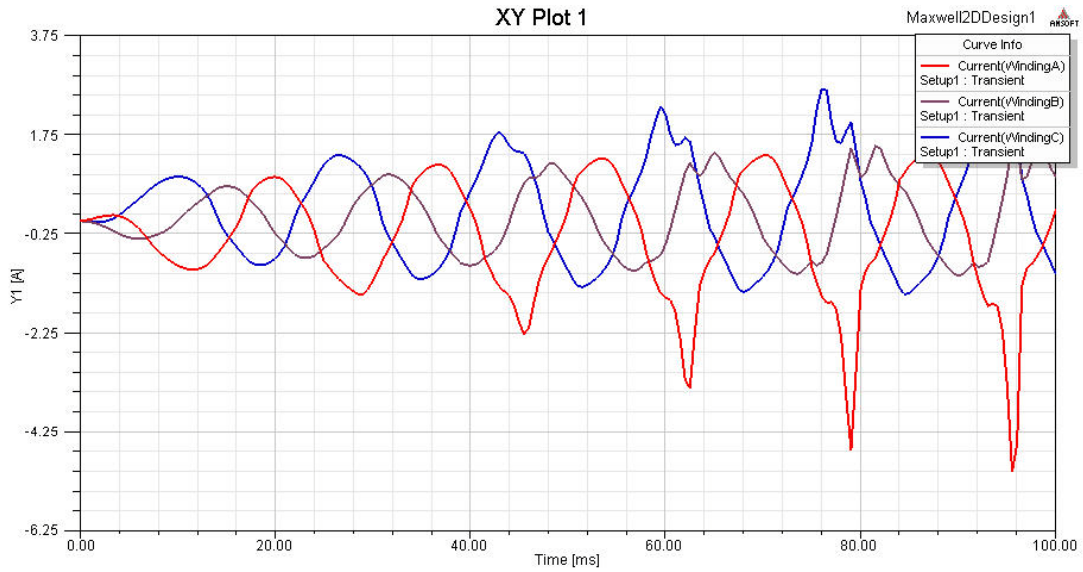


	Num Elements	Min edge length	Max edge length	RMS edge length	Min elem area	Max elem
core	2420	0.0305	0.0996799	0.0732129	0.000898698	0.00378664
LV_A	43	0.0511517	0.094578	0.0682597	0.00128375	0.0025675
LV_A_Separate1	41	0.0433333	0.0978738	0.0705433	0.00128375	0.0025675
LV_B	40	0.0433333	0.0901043	0.0673683	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

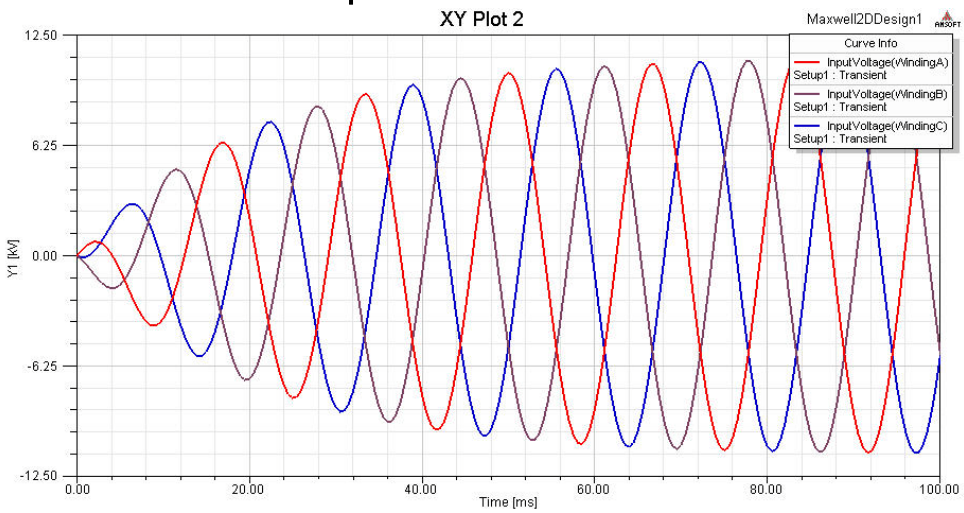
- **Plot Winding Currents Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In Report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **Current(WindingA)**, **Current(WindingB)** and **Current(WindingC)**
    - Select **New Report**
  - **Note:** Do not close Report window as we will create more plots using same window



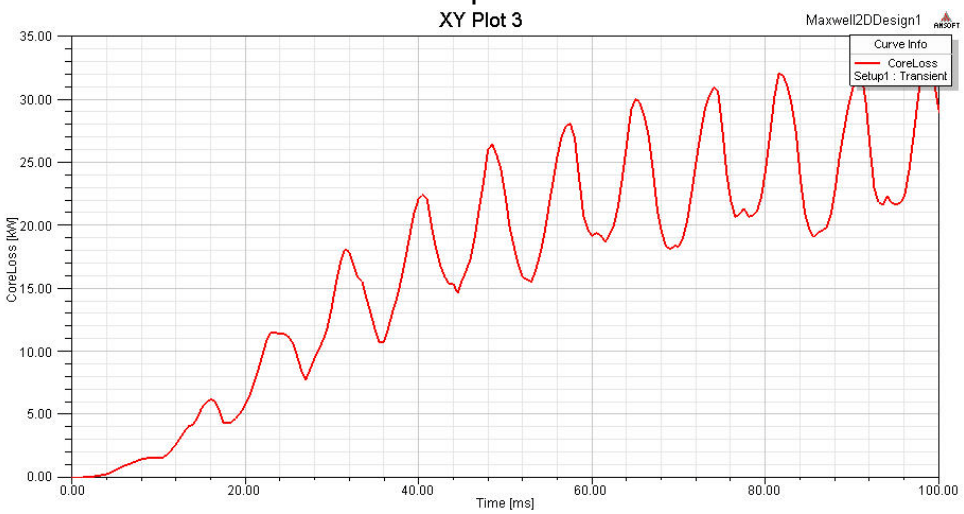


Example (2D/3D Transient) - Core Loss

- **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**

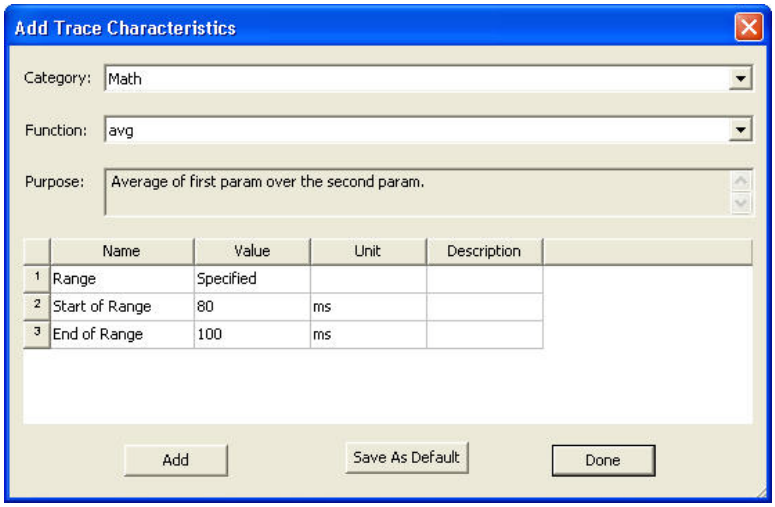


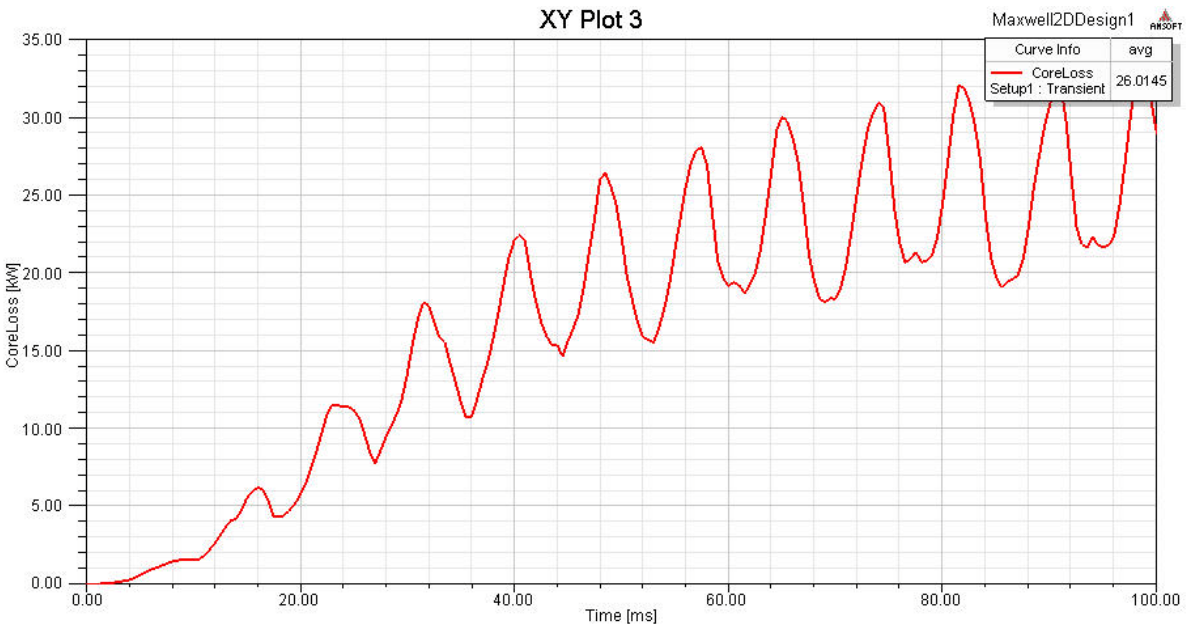
- **Plot Cores Loss vs Time**
  - In Report window,
    - Category: Change to **Loss**
    - Quantity: Select **CoreLoss**
    - Select **New Report**
    - Press **Close** to close report window



Example (2D/3D Transient) - Core Loss

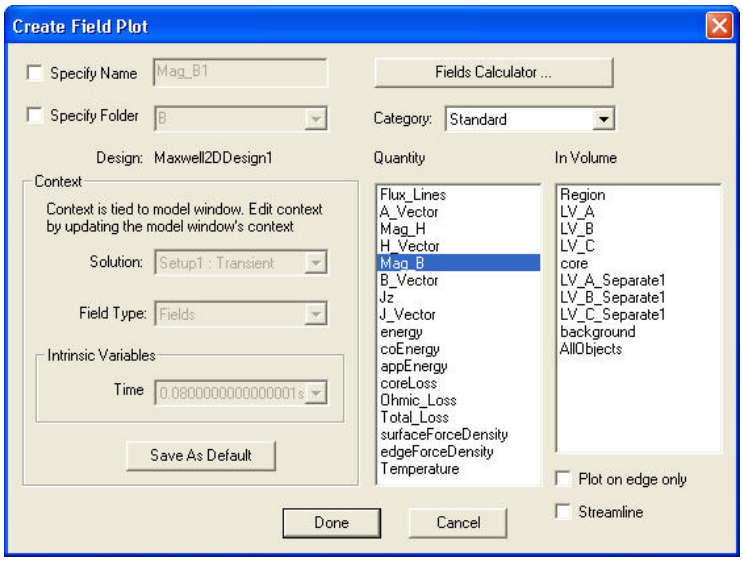
- **Calculate Avg Losses over a Time Range**
  - In XY Plot Corresponding to CoreLoss, right click on the Legend and select *Trace Characteristics > Add*
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **Avg**
    - Change the Range from Full to **Specified**
    - Start of Range: **80 ms**
    - End of Range: **100 ms**
    - Select **Add** and **Done**

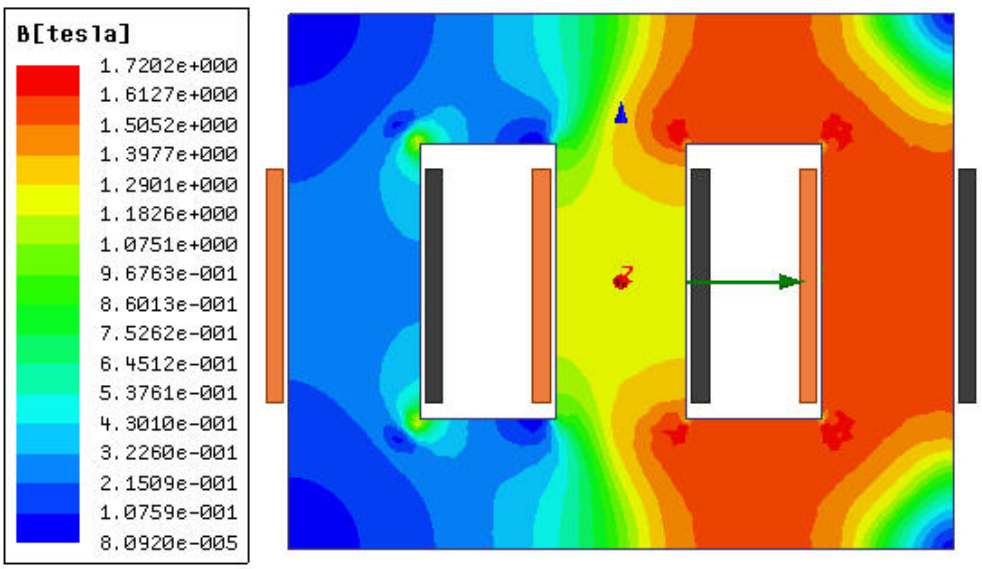




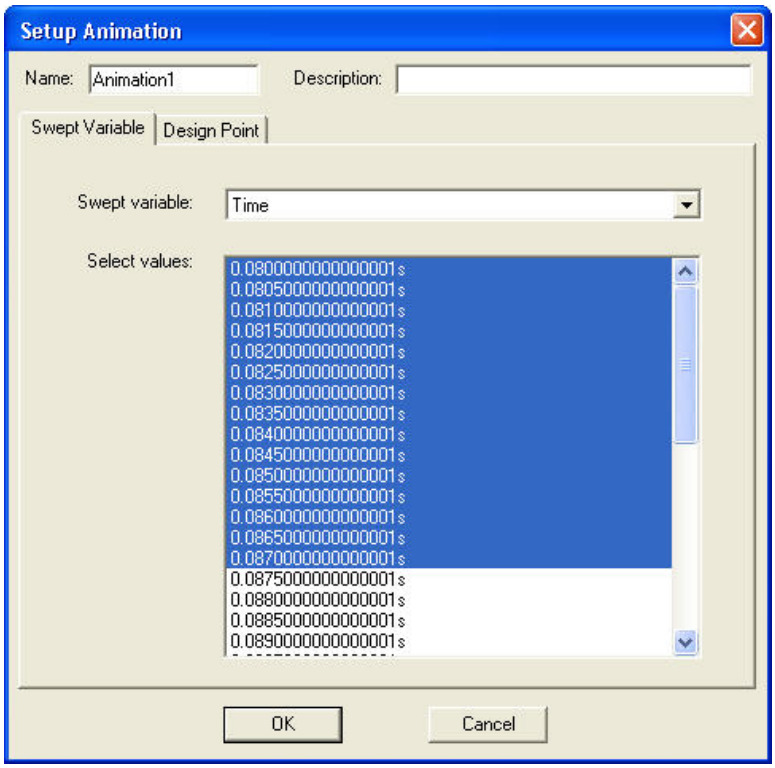
Create Flux Density Plot

- **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**

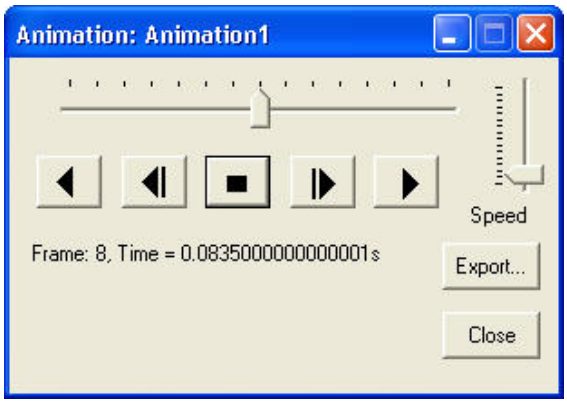




- **To Animate the Plot**
  - Select the menu item *Maxwell 2D > Fields > Animate*
  - In Setup Animation window,
    - Sweep Variable: **Time**
    - Select values: Select the time range from **0.0805s** to **0.087s**
    - Press **OK**



  - 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 9.0 - Basic Exercises

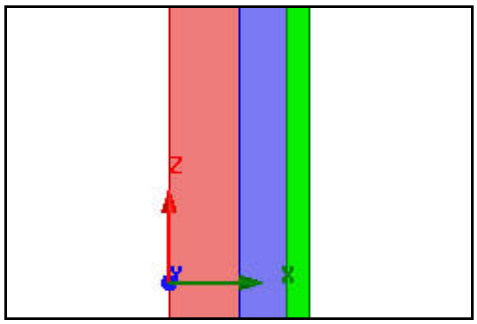
- 9.1 - Electrostatic
- 9.2 - DC Conduction
- 9.3 - Magnetostatic
- 9.4 - Parametric
- 9.5 - Transient
- 9.6 - Transient with Circuit Editor
- 9.7 - Post-processing
- 9.8 - Optimetrics
- 9.10 - Scripting
- 9.12 - Eddy Current
- 9.13 - Rotational Transient Motion
- 9.14 - Boundary Conditions
- 9.15 - Permanent Magnets Assignment

Introduction on the Electrostatic Solver

- This note introduces the Electro Static solver based on some simple examples. This solver is meant to solve the static electric field without current flowing in conductors (conductors are in electrostatic equilibrium). The conductors are considered perfect such that there is no electric field inside conductors.

Capacitance of a Cylindrical Capacitor in RZ

- 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. We use an R-Z representation. We will then solve the same problem using an XY representation.



Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Electric > Electrostatic**
    3. Click the **OK** button

Create Model

- **Create object Inner**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.6**, dY: **0**, dZ: **25**, Press the **Enter** key
  - Change the name of resulting sheet to **Inner** and color to **Light Red**
  - Change the material of the sheet to **Copper**
- **Create Air Gap**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0.6**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.4,** dY: **0**, dZ: **25**, Press the **Enter** key
  - Change the name of resulting sheet to **Air** and color to **Light Blue**
  - Change the material of the sheet to **Air**
- **Create object outer**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **1**, Y: **0**, Z: **-4**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.2**, dY: **0**, dZ: **25**, Press the **Enter** key
  - Change the name of resulting sheet to **Outer** and color to **Light Green**
  - Change the material of the sheet to **Copper**

Define Region

- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad individual directions: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
       - +R = **300**
       - Specify rest to **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. In other words, we do not solve inside conductors, we assume that all the conductor parts are at the same potential.
- **To Assign Excitation to object Inner**
  - Select the sheet **Inner** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Name: **Voltage_Inner**
    - Value: **-1kV**
    - Press **OK**
- **To Assign Excitation to object Outer**
  - Select the sheet **Outer** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Name: **Voltage_Outer**
    - Value: **1kV**
    - Press **OK**

Assign Executive Parameters

- In addition to the fields, we are interested by the Capacitance value as well as the force applied to the inner armature.
- **To Assign Capacitance Computation**
  - Select the menu item *Maxwell2D > Parameters > Assign >Matrix*
  - In Matrix window
    1. Voltage_Inner and Voltage_Outer
       - Signal Line: ☑ **Checked**
    2. Press **OK**
- **To Assign Force Computation**
  - Select the sheet **Inner** from history tree
  - Select the menu item *Maxwell 2D > Parameters > Assign > Force*
  - In Force Setup window, press **OK**

Analysis Setup

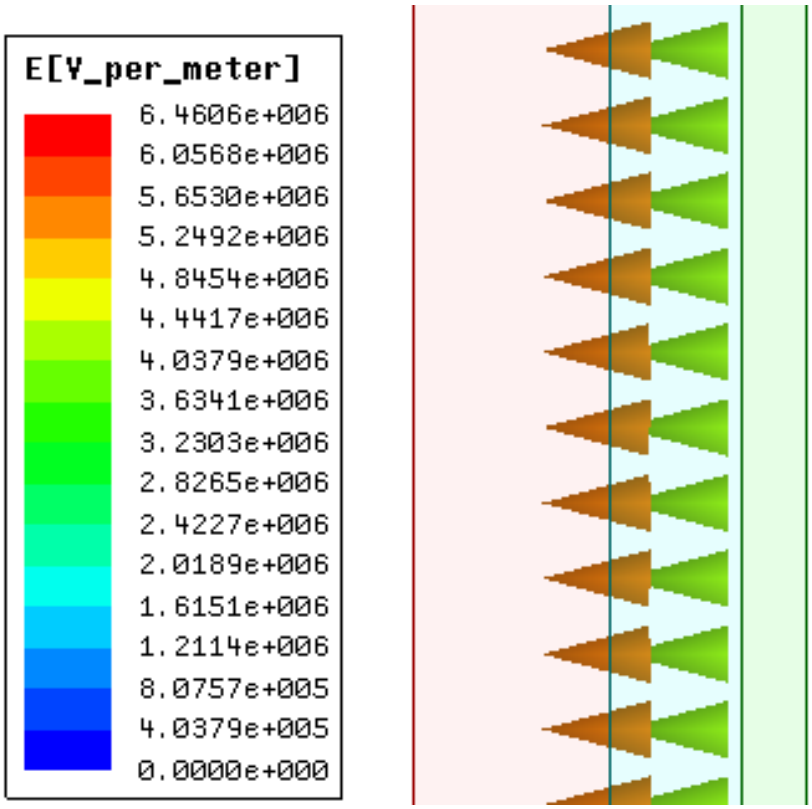
- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.5**
    2. **Convergence** Tab
       - Refinement Per Pass: **50%**
    3. Click the **OK** button

Analyze

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

Plot Electric Field Vectors

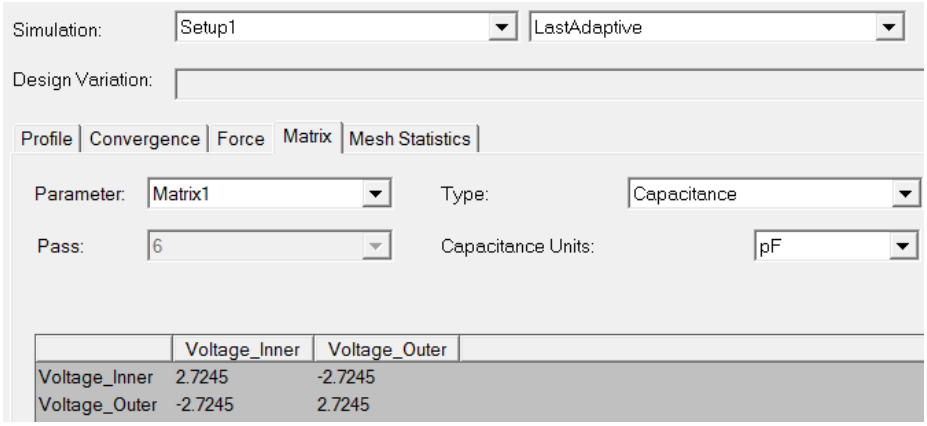
- **To Plot Electric Field Vectors**
  - Select the Plane **Global:XZ** from history treee
  - Select the menu item *Maxwell 2D > Fields > Fields > E > E_Vector*
  - In Create Field Plot window,
    - Press **Done**
  - To adjust spacing and size of arrows, double click on the legend and then go to **Marker/Arrow** and **Plots** tabs



View Results

To View Results

- Select the menu item *Maxwell 2D > Results > Solution Data*
- In Solutions window
  - Select **Matrix** tab
    - In our problem, we only have two conductors, therefore the capacitance values are symmetrical.
  - Select **Force** tab
    - **Note:** The force is essentially zero since the model is magnetically balanced.



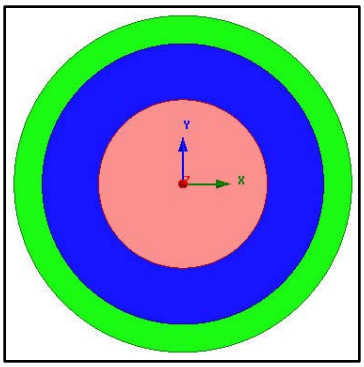
- The analytical value of the capacitance per meter for an infinite long coaxial wire is given by the following formula:

  $C = 2\pi\varepsilon_0 / \ln(b/a)$ (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. **2.723pF.** We obtain a good agreement with the obtained result. **2.745 pF**.
- **Note**: in the Convergence tab, you have access to the total energy of the system. We find **5.4489e-6** J. It is exactly 2000 times the capacitance (2000V being the difference of potential).

Capacitance of a Cylindrical Capacitor in XY

- The same problem is now solved using an XY representation



Create New Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cartesian, XY**
    2. Choose **Electric > Electrostatic**
    3. Click the **OK** button

Create Model

- **Create Object Inner**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **0.6**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Change the name of resulting sheet to **Inner** and color to **Light Red**
  - Change the material of the sheet to **Copper**

- **Create Object Outer**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **1.2**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Change the name of resulting sheet to **Outer** and color to **Light Green**
  - Change the material of the sheet to **Copper**
- **Create Air Gap**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **1.0**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Change the name of resulting sheet to **Air** and color to **Light Blue**
  - Change the material of the sheet to **Air**

Assign Excitations

- Based on the assumptions that the conductors are in electrostatic equilibrium, we assign voltage potential on the object itself. In other words, we do not solve inside conductors, we assume that all the conductor parts are at the same potential.
- **To Assign Excitation to object Inner**
  - Select the sheet **Inner** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Name: **Voltage_Inner**
    - Value: **-1kV**
    - Press **OK**

- **To Assign Excitation to object Outer**
  - Select the sheet **Outer** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Name: **Voltage_Outer**
    - Value: **1kV**
    - Press **OK**

Assign Executive Parameters

- In addition to the fields, we are interested by the Capacitance value
- **To Assign Capacitance Computation**
  - Select the menu item *Maxwell2D > Parameters > Assign >Matrix*
  - In Matrix window
    1. Voltage_Inner
       - Signal Line: ☑ **Checked**
    2. Voltage_Outer
       - Ground: ☑ **Checked**
    3. Press **OK**
  - We ground **Voltage_Outer**. We will obtain just a 1 by 1 matrix.

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **0.5 %**
    2. **Convergence** Tab
       - Refinement Per Pass: **50%**
    3. Click the **OK** button

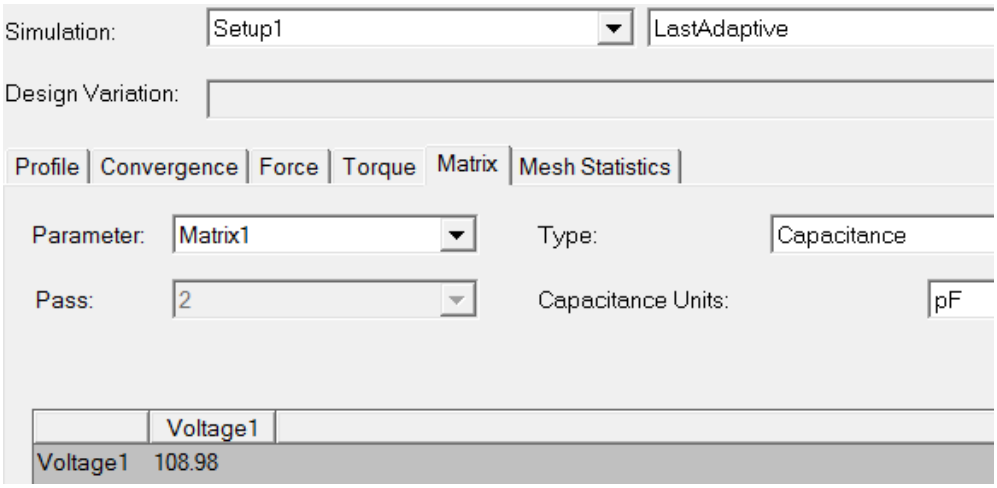
Analyze

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

View Results

To View Capacitance

- Select the menu item *Maxwell 2D > Results > Solution Data*
- In Solutions window
  - Select **Matrix** 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)
- This matches the obtained value.

Capacitance of a Planar Capacitor

- In this example we illustrate how to simulate a simple planar capacitor made of two parallel plates. The bottom plate is modeled and the top plate is considered by using only the edge of the dielectric (air).



Create New Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cartesian, XY**
    2. Choose **Electric > Electrostatic**
    3. Click the **OK** button

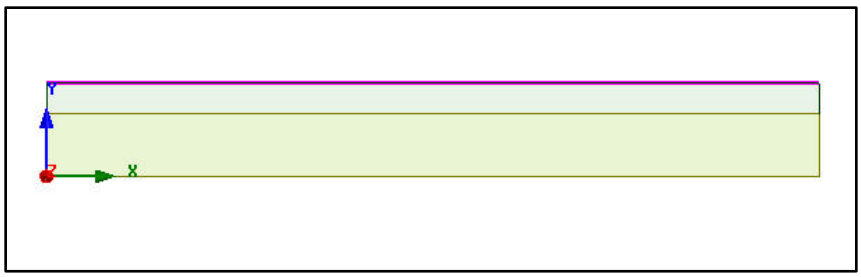
Create Model

- **Create Object DownPlate**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **25**, dY: **2**, dZ: **0**, Press the **Enter** key
  - Change the name of resulting sheet to **DownPlate** and color to **Yellow**
  - Change the material of the sheet to **Copper**

- **Create Object Region**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **25**, dY: **3**, dZ: **0**, Press the **Enter** key
  - Change the name of resulting sheet to **Region** and color to **Green**
  - Change the material of the sheet to **Air**

Assign Excitations

- **To Assign Excitation to object DownPlate**
  - Select the sheet **DownPlate** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Value: **0 V**
    - Press **OK**
- **To Assign Excitation to Region**
  - Select the menu item *Edit > Select > Edges*
  - Select the top edge of the **Region** as shown in below image
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    - Value: **1 V**
    - Press **OK**



Assign Executive Parameters

- In addition to the fields, we are interested by the Capacitance value as well as the force applied to the inner armature.
- **To Assign Capacitance Computation**
  - Select the menu item *Maxwell2D > Parameters > Assign >Matrix*
  - In Matrix window
    1. Voltage1
       - Signal Line: ☑ **Checked**
    2. Voltage2
       - Ground: ☑ **Checked**
    3. Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **1**
    2. **Convergence** Tab
       - Refinement Per Pass: **50%**
    3. Click the **OK** button

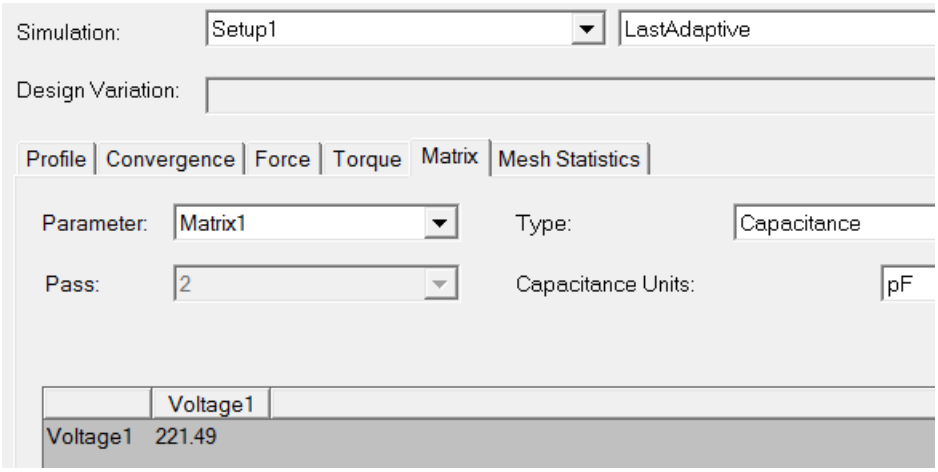
Analyze

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

View Results

To View Capacitance

- Select the menu item *Maxwell 2D > Results > Solution Data*
- In Solutions window
  - Select **Matrix** tab



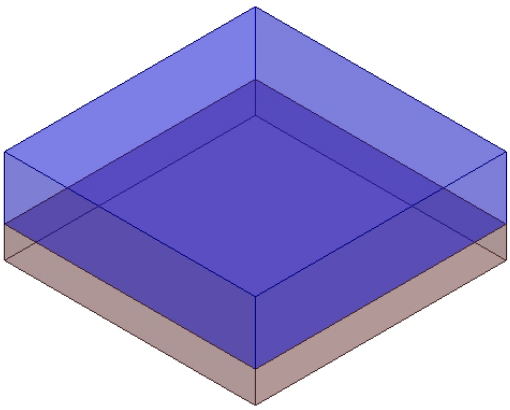
- The analytical value of the capacitance for two parallel plates is given by:
  $C = A / d * \varepsilon_0$ (A is the area of the plate and d is the thickness of the di electrics)
- If we consider the plate to be 25mm by 25 mm, using the above formula, we obtain **5.53 pF** (the dielectric is 1mm thick).
- We obtain **221.49pF**. This value should be considered as the capacitance of the two parallel plates with a 1 meter depth. If we rescale this value by multiplying by 0.025m (25 mm) we find **5.53pF** as well.

Introduction on the DC Conduction Solver

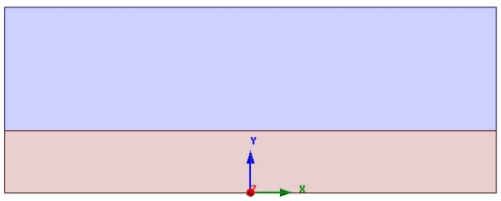
- This note introduces the DC conduction solver. Only conductors are considered in the process.

Example 1: Parallel Plates with Non-uniform media in XY Plane

- 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. The Problem is solved with 2D XY representation.



3D Geometry



2D Representation in XY Plane

Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon
  - Change the name of the design to **PlaneXY**

Set Solution Type

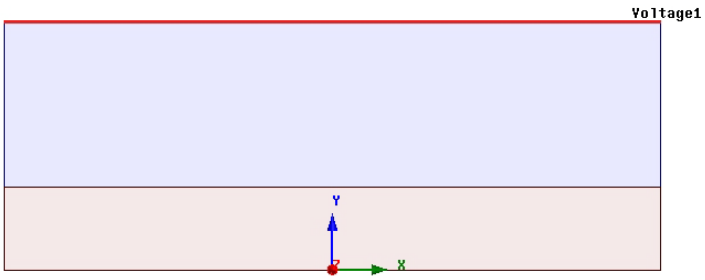
- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cartesian, XY**
    2. Choose **DC Conduction**
    3. Click the **OK** button

Create Geometry

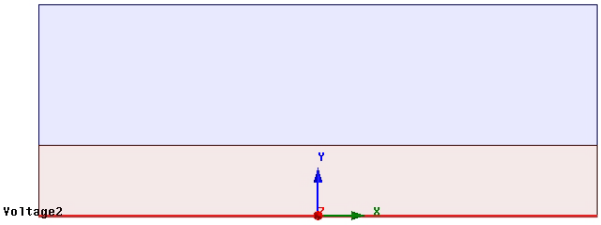
- **Create Solid**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the box position
       - X: **-2**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **4**, dY: **0.5**, dZ: **0**, Press the **Enter** key
  - Change the name of the sheet to **Solid** and material to **graphite**
- **Create Liquid**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the box position
       - X: **-2**, Y: **0.5**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **4**, dY: **1**, dZ: **0**, Press the **Enter** key
  - Change the name of the sheet 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 Edges**
  - Select the menu item *Edit > Select > Edges* or press **E** from the keyboard
- **Assign Excitation for Top**
  - Select the top edge of the sheet **Liquid**
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    1. Set Value to **10 V**
    2. Press **OK**



- **Assign Excitation for Top**
  - Select the bottom edge of the sheet **Solid**
  - Select the menu item *Maxwell 2D > 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 2D > 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 2D > Analyze All*

Compute the DC Resistance

- DC Resistance is not calculated automatically by Maxwell. But since we already know the voltages applied, Resistance can be calculated. 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 will create a line at the middle of Liquid
- **Create Line**
  - Select the menu item *Draw > Line*
    - A massage will pop up asking if the geometry needs to be created as a non-model object, press **Yes**
    1. Using the coordinate entry fields, enter the first point
       - X: **-2**, Y: **1**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - X: **2**, Y: **1**, Z: **0**, Press the **Enter** key
    3. Press **Enter** again to exit the drawing
  - Change the name of the line to **Sec_Line**

- **Calculate Resistance**
  - Select the menu item *Maxwell 2D > 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 line Sec_Line multiplied by depth of the model in Z direction (assuming dimensions of the planes in Z are 4mm)
    1. Select **Input > Quantity > J**
    2. Select **Vector > Scal? > ScalarY**
    3. Select **Input > Geometry**
       - Select **Line**
       - Select **Sec_Line** from the list
       - Press **OK**
    4. Select **Scalar > Integrate**
    5. Press **Eval**
  - The reported value is around **-320.** The total current appears, close to **1.28A**(-320*0.004).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

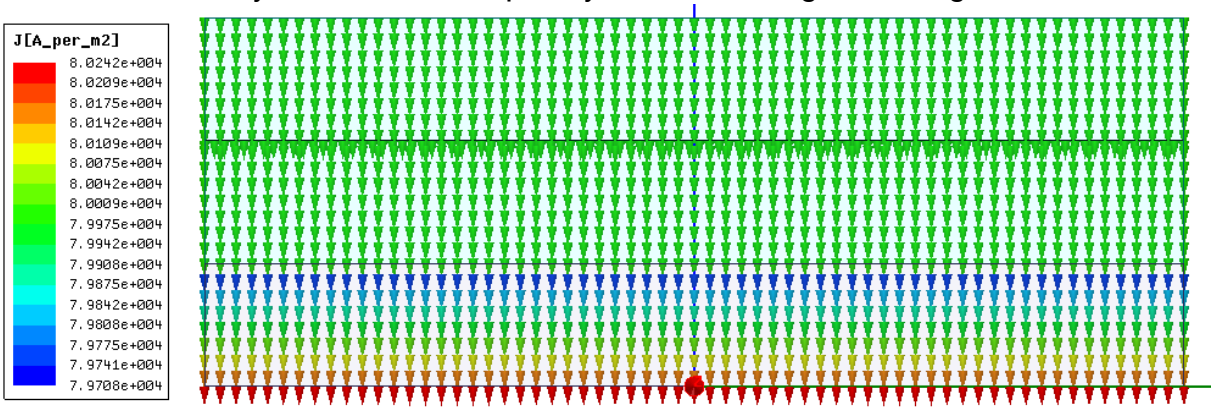
    $$R = \sigma_2 h_1 + \sigma_1 h_2 / (\sigma_1 \sigma_2 A)$$

    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 =16 e-6 m^2.
  - The value of resistance comes out to be:

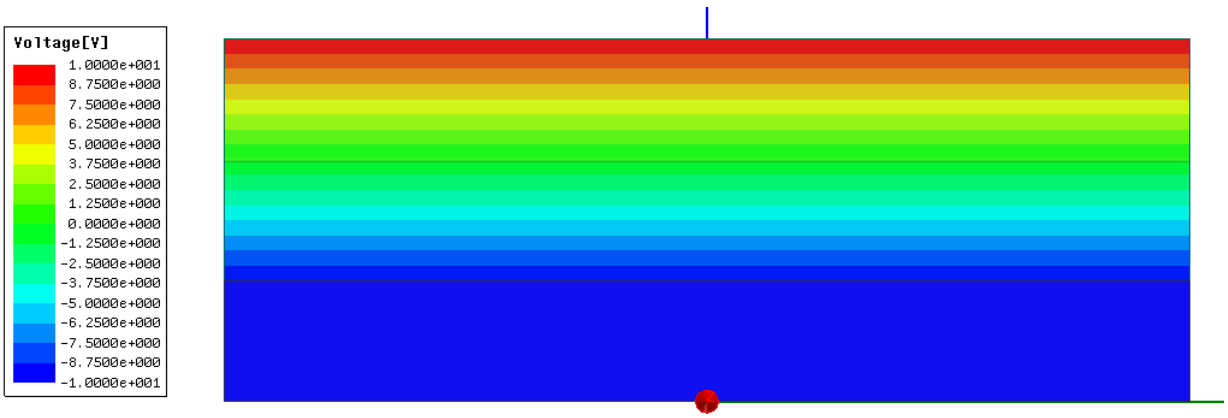
    R = 15.625 Ohm
  - The values are similar to what we obtained using Maxwell

Field Plots

- **Change Selection to Objects**
  - Select the menu item *Edit > Select > Objects* or press **O** from the keyboard
- **Plot Current Density Vectors**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > J > J_Vector*
  - In Create Field Plot window,
    - Press **Done**
  - Modify attributes of the plot by double clicking on the legend

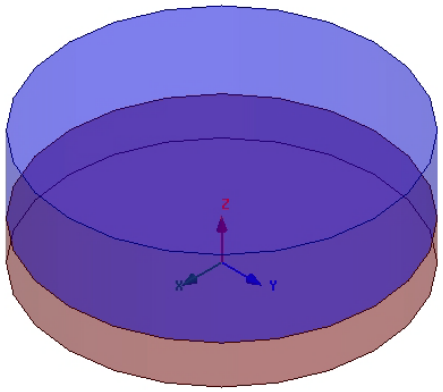


- **Plot Voltage**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > Voltage*
  - In Create Field Plot window,
    - Press **Done**

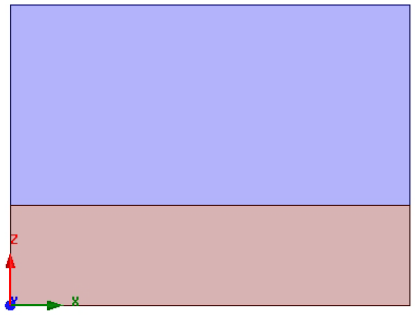


Example 2: Parallel Plates with Non-uniform media in RZ Plane

- In this example, we will solve same example as Example1 with RZ representation. Hence the plates are cylindrical in shape instead of rectangular as in previous example



3D Geometry



2D Representation in R-Z Plane

Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon
  - Change the name of the design to **Plates**

Set Solution Type

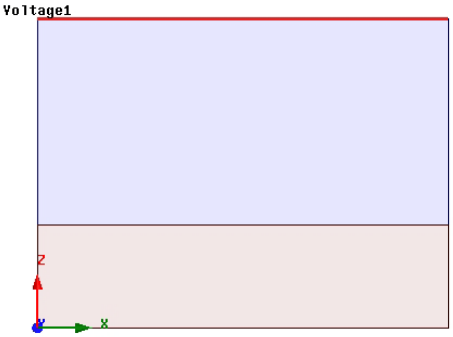
- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **DC Conduction**
    3. Click the **OK** button

Create Geometry

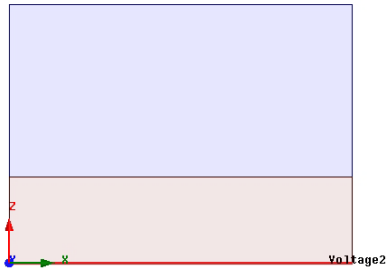
- **Create Solid**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the box position
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **0**, dZ: **0.5**, Press the **Enter** key
  - Change the name of the sheet to **Solid** and material to **graphite**
- **Create Liquid**
  - Select the menu item *Draw > Rectangle*
    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: **2**, dY: **4**, dZ: **1**, Press the **Enter** key
  - Change the name of the sheet 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 Edges**
  - Select the menu item *Edit > Select > Edges* or press **E** from the keyboard
- **Assign Excitation for Top**
  - Select the top edge of the sheet **Liquid**
  - Select the menu item *Maxwell 2D > Excitations > Assign > Voltage*
  - In Voltage Excitation window,
    1. Set Value to **10 V**
    2. Press **OK**



- **Assign Excitation for Top**
  - Select the bottom edge of the sheet **Solid**
  - Select the menu item *Maxwell 2D > 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 2D > 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 2D > Analyze All*

Compute the DC Resistance

- DC Resistance is not calculated automatically by Maxwell. But since we already know the voltages applied, Resistance can be calculated. 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 will create a line at the middle of Liquid
- **Create Line**
  - Select the menu item *Draw > Line*
    - A massage will pop up asking if the geometry needs to be created as a non-model object, press **Yes**
    1. Using the coordinate entry fields, enter the first point
       - X: **0**, Y: **0**, Z: **1**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the second point
       - X: **2**, Y: **0**, Z: **1**, Press the **Enter** key
    3. Press **Enter** again to exit the drawing
  - Change the name of the line to **Sec_Line**

- **Calculate Resistance**
  - Select the menu item *Maxwell 2D > 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 RZ integral of the current J on the line Sec_Line
    1. Select **Input > Quantity > J**
    2. Select **Vector > Scal? > ScalarZ**
    3. Select **Input > Geometry**
       - Select **Line**
       - Select **Sec_Line** from the list
       - Press **OK**
    4. Select **Scalar > Integral >RZ**
    5. Press **Eval**
  - The total current appears, close to **-1.005A**. 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 = 19.9 Ohm**

- **Analytical value of Resistance**
  - The analytical value of the resistance is given by the following formula

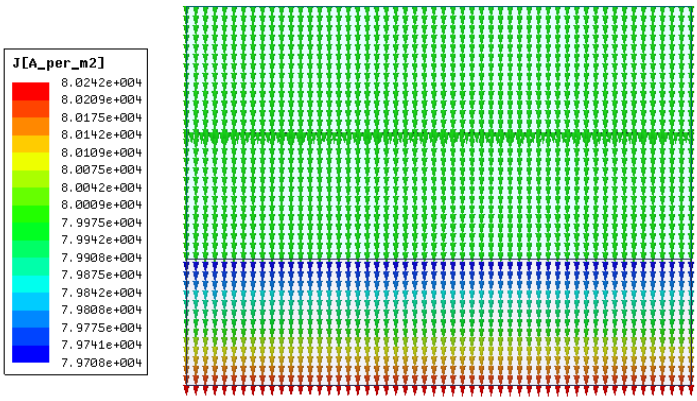
    $$R = \sigma_2 h_1 + \sigma_1 h_2 / (\sigma_1 \sigma_2 A)$$

    where σ_1, σ_2 are the conductivity of the two medium, h_1,h_2 the thickness of the two medium R is radius of plates 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) ;
    - R = 2e-3 m
    - A = πR^2 =12.566 e-6 m^2.
  - The value of resistance comes out to be:

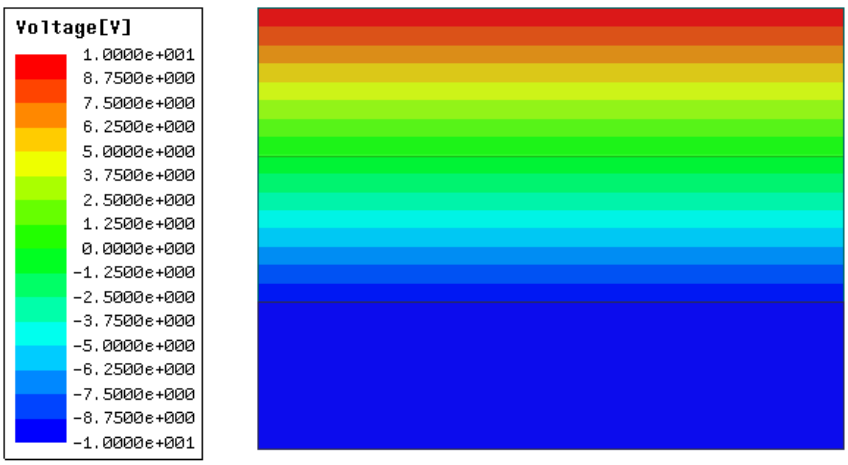
    R = 19.9 Ohm
  - The values are similar to what we obtained using Maxwell

Field Plots

- **Change Selection to Objects**
  - Select the menu item *Edit > Select > Objects* or press **O** from the keyboard
- **Plot Current Density Vectors**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > J > J_Vector*
  - In Create Field Plot window,
    - Press **Done**
  - Modify attributes of the plot by double clicking on the legend



- **Plot Voltage**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > Voltage*
  - In Create Field Plot window,



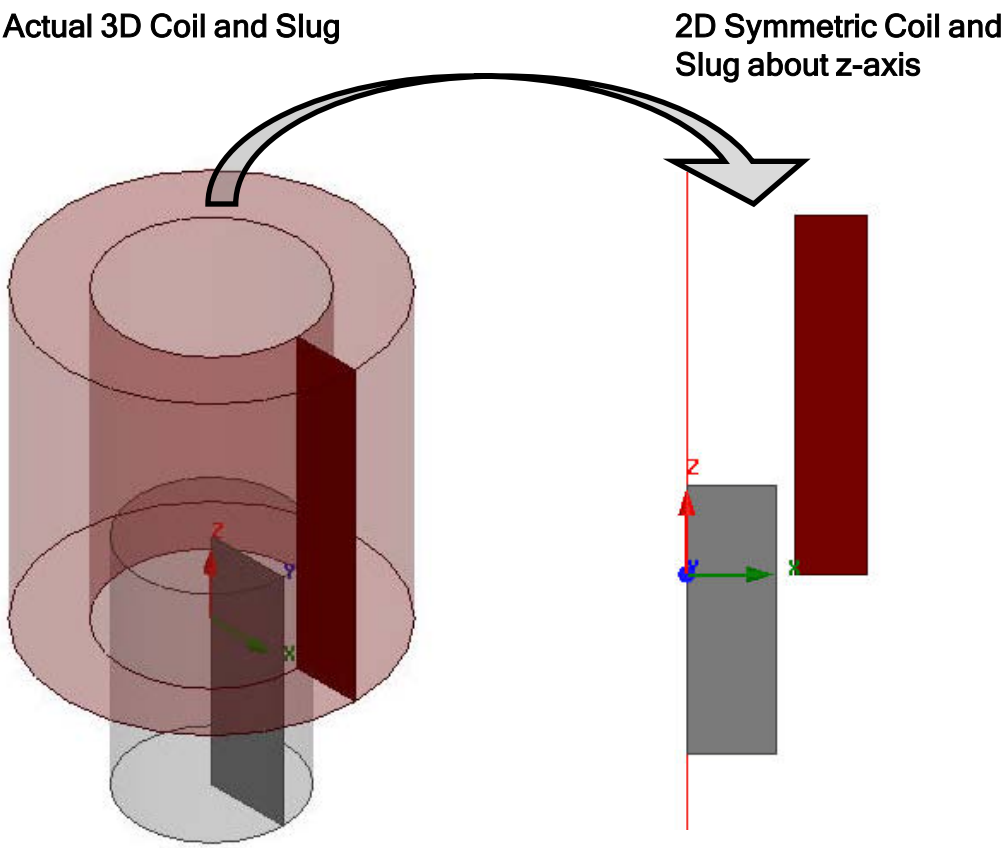
Basic Exercises – Magnetostatic Force Calculation

Force calculation in Magnetostatic Solver

- This exercise will discuss how to set up a force calculation in the 2D Magnetostatic Solver.

Problem Description

- As shown in the following picture, a coil and slug are drawn in a plane using RZ symmetry. The coils carry a current that exert a vertical force on the ferromagnetic slug.



Basic Exercises - Magnetostatic Force Calculation

Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the  icon

Set Solution Type

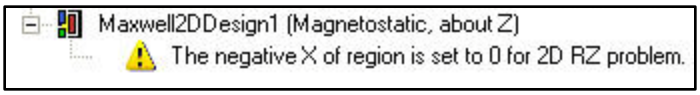
- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Magnetostatic**
    3. Click the **OK** button

Create Model

- **Create Slug**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **-10**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **5**, dY: **0**, dZ: **15**, Press the **Enter** key
  - Change the name of resulting sheet to **Slug** and color to **Gray**
  - Change the material of the sheet to **Steel1008**
- **Create Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **6**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **4**, dY: **0**, dZ: **20**, Press the **Enter** key
  - Change the name of resulting sheet to **Coil** and color to **Brown**
  - Change the material of the sheet to **Copper**

Define Region

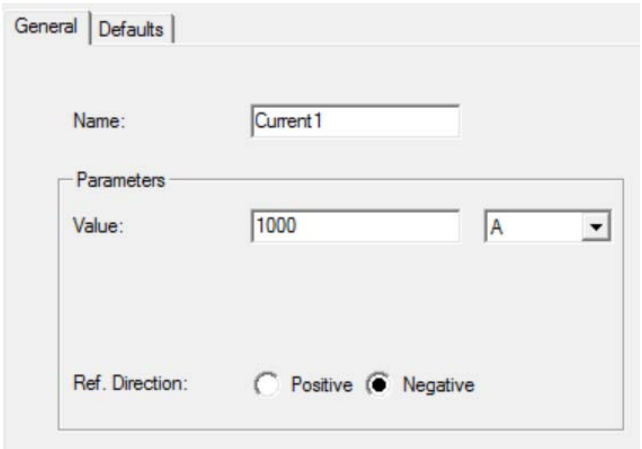
- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **100**
    4. Press **OK**
  - You should see a message indicating that the -X direction is set to zero due to RZ-symmetry about the Z-axis.

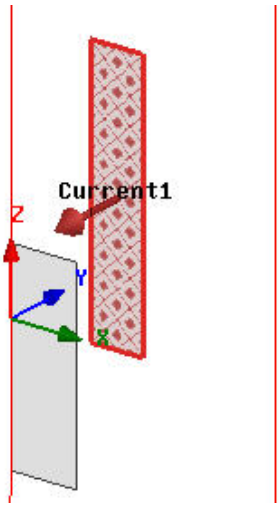


  - Select the object **Region** from the history tree.
  - Select the menu item *View > Hide Selection > Active View*

Assign Excitations

- **To Assign Excitations**
  - Select the sheet **Coil** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
    - Name: **Current1**
    - Value: **1000**
    - Ref. Direction: **Negative** (current will be in the negative Y direction)





Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - Note: On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window,
    - Press **OK**

Assign Force Calculation

- **To Assign Force Parameter**
  - Select the object **Coil** from history tree
  - Select the menu item *Maxwell 2D > Parameters > Assign > Force*
  - In Force Setup window,
    - Press **OK**

Analysis Setup

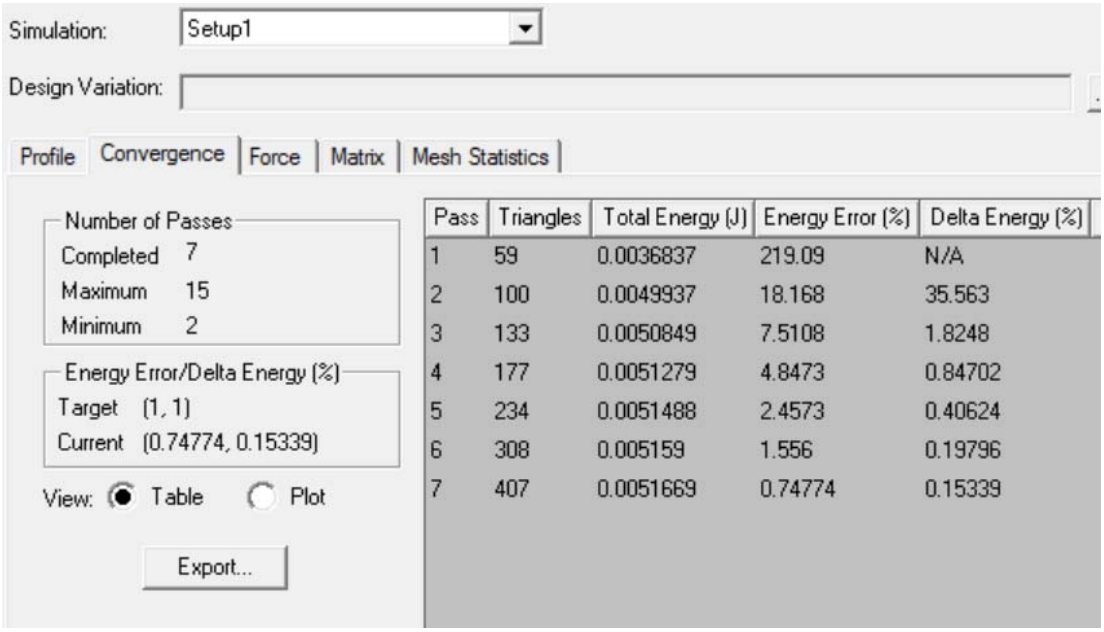
- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Maximum Number of Passes: **15**
    2. **Convergence** Tab
       - Refinement Per Pass: **30%**
    3. Click the **OK** button

Analyze

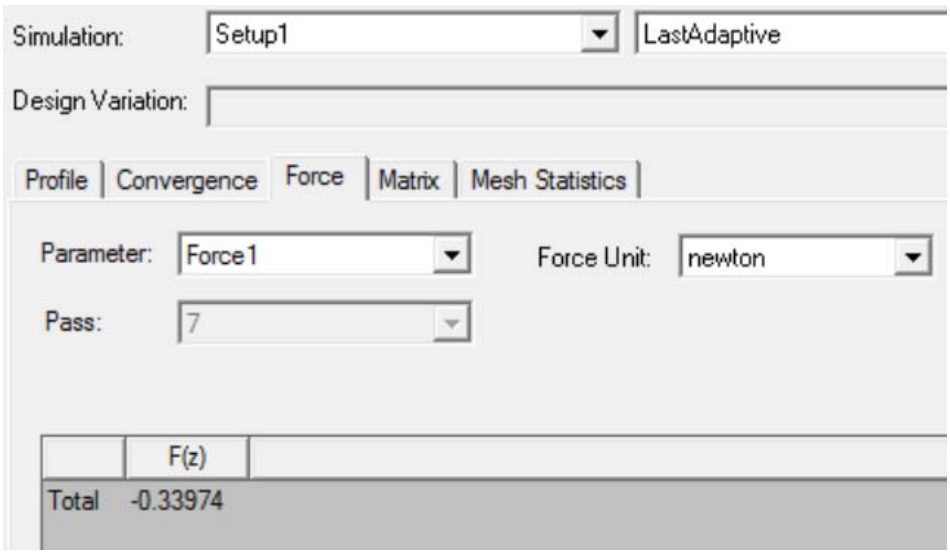
- **To start the solution process:**
  1. Select the menu item *Maxwell 2D > Analyze All*

Solution Data

- **To View Solution Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
    - To View Convergence
      - Select the **Convergence** tab

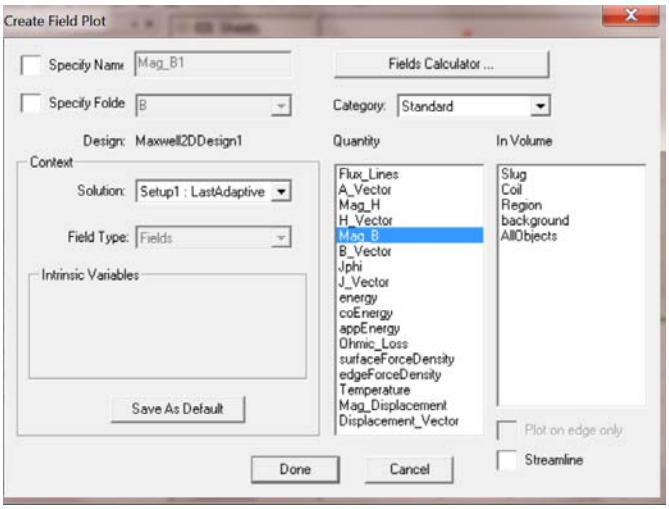


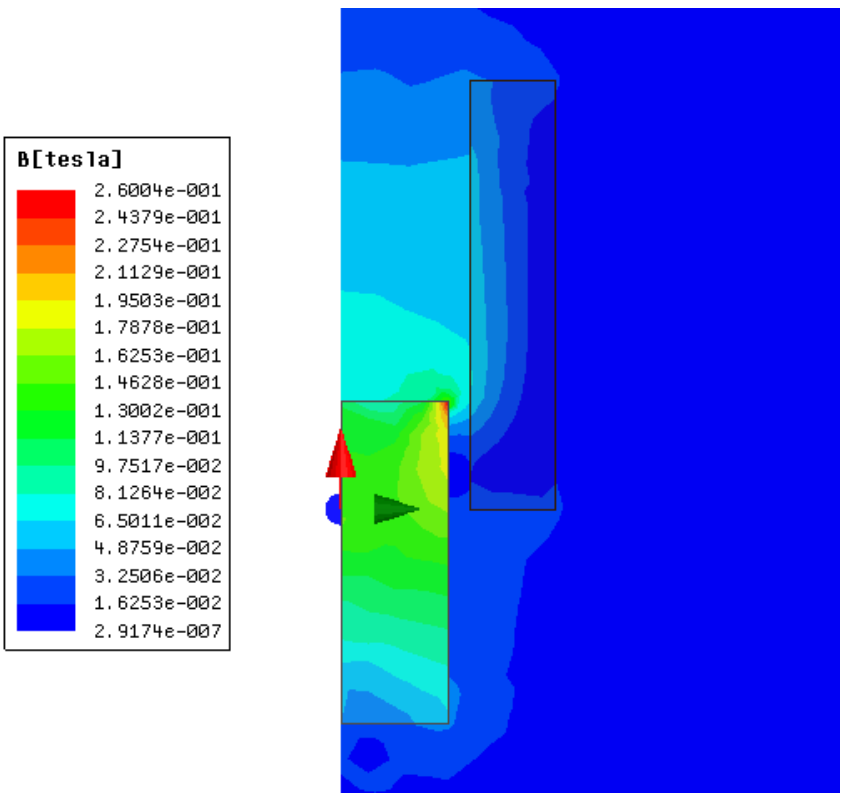
    - To View Calculated Force Value
      - Select the **Force** tab



Plot Magnetic Flux Density

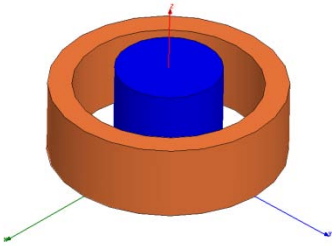
- **To Plot Magnetic Flux Density**
  - Expand the history tree for Planes
  - Select the plane **Global:XZ**
  - Select the menu item ***Maxwell 2D > Fields > Fields > B > Mag_B***
  - In Create Field Plot window,
    - Press **Done**





2D Parametric study using a coil and iron slug.

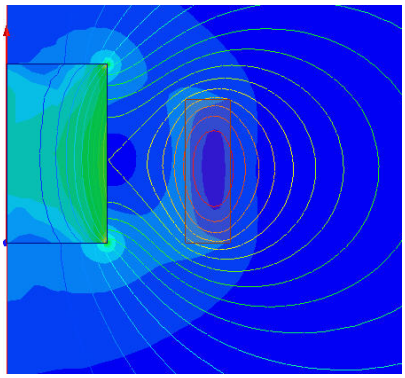
- An RZ Magnetostatic problem will be used to demonstrate the setup of a parametric solution using Optimetrics in Maxwell 2D. The coil current and the dimensional length of an iron slug will be varied and the force on the slug will be observed.



3D Geometry: Coil and Iron Slug

Coil OR = 1.25mm
Coil IR = 1mm
Coil Height = 0.8mm

Slug width = 1mm
Slug depth = 1mm
Slug Height = 1mm



2D Flux Lines and Flux Density

Create Design

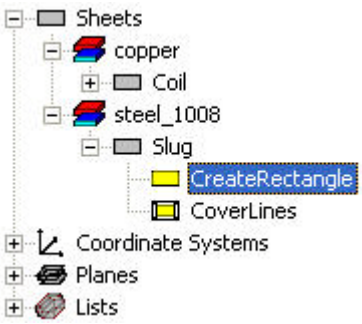
- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

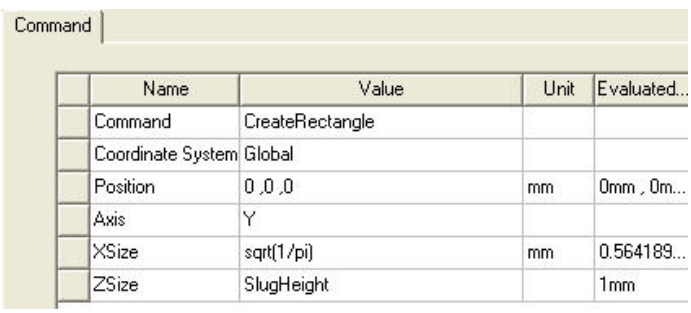
Set Solution Type

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

Create Model

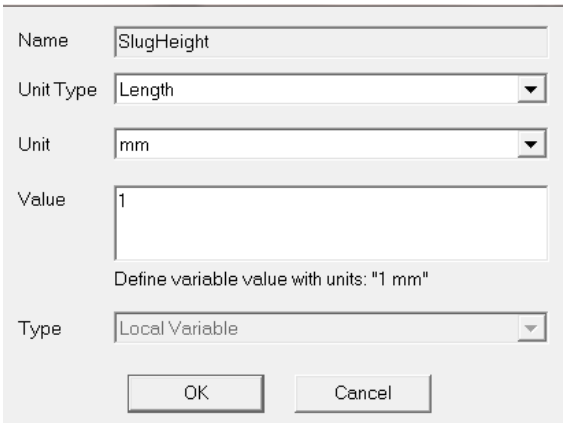
- **Create Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **1**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.25**, dY: **0**, dZ: **0.8**, Press the **Enter** key
  - Change the name of resulting sheet to **Coil** and color to **Orange**
  - Change the material of the sheet to **Copper**
- **Create Slug**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **0.5**, dY: **0**, dZ: **1**, Press the **Enter** key
  - Change the name of resulting sheet to **Slug** and color to **Blue**
  - Change the material of the sheet to **Steel_1008**
- **Parameterize Slug**
  - Expand the history tree for the object **Slug**
  - Double Click on the command **CreateRectangle** to open Properties window,
  - In Properties window,
    - Change Xsize to **sqrt(1/pi)**
    - Change Zsize to **SlugHeight**
    - Press **Enter**





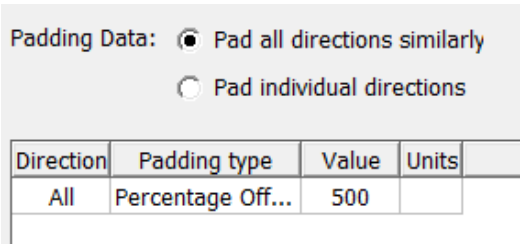
Name	Value	Unit	Evaluated...
Command	CreateRectangle		
Coordinate System	Global		
Position	0 ,0 ,0	mm	0mm , 0m...
Axis	Y		
XSize	sqrt(1/pi)	mm	0.564189...
ZSize	SlugHeight		1mm

- In Add Variable window,
  - Unit Type: **Length**
  - Unit: **mm**
  - Value: **1**
  - Press **OK**



- **Note:** By defining a variable name (SlugHeight) it becomes a design variable. Similarly, if an object is moved, it's move distance can be assigned a variable. The Design Variables are accessible in the Property window by clicking on the Design name in the Project Manager.
  Or they can be viewed by clicking: **Maxwell 2D > Design Properties . . .**

- **Note:** The parameter for Xsize is defined using the predefined constant, *pi,* and an equation that calculates the equivalent 2D cross-section of a $1mm^2$ slug which was used in the 3D Exercise. Other predefined constants can be found by selecting from the menu, **Project > Project Variables**, and selecting **Constants** tab.



Define Region

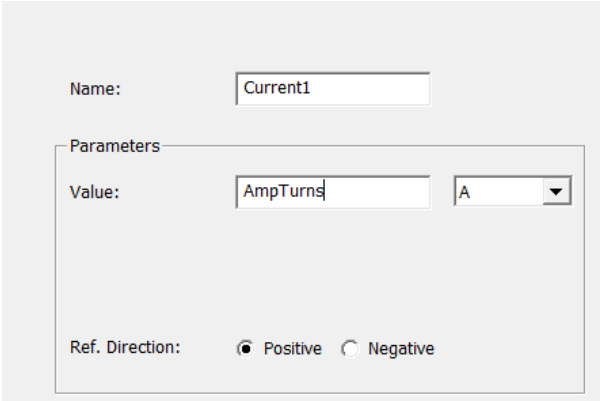
- **Create Simulation Region**
  - Select the menu item ***Draw > Region***
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **500**
    4. Press **OK**
  - **Note:** Region will not be created in Negative X direction due to RZ-symmetry about the Z-axis.

Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - Note: On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window, Press **OK**

Assign Excitations

- **To Assign Excitations**
  - Select the sheet **Coil** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
    - Name: **Current1**
    - Value: **AmpTurns**
    - Ref. Direction: **Positive**
    - Press **OK**
  - In Add Variable window,
    - Value: **100**
    - Press **OK**



Assign Force Calculation

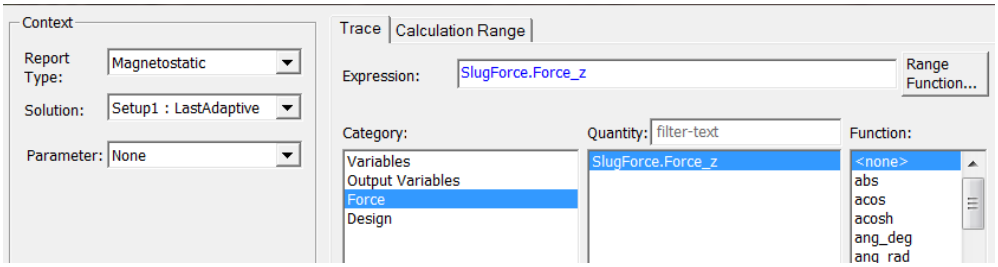
- **To Assign Force Parameter**
  - Select the object **Slug** from history tree
  - Select the menu item *Maxwell 2D > Parameters > Assign > Force*
  - In Force Setup window,
    - Name: **SlugForce**
    - Press **OK**

Analysis Setup

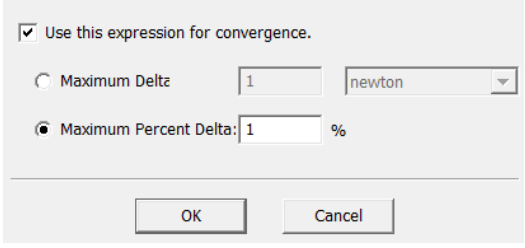
- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,
    - Press **OK**
  - **Note:** We have created Analysis Setup to enable Expression Cache tab. Parameters will be edited in next step

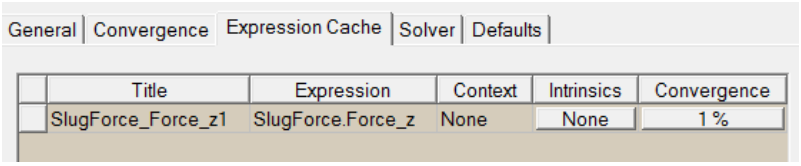
Modify Analysis Setup

- **To Modify Analysis Setup**
  - Expand the Project Manager tree to view Analysis
  - Double click on **Setup1** to edit its parameters
  - In Solve Setup window,
    - **General** tab
      - Maximum Number of Passes: **10**
      - Percentage Error: **0.5**
    - **Expression Cache** tab
      - Select **Add**
      - In Add to Expression Cache window,
        - Category: **Force**
        - Quantity: **SlugForce.Force_z**
        - Select **Add to Calculation** and **Done**



      - Change Title to **SlugForce**
      - Select the tab in **Convergence** column
      - In Adaptive Convergence window,
        - Use this expression for convergence: ☑ **Checked**
        - Maximum Percentage Delta: **1 %**
        - Press **OK**
      - Press **OK** to close Solve Setup window



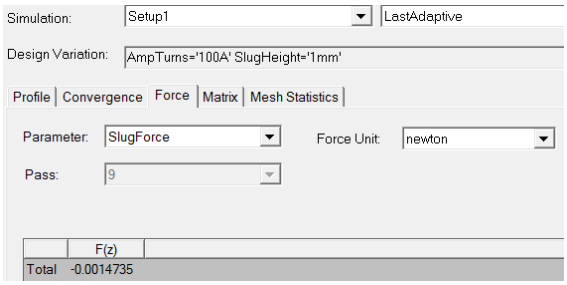


Analyze

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

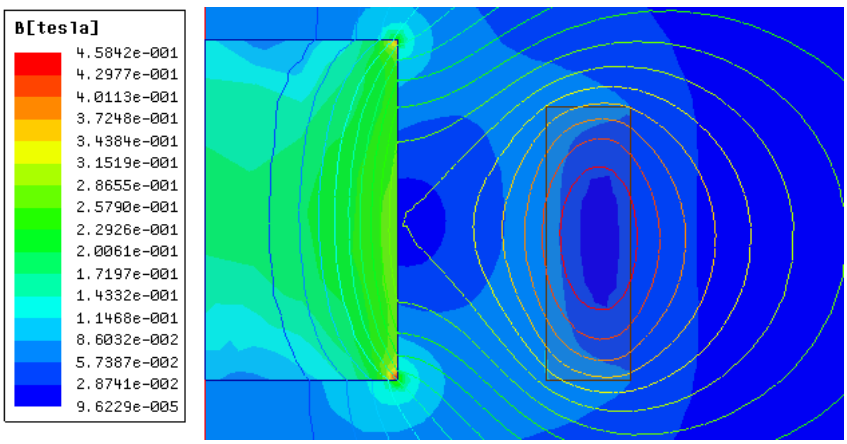
Solution Data

- **To View Solution Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
    - To View Convergence
      - Select **Convergence** tab
    - To View Force Values
      - Select the **Force** tab



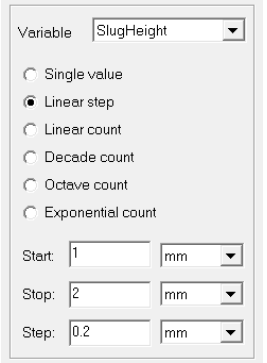
Plot Flux Results

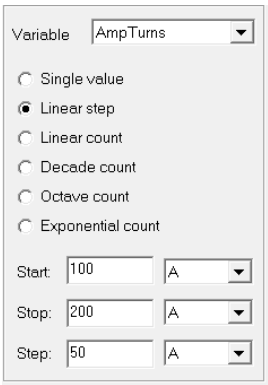
- **To Plot Flux**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    - Press **Done**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window,
    - Press **Done**

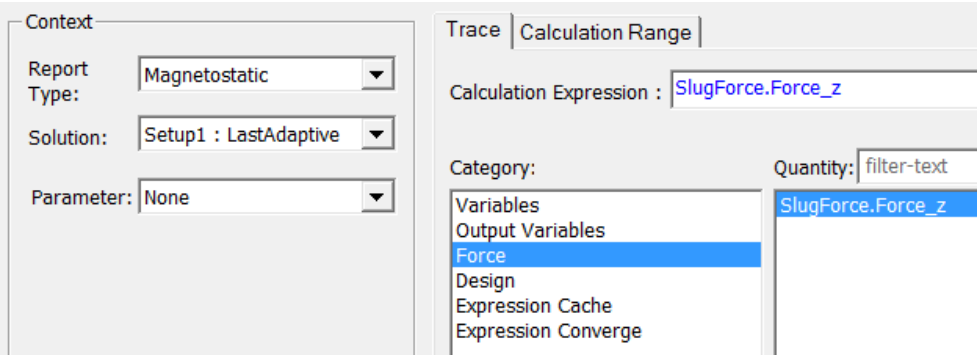


Create Parametric Setup

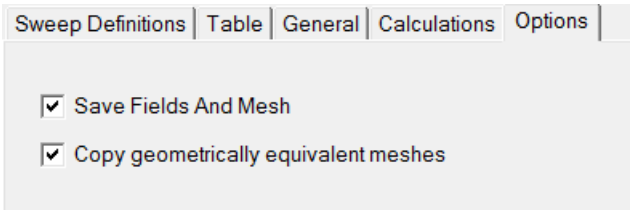
- **To Create Parametric Setup**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Add Parametric*
  - In Setup Sweep Analysis window, select **Add**
  - In Add/Edit Sweep window,
    - Variable: **SlugHeight**
    - Select **Linear Step**
    - Start: **1mm**
    - Stop: **2mm**
    - Step: **0.2mm**
    - Select **Add**
    - Change Variable to **AmpTurns**
    - Select **Linear Step**
    - Start: **100 A**
    - Stop: **200 A**
    - Step: **50 A**
    - Select **Add** and **OK**
  - In Setup Sweep Analysis window,
    - Select **Table** tab
      - There should be 18 solutions since we defined 6 variations of SlugHeight and 3 variations of AmpTurns.
    - Select **Calculations** tab
      - Select **Setup Calculations**
      - In Add/Edit Calculations window,
        - Category: **Expression Cache**
        - Quantity: **ExprCache(SlugForce)**
        - Select **Add Calculation** and **Done**







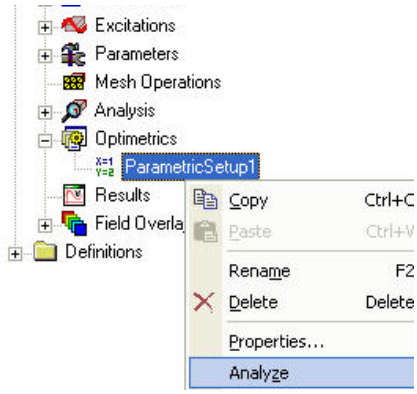
- Select **Options** tab
  - Save Fields And Mesh: ☑ **Checked**
  - Copy geometrically equivalent meshes: ☑ **Checked**



- Select **OK** to close Setup Sweep Analysis window

Solve Parametric Problem

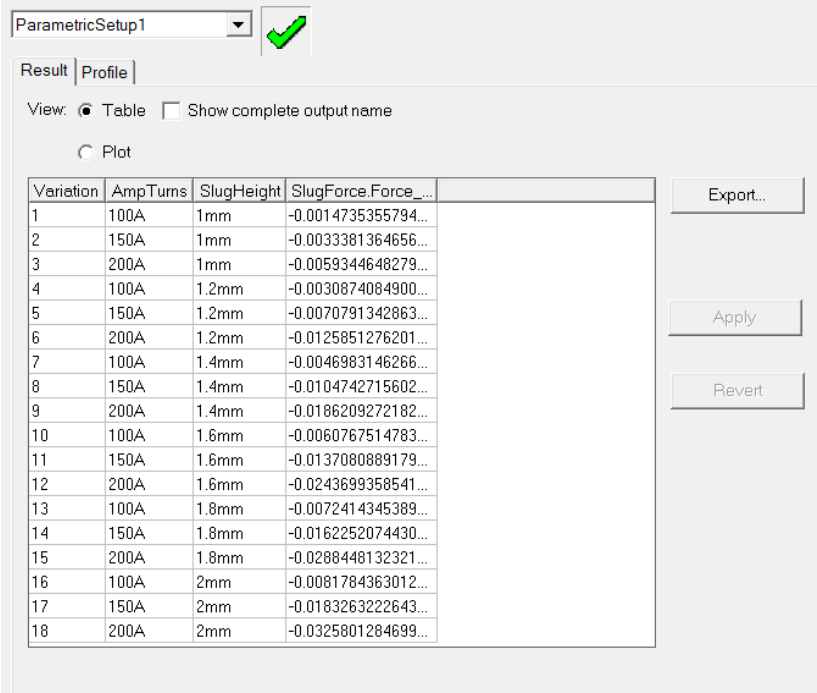
- **To Solve Parametric Analysis**
  - Expand the Project Manager tree for **Optimetrics**
  - Right click on the tab **Parametric Setup1** and select **Analyze**



  - **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 Solution Progress

- **To View Solution Progress**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Optimetrics Results*
  - In Post Analysis Display window,
    - Set the radio button to **Table** to view all the results in tablature form

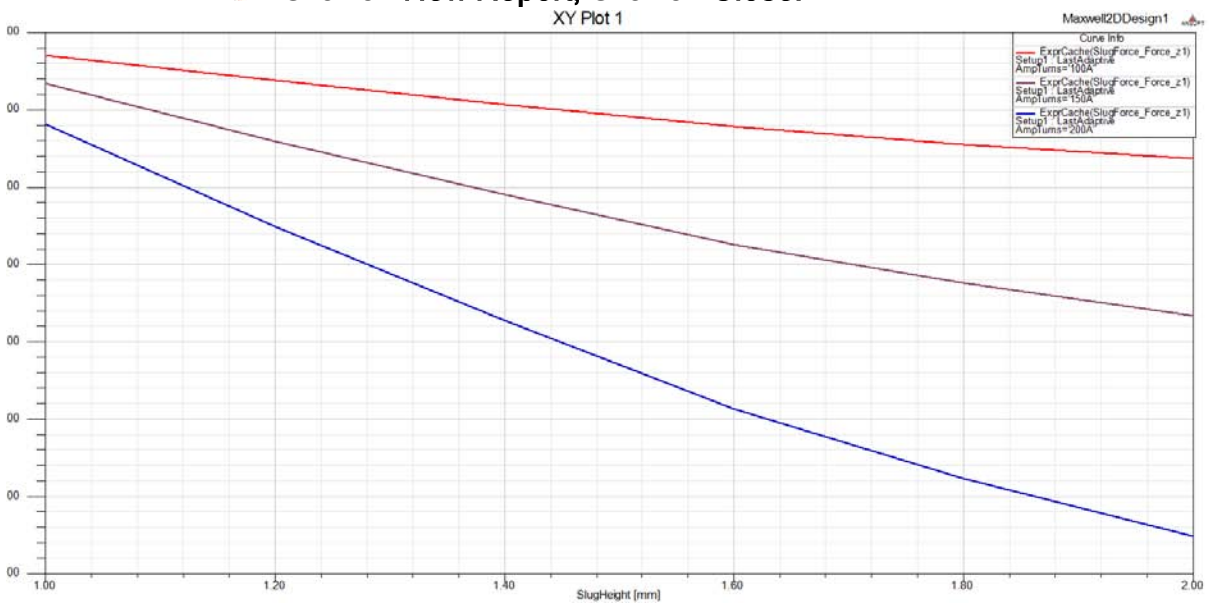


Variation	AmpTurns	SlugHeight	SlugForce.Force_...
1	100A	1mm	-0.0014735355794...
2	150A	1mm	-0.0033381364656...
3	200A	1mm	-0.0059344648279...
4	100A	1.2mm	-0.0030874084900...
5	150A	1.2mm	-0.0070791342863...
6	200A	1.2mm	-0.0125851276201...
7	100A	1.4mm	-0.0046983146266...
8	150A	1.4mm	-0.0104742715602...
9	200A	1.4mm	-0.0186209272182...
10	100A	1.6mm	-0.0060767514783...
11	150A	1.6mm	-0.0137080889179...
12	200A	1.6mm	-0.0243699358541...
13	100A	1.8mm	-0.0072414345389...
14	150A	1.8mm	-0.0162252074430...
15	200A	1.8mm	-0.0288448132321...
16	100A	2mm	-0.0081784363012...
17	150A	2mm	-0.0183263222643...
18	200A	2mm	-0.0325801284699...

Plot Force Vs AmpTurns Vs Time

- **To Create a Plot**
  - Select the menu item *Maxwell 2D > Results > Create Magnetostatic Report > Rectangular Plot*
  - In Reports window,
    - Primary Sweep: **SlugHeight**
    - X: **Default**
    - Category: **ExpressionCache**
    - Quantity: **ExprCache(SlugForce)**
  - Select **Done**

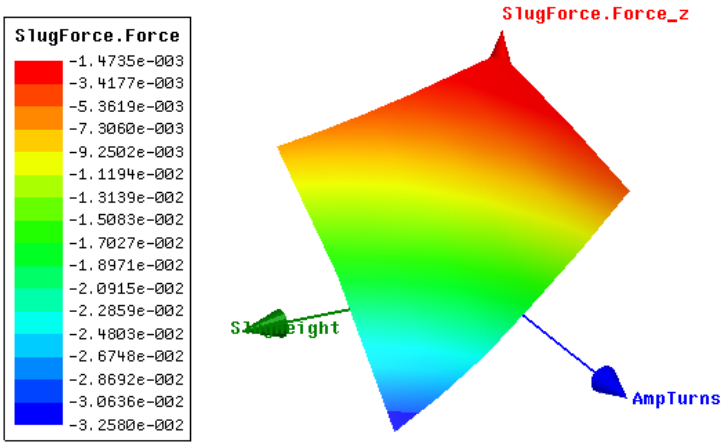
- Select the **Families** tab:
  - Ensure that that AmpTurns is selected as the Sweeps variable.
  - Click on **New Report,** Click on **Close.**



- To Export results, right click on the plot and select **Export**

Create 3D Plot

- **To Create a 3D plots**
  - Select the menu item *Maxwell 2D > Results > Create Magnetostatic Report > 3D Rectangular Plot*
  - In Reports window,
    - Press **New Report** and **Done**



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 2D Design*, or click on the  icon

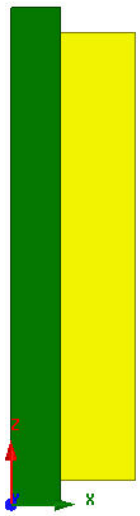
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Transient**
    3. Click the **OK** button

Create Model

- **Create Core**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **0**, dZ: **20**, Press the **Enter** key
  - Change the name of resulting sheet to **Core** and color to **green**
  - Change the material of the sheet to **Ferrite**
- **Create Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **1**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **5**, dY: **0**, dZ: **18**, Press the **Enter** key
  - Change the name of resulting sheet to **Coil** and color to **Yellow**
  - Change the material of the sheet to **Copper**

- **Subtract Sheets**
  - Press **Ctrl** and select the sheets **Core** and **Coil**
  - Select the menu item *Modeler > Boolean > Subtract*
    - Blank Part: **Coil**
    - Tool Part: **Core**
    - Clone tool objects before subtracting: ☑ **checked**

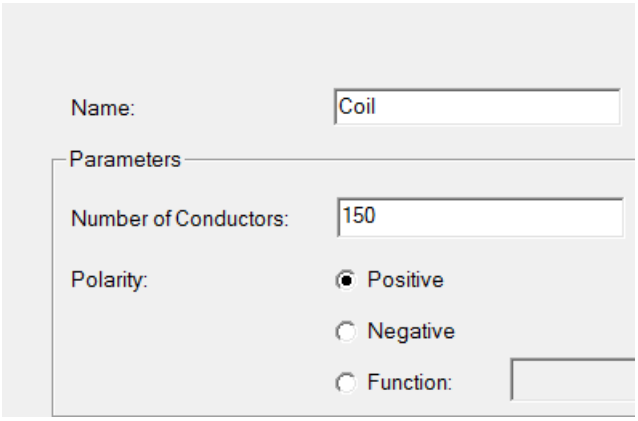


Define Region

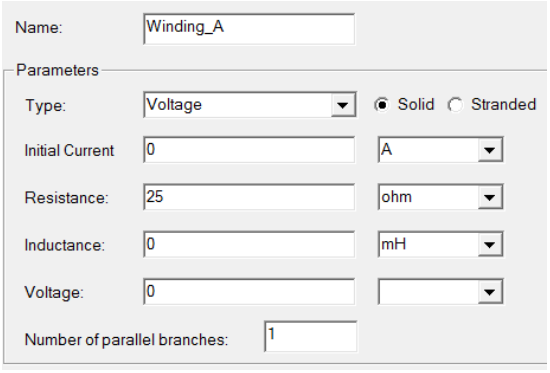
- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **500**
    4. Press **OK**
  - **Note:** Region will not be created in Negative X direction due to RZ-symmetry about the Z-axis.

Assign Excitations

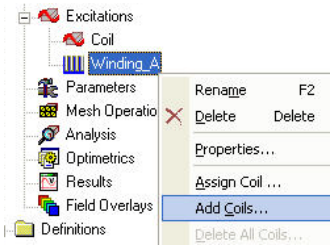
- **To Create Coil**
  - Select the object **Coil** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
    - Name: **Coil**
    - Number of Conductors: **150**
    - Polarity: **Positive** (into the screen)



- **To Add Winding**
  - Select the menu item *Maxwell 2D > Excitation > Add Winding*
  - In Winding window,
    - Name: **Winding_A**
    - Type: **Voltage**
    - Stranded: ☑ **Checked**
    - Initial Current: **0 A**
    - Resistance: **25 ohm**
    - Inductance: **0 H**
    - Voltage: **0 V**
      - Note: This will be changed later
    - Number of parallel branches: **1**

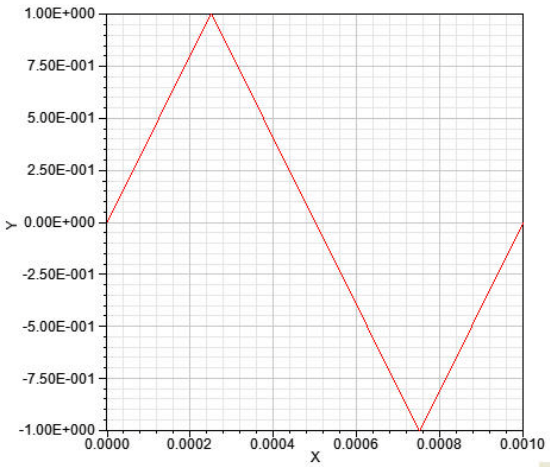


- **To Add Coil to Winding**
  - Expand the Project Manager tree to view **Excitations**
  - Right click on the tab **Winding_A** and select **Add Coil**
  - In Add Terminals window,
    - Select **Coil**
    - Press **OK**

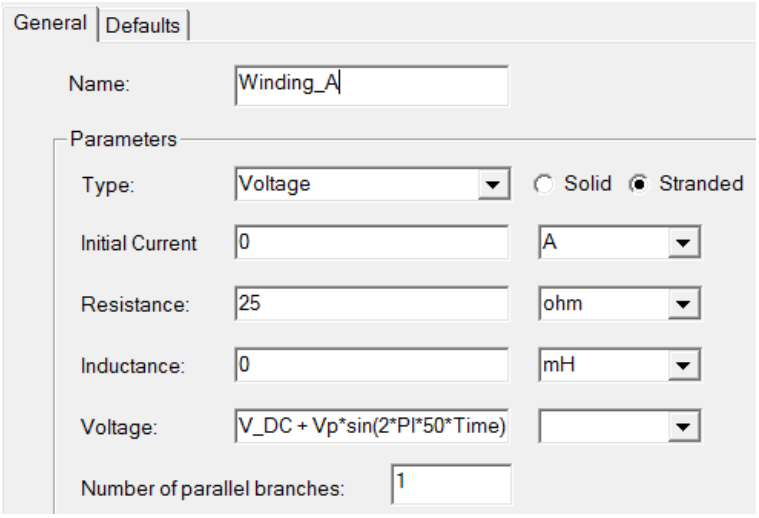


Specify Excitation

- The excitation for this problem will be a voltage source with a 1KHz triangular wave superimposed on a 50 Hz sine wave that has a 50 volt DC offset.
- **To Add a Dataset**
  - Select the menu item *Maxwell 2D > Design Datasets*
  - In Datasets window, select **Add**
  - In Add Dataset window,
  - Name: **DSet_A**
  - Coordinates:
    - X1 = 0 Y1 = 0
    - X2 = 250e-6 Y2 = 1
    - X3 = 750e-6 Y3 = -1
    - X4 = 1e-3 Y4 = 0
  - Click **OK** and **Done.**



- **Edit Winding Parameters**
  - Expand the Project Manager tree to view Excitations
  - Double click on **Winding_A** tab to edit it
  - In Winding window,
    - Change Voltage to **V_DC + Vp*sin(2*Pl*50*Time) + 5*pwl_periodic (DSet_A, Time)**
    - **Note:** The first term is the DC offset and the 2nd is peak voltage of the sine wave
    - Press **OK**
  - In Add Variable window for **V_DC**
    - Value: **50 V**
    - Press **OK**
  - In Add Variable window for **Vp**
    - Value: **25 V**
    - Press **OK**
  - Type: **Stranded**



Assign Boundary

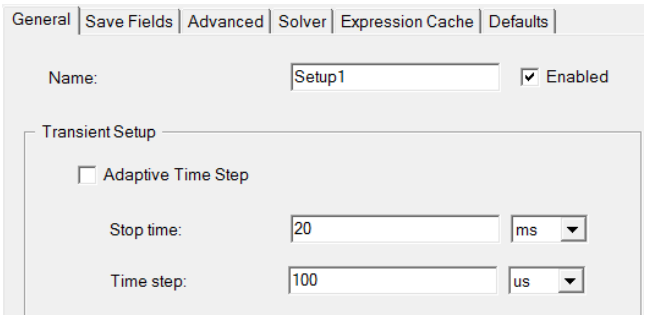
- **To Assign Balloon Boundary**
  - Select the object **Region** from history tree
  - Select the menu item ***Edit > Select > All Object Edges***
  - Select the menu item ***Maxwell 2D > Boundaries > Assign > Balloon***
  - **Note:** On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window,
    - Press **OK**

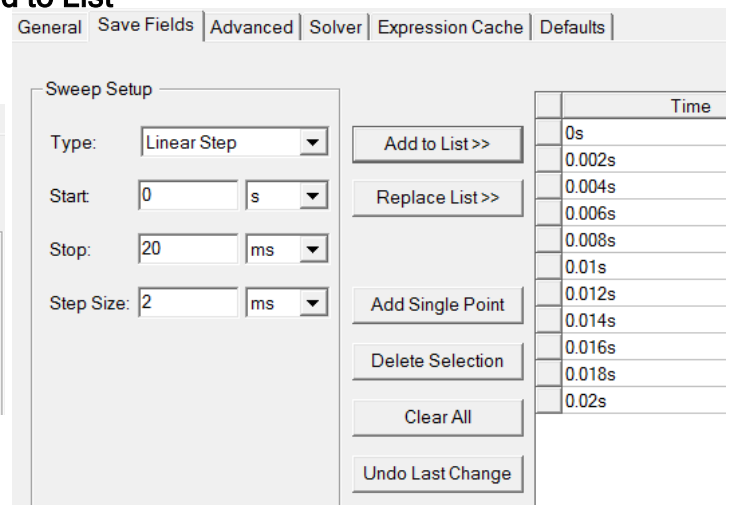
Assign Mesh Operations

- The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created.
- **To Assign Mesh Operation for Core**
  - Select the object **Core** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Core_Inside**
    - Restrict Length Of Elements: ☐**Unchecked**
    - Restrict Number of Elements: ☑**Checked**
    - Maximum Number of Elements: **250**
    - Press **OK**
- **To Assign Mesh Operation for Coil**
  - Select the object **Coil** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Coil_Inside**
    - Restrict Length Of Elements: ☐**Unchecked**
    - Restrict Number of Elements: ☑**Checked**
    - Maximum Number of Elements: **100**
    - 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: **20 ms**
      - Time Step: **100 us**
    - **Save Fields** Tab
      - Type: **Linear Count**
      - Start: **0 sec**
      - Stop: **20 ms**
      - Step Size: **2 ms**
      - Click on: **Add to List**
    - Press **OK**



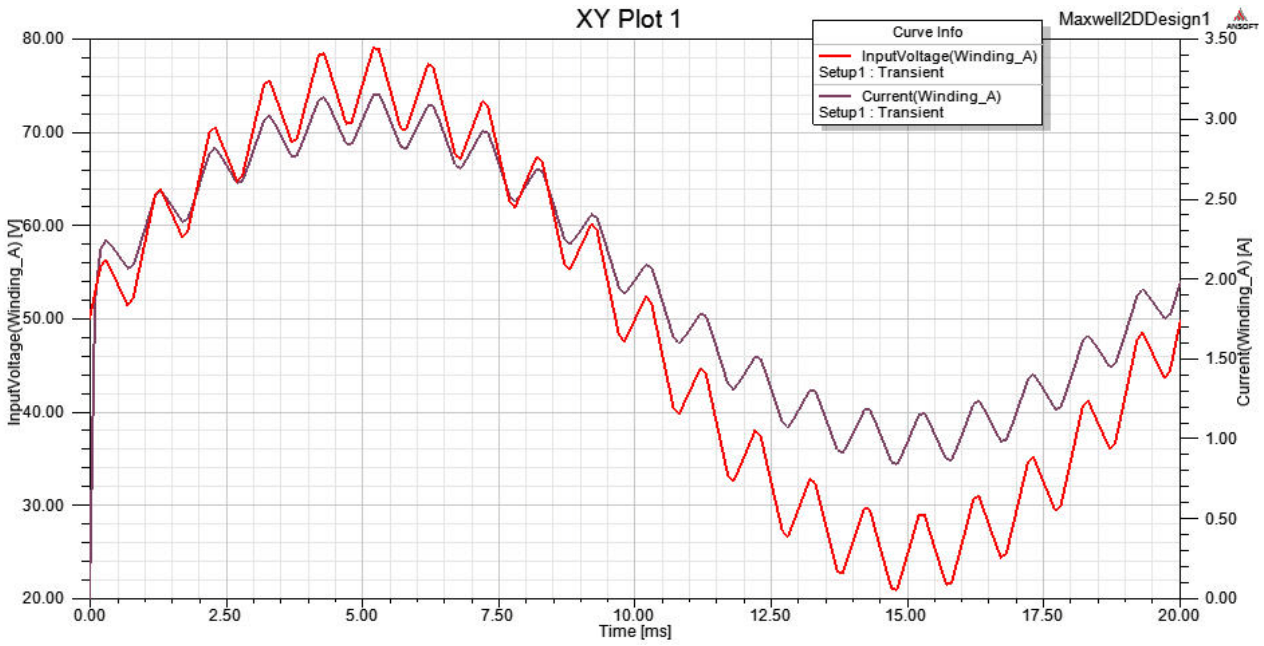


Analyze

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

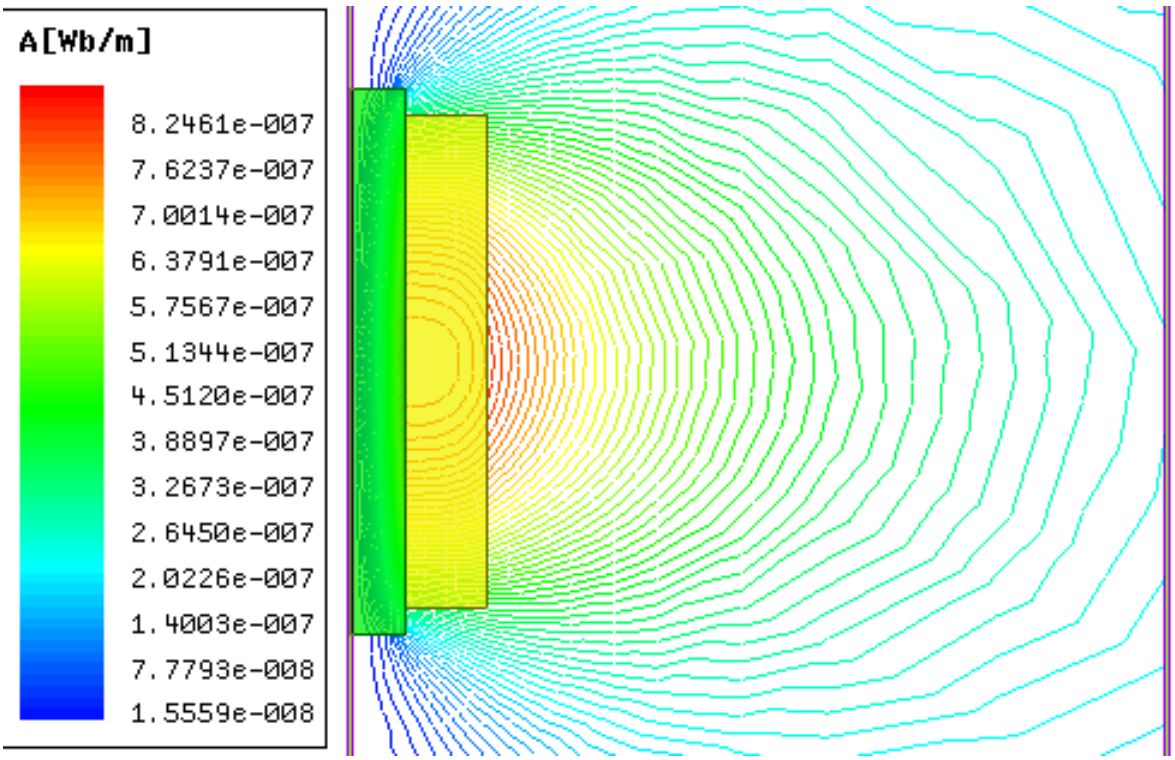
Plot the Voltage and Current

- **To Create a Plot**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular plot:*
  - In Reports window,
    - Select Category: **Winding**
    - Select Quantity: **InputVoltage(Winding_A)**
    - Click on: **New Report**
    - Select Quantity: **Current(Winding_A)**
    - Click on: **Add Trace**
    - Click on: **Close**



Plot Flux Lines

- **To Plot Flux Lines**
  - Be sure that the 2D Modeler window is in the active view window.
  - Select the menu item *View > Set Solution Context*
  - In Set view Context window,
    - Time: Set to **0.01 sec**
    - Press **OK**
  - Select the menu *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plots window,
    - Press **Done**
- **Modify Plot**
  - Double click on the legend to modify its attributes
  - In the window,
    - **Scale** tab
      - Num. Divisions: **56**
    - Press **Apply** and **Close**



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 2D Design*, or click on the  icon

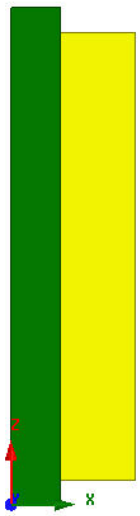
Set Solution Type

- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Transient**
    3. Click the **OK** button

Create Model

- **Create Core**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **0**, dZ: **20**, Press the **Enter** key
  - Change the name of resulting sheet to **Core** and color to **green**
  - Change the material of the sheet to **Ferrite**
- **Create Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **1**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **5**, dY: **0**, dZ: **18**, Press the **Enter** key
  - Change the name of resulting sheet to **Coil** and color to **Yellow**
  - Change the material of the sheet to **Copper**

- **Subtract Sheets**
  - Press **Ctrl** and select the sheets **Core** and **Coil**
  - Select the menu item *Modeler > Boolean > Subtract*
    - Blank Part: **Coil**
    - Tool Part: **Core**
    - Clone tool objects before subtracting: ☑ **checked**

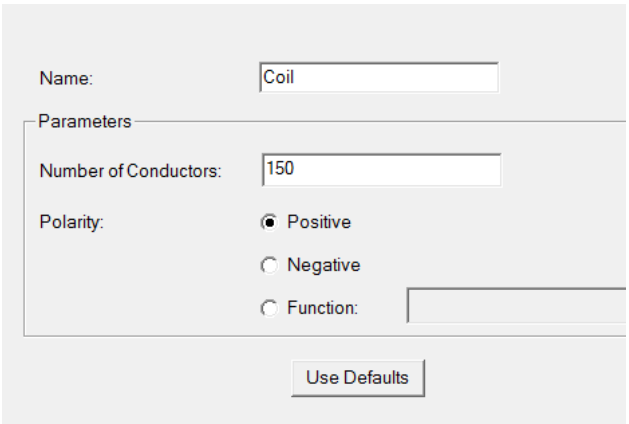


Define Region

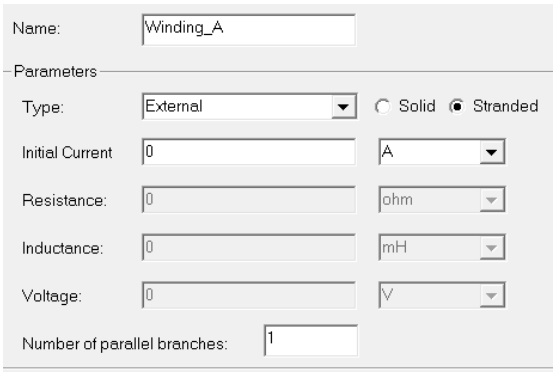
- **Create Simulation Region**
  - Select the menu item *Draw > Region*
  - In Region window,
    1. Pad all directions similarly: ☑ **Checked**
    2. Padding Type: **Percentage Offset**
    3. Value: **500**
    4. Press **OK**
  - **Note:** Region will not be created in Negative X direction due to RZ-symmetry about the Z-axis.

Assign Excitations

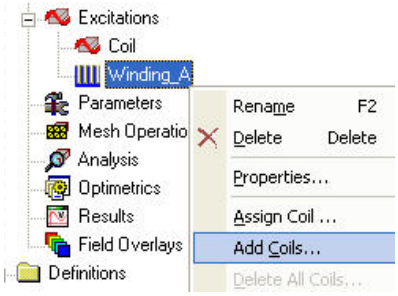
- **To Create Coil**
  - Select the object **Coil** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
    - Name: **Coil**
    - Number of Conductors: **150**
    - Polarity: **Positive** (into the screen)



- **To Add Winding**
  - Select the menu item *Maxwell 2D > Excitation > Add Winding*
  - In Winding window,
    - Name: **Winding_A**
    - Type: **External**
    - Stranded: ☑ **Checked**
    - Initial Current: **0 A**
    - Number of parallel branches: **1**
    - Type: **Stranded**
    - Press **OK**



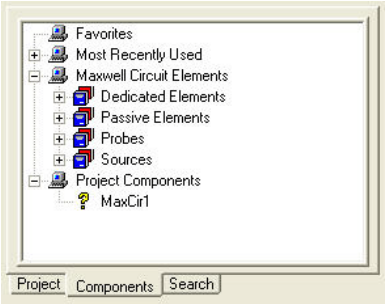
- **To Add Coil to Winding**
  - Expand the Project Manager tree to view **Excitations**
  - Right click on the tab **Winding_A** and select **Add Coil**
  - In Add Terminals window,
    - Select **Coil**
    - Press **OK**



Create External Circuit

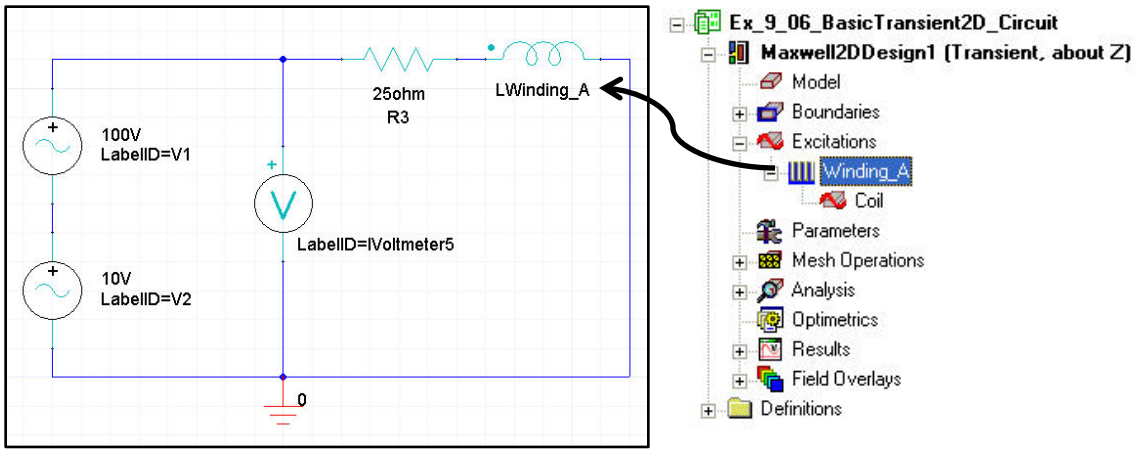
- **To Launch Maxwell Circuit Editor**
  - Select the menu item *Maxwell 2D > Excitations > External Circuit > Edit External Circuit*
  - In Edit External Circuit window,
    - Select **Edit Circuit**
  - Maxwell Circuit Editor will launch in separate window
- **To Create Maxwell Circuit Editor Project**
  - Select the menu item *File > New*

- **To Add Components to the Circuit**
  - In Maxwell Circuit Editor, select the tab **Components** in Project Manager window to get access to components

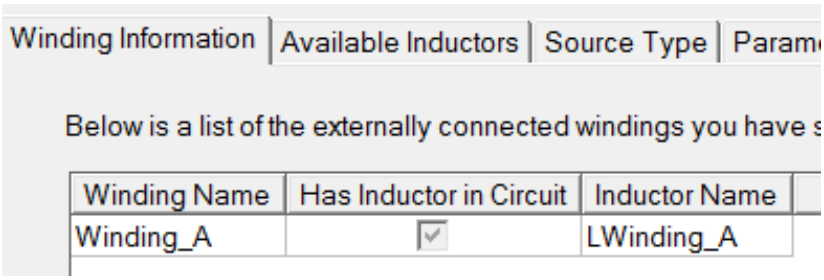


  - Add Winding
    - Select the tab **Maxwell Circuit Elements > Dedicated Elements > Winding** from the tree, drag and drop it on the Project Page
    - Press **Esc** to exit the component insertion
    - Double click on the added component to edit its properties
    - Change Properties as below
      - Name: **Winding_A**
        - **Note**: This name has to be **exactly** the same name as used in the Winding definition described previously in ***Maxwell 3D > Excitations > Add Winding***
      - Press **OK**
  - Similarly add below components
    - Add Voltage Source from **Sources > VSin**
      - Va = **100 V**
      - VFreq = **50 Hz**
    - Another Voltage Source from **Sources > VSin**
      - Va = **10 V**
      - VFreq = **1000 Hz**
    - Add Resistor from **Passive Elements > Resistor**
      - R = **25 ohm**
    - Add a Voltmeter from **Probes > Voltmeter**
  - Add Ground from menu item ***Draw > Ground*** *(ctrl+G brings ground)*

- **Build the Circuit**
  - Position the added components and connect them using wires to build a circuit as shown below



- **Save File**
  - Select the menu item *File > Save*
  - Save the file with the name **"Ex_9_06_BasicTransient2D_Circuit"**
- **Export Circuit to Maxwell**
  - Select the menu item *Maxwell Circuit > Export Netlist*
  - Save the file with the name **"Ex_9_06_BasicTransient2D_Circuit.sph"**
- **Import External Circuit**
  - In Maxwell 2D, say **Cancel** and select the tab **Import Circuit** from **Edit External Circuit** window
  - Select the file **"Ex_9_06_BasicTransient2D_Circuit.sph"** that was written in last step and select **Open**
  - Edit External circuit window will look as below
  - Close the circuit editor



Assign Boundary

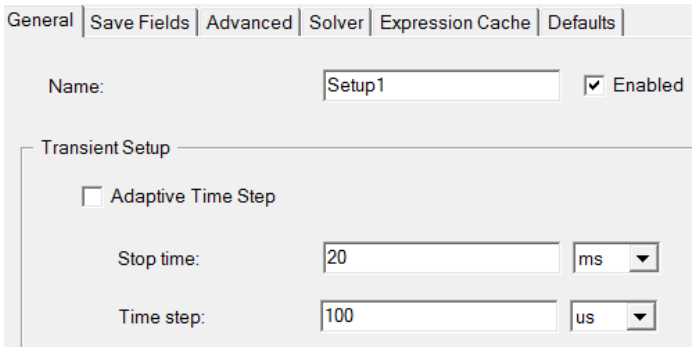
- **To Assign Balloon Boundary**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - Note: On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window,
    - Press **OK**

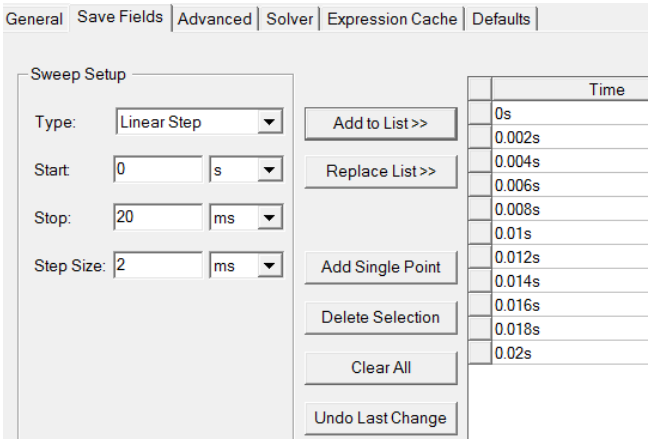
Assign Mesh Operations

- The transient solver does not use the automatic adaptive meshing process, so a manual mesh needs to be created.
- **To Assign Mesh Operation for Core**
  - Select the object **Core** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Core_Inside**
    - Restrict Length Of Elements: ☐**Unchecked**
    - Restrict Number of Elements: ☑**Checked**
    - Maximum Number of Elements: **250**
    - Press **OK**
- **To Assign Mesh Operation for Coil**
  - Select the object **Coil** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Coil_Inside**
    - Restrict Length Of Elements: ☐**Unchecked**
    - Restrict Number of Elements: ☑**Checked**
    - Maximum Number of Elements: **100**
    - 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: **20 ms**
      - Time Step: **100 us**
    - **Save Fields** Tab
      - Type: **Linear Count**
      - Start: **0 sec**
      - Stop: **20 ms**
      - Step Size: **2 ms**
      - Click on: **Add to List**
    - Press **OK**



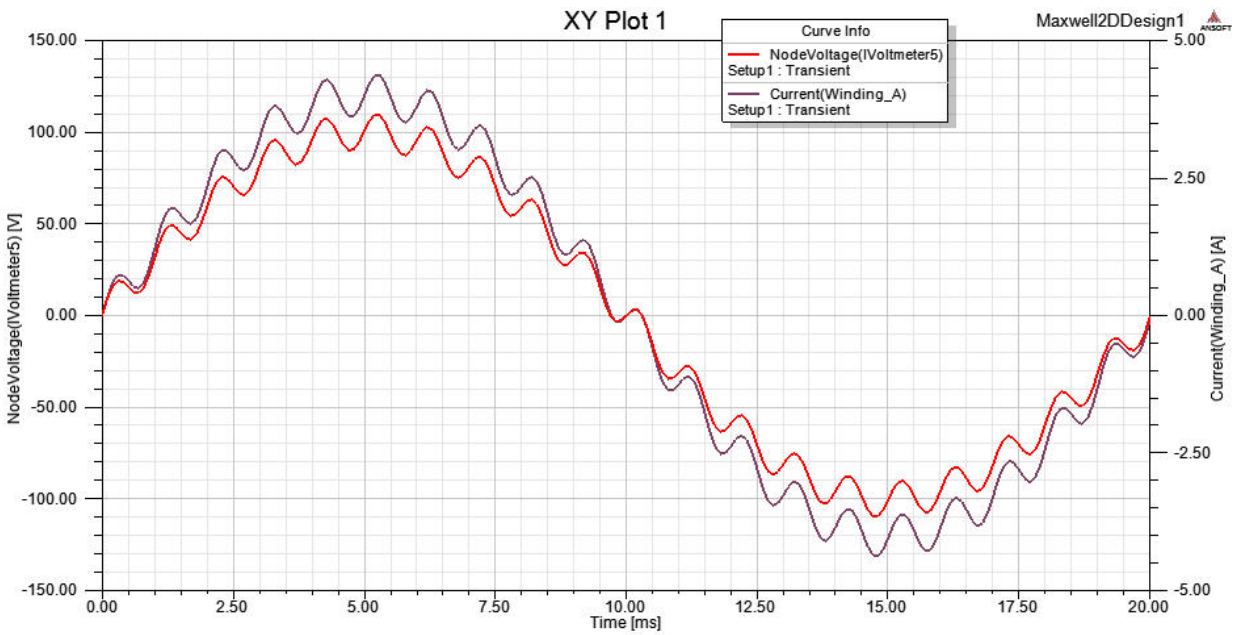


Analyze

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

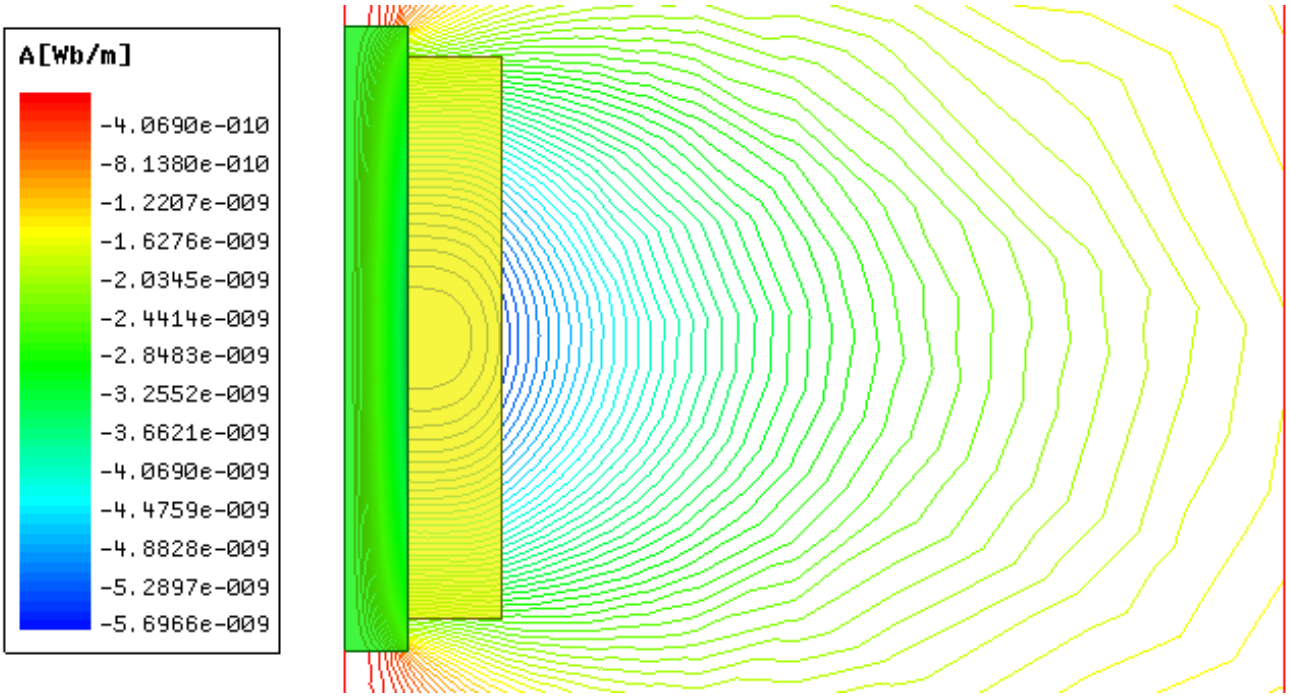
Plot the Voltage and Current

- **To Create a Plot**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular plot:*
  - In Reports window,
    - Select Category: **NodeVoltage**
    - Select Quantity: **NodeVoltage(IVoltmeter)**
    - Click on: **New Report**
    - Select Category: **Winding**
    - Select Quantity: **Current(Winding_A)**
    - Click on: **Add Trace**
    - Click on: **Close**
  - To View Results better, double click on Y Axis corresponding to NodeVoltage and change its range to **-150** to **150 V**



Plot Flux Lines

- **To Plot Flux Lines**
  - Be sure that the 2D Modeler window is in the active view window.
  - Select the menu item *View > Set Solution Context*
  - In Set view Context window,
    - Time: Set to **0.01 sec**
    - Press **OK**
  - Select the menu *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plots window,
    - Press **Done**
- **Modify Plot**
  - Double click on the legend to modify its attributes
  - In the window,
    - **Scale** tab
      - Num. Divisions: **56**
    - Press **Apply** and **Close**



Post Processing in Maxwell 2D

- This exercise will discuss how to use the Maxwell 2D Post Processor. Field plots and calculator operations will demonstrated on an Eddy Current project. The following tasks will be performed:
  - Plot the mesh
  - Plot of MagB
  - Plot of B_vector
  - Plot of MagH along a line
  - Create a table of MagH along a line
  - Calculate average MagB in an object
  - Verify DivB = 0
  - 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 2D Design*, or click on the  icon
  - Change the name of the Design to **Post_exercise**

Set Solution Type

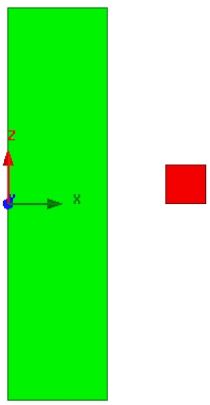
- **To set the Solution Type:**
  - Select the menu item *Maxwell 2D > Solution Type*
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Eddy Current**
    3. Click the **OK** button

Create Core

- **Create Rectangle**
  - Select the menu item *Draw > Rectangle*
    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: **5**, dY: **0**, dZ: **20**, Press the **Enter** key
  - Change the name of the sheet to **Core** and material to **iron**

Create Coil

- **Create Rectangle**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the center of the base
       - X: **8**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **2**, dY: **0**, dZ: **2**, Press the **Enter** key
  - Change the name of the sheet to **Coil** and material to **copper**



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

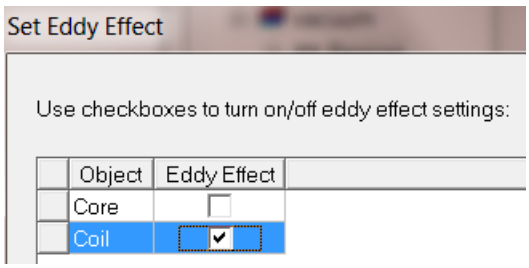
- **To Assign Excitation**
  - Select the sheet **Coil** from the history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **Current1**
    2. Value: **100 A**
    3. Phase: **0**
    4. Type: **Solid** (eddy effects can only be determined in solid sources)
    5. Ref. Direction: **Positive**
    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**

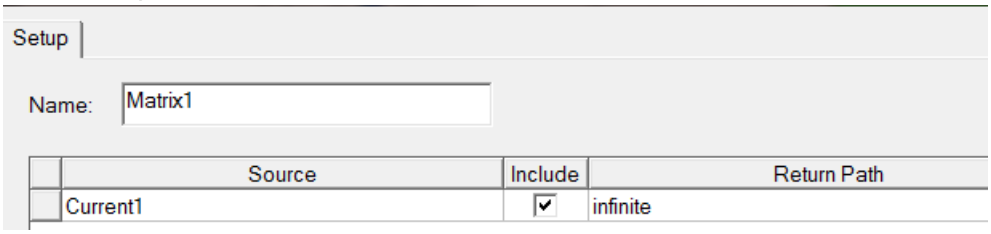
Set Eddy Effect

- **To Set Eddy Calculations**
  - Select the menu item *Maxwell 2D > 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 2D > Parameters > Assign > Matrix*
  - In Matrix window,
    1. Current1:
       - Include: ☑ **Checked**
    2. Press **OK**



Apply Mesh Operations

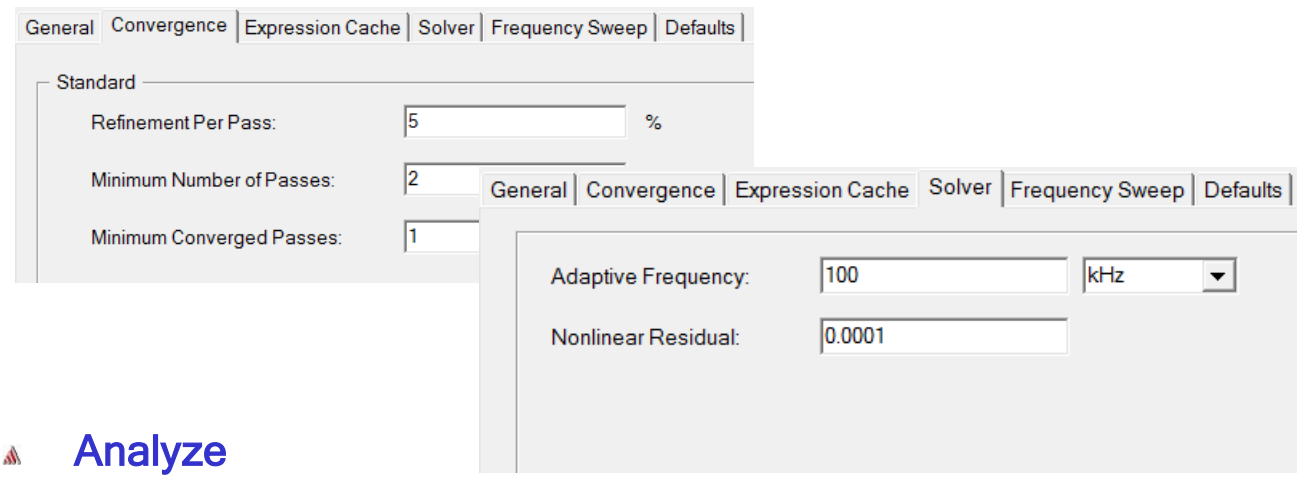
- In order to create a fine initial mesh, several mesh operation will be performed.
- **To Apply Mesh Operations for Core**
  - Select the sheet **Core** from the history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    1. Name: **Length1**
    2. Restrict Length Of Elements: ☐ **Unchecked**
    3. Restrict the Number of Elements: ☑ **Checked**
    4. Maximum Number of Elements: **500**
    5. Press **OK**
- **To Apply Mesh Operations for Coil**
  - Select the sheet **Coil** from the history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside 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: **500**
    5. Press **OK**

Assign Boundary

- **To Assign Boundary**
  - Select the sheet **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - Note: On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window,
    - Press **OK**

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **Convergence** tab
       - Refinement Per Pass: **5 %**
    2. **Solver** tab
       - Adaptive Frequency: **100 kHz**
    3. Click the **OK** button

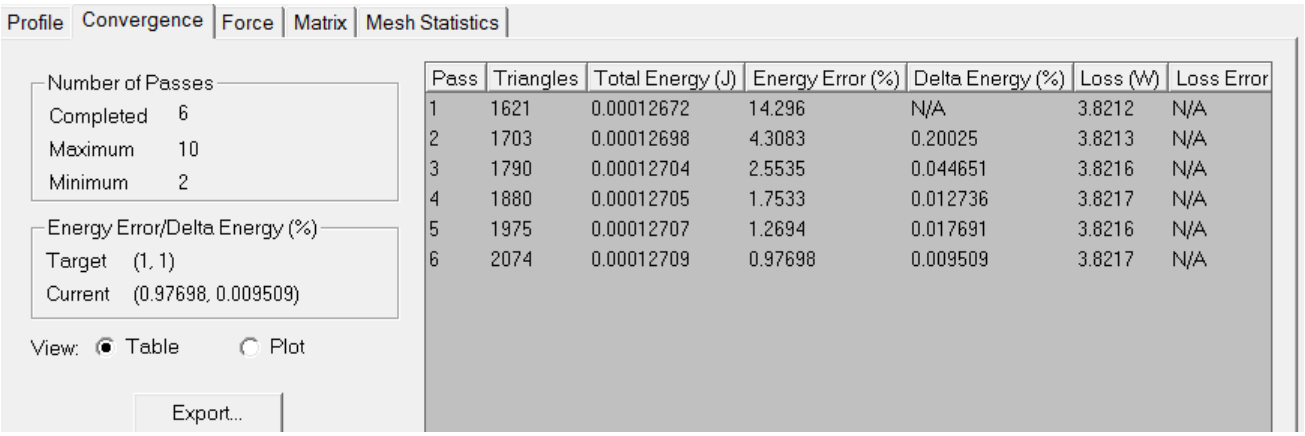


Analyze

- **To run Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

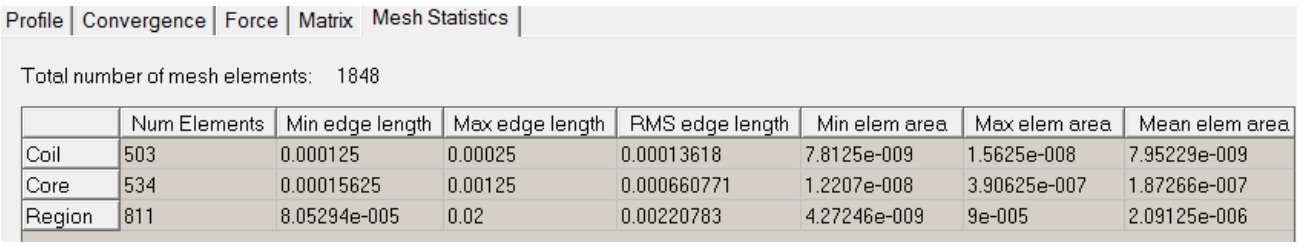
Solution Data

- **To view the Solution Data:**
  - Select the menu item *Maxwell 2D > 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.



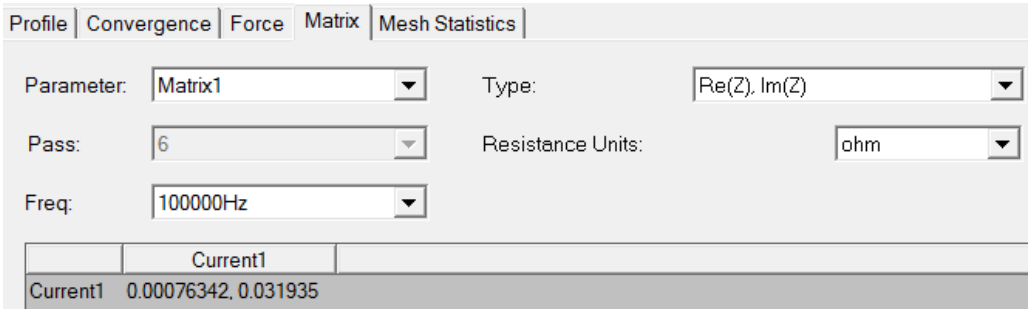
Pass	Triangles	Total Energy (J)	Energy Error (%)	Delta Energy (%)	Loss (W)	Loss Error
1	1621	0.00012672	14.296	N/A	3.8212	N/A
2	1703	0.00012698	4.3083	0.20025	3.8213	N/A
3	1790	0.00012704	2.5535	0.044651	3.8216	N/A
4	1880	0.00012705	1.7533	0.012736	3.8217	N/A
5	1975	0.00012707	1.2694	0.017691	3.8216	N/A
6	2074	0.00012709	0.97698	0.009509	3.8217	N/A

- To View Mesh information
  1. Click **Mesh Statistics** Tab

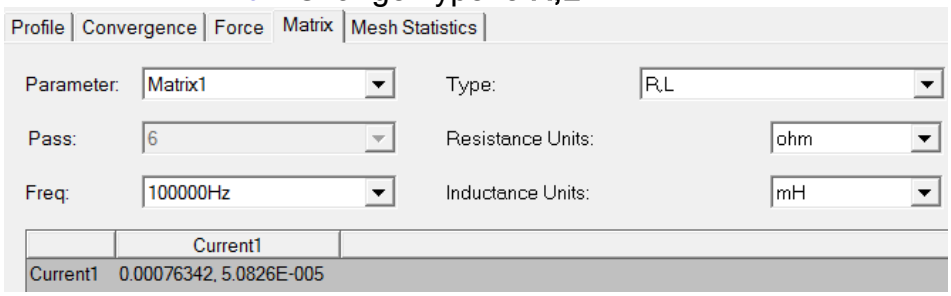


	Num Elements	Min edge length	Max edge length	RMS edge length	Min elem area	Max elem area	Mean elem area
Coil	503	0.000125	0.00025	0.00013618	7.8125e-009	1.5625e-008	7.95229e-009
Core	534	0.00015625	0.00125	0.000660771	1.2207e-008	3.90625e-007	1.87266e-007
Region	811	8.05294e-005	0.02	0.00220783	4.27246e-009	9e-005	2.09125e-006

- To View Impedance values
  1. Click **Matrix** tab
  2. Set Type to **Re(Z), Im(Z)**



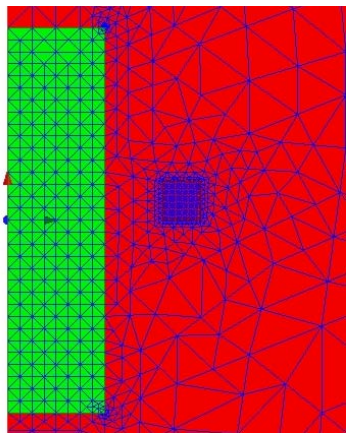
- **Note:** The imaginary term of the matrix includes both the inductive and capacitive reactance are reported in Ohms.
  3. Change Type to **R,L**



- **Note:** The Imaginary term of the matrix includes only the inductance and is reported in Ohms and Henries

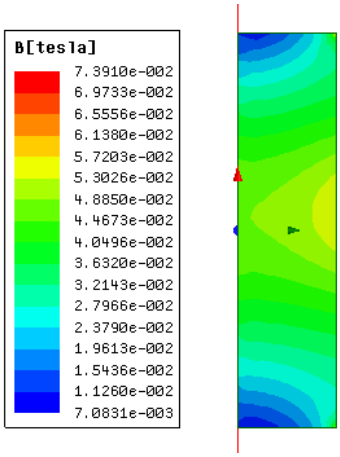
Plot Mesh

- **To Plot Mesh**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Maxwell 2D > Fields > Plot Mesh***
  - In Create Mesh Plot window,
    1. Press **Done**



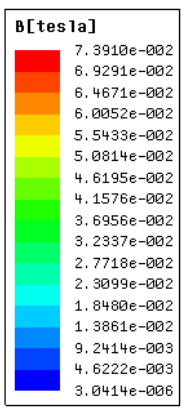
Plot Mag_B on Core

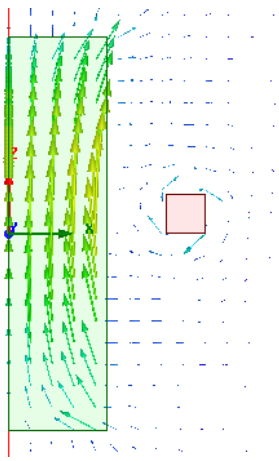
- **To Plot Mag_B**
  - Select the sheet **Core** from the history tree
  - Select the menu item *Maxwell 3D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    1. Press **Done**



Plot Vector_B

- **To Plot Vector_B**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > B > B_Vector*
  - In Create Field Plot window
    1. Press **Done**
- **To Modify Attributes**
  - Double click on the legend to change plot properties
  - In the window
    1. **Plots** tab
       - Plot: **B_Vector1**
       - Select the tab **OnSurface**
       - Vector Plot
         1. Spacing: **Five Spaces**
         2. Min: **0.5**
         3. Max: **2**
    2. Press **Apply** and **Close**





Plot Mag_H along a Line

- To Plot Mag_H
  - Select the menu item *Maxwell 2D > 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**

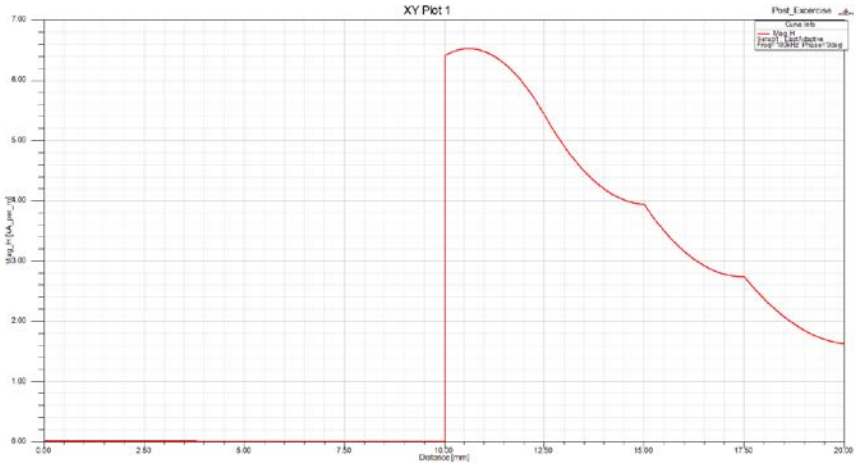


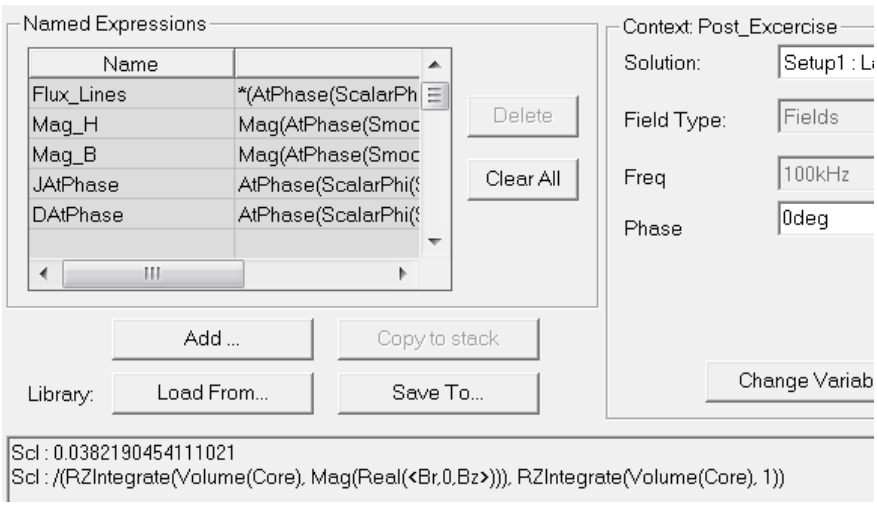
Table of Mag_H along a Line

- Select the menu item *Maxwell 2D > 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_m] Setup1 : LastAdaptive Freq='100kHz' Phase='0deg'
1	0.000000	0.009608
2	0.020000	0.009614
3	0.040000	0.009619
4	0.060000	0.009624
5	0.080000	0.009629
6	0.100000	0.009634
7	0.120000	0.009638
8	0.140000	0.009643
9	0.160000	0.009647
10	0.180000	0.009650
11	0.200000	0.009654
12	0.220000	0.009657
13	0.240000	0.009660
14	0.260000	0.009663

Calculate Average Mag_B in an Object

- **To Calculate Mag_B Over a Volume**
  - Select the menu item *Maxwell 2D > 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 > Integral > RZ**
    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 > Integral > RZ**
    9. Select **General > /** (Divide)
    10. Select **Output > Eval**
  - The average value of flux density calculated is around **0.038 Tesla**



Calculate DivB = 0

- To Calculate DivB
  - Select the menu item *Maxwell 2D > 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 > Integral > RZ**
    6. Select **Output > Eval**
  - The divergence of B is approximately zero

```
Scl : 1.57559485345854E-009
Scl : RZIntegrate(Volume(Core), Divg(Real(<Br,0,Bz>)))
```

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 > Integral > RZ**
    4. Select **Output > Eval**
  - The ohmic loss in the coil at 60Hz is approximately 3.82 Watts.

```
Scl : 3.82168490822298
Scl : RZIntegrate(Volume(Coil), Ohmic-Loss)
```

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 **Vector > Scal? > ScalarPhi**
  4. Select **Input > Geometry**
     - Surface: ☑ **Checked**
     - Select **Coil** from the list
     - Press **OK**
  5. Select **Scalar > Integral >XY**
  6. Select **Output > Eval**
- **Note:** Since total current needs to be calculated over the 2D section of Coil, Integration is done with XY integral and not RZ
- The net current in the coil is equal to the source amp-turns = 100

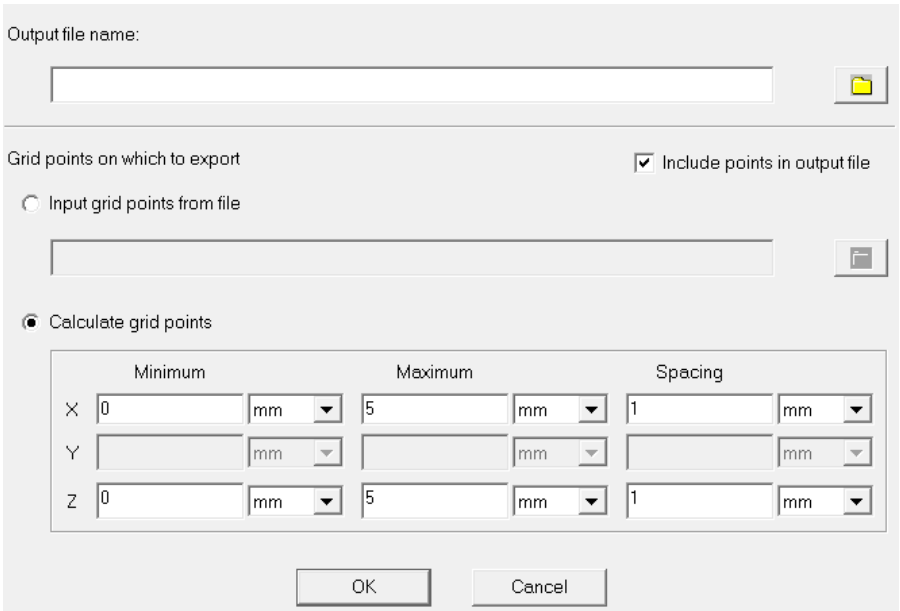
Calculate Total Current including Eddy Component

- To Calculate Net Current in Coil
  1. Select **Input > Quantity > J**
  2. Select **General > Complex > Cmplxmag**
  3. Select **Vector > Scal? > ScalarPhi**
  4. Select **Input > Geometry**
     - Surface: ☑ **Checked**
     - Select **Coil** from the list
     - Press **OK**
  5. Select **Scalar > Integral >XY**
  6. Select **Output > Eval**
- Calculated value of current comes out to be around 130 A

Export field results to a file

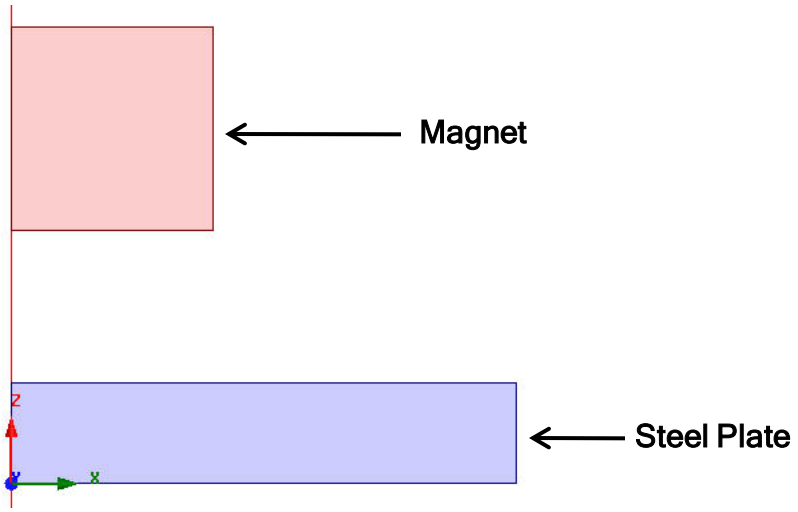
- In order to use the field results in another software program, you can export the fields on a uniform 2-dimensional grid.
- **To Export Results**
  1. Select **Input > Quantity > H**
  2. Select **Output > Export**
  3. In Export Solution window,
     - Output file name: **H_Vector**
     - Calculate grid points: ☑ **Checked**
     - Minimum: **(X:0, Z:0)**
     - Maximum: **(X:5, Z:5)**
     - Spacing: **(X:1, Z: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 2D RZ Magnetostatic solver and Optimetrics in the ANSYS Maxwell 2D Design Environment.
- The optimization obtains the desired force = 0.25N by varying the air gap between the plate and the puck using a local variable.



Create Design

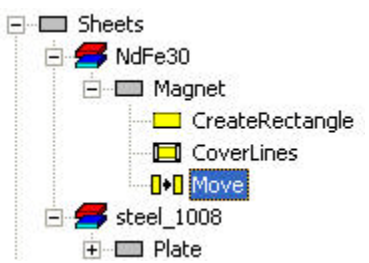
- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

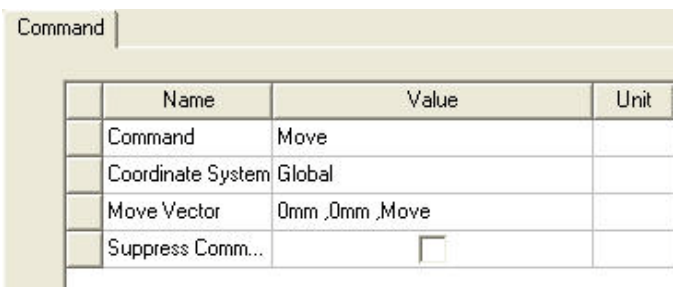
Set Solution Type

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

Create Model

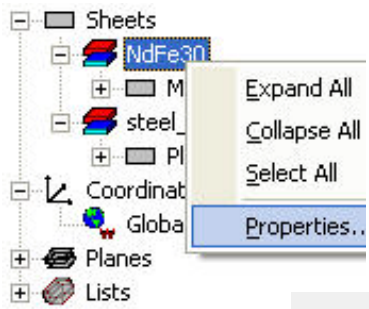
- **Create Plate**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **5**, dY: **0**, dZ: **1**, Press the **Enter** key
  - Change the name of resulting sheet to **Plate** and color to **Blue**
  - Change the material of the sheet to **Steel 1008**
- **Create Magnet**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **0**, dZ: **2**, Press the **Enter** key
  - Change the name of resulting sheet to **Magnet** and color to **Red**
  - Change the material of the sheet to **NdFe30**
- **Parameterize Magnet**
  - Select the object **Magnet** from history tree
  - Select the menu item *Edit > Arrange > Move*
    1. Using the coordinate entry fields, enter the reference point
       - X: **0**, Y: **0**, Z: **2**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the target point
       - dX: **0**, dY: **0**, dZ: **0**, Press the **Enter** key
  - Expand the history tree for the object **Magnet**
  - Double click on the command **Move** from the tree
  - In Properties window,
    - Change Move Vector to **0, 0, Move**

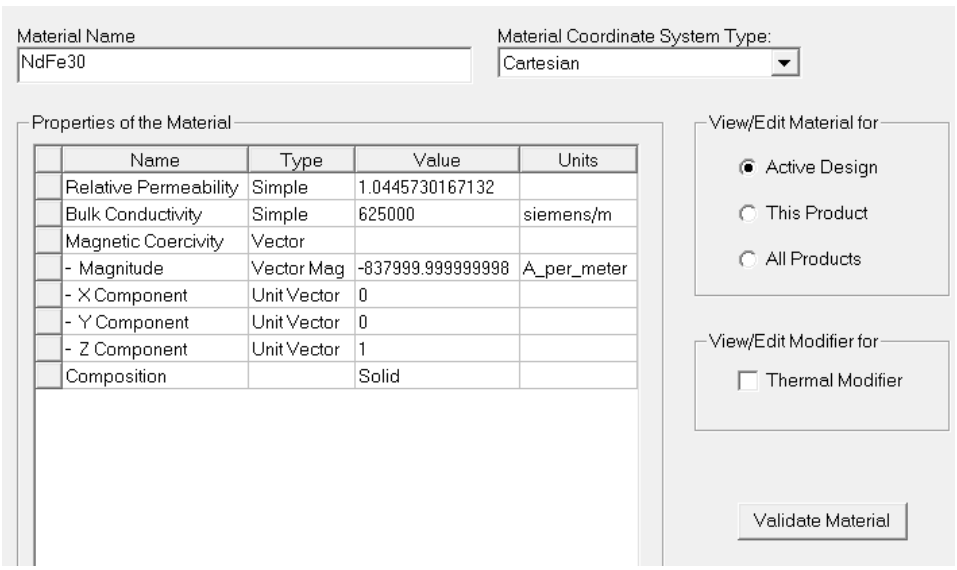




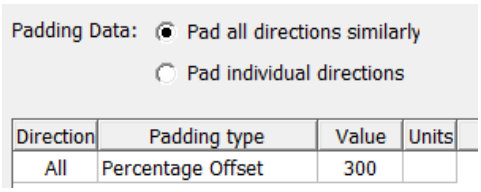
Modify Magnetization Direction for Puck

- The default magnetization direction for NdFe30 is in the X-direction. Since magnetization in the Z-direction is desired for this example, material properties will be changed accordingly
- **To Modify Magnetization Direction**
  - Right click on the material **NdFe30** from history tree and select **Properties**
  - In Select Definition window,
    - Ensure that material **NdFe30** is selected
    - Select the button **View/Edit Material**
    - In View/Edit Material window,
      - Change X Component to **0**
      - Change Z Component to **1**
      - Select **Validate Material** and Press **OK**
    - Select **OK** to close Select Definition window





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:** Region will not be created in Negative X direction due to RZ-symmetry about the Z-axis.

Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - **Note:** On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window, Press **OK**

Assign Force Calculation

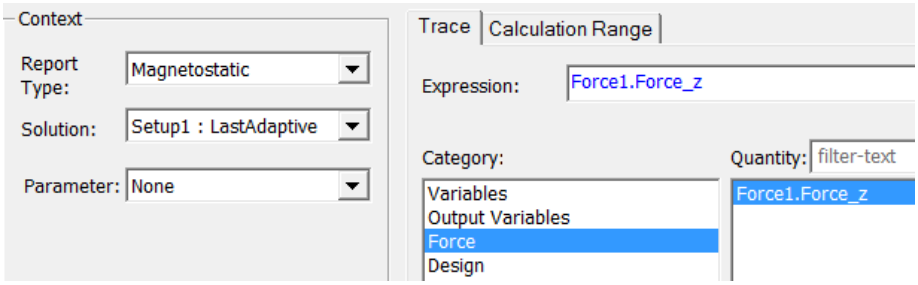
- **To Assign Force Parameter**
  - Select the object **Plate** from history tree
  - Select the menu item *Maxwell 2D > Parameters > Assign > Force*
  - In Force Setup window,
    - Name: **Force1**
    - Press **OK**

Analysis Setup

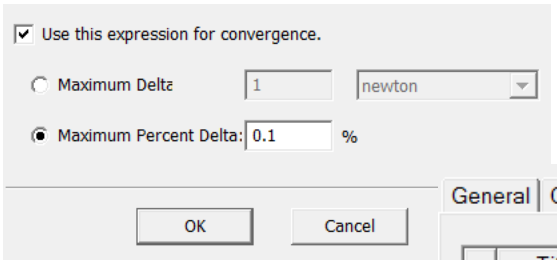
- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,
    - Press **OK**
  - **Note:** We have created Analysis Setup to enable Expression Cache tab. Parameters will be edited in next step

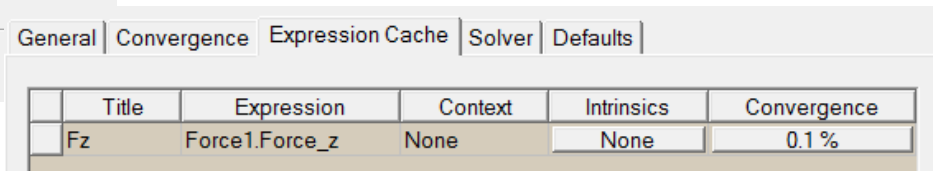
Modify Analysis Setup

- **To Modify Analysis Setup**
  - Expand the Project Manager tree to view Analysis
  - Double click on **Setup1** to edit its parameters
  - In Solve Setup window,
    - **General** tab
      - Maximum Number of Passes: **15**
      - Percentage Error: **0.1**
    - **Expression Cache** tab
      - Select **Add**
      - In Add to Expression Cache window,
        - Category: **Force**
        - Quantity: **Force1.Force_z**
        - Select **Add to Calculation** and **Done**



      - Change Title to **Fz**
      - Select the tab in **Convergence** column
      - In Adaptive Convergence window,
        - Use this expression for convergence: ☑ **Checked**
        - Maximum Percentage Delta: **0.1**
        - Press **OK**
      - Press **OK** to close Solve Setup window



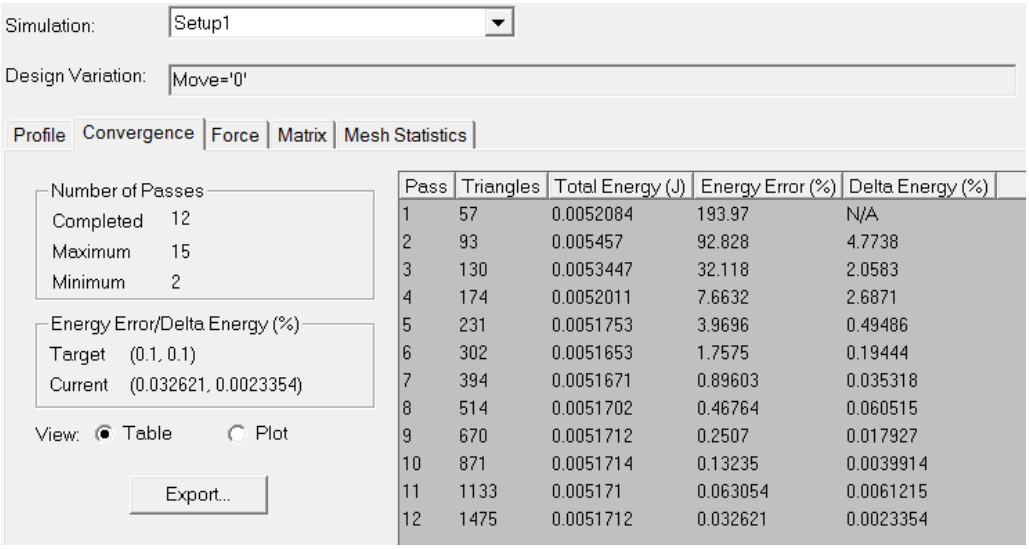


Analyze

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

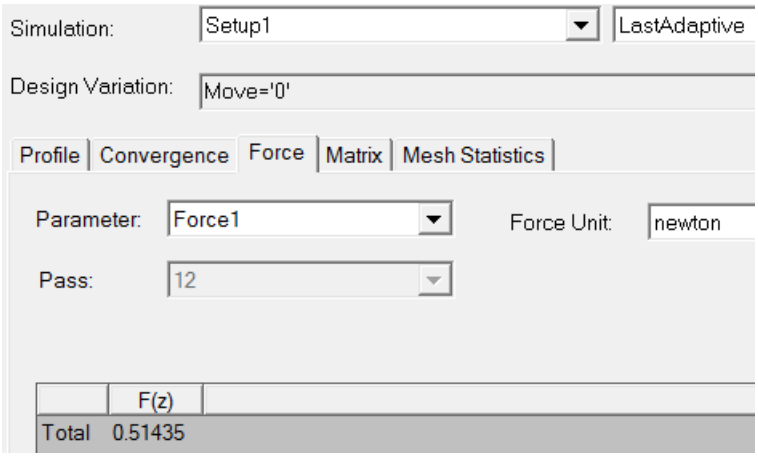
Solution Data

- **To View Solution Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
    - To View Convergence
      - Select **Convergence** tab



Pass	Triangles	Total Energy (J)	Energy Error (%)	Delta Energy (%)
1	57	0.0052084	193.97	N/A
2	93	0.005457	92.828	4.7738
3	130	0.0053447	32.118	2.0583
4	174	0.0052011	7.6632	2.6871
5	231	0.0051753	3.9696	0.49486
6	302	0.0051653	1.7575	0.19444
7	394	0.0051671	0.89603	0.035318
8	514	0.0051702	0.46764	0.060515
9	670	0.0051712	0.2507	0.017927
10	871	0.0051714	0.13235	0.0039914
11	1133	0.005171	0.063054	0.0061215
12	1475	0.0051712	0.032621	0.0023354

    - To View Force Values
      - Select the **Force** tab

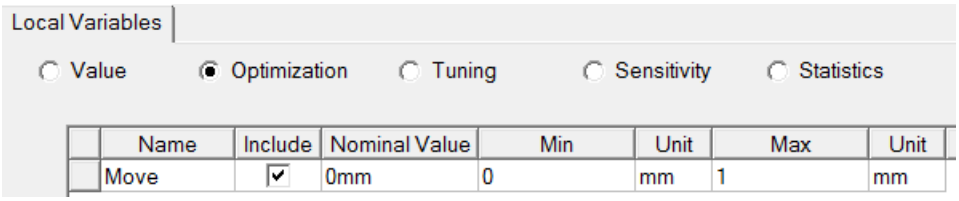


Optimetrics Setup and Solution

- 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.

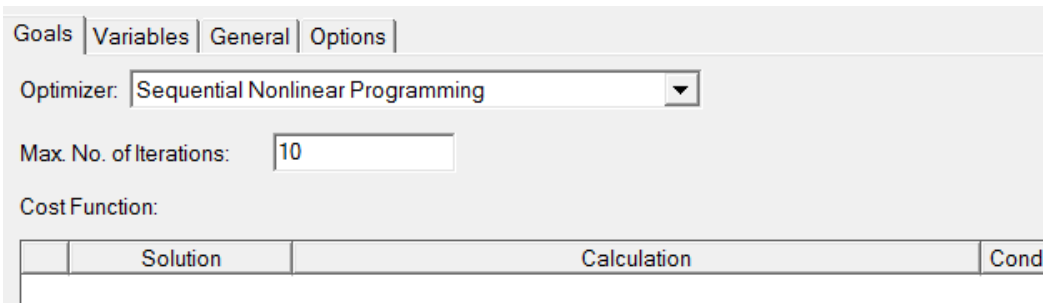
Specify the Optimization Variables

- Before starting the optimization setup, the appropriate variables must be included in the optimization.
- **To Specify Optimization Variables**
  - Select the menu item *Maxwell 2D > Design Properties*
  - In Properties window,
    - Select **Optimization**
    - Select the check box to Include the parameter **Move**
    - Min: **0 mm**
    - Max: **1 mm**
    - Press **OK**

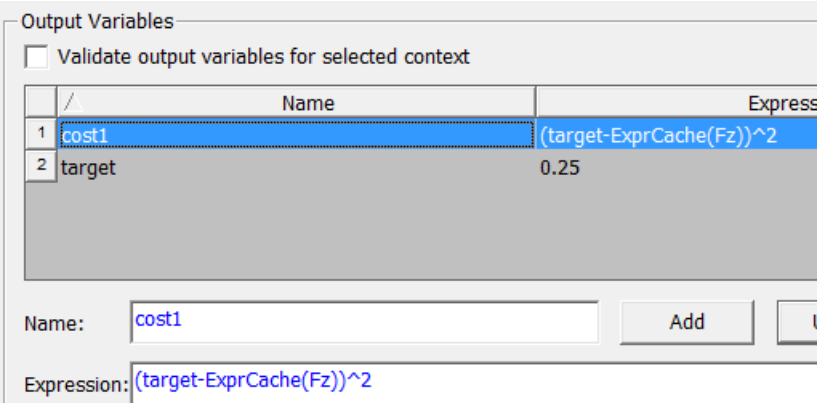


Setup Optimization Analysis

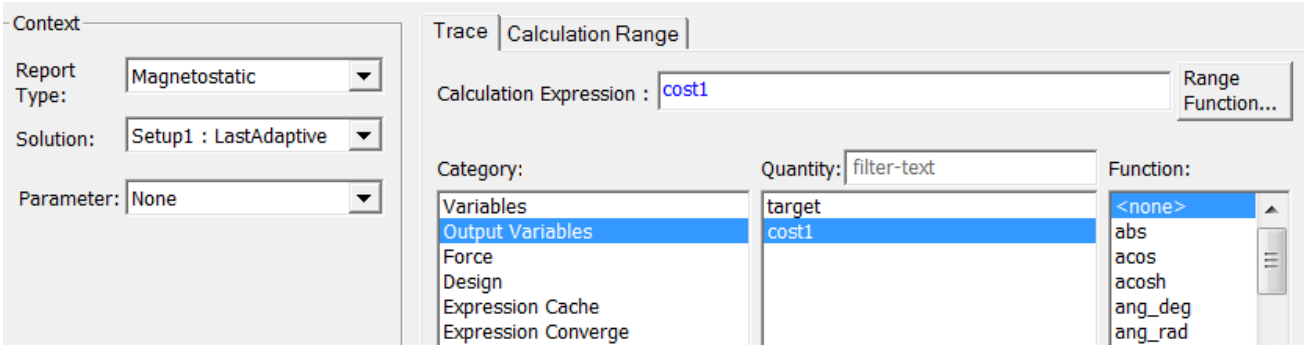
- **To Define Optimization Analysis**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Add Optimization*
  - In Setup Optimization window,
    - Optimizer: **Sequential Nonlinear Programming**
    - Max. No. of Iterations: **10**



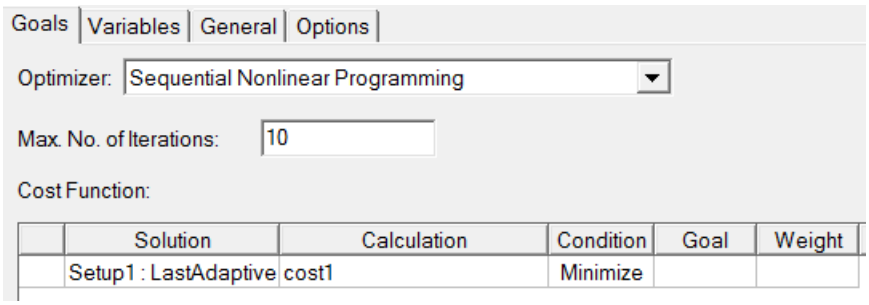
- Select the button **Setup Calculations**
- In Add/Edit Calculations window, select **Output Variables**
- In Output Variables window,
  - Name: **target**
  - Expression: **0.25**
  - Select **Add**
  - Name: **cost1**
  - Expression: **(target - ExprCache(Fz))^2**
  - Select **Add**
  - Select **Done** to close Output variables window



- In Add/Edit Calculations window
  - Category: **Output Variables**
  - Quantity: **cost1**
  - Select **Add Calculation**
  - Press **Done**



- In Setup Optimization window,
  - Condition: **Minimize**
  - Press **OK**



Solve the Optimization Analysis

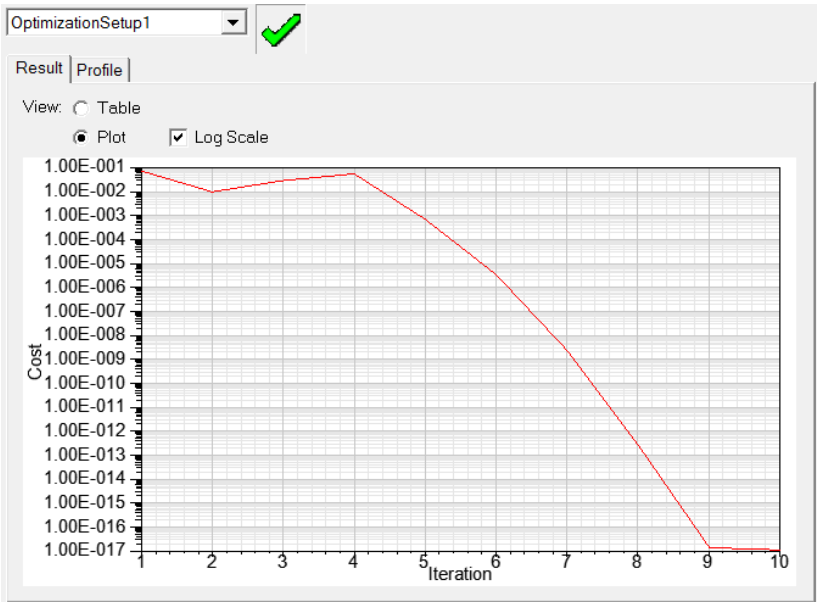
To Run Optimization Analysis

- Expand the Project Manager tree to view **Optimetrics**
- Right click on the tab **Optimization Setup1** and select **Analyze**

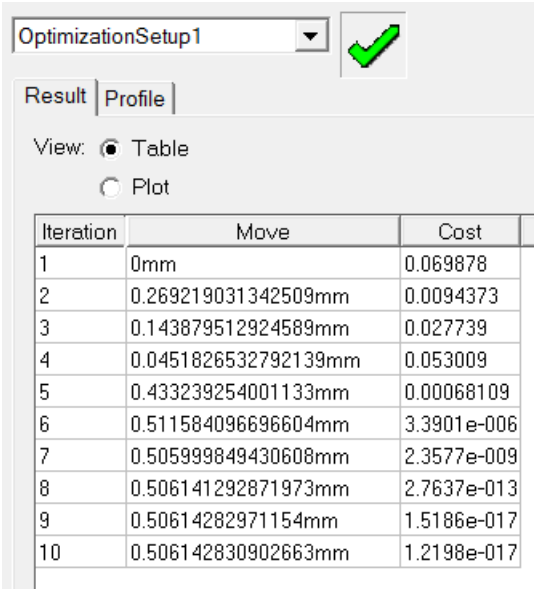
View Optimization Results

To View Optimization Results

- Select the menu item *Maxwell 2D > Optimetrics Analysis > Optimetrics Results*
- In Post Analysis Display window,
  - Set Plot to **Log Scale**



- Change View Type to **Table** to display results in tabular form

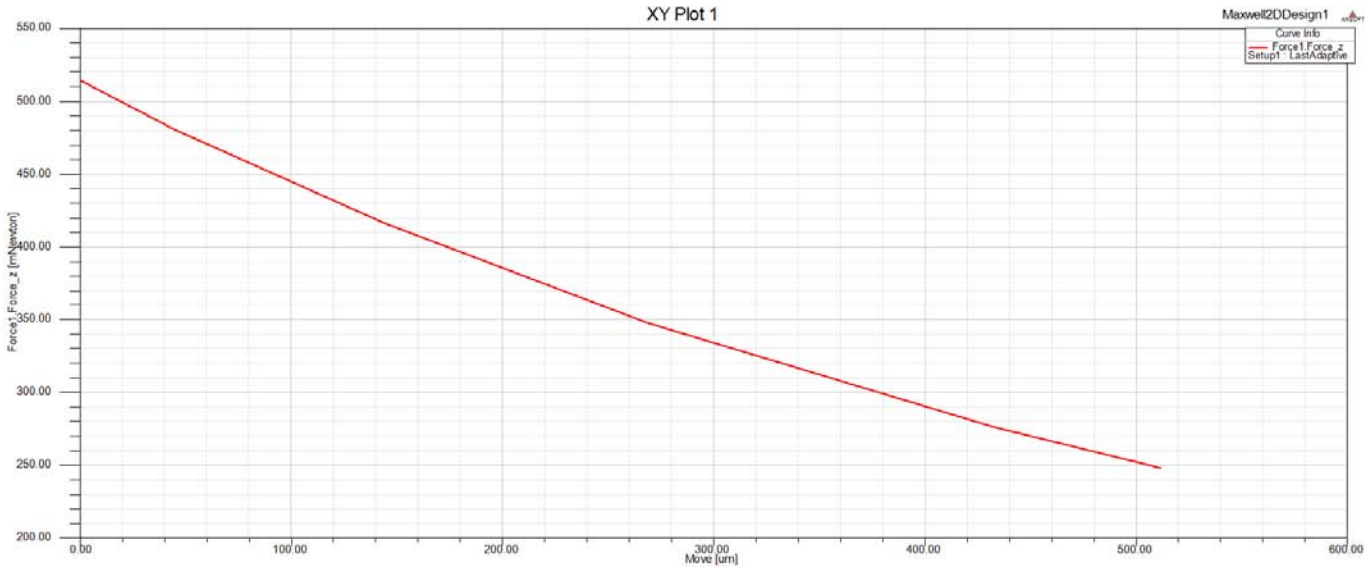


Iteration	Move	Cost
1	0mm	0.069878
2	0.269219031342509mm	0.0094373
3	0.143879512924589mm	0.027739
4	0.0451826532792139mm	0.053009
5	0.433239254001133mm	0.00068109
6	0.511584096696604mm	3.3901e-006
7	0.505999849430608mm	2.3577e-009
8	0.506141292871973mm	2.7637e-013
9	0.50614282971154mm	1.5186e-017
10	0.506142830902663mm	1.2198e-017

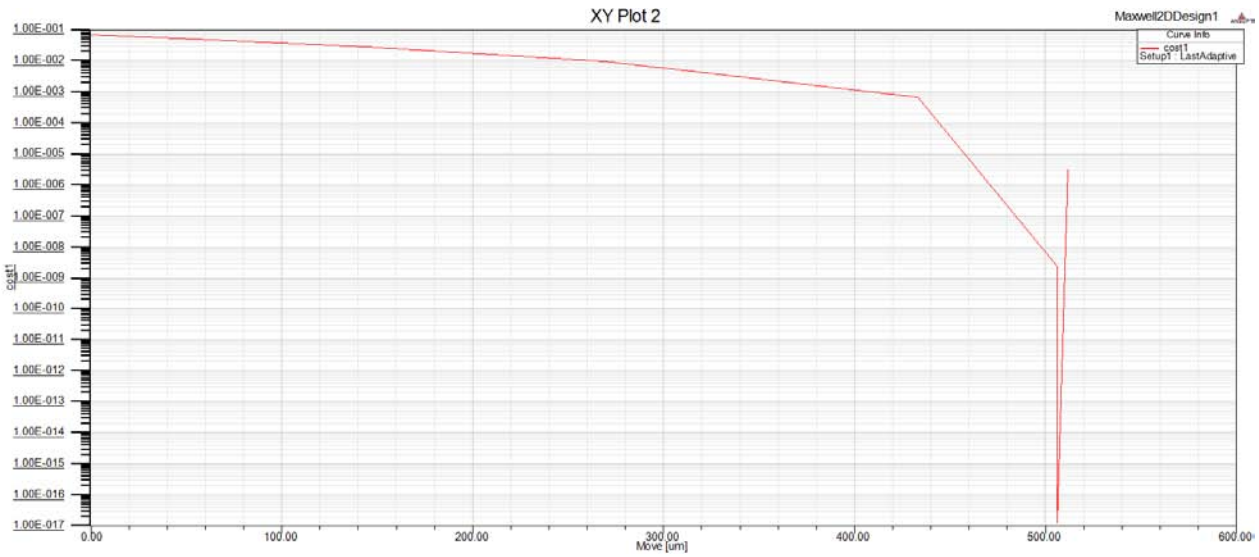
Create Field Plots

To Plot Move Vs Force

- Select the menu item *Maxwell 2D > Results > Create Magnetostatic Report > Rectangular Plot*
- In Report window,
  - Category: **Force**
  - Quantity: **Force1.Force_z**
  - Select **New Report**

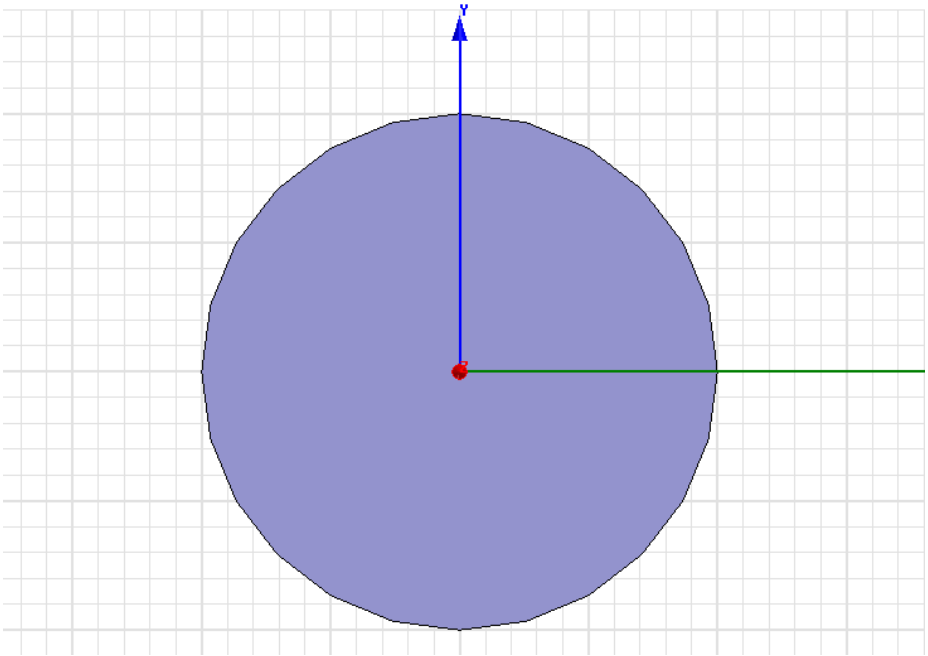


- **To Plot Move Vs Cost**
  - Select the menu item *Maxwell 2D > Results > Create Magnetostatic Report > Rectangular Plot*
  - In Report window,
    - Category: **Output Variables**
    - Quantity: **Cost1**
    - Select **New Report**
  - Double click on Y Axis to modify its properties
  - In Properties window,
    - **Scaling** tab
      - Axis Scaling: **Log**
      - Press **OK**



Scripting the Creation of a Model Object

- This exercise will discuss how to record, modify and run a script for automating generation of a circle. The following tasks will be performed:
  - Record a script in which a circle is created.
  - Modify the script to change the circle's radius and height.
  - Run the modified script.



Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

Save Project

- **To Save Project**
  - Select the menu item *File > Save*
  - Save the file with the name **"Ex_9_10_BasicScripting2D.mxwl"**

Record Script

- **To Start Script Recording**
  - Select the menu item *Tools > Record Script To File*
  - Save the script file with the name **"Ex_9_10_BasicScripting2D.vbs"**

Draw Circle

- **To Draw a Circle**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - 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

Stop Script Recording

- **To Stop the Script Recording**
  - Select the menu item *Tools > Stop Script Recording*

Delete Circle

- **To Delete the Circle**
  - Select the sheet **Circle1** from history tree
  - Select the menu item *Edit > Delete*

Run the Script

- **To Run Recorded Script**
  - Select the menu item *Tools > Run Script*
  - Browse to the location of the script file and **Open it**
  - If successful, the original circle, **Circle1**, should be back.
  - We can now explore the contents of the script file.

Script Content

To open the Script in Editor

- Locate the file on the Hard Disk and open it with Text Editor (say Textpad)

```
' ----------------------------------------------
' Script Recorded by Ansoft Maxwell Version 15.0.0
' 10:23:17 PM  Dec 18, 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_BasicScripting2D")
Set oDesign = oProject.SetActiveDesign("Maxwell2DDesign1")
Set oEditor = oDesign.SetActiveEditor("3D Modeler")
oEditor.CreateCircle Array("NAME:CircleParameters", "IsCovered:=", true, "XCenter:=", _ "0mm", "YCenter:=", "0mm", "ZCenter:=", "0mm", "Radius:=", "1mm", "WhichAxis:=", _ "Z", "NumSegments:=", "0"), Array("NAME:Attributes", "Name:=", "Circle1", "Flags:=", _ "", "Color:=", "(132 132 193)", "Transparency:=", 0.25, "PartCoordinateSystem:=", _ "Global", "UDMId:=", "", "MaterialValue:=", "" & Chr(34) & "vacuum" & Chr(34) & "", "SolveInside:=", _ true)
```

- **Definition of environment variables**
  - **Dim** is the generic visual basic variable type.

```
' ----------------------------------------------
' Script Recorded by Ansoft Maxwell Version 15.0.0
' 10:23:17 PM  Dec 18, 2011
' ----------------------------------------------
Dim oAnsoftApp
Dim oDesktop
Dim oProject
Dim oDesign
Dim oEditor
Dim oModule
```

- **Reference defined environment variables**
  - Defined using **Set**.

```
Set oAnsoftApp = CreateObject("AnsoftMaxwell.MaxwellScriptInterface")
Set oDesktop = oAnsoftApp.GetAppDesktop()
oDesktop.RestoreWindow
Set oProject = oDesktop.SetActiveProject("Ex_9_10_BasicScripting2D")
Set oDesign = oProject.SetActiveDesign("Maxwell2DDesign1")
Set oEditor = oDesign.SetActiveEditor("3D Modeler")
```

- **oAnsoftApp**

  The oAnsoftApp object provides a handle for VBScript to access Maxwell
- **oDesktop**

  The oDesktop object is used to perform desktop-level operations, including project management.
- **oProject**

  The oProject object corresponds to one project open in the product. It is used to manipulate the project and its data.
- **oDesign**

  The oDesign object corresponds to an instance of a design in the project.
- **oEditor**

  The oEditor object corresponds to an editor, such as the 3D Modeler. This object is used to add and modify data in the editor.

- **Circle Creation**
  - All of the parameters needed to create the circle are defined in this line of code. Here we will modify the **Radius** of the circle by changing the appropriate text.

```
oEditor.CreateCircle Array("NAME:CircleParameters", "IsCovered:=", true,
"XCenter:=", _  "0mm", "YCenter:=", "0mm", "ZCenter:=", "0mm",
"Radius:=", "1mm", "WhichAxis:=", _  "Z", "NumSegments:=", "0"),
Array("NAME:Attributes", "Name:=", "Circle1", "Flags:=", _  "", "Color:=",
"(132 132 193)", "Transparency:=", 0.800000011920929,
"PartCoordinateSystem:=", _  "Global", "UDMId:=", "", "MaterialValue:=", ""
& Chr(34) & "vacuum" & Chr(34) & "", "SolveInside:=", _  true)
```

- **Modify Script**
  - Locate the line containing the **Radius** and change the numerical values to **5mm**:

```
"Radius:=", "1mm", "WhichAxis:=", _  "Z",
"Radius:=", "5mm", "WhichAxis:=", _  "Z",
```

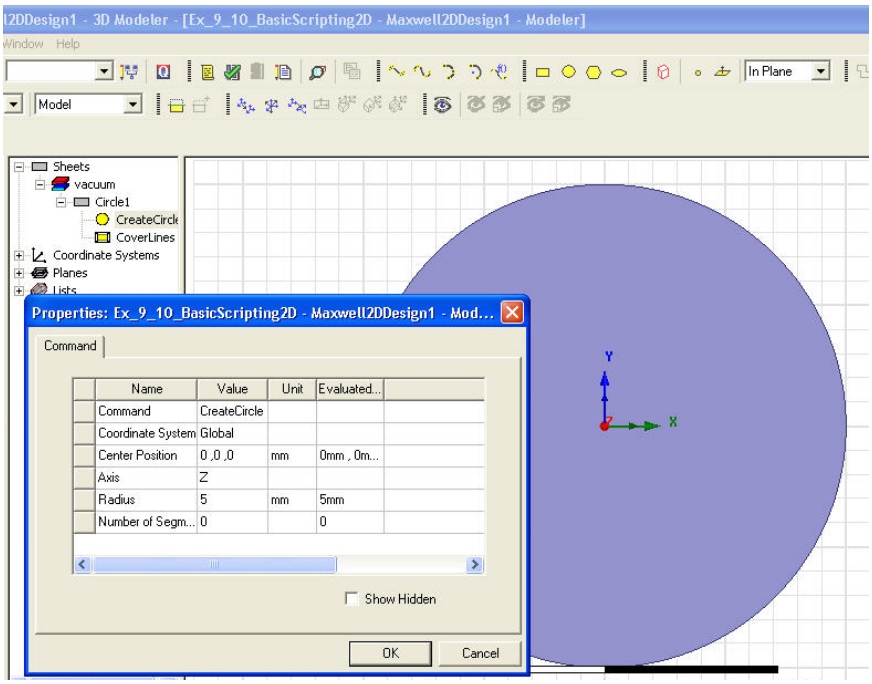
  - Save the file and return to Maxwell.

Delete Circle

- **To Delete the Circle**
  - Select the sheet **Circle1** from history tree
  - Select the menu item ***Edit > Delete***

Run the Script

- **To Run Recorded Script**
  - Select the menu item *Tools > Run Script*
  - Browse to the location of the script file and **Open it**
  - If successful, the original circle, **Circle1**, should be back.



Generalize the script to run in any Project and Design

- To run the script in order to create your circle in a different project. Change the following lines in the script.

```
Set oProject = oDesktop.SetActiveProject("scripting_example")
Set oDesign = oProject.SetActiveDesign("MaxwellDesign1")

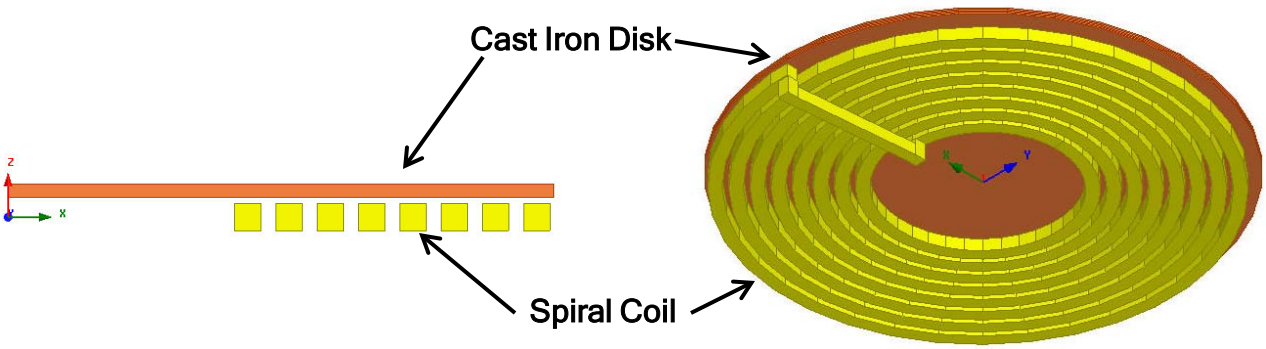
Set oProject = oDesktop.GetActiveProject()
Set oDesign = oProject.GetActiveDesign()
```

Introduction to the Eddy Current Solver

- This example introduces the Eddy Current solver based on a simple example with a disk above a coil. This solver calculates the magnetic fields at a specified sinusoidal frequency. Both linear and nonlinear (for saturation effects) magnetic materials can be used. Also, eddy, skin and proximity effects are considered.

2D Geometry: Iron Disk above a Spiral Coil

- A sinusoidal 500 Hz current will be assigned to an eight turn spiral coil underneath of a cast iron disk. The coil induces eddy currents and losses in plate. The 2D model will be setup as shown below using the 2D RZ axisymmetric solver.



Simulated 2D model

Actual 3D model

Create Design

- **To Create Design**
  - Select the menu item ***Project > Insert Maxwell 2D Design***, or click on the icon

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - Solution Type Window:
    1. Geometry Mode: **Cylindrical about Z**
    2. Choose **Magnetic > Eddy Current**
    3. Click the **OK** button

Set Default Units

- **To Set Units**
  - Select the menu item *Modeler > Units*
    - Set units to **cm (centimeters)** and press **OK**

Draw Solution Region

- **To Create Solution Region**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **-100**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **120**, dY: **0**, dZ: **200**, Press the **Enter** key
  - Change the name of resulting sheet to **Region**

Create Model

- **Create Coil**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **17**, Y: **0**, Z: **-1**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **2**, dY: **0**, dZ: **2**, Press the **Enter** key
  - Change the name of resulting sheet to **Coil** and color to **Yellow**
  - Change the material of the object to **Copper**
- **Duplicate Coil**
  - Select the sheet **Coil** from 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: **3.1**, dY: **0**, dZ: **0**, Press the **Enter** key
    - Total Number: **8**
    - Press **OK**

- **To Create Plate**
  - Select the menu item *Draw > Rectangle*
    1. Using the coordinate entry fields, enter the position of rectangle
       - X: **0**, Y: **0**, Z: **1.5**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the opposite corner
       - dX: **41**, dY: **0**, dZ: **1**, Press the **Enter** key
  - Change the name of resulting sheet to **Plate** and color to **Orange**
  - Change the material of the object to **cast_iron**

Assign Excitations

- A current of 125A will be assigned to each coil. This will result in a total of 1000 A-turns being assigned to the complete winding.
- **To Assign Excitation**
  - Press **Ctrl** and select all Coils from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    - Base Name: **Current**
    - Value: **125 A**
    - Type: **Solid**
    - Ref. Direction: **Positive**
    - Press **OK**
  - Note: Choosing **Solid** specifies that the eddy effects in the coil will be considered. On the other hand, if **Stranded** had been chosen, only the DC resistance would have been calculated and no AC effects in the coil would have been considered. **Stranded** is appropriate when the skin depth is much larger than the stranded conductor thickness, for example when using Litz wire. Note that the induced eddy effects in the plate will be calculated in either case.

Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - Note: On symmetry axis, "Balloon Boundary" assignment is automatically skipped, This can also be achieved by selecting the edges of region which are not on symmetry axis.
  - In Balloon Boundary window,
    - Press **OK**

Assign Matrix Parameters

- In this example, the compete [8x8] impedance matrix will be calculated. This is done by setting a parameter.
- **To Calculate Impedance Matrix**
  - Select the menu item *Maxwell 2D > Parameters > Assign > Matrix*
  - In Matrix window,
    - For all current Sources
      - Include: ☑ **Checked**
    - Press **OK**

Compute the Skin Depth

- Skin depth is a measure of how current density concentrates at the surface of a conductor carrying an alternating current. It is a function of the permeability, conductivity and frequency
- Skin depth in meters is defined as follows:

$$\delta = \sqrt{\frac{2}{\omega \mu_o \mu_r \sigma}}$$

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 $4\pi \times 10^{-7}$ A/m.

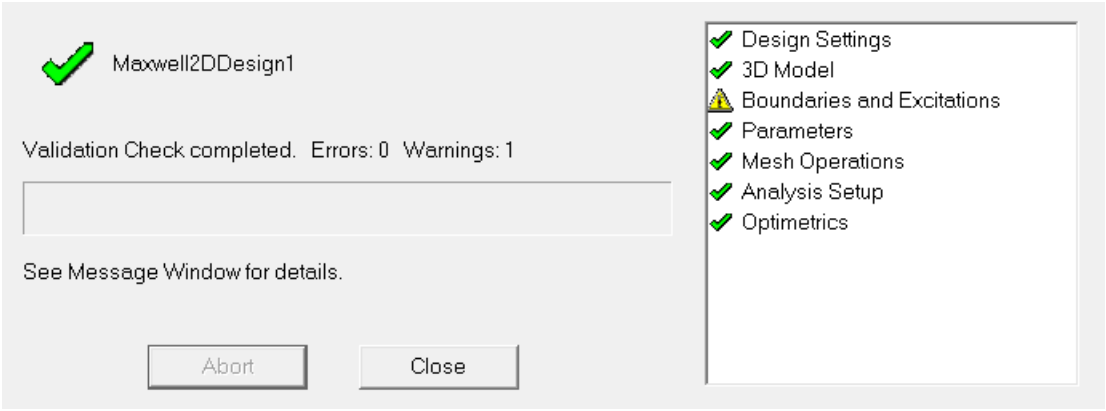
- For cast iron the plate the skin depth is approximately 0.24 cm.
- After three skin depths, the induced current will become almost negligible. The automatic adaptive meshing in Maxwell 2D does an excellent job of refining the mesh in the skin depth, so that mesh operations are not needed.

Analysis Setup

- **To create an analysis setup:**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - Solution Setup Window:
    1. **General** Tab
       - Maximum Number of Passes: **15**
    2. **Solver** Tab
       - Adaptive Frequency: **500 Hz**
    3. Click the **OK** button

Validation Check

- **To Validate the Case**
  - Select the menu item *Maxwell 2D > Validation Check*
  - You will get a warning about Boundaries and Excitations. To clear this warning, simply return to the eddy effect screen by choosing: *Maxwell 2D > Excitations > Set Eddy Effects > OK*. This tells the solver that you have checked the eddy setup and that you have correctly set the eddy effect on the appropriate objects.



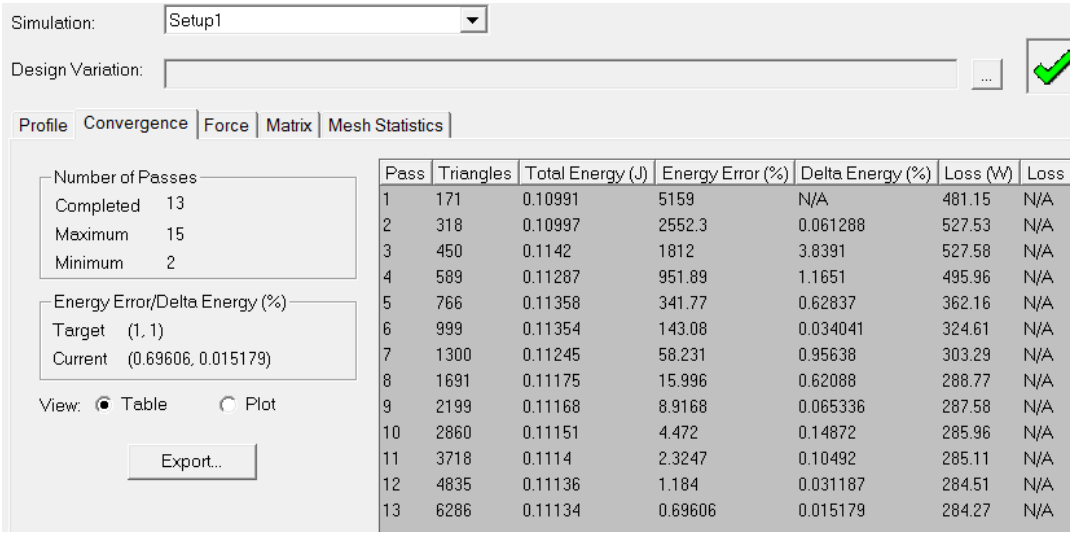
Analyze

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

Solution Data

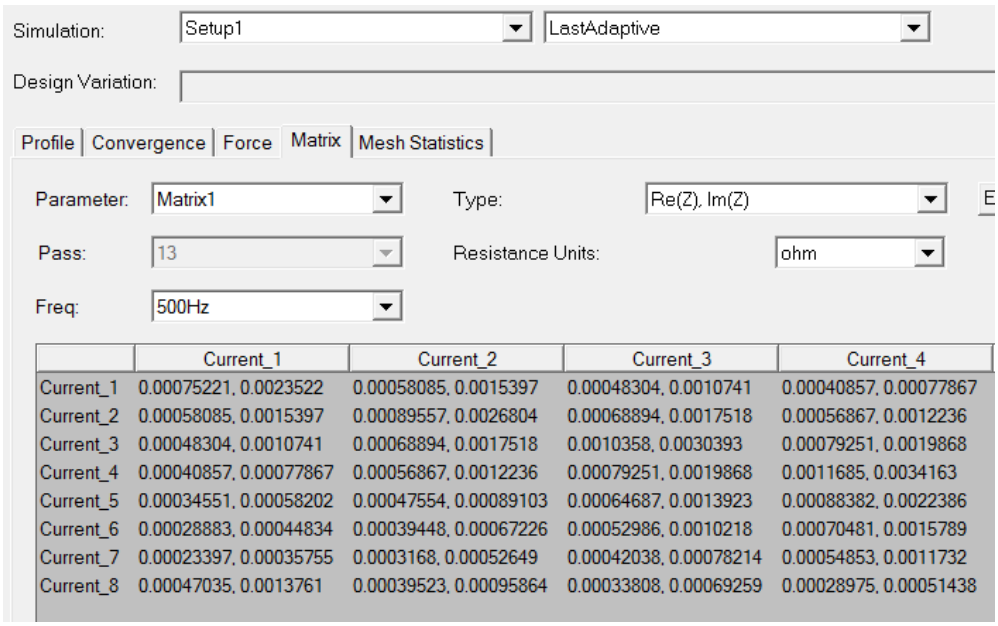
To View Solution Information

- Select the menu item *Maxwell 2D > Results > Solution Data*
  - To view Convergence
    - Select the **Convergence** tab



Pass	Triangles	Total Energy (J)	Energy Error (%)	Delta Energy (%)	Loss (W)	Loss
1	171	0.10991	5159	N/A	481.15	N/A
2	318	0.10997	2552.3	0.061288	527.53	N/A
3	450	0.1142	1812	3.8391	527.58	N/A
4	589	0.11287	951.89	1.1651	495.96	N/A
5	766	0.11358	341.77	0.62837	362.16	N/A
6	999	0.11354	143.08	0.034041	324.61	N/A
7	1300	0.11245	58.231	0.95638	303.29	N/A
8	1691	0.11175	15.996	0.62088	288.77	N/A
9	2199	0.11168	8.9168	0.065336	287.58	N/A
10	2860	0.11151	4.472	0.14872	285.96	N/A
11	3718	0.1114	2.3247	0.10492	285.11	N/A
12	4835	0.11136	1.184	0.031187	284.51	N/A
13	6286	0.11134	0.69606	0.015179	284.27	N/A

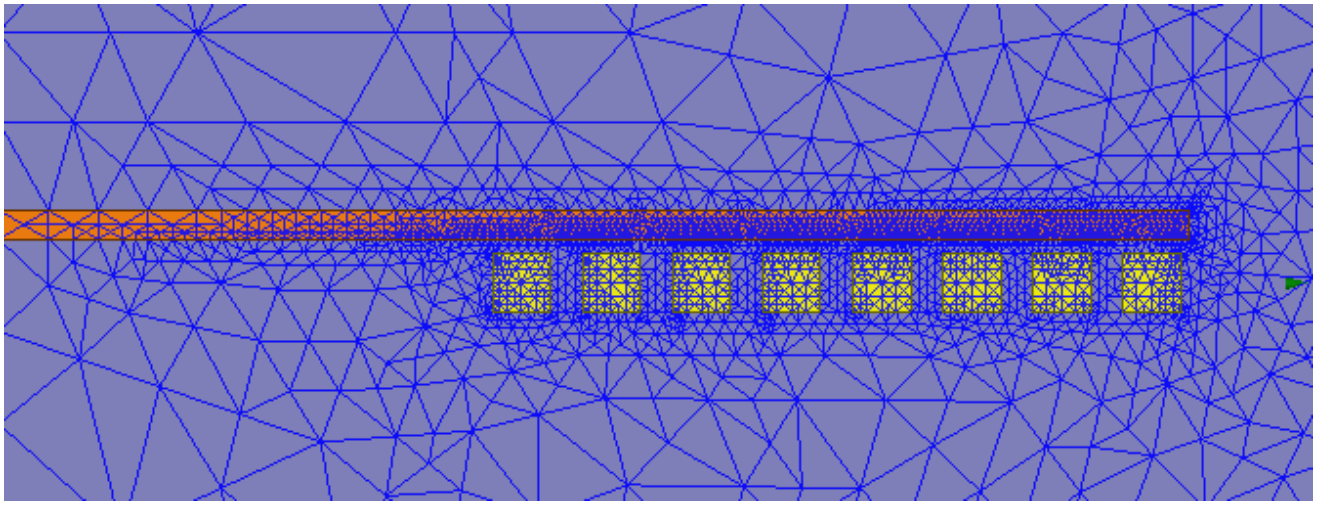
  - To View Impedance matrix
    - Select **Matrix** tab
    - By default, the results are displayed as [R, Z] but can be also shown as [R, L] or as coupling coefficients.



	Current_1	Current_2	Current_3	Current_4
Current_1	0.00075221, 0.0023522	0.00058085, 0.0015397	0.00048304, 0.0010741	0.00040857, 0.00077867
Current_2	0.00058085, 0.0015397	0.00089557, 0.0026804	0.00068894, 0.0017518	0.00056867, 0.0012236
Current_3	0.00048304, 0.0010741	0.00068894, 0.0017518	0.0010358, 0.0030393	0.00079251, 0.0019868
Current_4	0.00040857, 0.00077867	0.00056867, 0.0012236	0.00079251, 0.0019868	0.0011685, 0.0034163
Current_5	0.00034551, 0.00058202	0.00047554, 0.00089103	0.00064687, 0.0013923	0.00088382, 0.0022386
Current_6	0.00028883, 0.00044834	0.00039448, 0.00067226	0.00052986, 0.0010218	0.00070481, 0.0015789
Current_7	0.00023397, 0.00035755	0.0003168, 0.00052649	0.00042038, 0.00078214	0.00054853, 0.0011732
Current_8	0.00047035, 0.0013761	0.00039523, 0.00095864	0.00033808, 0.00069259	0.00028975, 0.00051438

Plot Mesh

- **To Plot Mesh**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  - In Create Mesh Plot window, press **Done**



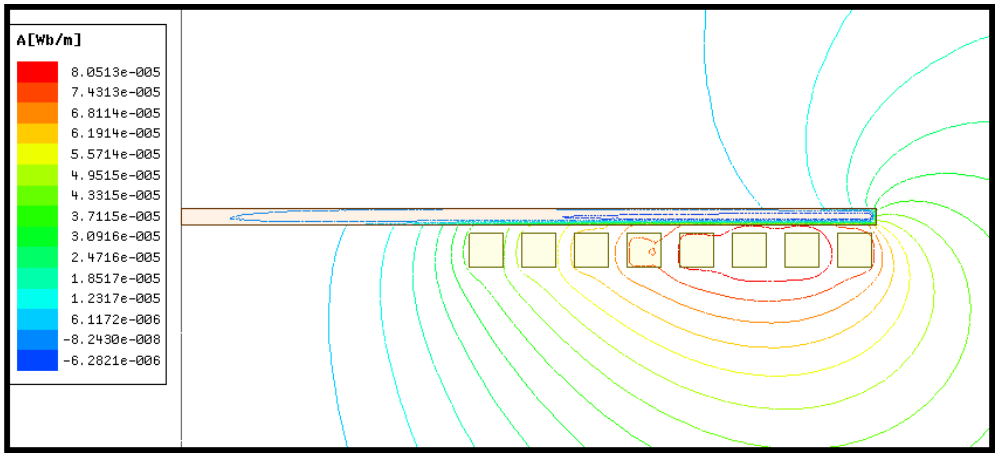
Compute Total Power Loss in the Plate

- **To Compute the Power Loss**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > OhmicLoss**
    - Select **Input > Geometry**
      - Select **Volume**
      - Select **Plate**
      - Press **OK**
    - Select **Scalar > Integral > RZ**
    - Select **Output > Eval**
  - Note: The evaluated loss in the **Plate** should be about **260 W.**

Plot Flux Lines

To Plot Flux Lines

- Select the menu item *Edit > Select All*
- Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
- In Create Field Plot window, Press **Done**

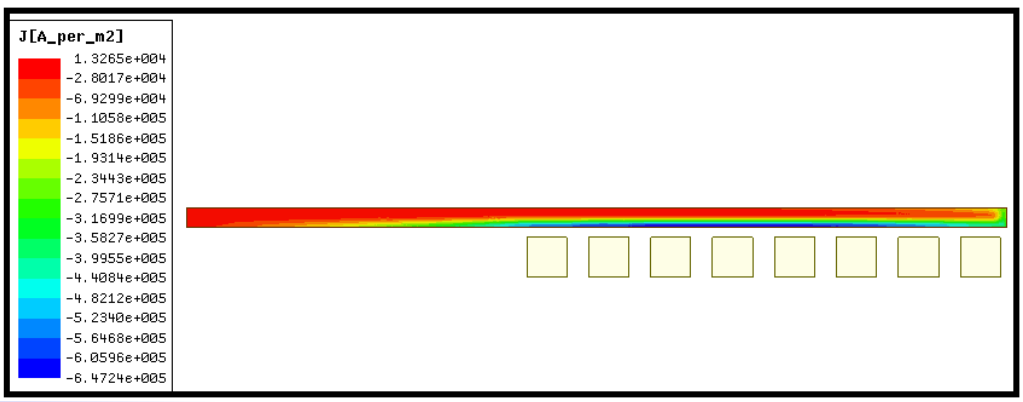


- Note that the flux lines are attracted to the plate since it is magnetic. Also, skin effects are present in the plate since there are eddy currents flowing in it. This can be seen best if you zoom into the plate

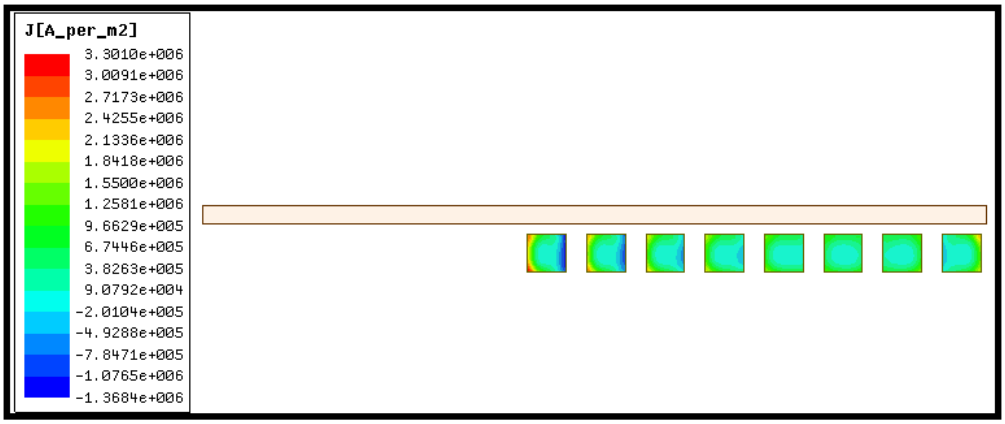
Plot Current Density Scalar

To Plot Current Density Scalar on Plate

- Hide previous plots by selecting *View > Active View Visibility > Fields Reporter* and unchecking the previous plots.
- Select the sheet **Plate** from history tree
- Select the menu item *Maxwell 2D > Fields > Fields > J >JAtPhase*
- In Create Field Plot window, Press **Done**

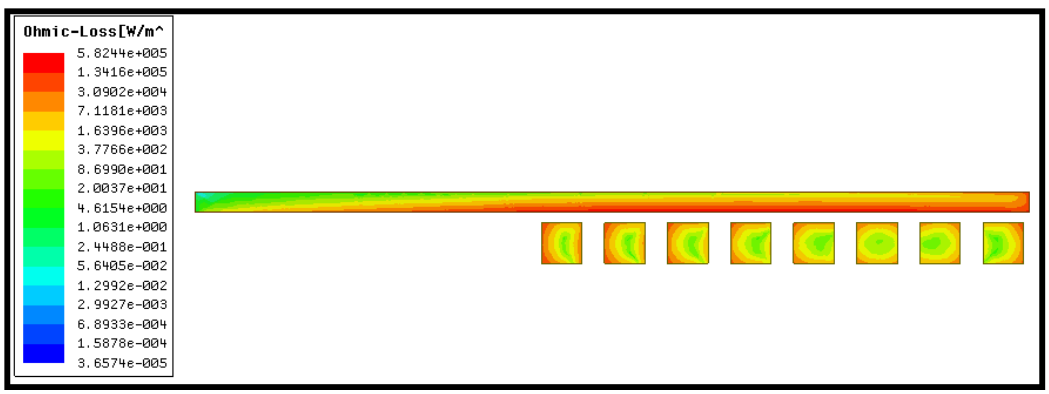


- **To Plot Current Density Scalar on Plate**
  - Press **Ctrl** and select all coils from history tree
  - Select the menu item *Maxwell 2D > Fields > Fields > J >JAtPhase*
  - In Create Field Plot window, Press **Done**



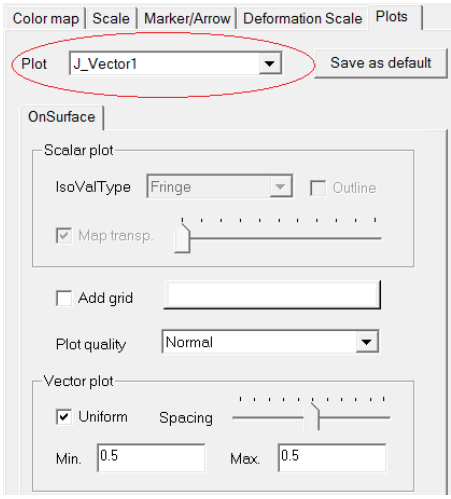
Plot Ohmic Loss Distribution

- **To Plot Ohmic Losses**
  - Hide previous plots by selecting *View > Active View Visibility > Fields Reporter* and unchecking the previous plots.
  - Select the menu item *Edit > Select All Visible*
  - Select the menu item *Maxwell 2D > Fields > Fields > Other > Ohmic_Loss*
  - In Create Field Plot window, Press **Done**
- **To Modify Plot Attributes**
  - Double click on the Legend to modify plot
  - In the window,
    - **Scale** tab
      - Select **Log**
    - Press **Apply** and **Close**

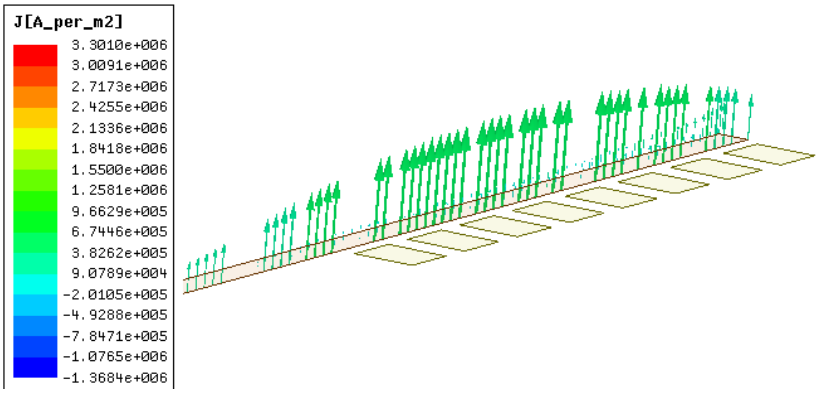


Plot Current Density Vectors

- **To Plot Current Density vectors**
  - Select the sheet **Plate** from history tree
  - Select the menu item *Maxwell 2D > Fields > Fields > J > J_Vector*
  - In Create Field Plot window, Press **Done**
- **To Modify Plot Attributes**
  - Double click on the Legend to modify plot
  - In the window,
    - **Plots** tab
      - Plot: Change to **J_Vector1**
      - Change Vector plot spacing
        - Min: **0.5**
        - Max: **0.5**
    - Press **Apply** and **Close**
  - Rotate geometry to view vector plot

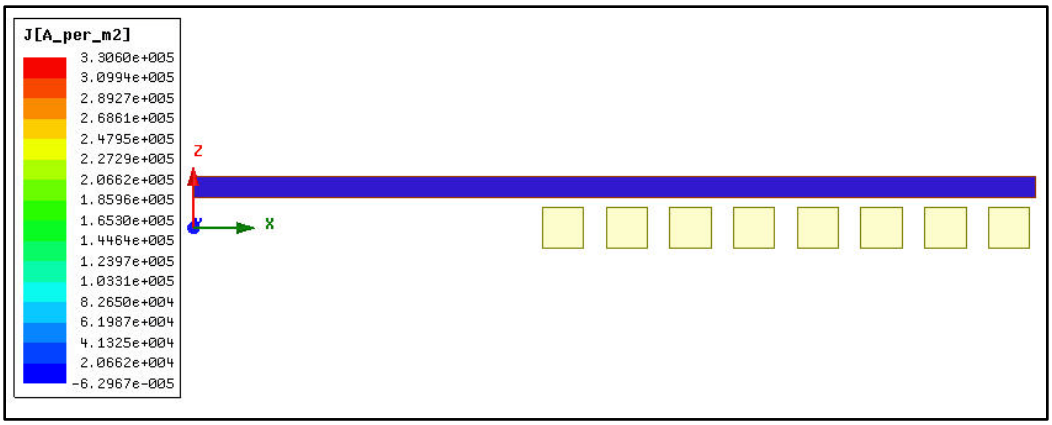


- **To A**

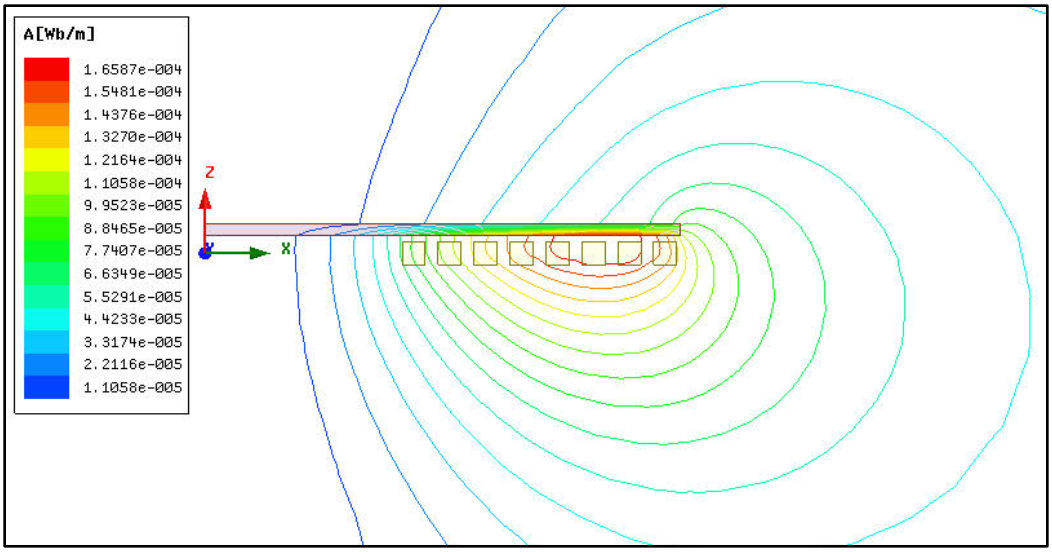


  - Select the Vector plot from Project Manager tree, right click and select **Animate**
  - In Setup Animation window,
    - Press **OK** with default settings
  - A window will appear to start, stop, pause or save the animation

- Solve a DC Problem
- Copy Design
  - **To Copy Design**
    - Select the Maxwell2D Design from Project Manager tree, right click and select **Copy**
    - Right click on the Project name in Project Manager tree and select **Paste**
- Change Analysis Setup
  - **To Change Analysis Setup**
    - Expand the project Manager tree for newly created design
    - Double click on **Setup1** under Analysis in the tree
    - In Solve Setup window,
      - **Solver** tab
        - Adaptive Frequency: Change to **0.001 Hz**
- Analyze
  - **To start the solution process:**
    - Select the menu item *Maxwell 2D > Analyze All*
- View Results
  - **To View Current Density Plots**
    - Plots are already copied from previous design
    - Double click on the corresponding plot from Project manager tree to view
    - Note that there is no significant current induced in the plate at 0.001 Hz.



- **View Flux Lines**
  - Note that the flux lines penetrate in and through the plate. While saturation is considered at DC, no AC skin effects or shielding occurs.



Basic Exercises – Transient – Large Motion - Rotational

Large Motion – its Quick Implementation Using the Maxwell 2D Transient Solver

- Maxwell Transient is able to consider *interactions* between *transient electromagnetic fields* and *mechanical motion* of objects.
- Maxwell Transient (with motion) includes $d\vec{B}/dt$ 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 2D 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

- 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

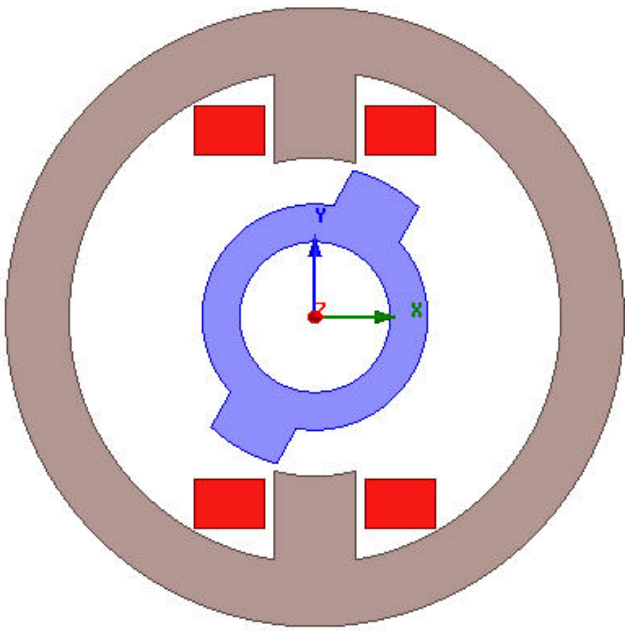
Open the Rotational Actuator Model

To Open the Input File

- Select the menu item ***File > Open***
- Browse to the location of the file **"Ex_09_13_BasicTransient_MotionRotational_M2dTrs120.mxwl"** and Open it

Activate Design

- Double click on the design **"00_Template"** to activate it and start working from there.
- You can copy/paste **00_Template** into your own working project. The other designs show the fully setup models we will be working on.
- The model should look like this:

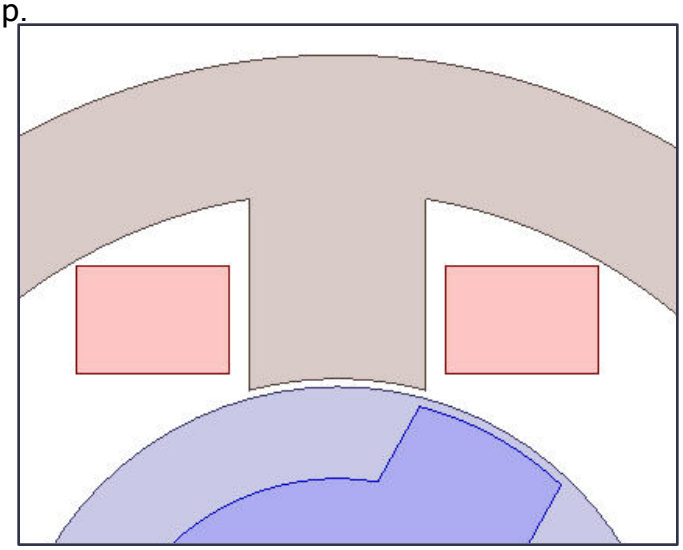


Verify the Setup

- 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, but rather work in steps. Especially in cases eddy current effects, external circuits, and large motion are included, the correctness of the setup for each individual property should be verified. After that, all properties can be considered together.
- For this quickstart, please study the winding setup and background.
- 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.
- **Verify Symmetry Multiplier**
  - Select the menu item *Maxwell 2D > Design Settings*
  - In Design Settings window,
    - Select **Symmetry Multiplier** tab
      - Verify that the value is set to **1**
- **Verify Model Depth**
  - In Design Settings window,
    - Select **Model Depth** tab
      - Verify that the value is set to **25.4 mm**
- 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 (**00_Template**).

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 i.e. a regular cylinder, but segmented preferably.
- Maxwell considers all moving objects (inside the *band*) to form one single moving object group.



Band object separating rotor from stator

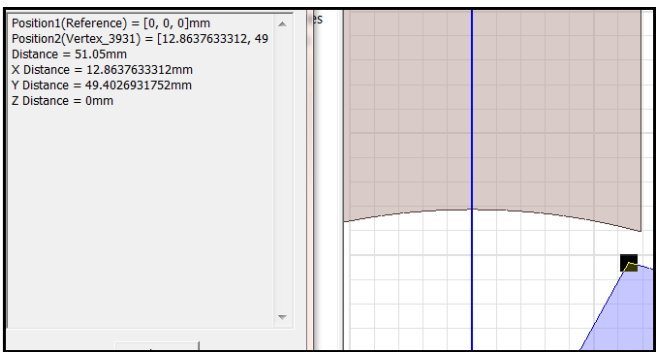
- *Constant Speed* mode:
  - If the model is setup to operate in constant speed mode (see below), Maxwell will not compute mechanical transients.
  - However, changing magnetic fields owing to speed ω_m, i. e. d*B*/d*t* effects are included in the field solution.
- *Mechanical Transient* mode:
  - In case inertia was specified, Maxwell will compute the motion equation in each time step.

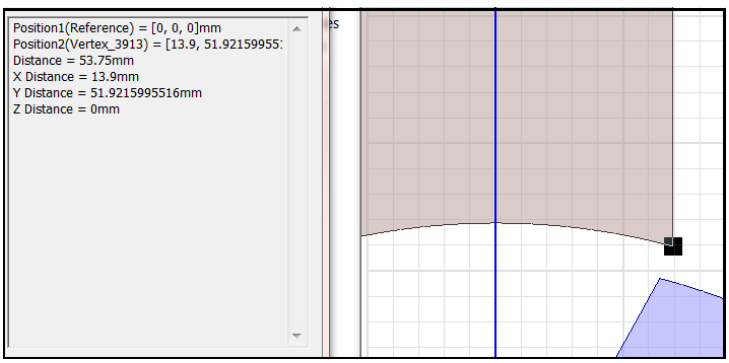
$$J_m \cdot \mathrm{d}^2\varphi_m(t) / \mathrm{d}t^2 + k_D(t) \cdot \mathrm{d}\varphi_m(t) / \mathrm{d}t = T_\psi(t) + T_m(t)$$

  - See Appendix A for a variable explanation.

Create Band Object

- **Analyze Geometry**
  - 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.
  - We want a regular polyhedron that encloses all moving objects.
  - 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.
- **Determine Size of Band Object**
  - Select the menu item ***View > Visibility> Active View Visibility***
  - In Active View Visibility window,
    - Uncheck the Visibility for all parts apart from **Stator1** and **Rotor1**
  - Select the menu item ***Modeler > Measure > Position***
  - Move the mouse curser to the end vertex of the **Rotor1** and just place on it (Do not click)
    - Read the *Distance* value from the Measura Data window (51.05 mm).
  - Move the mouse curser to the inner vertex of the **Stator1** and just place on it (Do not click)
    - Read the *Distance* value (53.75 mm).
  - Press **Esc** to exit measure
  - Thus, *band* should have a radius of 52.4 mm. Here, 52.5 mm was used.





- **Draw Band Object**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius
       - dX: **52.5**, dY: **0**, dZ: **0**, Press the **Enter** key
- **Change Attributes**
  - Change the name of resulting sheet to **Band1**
  - Change the transparency of the sheet to **0.9**
  - Select the menu item *View > Visibility> Show All > Active View*

Perform Large Rotational Standstill Test

Copy Design

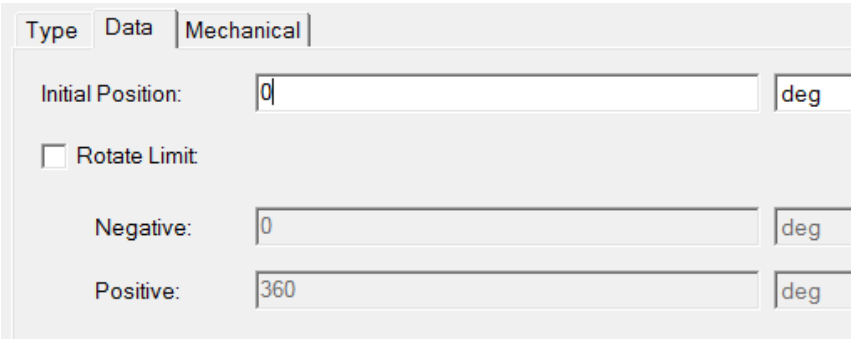
- **To Create a Copy of the Design**
  - Select the design **"00_Template"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"11_GeoFull_MagI_MchStandstill"**

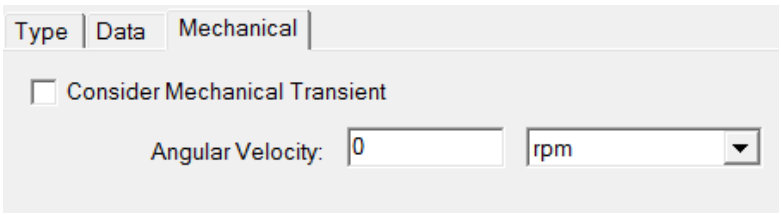
Assign motion

- **To Setup motion**
  - Select the object **Band1** from history tree
  - Select the menu item *Maxwell 2D > Model > Motion Setup > Assign Band*
  - In Motion Setup window,
    - **Type** tab
      - Motion type: **Rotational**
      - Rotation Axis: **Global::Z**
      - Select **Positive**



- **Data** tab
  - Initial position: **0 deg**
    - Motion will start at $t = 0$ with the rotor position being as drawn. Applying $\varphi_{m0} \neq 0$ would start with Rotor1 rotated by φ_{m0} from the drawn position.
  - Leave Rotate Limit unchecked (allowing the rotor to spin continuously)
- **Mechanical** tab
  - Angular Velocity: **0 rpm**
    - Standstiill Condition





- Now, we have setup “large rotational standstill”. Positive magnetic torque is generated around the positive z-axis.
- Expand the Project Manager tree for **Model.** Under MotionSetp1, a tab will be created as **Moving1.** Selecting this tab will highlight all objects which are considered to under motion.

Specify Mesh Parameters

- 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.
- **Assign Mesh Operation for Band**
  - For torque computation, the most critical areas are the airgap and its immediate proximity. Thus, the *band* mesh is crucial for accurate results.
  - Select the sheet **Band** from the history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Band_Length**
    - Restrict Length of Elements: ☐ **Unchecked**
    - Restrict the Number of Elements: ☑ **Checked**
    - Maximum Number of Elements: **5000**
    - Press **OK**
- **Assign Mesh Operation for Stator and Rotor**
  - Similarly Assign Mesh operations for other objects as mentioned below
  - For the sheet **Rotor1**
    - Name: **Rotor_Length**
    - Maximum Number of Elements: **1000**
  - For the sheet **Stator1**
    - Name: **Stator_Length**
    - Maximum Number of Elements: **1000**
  - For **Background1**
    - Name: **Background_Length**
    - Maximum Number of Elements: **1000**
  - Press **Ctrl** and select all coils
    - Name: **Coils_Length**
    - Maximum Number of Elements: **100**
    - Simultaneously selecting CoilA, CoilA_Neg, CoilB, and CoilB_Neg will try to assign 100 triangles to the group, i. e. about 25 triangle in each coil will result

Define Analysis Setup

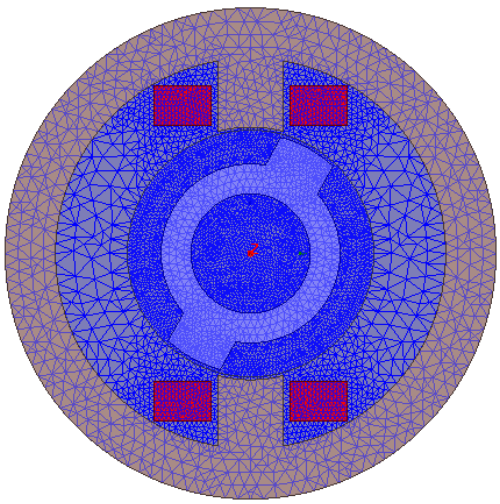
- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **20 ms**
      - Time Step: **5 ms**
    - Press **OK**
  - Add the same time values to **Save Fields** tab

Analyze

- **To Run Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

Plot Mesh

- **To plot Mesh**
  - Select the menu item *View > Set Solution Context*
  - In Set view Context window,
    - Change Time to **0.01 s**
    - Press **OK**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > Plot Mesh*



Perform Large Rotational Constant Speed Test

- 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 are switched off for all objects.
- We can now use Transient with Large Motion to monitor *cogging torque* effects.

Copy Design

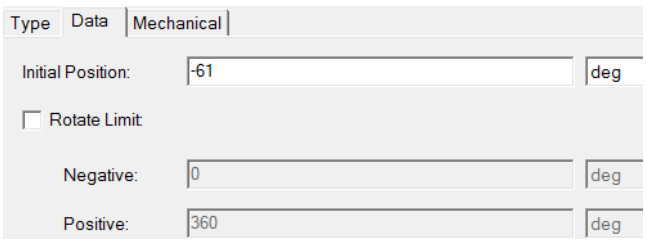
To Create a Copy of the Design

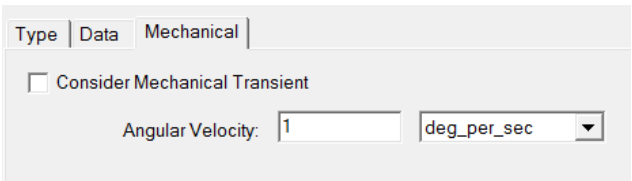
- Select the design **"11_GeoFull_MagI_MchStandstill"** from Project Manager tree, right click and select **Copy**
- Right click on the Project name in Project Manager tree and select **Paste**
- Rename the newly created design to **"12_GeoFull_MagI_MchSpeedslow"**

Modify Motion Setup

To Modify Motion Setup

- Expand the Project Manager tree to view **Model,** double click on the tab **MotionSetup1**
- In Motion Setup window,
  - **Data** tab
    - Initial Position: **-61 deg**
  - **Mechanical** tab
    - Angular Velocity: **1 deg_per_sec**
    - Rotor1 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: -61 -29 = -90°.
  - Press **OK**





Modify Analysis Setup

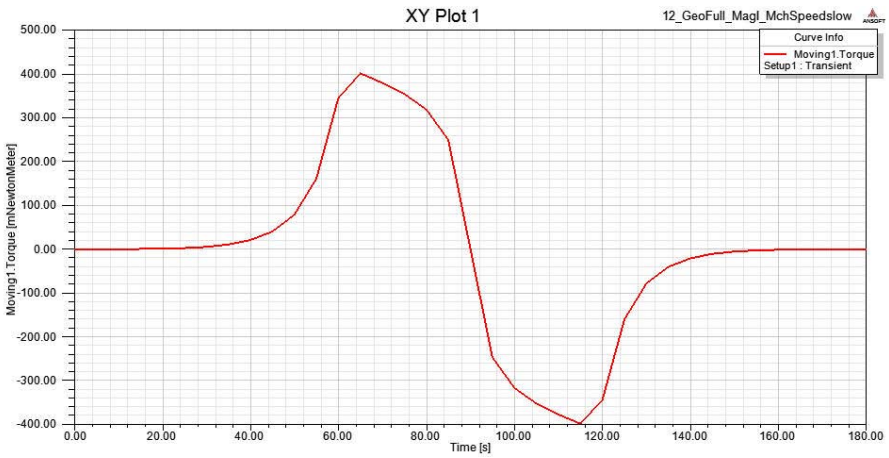
- **To Modify Analysis Setup**
  - Expand the Project manager tree to view **Analysis.** Double click on the tab **Setup1** to modify its properties
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **180 s**
      - Time Step: **5 s**
      - By rotating at a speed of 1 °/s 180 s long, Rotor1 will move 180°, i. e. from -90° to +90°, at 5°/step.
    - Press **OK**

Analyze

- **To Run Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

Plot Torque Vs Time

- **To Create a Plot**
  - Select the menu item *Maxwell 2D > Results > Create Transient Reports > Rectangular Plot*
  - In Reports window,
    - Category: **Torque**
    - Quantity: **Moving1.Torque**
    - Select **New Report**



Perform Large Rotational Transient MotionTest

- We will now operate the actuator as a one-body oscillator.
- Inertia will be specified as well as some damping.
- We can expect Rotor1 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 Rotor1.
- c_ψ in Nm/rad is the magnetic rigidity. As an analogy it can be understood as a mechanical spring spanned between Rotor1 and Stator1, whose force coming from the magnetic field.
- We can roughly calculate rigidity c from the cogging torque function (stable limb):

$$c_\psi = \frac{\Delta T_\psi}{\Delta \varphi_m} \approx \frac{400\ \text{mNm}}{\text{rad}(10°)} = 2.3\ \text{Nm/rad}$$

- 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.

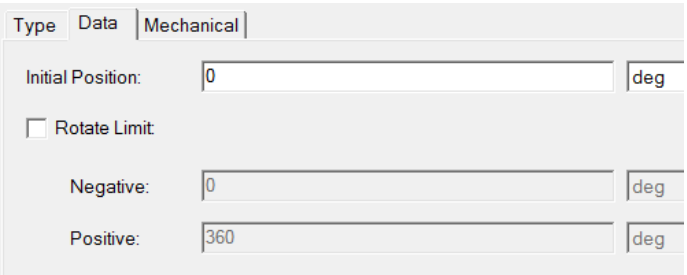
Copy Design

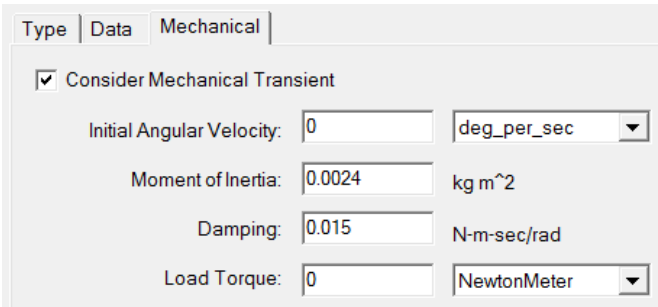
- **To Create a Copy of the Design**
  - Select the design **"12_GeoFull_MagI_MchSpeedslow"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"13_GeoFull_MagI_MchTransient"**

Basic Exercises – Transient – Large Motion - Rotational

Modify Motion Setup

- **To Modify Motion Setup**
  - Expand the Project Manager tree to view **Model,** double click on the tab **MotionSetup1**
  - In Motion Setup window,
    - **Data** tab
      - Initial Position: **0 deg**
    - **Mechanical** tab
      - Consider Mechanical Transient: ☑ **Checked**
      - Initial Angular Velocity: **0 deg_per_sec**
      - Moment of Inertia: **0.0024 kgm²**,
      - Damping: **0.015 Nm·s/rad**
    - 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.

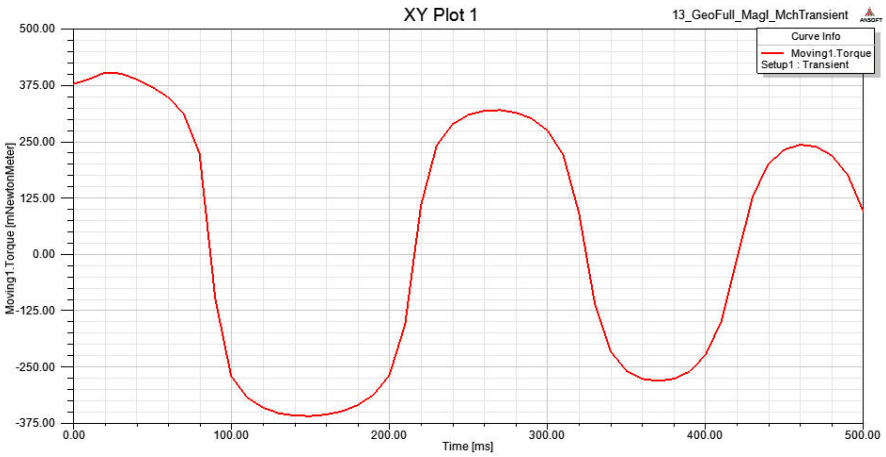




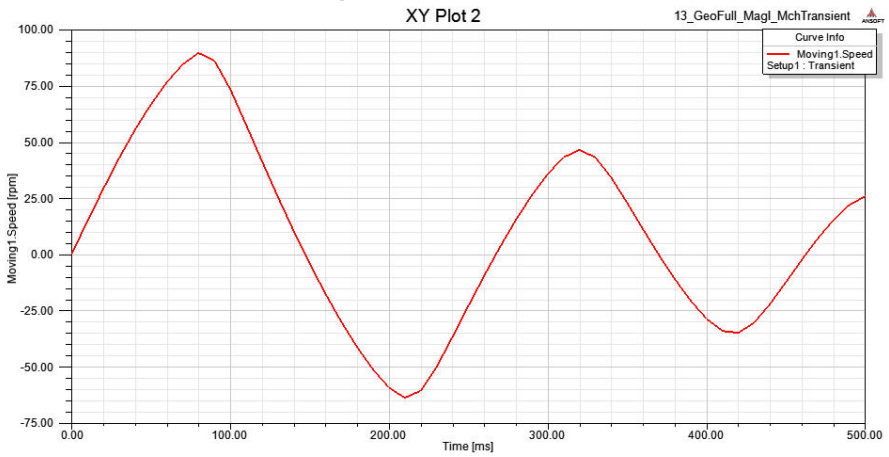
Modify Analysis Setup

- **To Modify Analysis Setup**
  - Expand the Project manager tree to view **Analysis.** Double click on the tab **Setup1** to modify its properties
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **0.5 s**
      - Time Step: **0.01 s**
      - From f_0, we can expect a >200 ms cycle. At 10 ms timestep we will sample one cycle >20 times.
    - Press **OK**

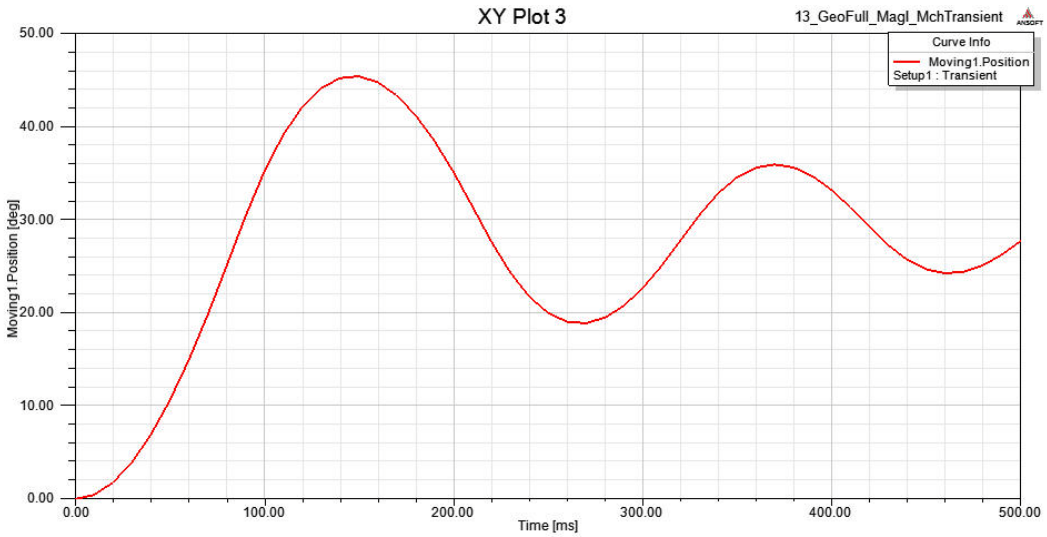
- Analyze
  - To Run Solution
    - Select the menu item *Maxwell 2D > Analyze All*
- Results
  - Torque Vs Time Plot
    - Plot already exist in the design since it is copied from pervious design
    - Double click on it to view results



  - Plot Mechanical Speed Vs Time
    - Select the menu item *Maxwell 2D > Results > Create Transient Reports > Rectangular Plot*
    - In Reports window,
      - Category: **Speed**
      - Quantity: **Moving1.Speed**
      - Select **New Report**



- **Plot Mechanical Position Vs Time**
  - In Reports window,
    - Category: **Position**
    - Quantity: **Moving1. Position**
    - Select **New Report**



- From Results, it can be seen that
  - T_ψ looks as expected from previous simulations.
  - ω_m corresponds to T_ψ's first derivative and is correct.
  - φ_m oscillates around +29°, which is the stator flux axis (y) with respect to the initial position.

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 $\varphi_{m0} = 0$.
$d\varphi_m(t) / dt$, $\omega_m(t)$	Mechanical angular speed in rad/s.
ω_{m0}	Initial ω_m in rad/s.
$d^2\varphi_m(t) / dt^2$	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.

Basic Exercises – 2D Boundary Conditions

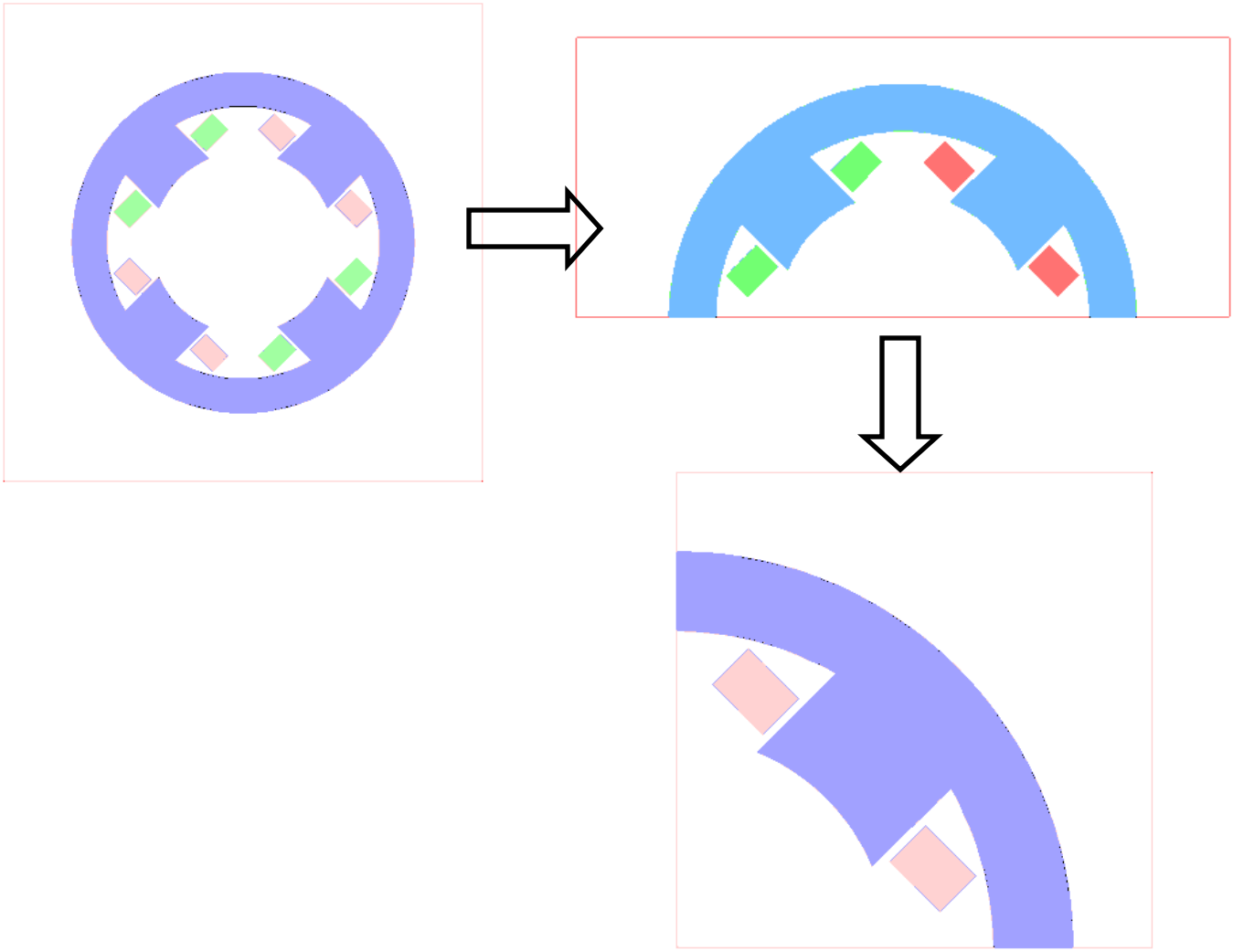
Assigning Boundary Conditions

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.

Purpose of the Exercise

- This exercise introduces various boundary conditions used in **Maxwell 2D** based on a simple example with coils and steel core. The user will learn how to use **Vector Potential, Balloon, Symmetry and Matching Boundary (Master and Slave)**.



ANSYS Maxwell Design Environment

- The following features of the ANSYS Maxwell Design Environment are used to create the models covered in this topic
  - 2D Sheet Modeling
    - User Defined Primitives (UDPs): **SRMCore**
    - Boolean Operations: **Separate Bodies**
  - Boundaries/Excitations
    - Current: **Stranded**
    - Boundaries: **Vector Potential, Balloon, Symmetry, Master/Slave,**
  - Analysis
    - **Magnetostatic**
  - Field Overlays:
    - **H Vector**

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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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 2D Design***, or click on the  icon

Change Design name

- **To Change Design Name:**
  - Right-click **Maxwell2D Design1** at the project manager window and select **Rename.**
  - Change the Design Name to **1_VectorPotential**

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 Filename: **Ex_9_14_BasicBoundaryCondictions**
  - Click the **Save** button

Set Solution Type

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

Create Model

Create Stator and Coils

- Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SRMCore*
- In User Defined Primitive Operation window,
  - DiaGap: **80**
  - DiaYoke: **150**
  - Length: **0**
  - Poles: **4**
  - LenRegion: **0**
  - InfoCore: **1**
  - Press **OK**

Parameters | Info

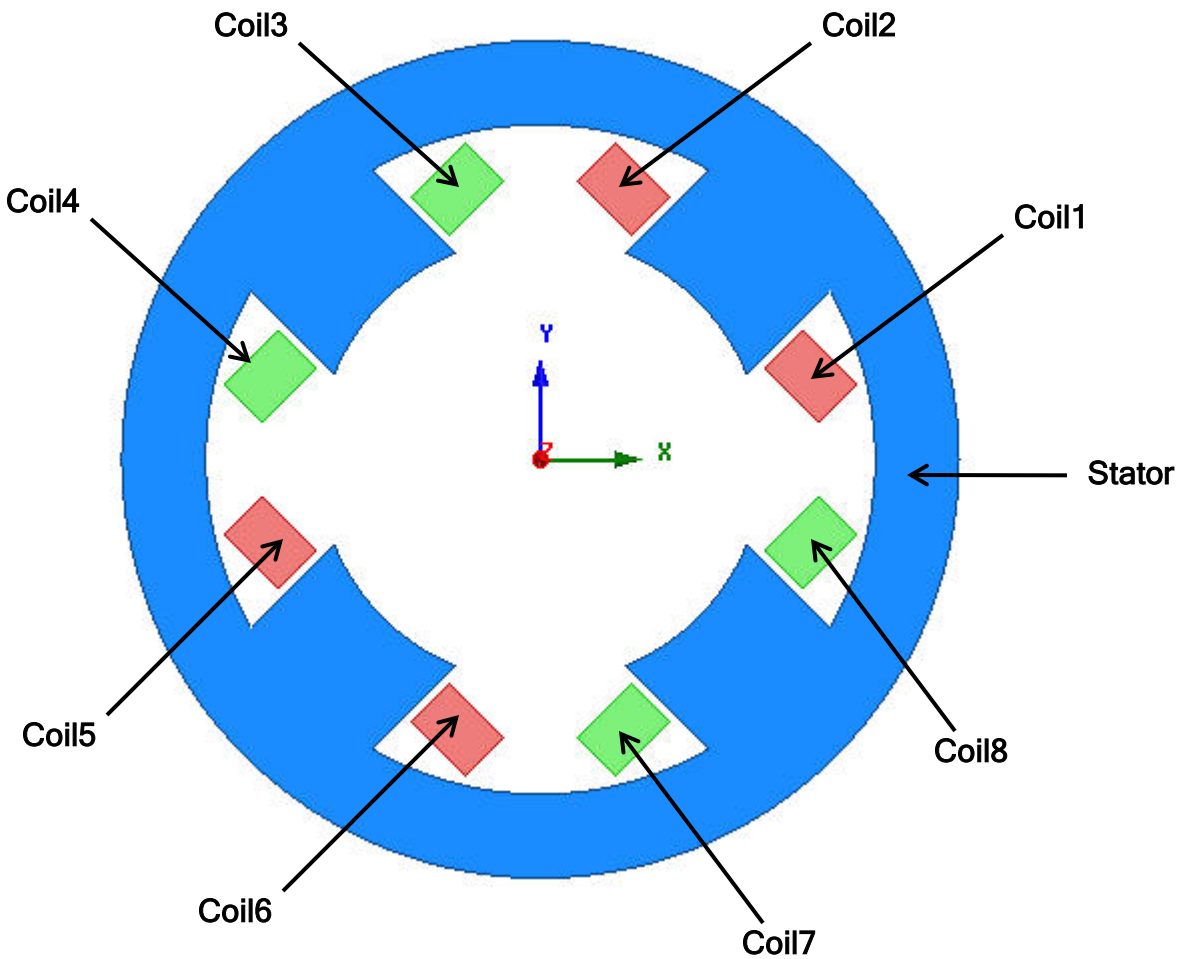
Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate Sys...	Global			
DLL Name	RMxprt/SRMCore			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	80	mm	80mm	Core diameter on gap si...
DiaYoke	150	mm	150mm	Core diameter on yoke ...
Length	0	mm	0mm	Core length
Poles	4		4	Number of poles
ThkYoke	15	mm	15mm	Yoke thickness
Embrace	0.5		0.5	Pole embrace (the ratio...
EndExt	5	mm	5mm	Coil one-side end exten...
LenRegion	0	mm	0mm	Region length
InfoCore	1		1	0: core; 1: core & coils; ...

Show Hidden

Change Attributes

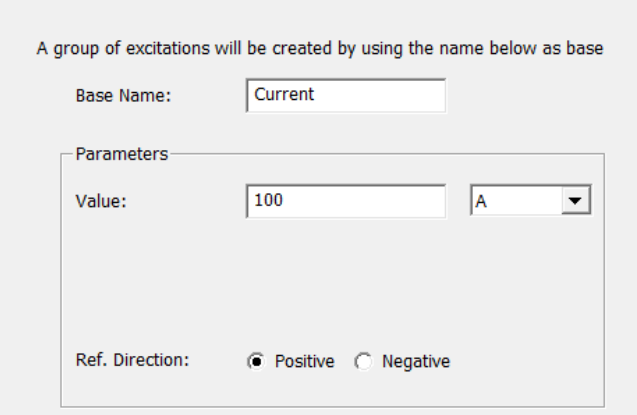
- Change the name of resulting sheet to **Stator** and color to **Blue**
- Change the material of resulting sheet to **Nickel**

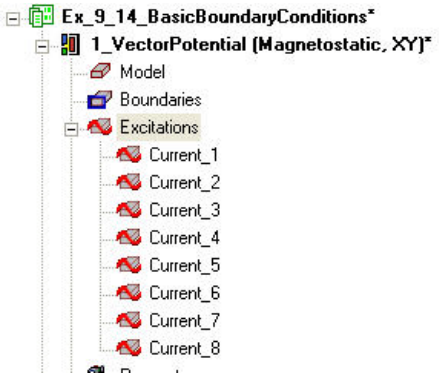
- **Separate Coils from Stator**
  - Select the sheet **Stator** from history tree
  - Select the menu item *Modeler > Boolean > Separate Bodies*
- **Change Attributes**
  - Change the name and color of the resulting sheets as shown in below image
  - Change the material of all coils to **Copper**



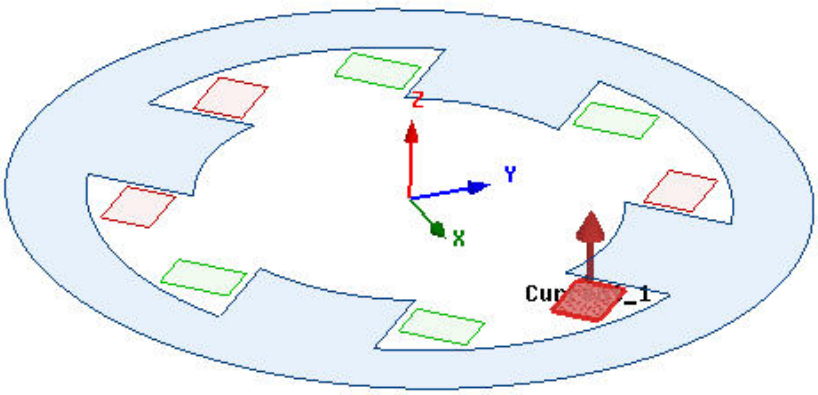
Assign Excitation

- **To Assign Current Excitations to Coils**
  - Press **Ctrl** and select all the coils sequentially starting with coil1 to coil6 from history tree
    - **Note:** Excitation numbering might alter if coils are not selected sequentially
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    - Base Name: **Current**
    - Value: **100 A**
    - Ref. Direction: **Positive**
    - Press **OK**





- Excitations are created for each of the coil separately
- At Present direction of excitation for all coils is pointing out of the plane (Positive Z direction)

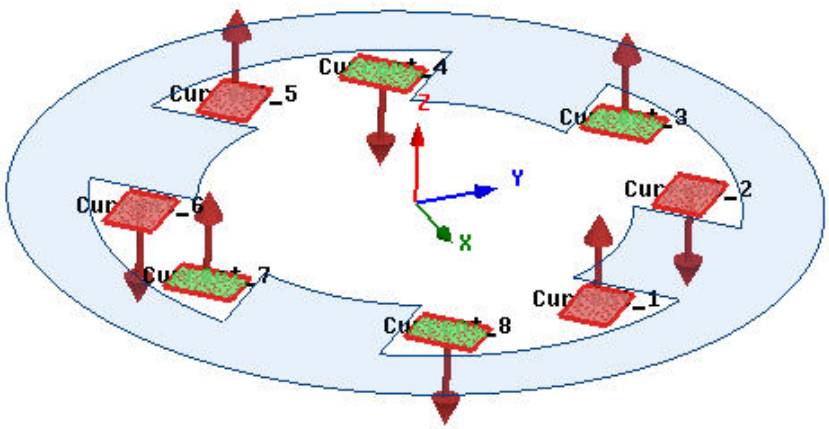


- **Alter Direction of Excitation for Coils**
  - Select the menu item *Maxwell 2D > List*
  - In Design List window,
    - **Excitations** tab
      - Press **Ctrl** and select **Current_2, Current_4, Current_6** and **Current_8**
      - Select the button **Properties**
      - In Properties window,
        - Change Direction to **Negative**
        - Press **OK** and **Done** to exit definition

Model | Boundaries | Excitations | Parameters | Mesh Operations | Analysis Setup

Name	Type	Description
Current_1	Current	100A
Current_2	Current	100A
Current_3	Current	100A
Current_4	Current	100A
Current_5	Current	100A
Current_6	Current	100A
Current_7	Current	100A
Current_8	Current	100A

  - Now the direction of excitation is as shown below

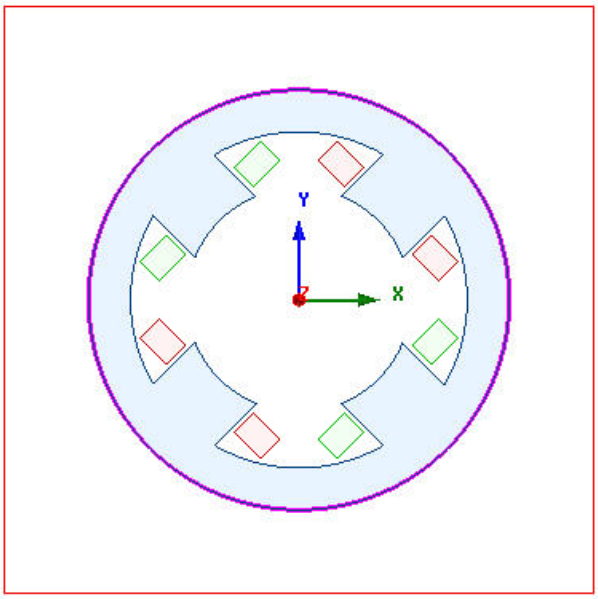


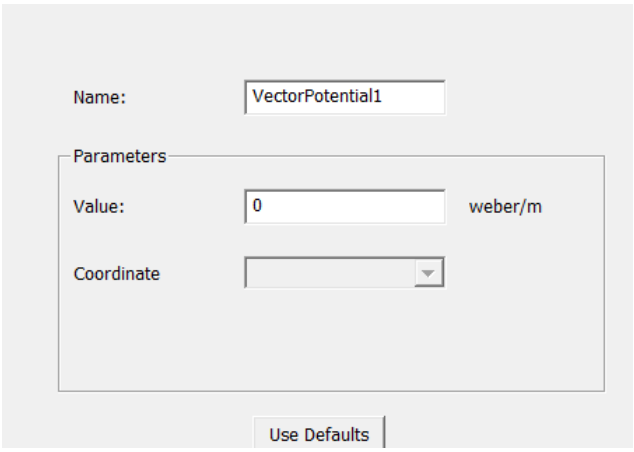
Define Region

- **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: **20**
    4. Press **OK**

Assign Boundary

- **To Assign Boundary**
  - Select the menu item *Edit > Select > Edges*
  - Select the outer edges of the sheet **Stator**
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Vector Potential*
  - In Vector Potential Boundary window,
    - Leave the Value to **0**
    - Press **OK**





Analysis Setup

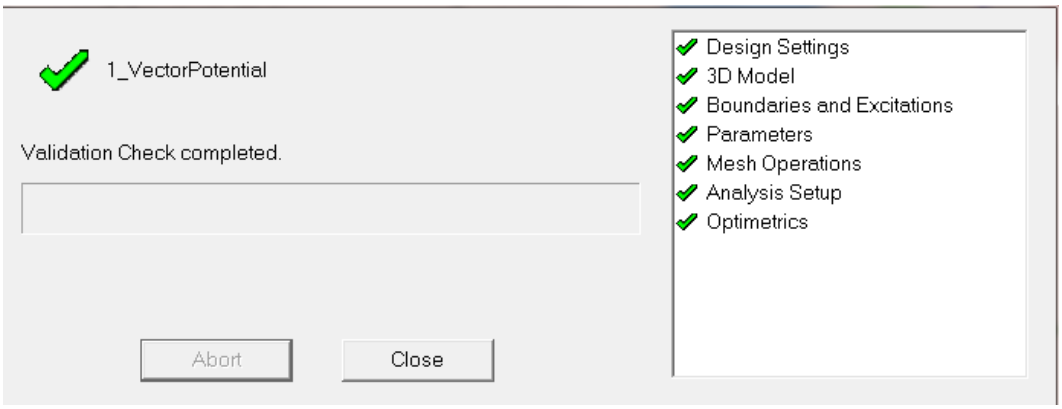
- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,
    - Press **OK**

Save File

- **To Save File**
  - Select the menu item *File > Save*

Validation Check

- **To Check the Validity of the Setup**
  - Select the menu item *Maxwell 2D > Validation Check*



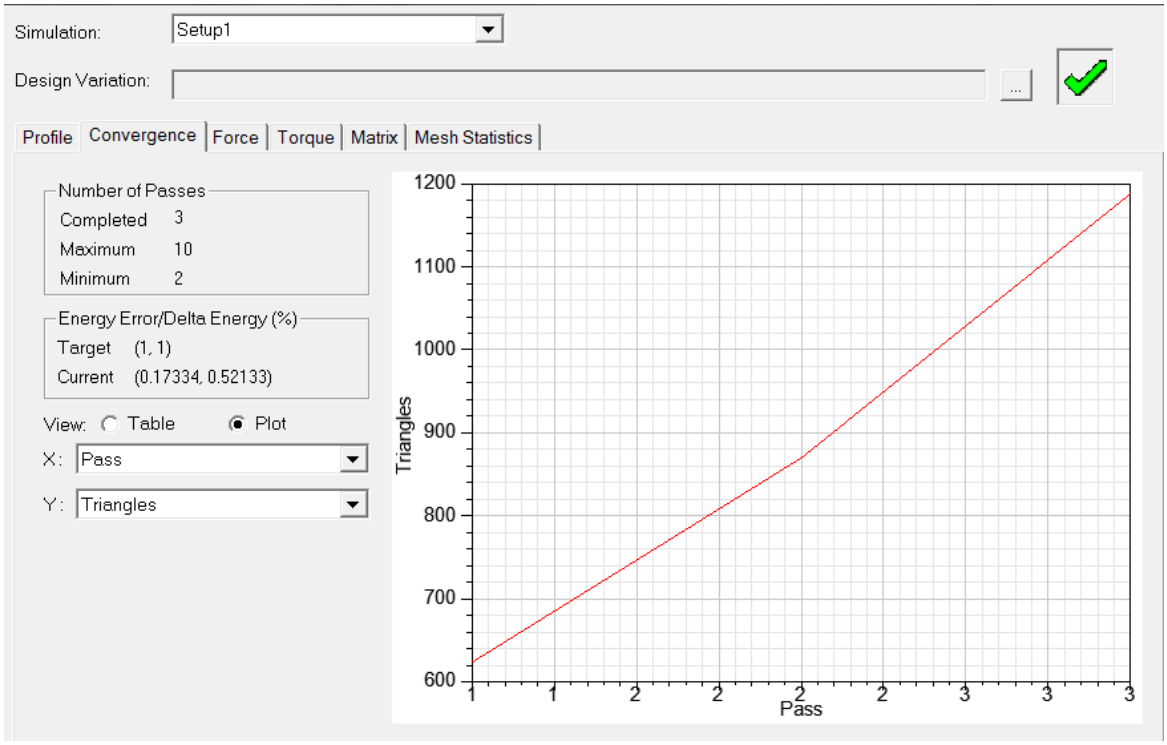
  - The problem won't solve unless each item has a check mark.

Analyze

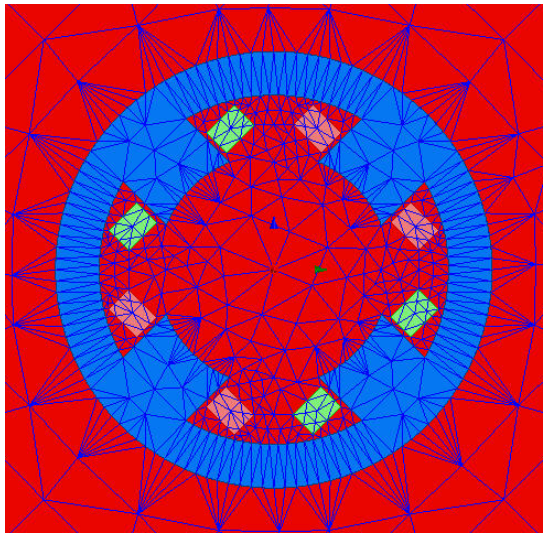
- **To start the solution process:**
  - Select the menu item *Maxwell 2D > Analyze All*

Solution Data

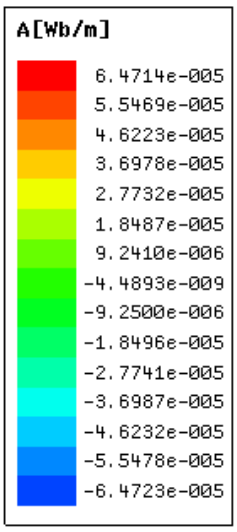
- **To View Solution Information**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
  - In Solutions window,
    - To View Profile of the solution
      - Select **Profile** tab
    - To View Solution Convergence
      - Select **Convergence** tab
      - Note: The default view for convergence is **Table**. Click on the **Plot** radio button to view a graphical representations of the convergence data.
  - Note: You don't have to wait for the solution to be done to do this. You can do this while the simulation is running, all information will update automatically after each pass is done.

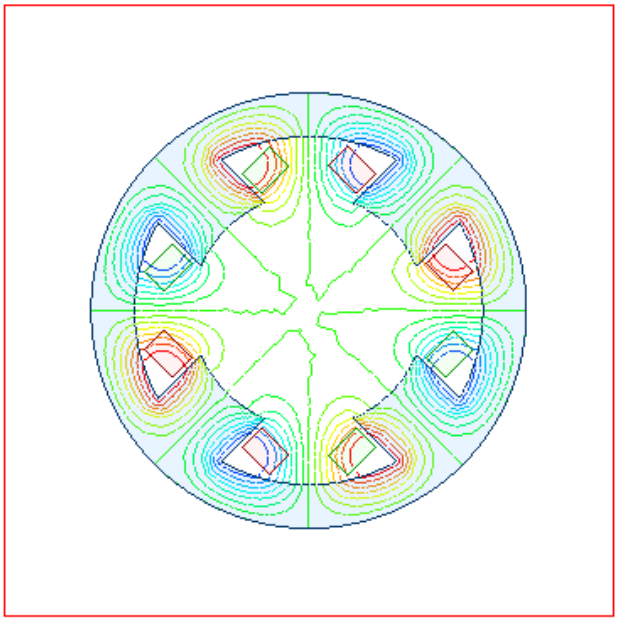


- PostProcess
  - To Plot Mesh
    - Select the menu item *Edit > Select All*
    - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
    - In Create Mesh Plot window,
      - Press **Done**

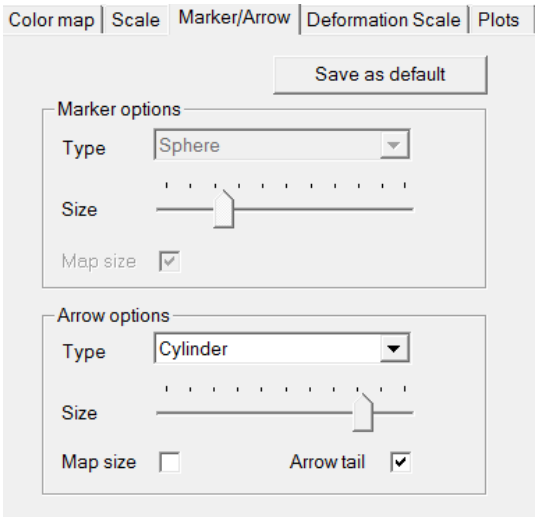


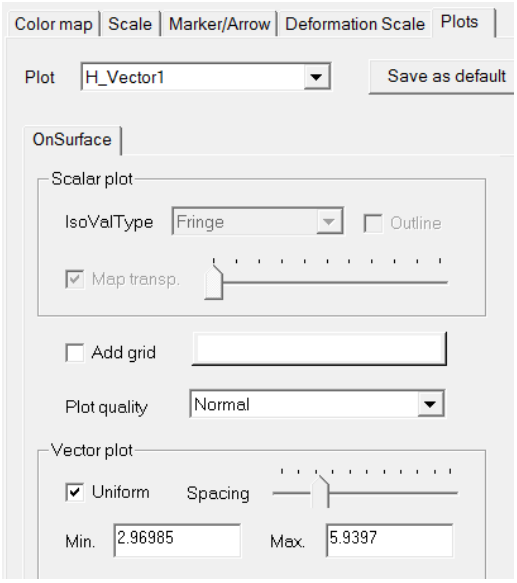
  - Plot Flux Lines
    - Select the menu item *Edit > Select All*
    - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
    - In Create Field Plot window,
      - Press **Done**
    - Double click on the legend to modify plot attributes

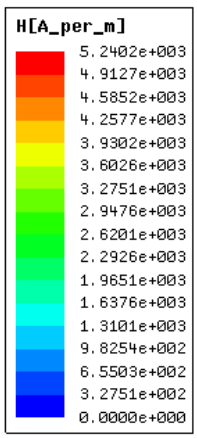


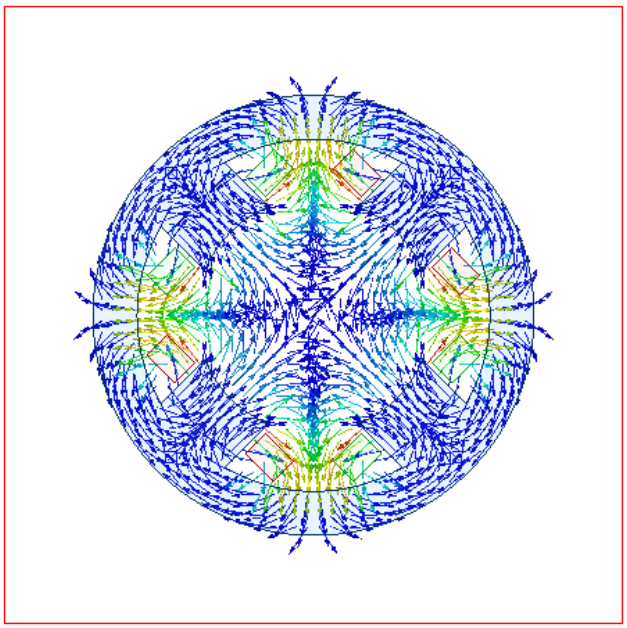


- **Plot H Vector**
  - Select the menu item ***Edit > Select All***
  - Select the menu item ***Maxwell 2D > Fields > Fields > H > H_Vector***
  - In Create Field Plot window,
    - Press **Done**
  - Double click on the legend to modify attributes
  - Change the settings as shown in below images









Example 2: Balloon Boundary

Copy Design

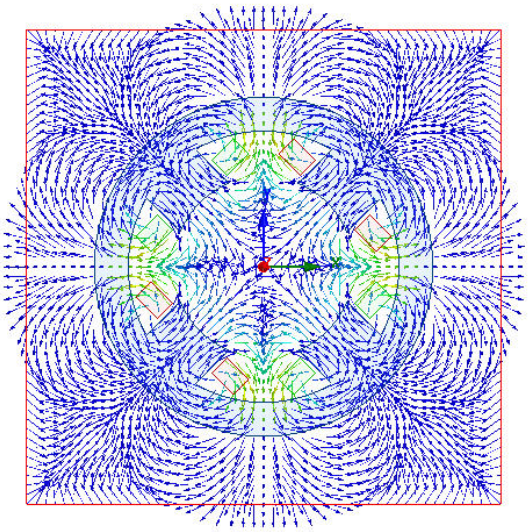
- **To Create a Copy of the Design**
  - Select the design **"1_VectorPotential"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"2_Balloon"**

Modify Boundary Conditions

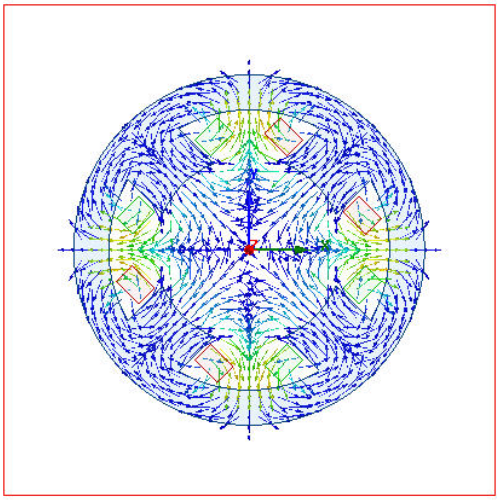
- **Specify Balloon Boundary**
  - Select the menu item *Maxwell 2D > Boundaries > Delete All* to remove existing boundary specifications
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Balloon*
  - In Balloon Boundary window, Press **OK**

Analyze

Examine H Vector Plot



Balloon Boundary on all edges of the Region



Vector Potential Boundary on outer edges of Stator

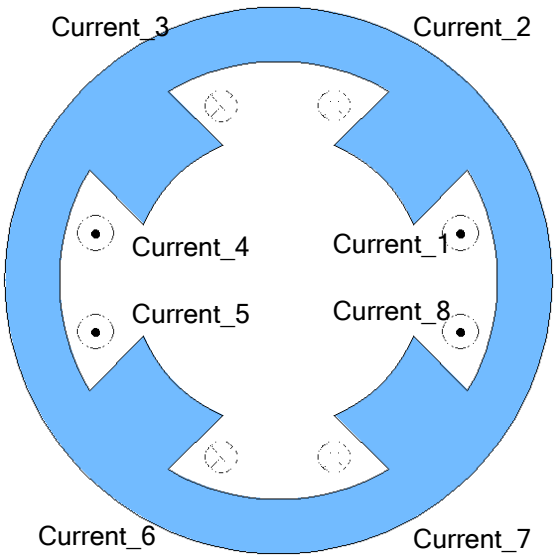
Example 3: Balloon Boundary with Changed Excitations

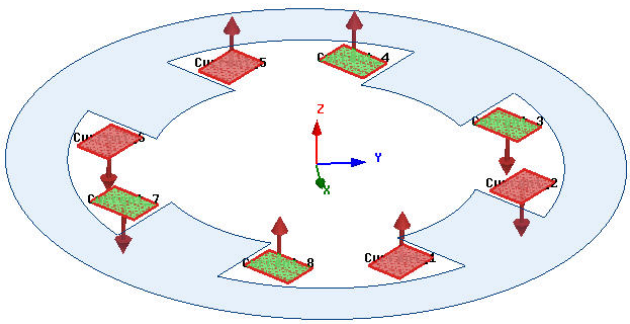
Copy Design

- **To Create a Copy of the Design**
  - Select the design **"2_Balloon"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"3_Balloon_ChangeExcitation"**

Change Direction of Excitations

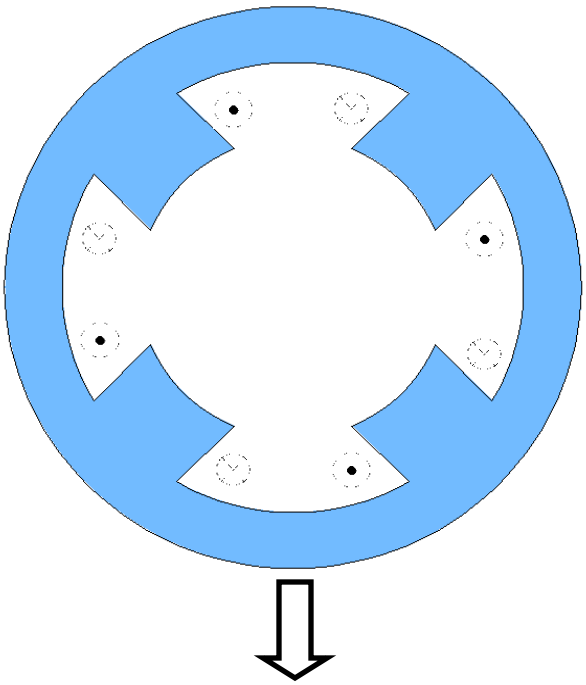
- **To Change Excitation directions**
  - Expand the Project Manager tree to view **Excitations**
  - Double click on the excitation **Current_3**
    - Change the direction of excitation to **Negative**
  - Similarly Change the direction of excitation for **Current_7** to **Negative**
  - Change the direction of excitation for **Current_4** and **Current_8** to **Positive**
  - The idea is to achieve the configuration shown in below image

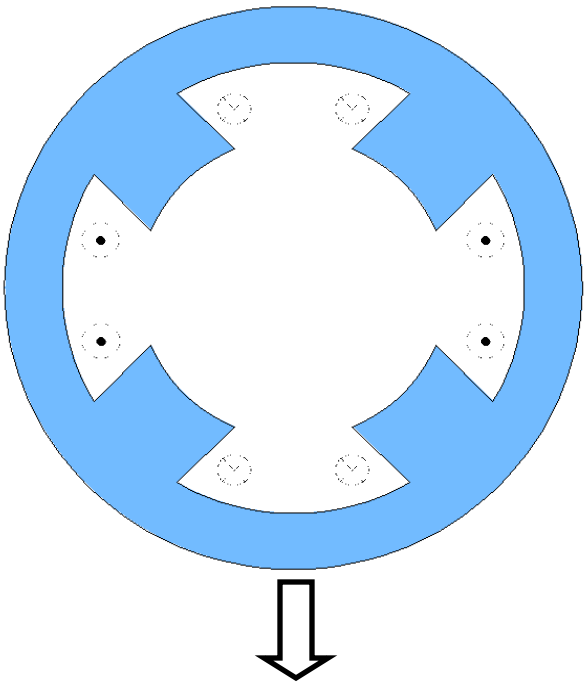


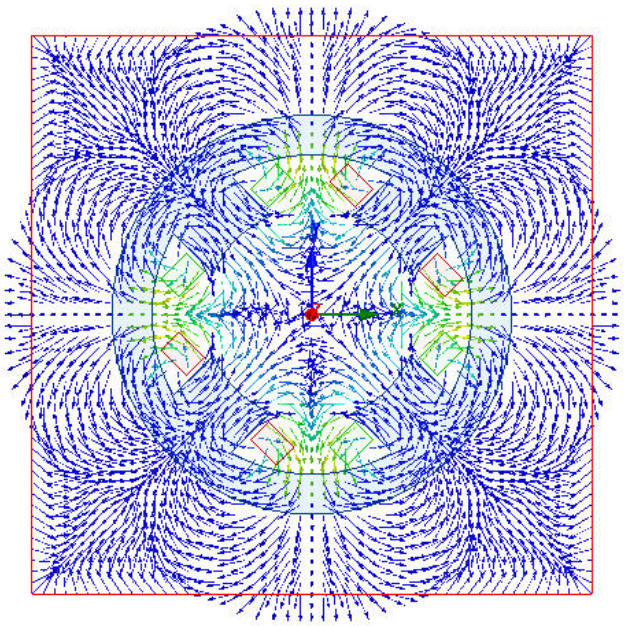


Current_1	Positive
Current_2	Negative
Current_3	Negative
Current_4	Positive
Current_5	Positive
Current_6	Negative
Current_7	Negative
Current_8	Positive

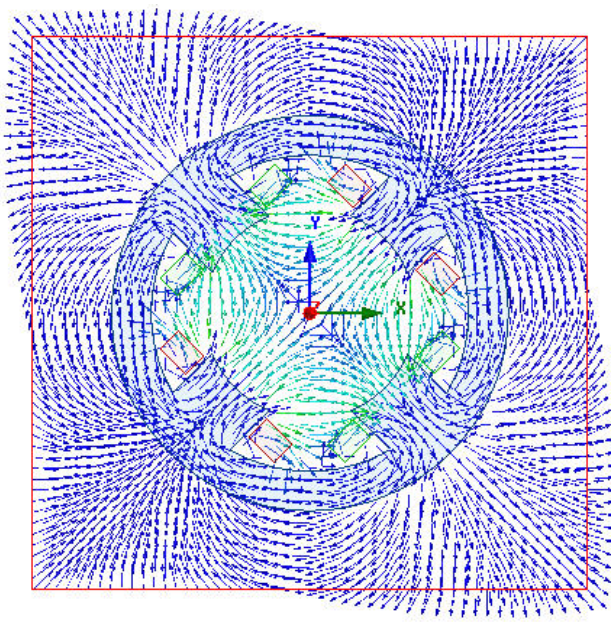
- Analyze
- Examine H Vector Plot







Design2: **2_Balloon**



Design3: **3_Balloon_ChangeExcitation**

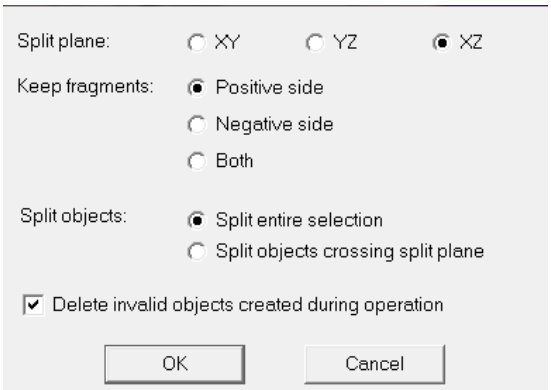
Example 4: Symmetry Boundary - Odd

Copy Design

- **To Create a Copy of the Design**
  - Select the design **"2_Balloon"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"4_Symmetry_Odd"**

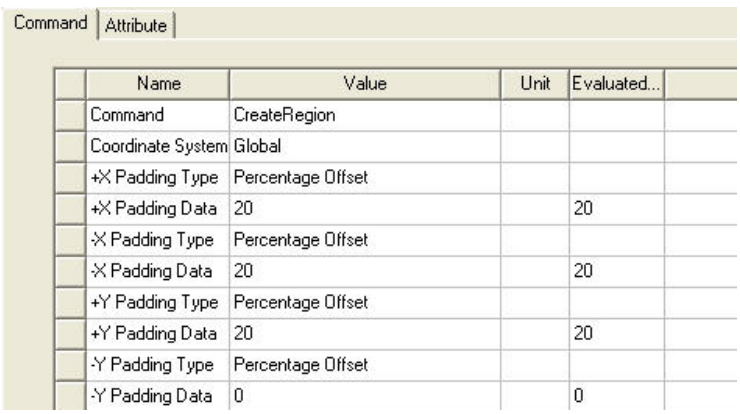
Split Model

- **To Create Half Model**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Modeler > Boolean > Split*
  - In Split window,
    - Set Split Plane to **XZ**
    - Press **OK**



Modify Region

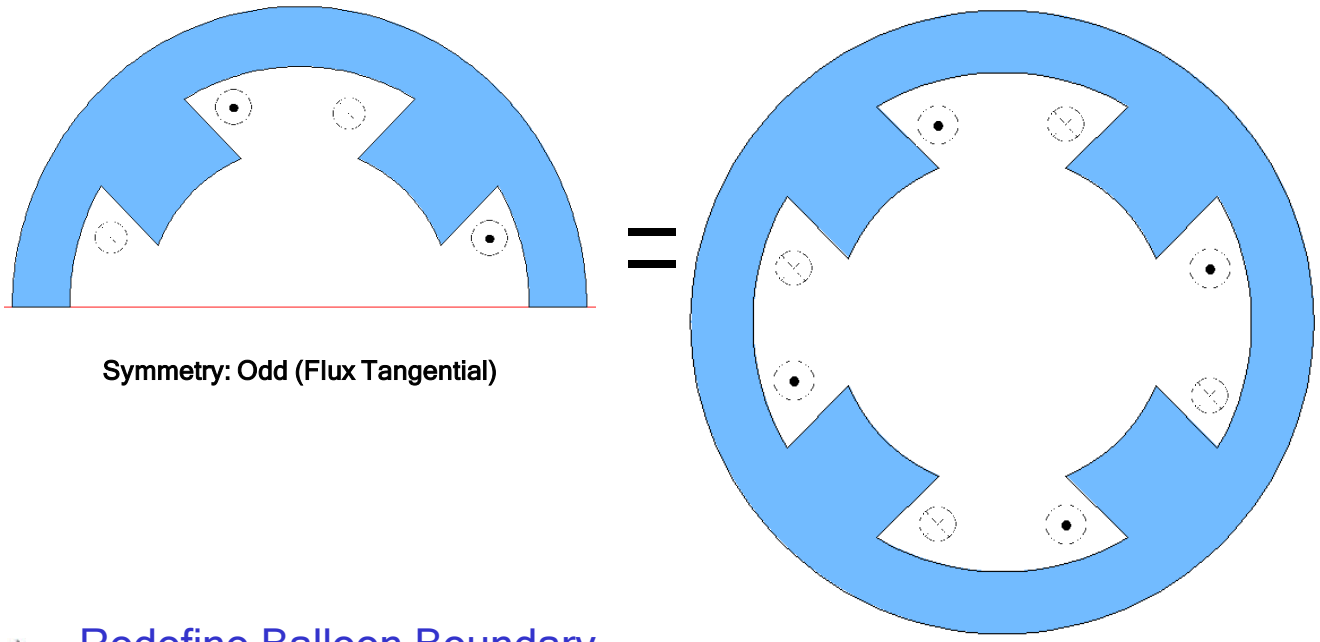
- **To Modify Region**
  - Select the menu item *CreateRegion*
  - In Properties window,
    - Change Padding Data for -Y to **0**
    - Press **OK**



Name	Value	Unit	Evaluated...
Command	CreateRegion		
Coordinate System	Global		
+X Padding Type	Percentage Offset		
+X Padding Data	20		20
-X Padding Type	Percentage Offset		
-X Padding Data	20		20
+Y Padding Type	Percentage Offset		
+Y Padding Data	20		20
-Y Padding Type	Percentage Offset		
-Y Padding Data	0		0

Define Symmetry Boundary

- **To Assign Symmetry**
  - Delete all the boundary conditions defined before
  - Select the menu item *Edit > Select > Edges*
  - Select the bottom edge of the **Region** that coincides with X axis
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Symmetry*
  - In Symmetry Boundary window,
    - Set Symmetry to **Odd (Flux Tangential)**
    - Press **OK**
  - This symmetry definition will be equivalent to full model with excitations as shown in below image



Symmetry: Odd (Flux Tangential)

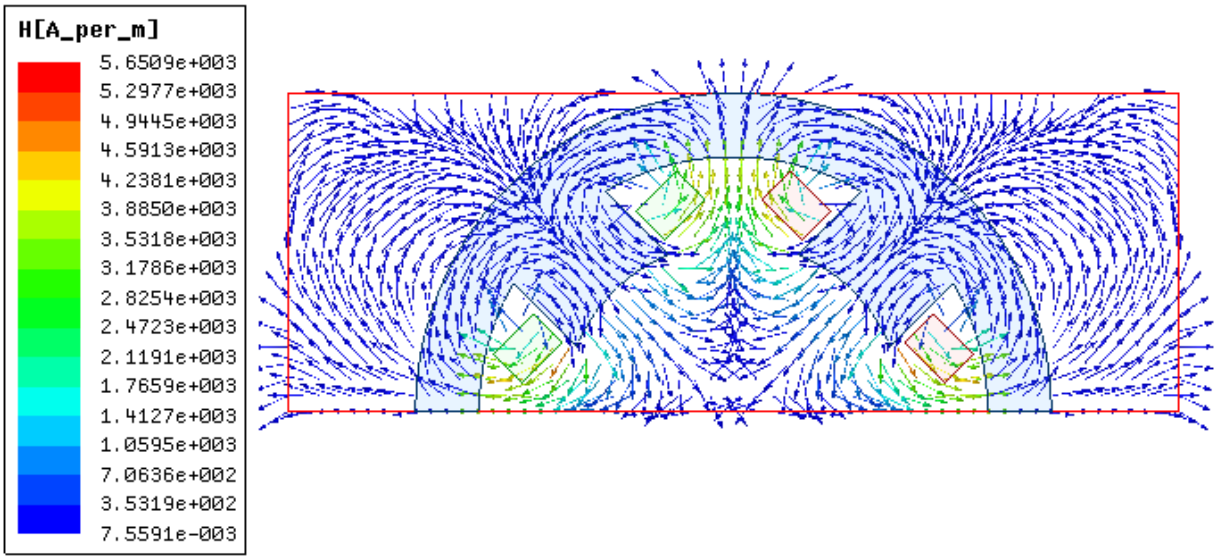
Redefine Balloon Boundary

- **To Specify Balloon**
  - Select all edges of the region except the one to which Symmetry is defined
  - Expand the Project Manager tree to view **Boundaries**
  - Right click on the tab **Balloon1** and select Reassign
  - This will assign selected edges to the Balloon boundary and symmetry edge will not be specified as balloon

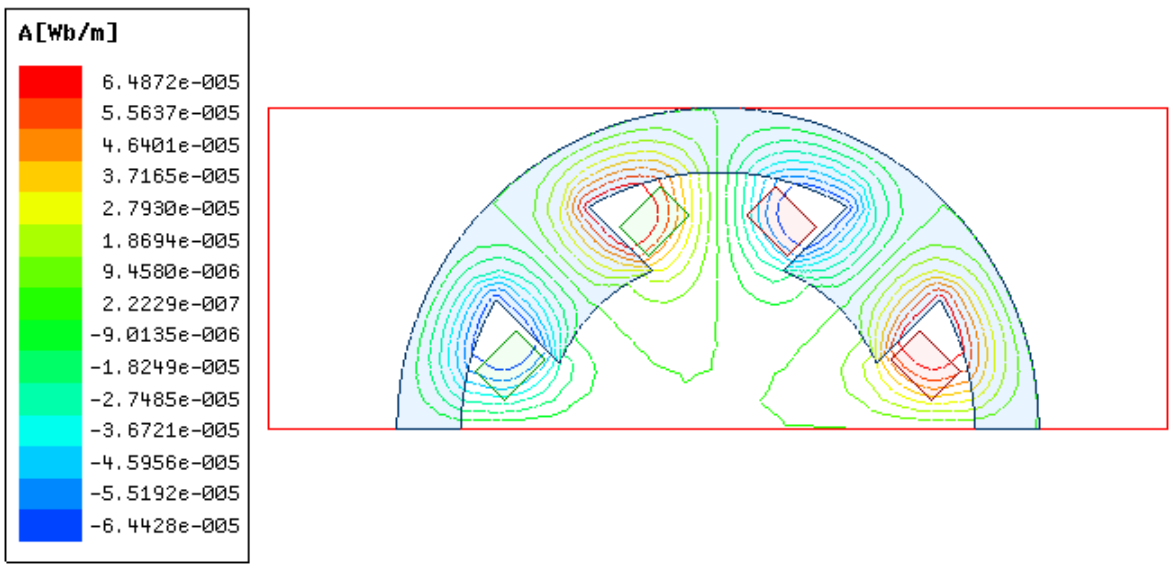
Analyze

Results

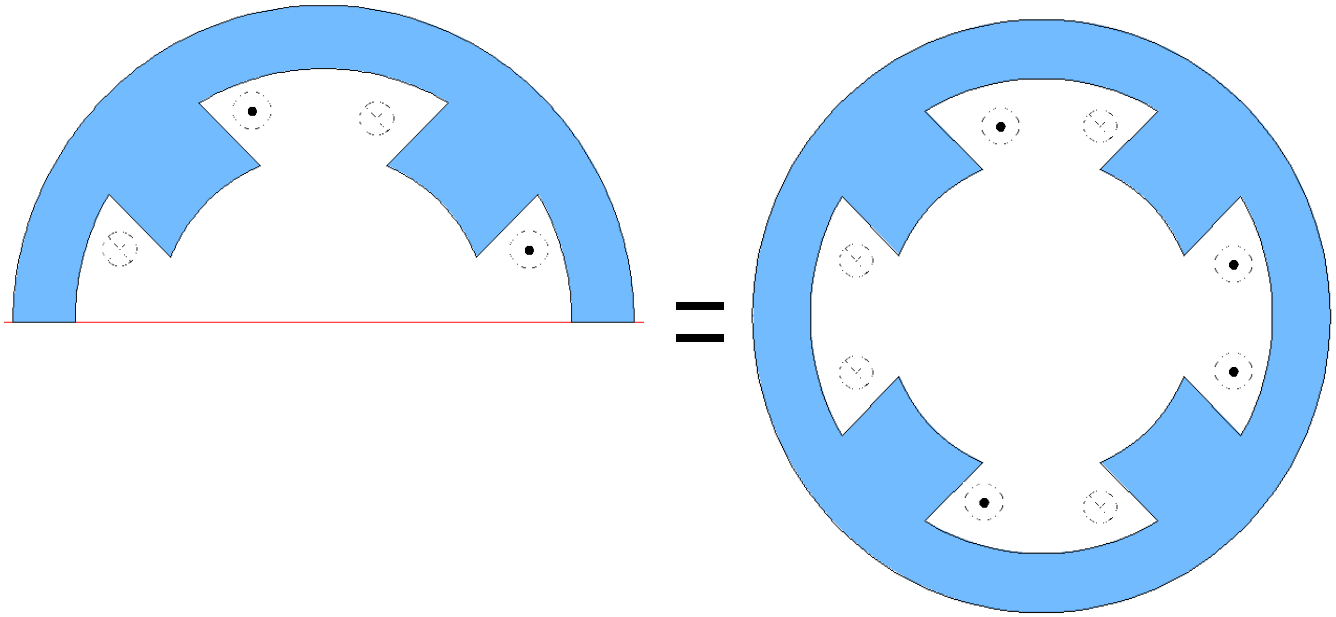
H Vector



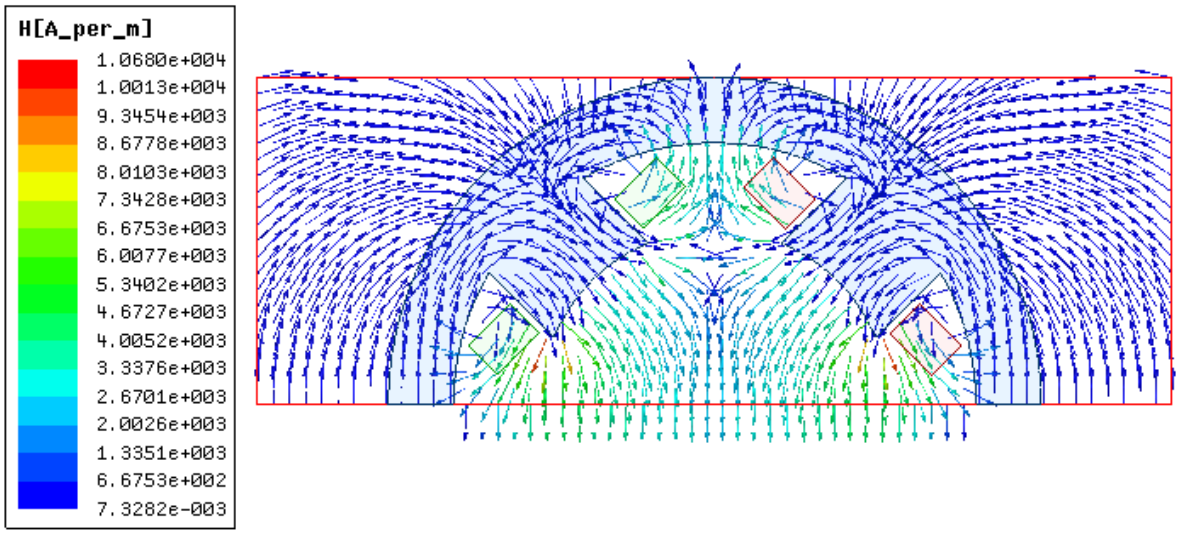
Field Lines



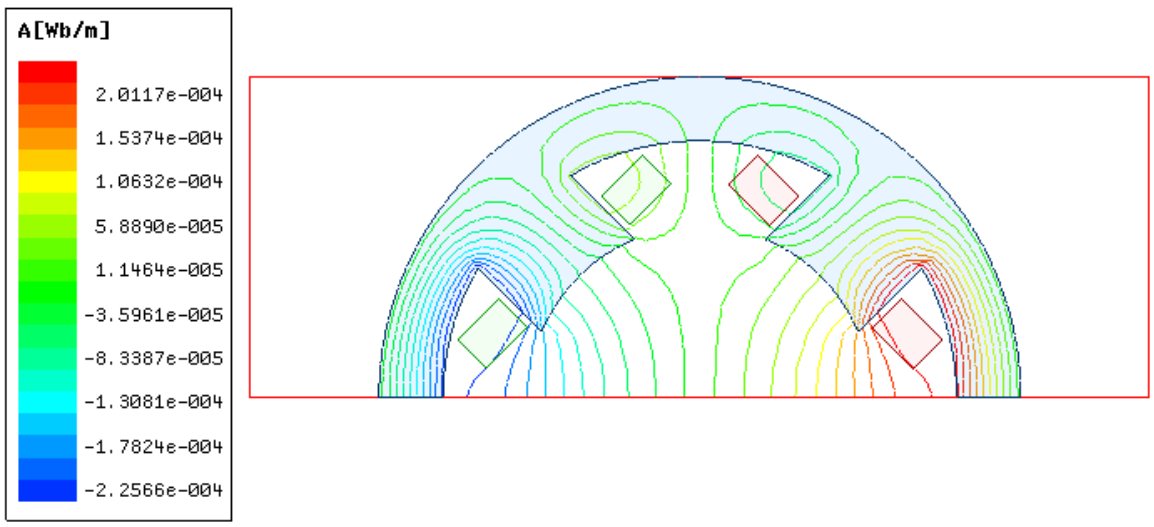
- Example 5: Symmetry Boundary - Even
- Copy Design
  - **To Create a Copy of the Design**
    - Select the design **"4_Symmetry_Odd""** from Project Manager tree, right click and select **Copy**
    - Right click on the Project name in Project Manager tree and select **Paste**
    - Rename the newly created design to **"5_Symmetry_Even"**
- Modify Symmetry
  - **To Modify Symmetry**
    - Expand the Project Manager tree to view **Boundaries**
    - Double click on the tab **Symmetry1**
    - In Symmetry Boundary window,
      - Change Symmetry to **Even (Flux Normal)**
      - Press **OK**
    - This symmetry definition will be equivalent to full model with excitations as shown in below image



- Analyze
- Results
  - H Vector



  - Field Lines



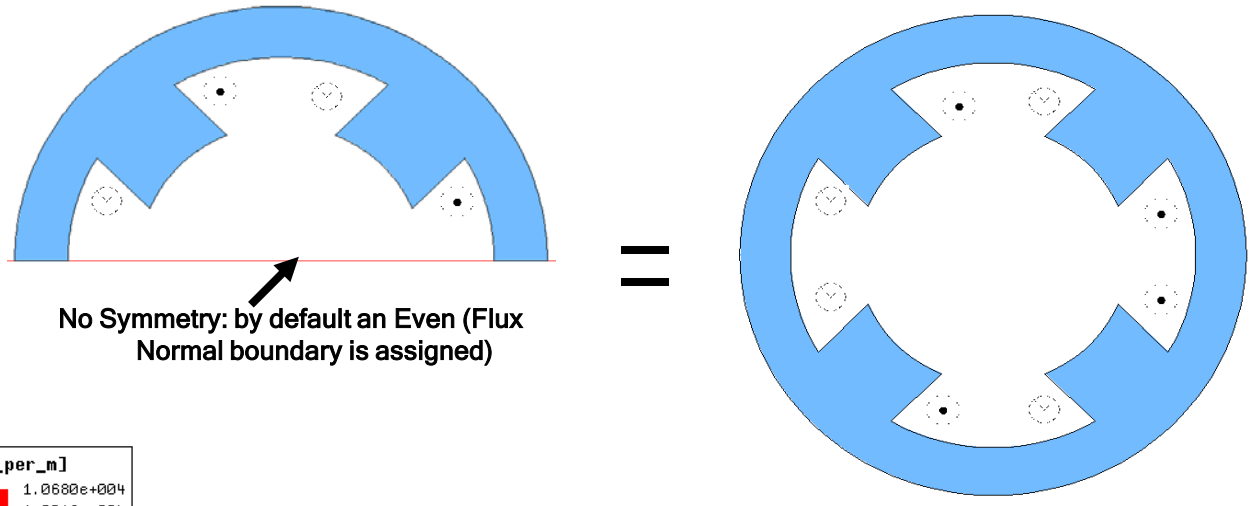
Example 6: No Symmetry

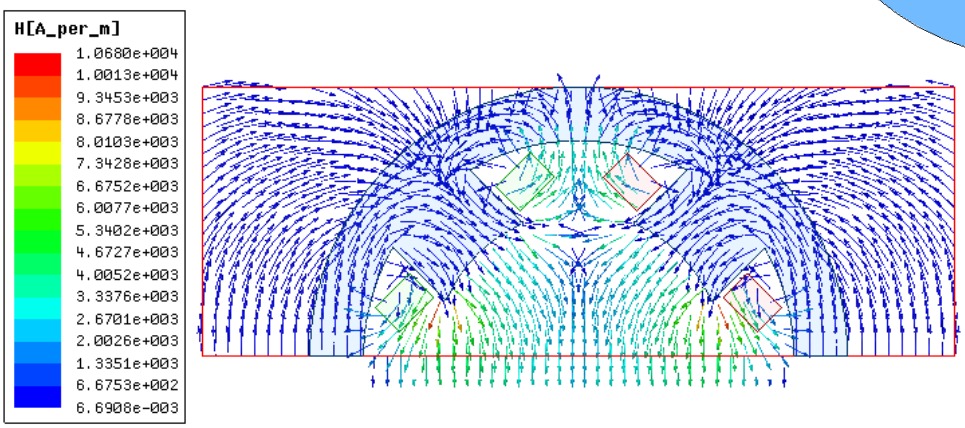
Copy Design

- **To Create a Copy of the Design**
  - Select the design **"4_Symmetry_Odd"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"6_NoSymmetry"**

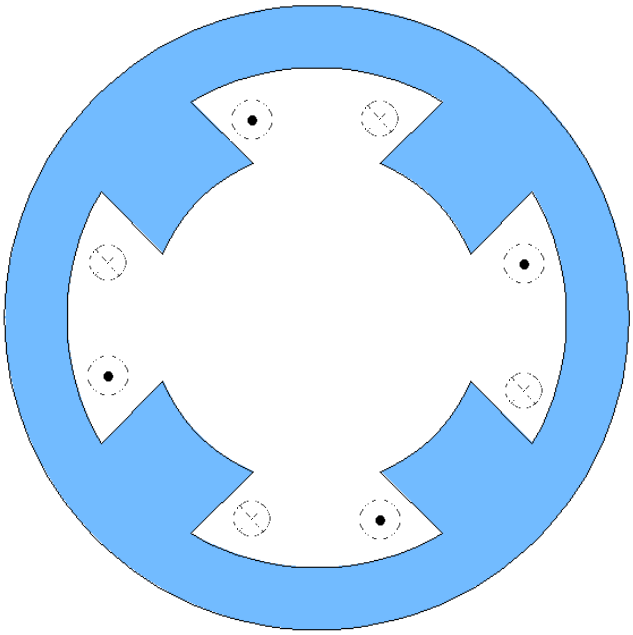
Delete Symmetry

- **To Delete Symmetry**
  - Expand the Project Manager tree to view **Boundaries**
  - Double click on the tab **Symmetry1**
  - Press **Delete** button from keyboard
- The result should look the same as Design4: **4_Symmetry_Even** because Even symmetry or flux normal is the default boundary condition.

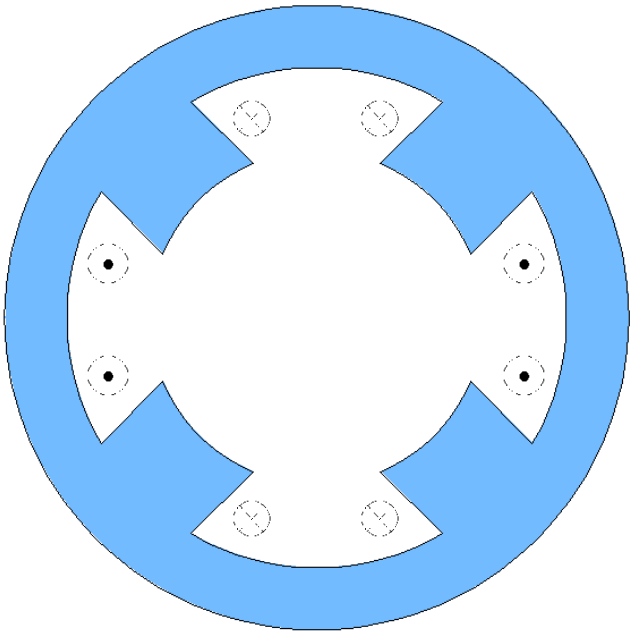




Matching Boundaries (Master/Slave).

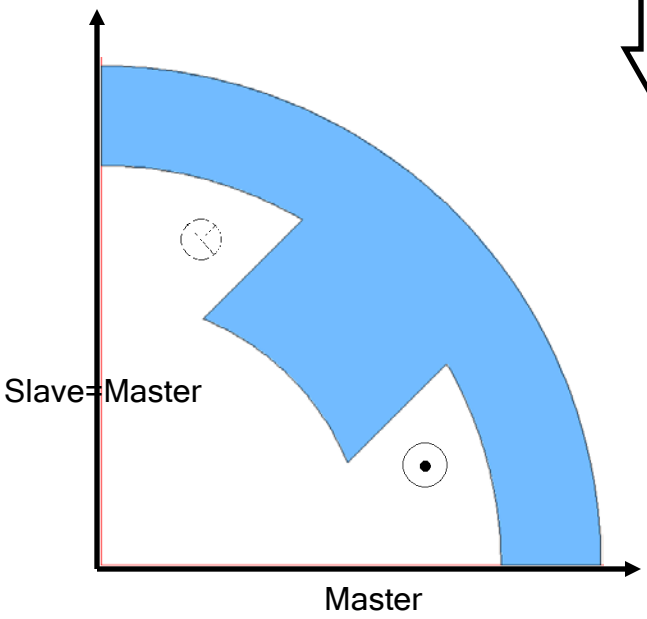


Design2: **2_Balloon**

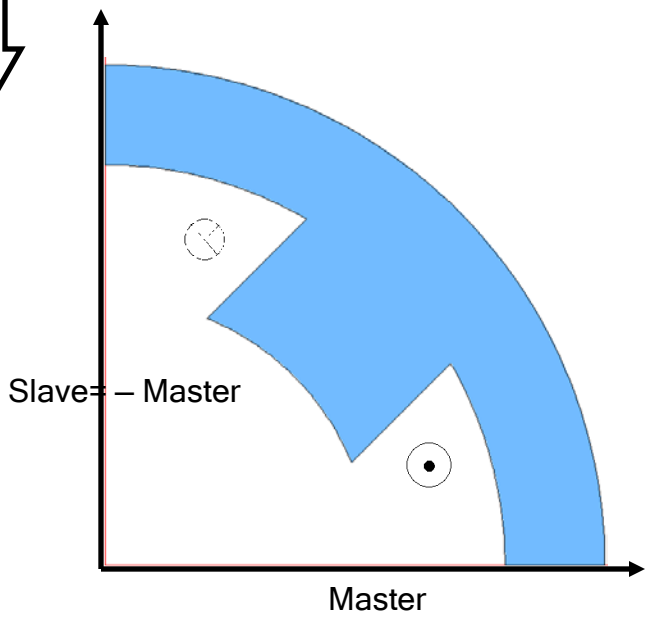


Design3: **3_Balloon_ChangeExcitation**

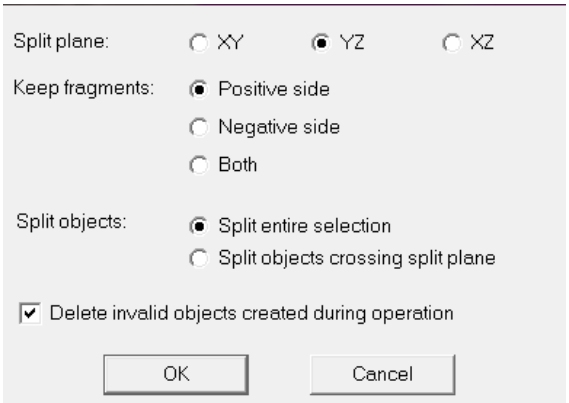
Design7: **7_Matching_Positive**



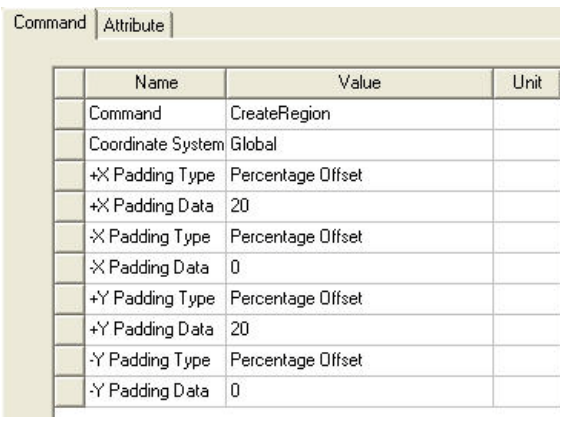
Design8: **8_Matching_Negative**



- Example 7: Matching Boundaries (Master = Slave)
- Copy Design
  - **To Create a Copy of the Design**
    - Select the design **"6_NoSymmetry"** from Project Manager tree, right click and select **Copy**
    - Right click on the Project name in Project Manager tree and select **Paste**
    - Rename the newly created design to **"7_Matching_Positive"**
    - Delete all previous boundary assignments
- Split Model
  - **To Create Half Model**
    - Select the menu item *Edit > Select All*
    - Select the menu item *Modeler > Boolean > Split*
    - In Split window,
      - Set Split Plane to **YZ**
      - Press **OK**

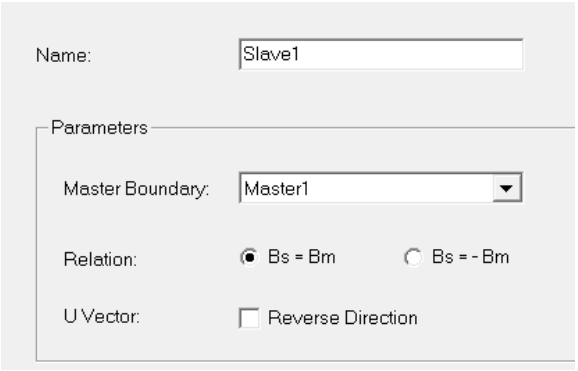


- Modify Region
  - **To Modify Region**
    - Select the menu item *CreateRegion*
    - In Properties window,
      - Change Padding Data for -X to **0**
      - Press **OK**



Define Master/Salve

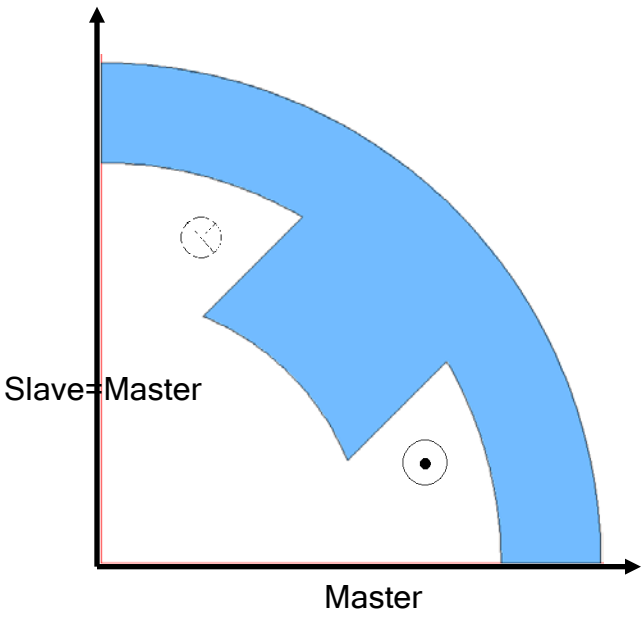
- **To Define Master Boundary**
  - Select the menu item *Edit > Select > Edges*
  - Select the edge of the **Region** that coincides with X axis
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Master*
  - In Master Boundary window,
    - Ensure U Vector is pointing in positive X direction.
    - If not use **Reverse Direction**
    - Press **OK**
- **To Define Slave Boundary**
  - Select the edge of the **Region** that coincides with Y axis
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Slave*
  - In Master Boundary window,
    - Specify Master Boundary as **Master1**
    - Set Relation to **Bs= Bm**
    - Ensure U Vector is pointing in positive Y direction.
    - If not use **Reverse Direction**
    - Press **OK**

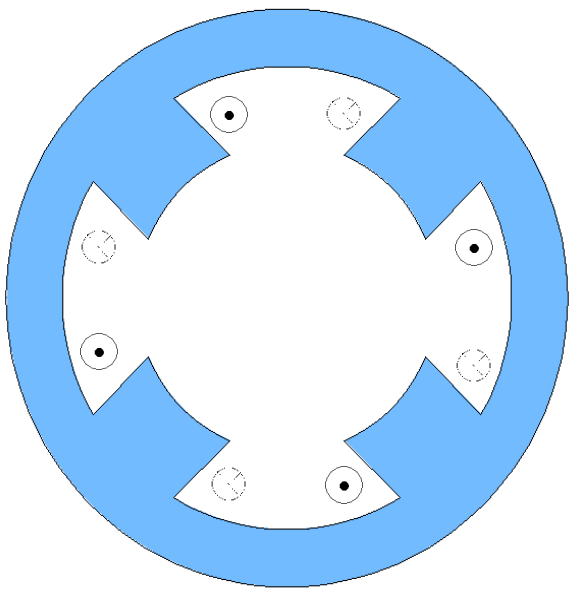


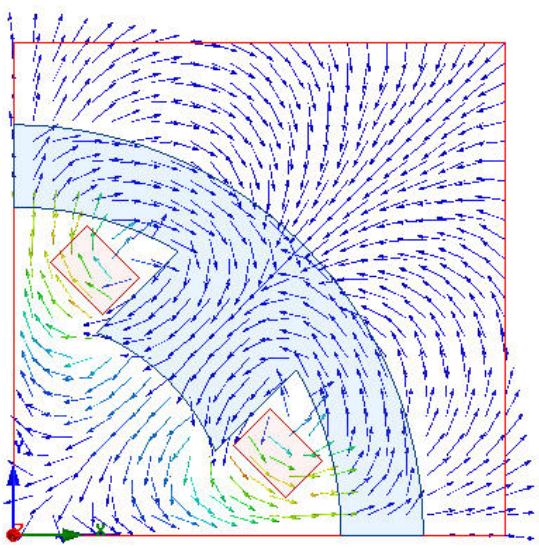
Redefine Balloon Boundary

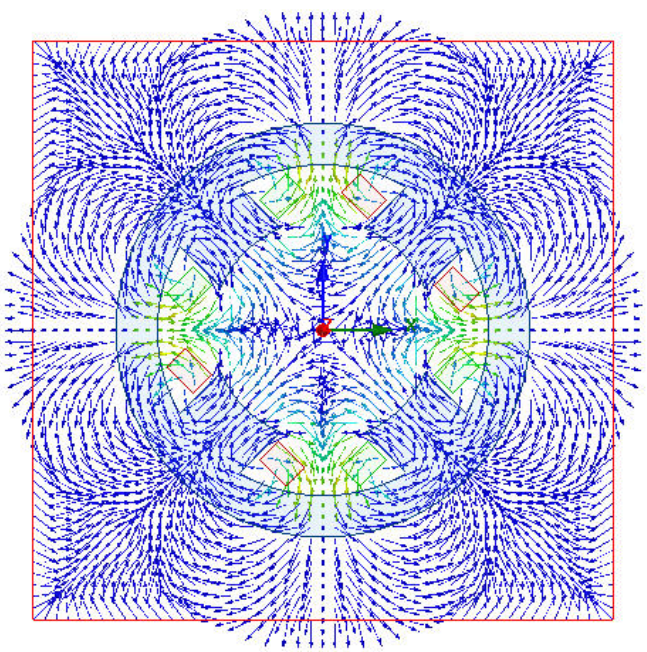
- **To Specify Balloon**
  - Select all edges of the region except the ones to which Master or Salve boundary is defined
  - Expand the Project Manager tree to view **Boundaries**
  - Right click on the tab **Balloon1** and select Reassign
  - This will assign selected edges to the Balloon boundary and symmetry edge will not be specified as balloon

- Analyze
- Results

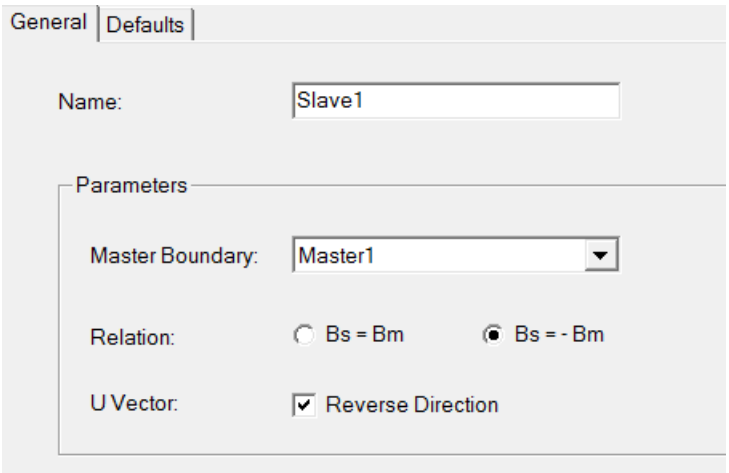






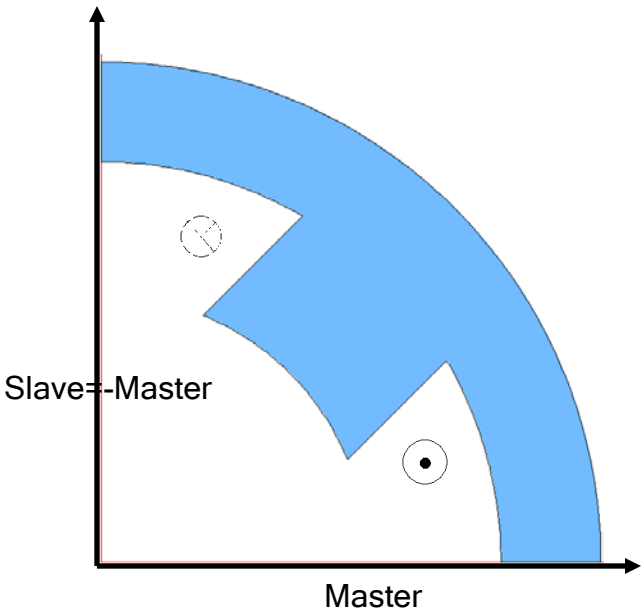


- Example 8: Matching Boundaries (Master = - Slave)
- Copy Design
  - **To Create a Copy of the Design**
    - Select the design **"7_Matching_Positive"** from Project Manager tree, right click and select **Copy**
    - Right click on the Project name in Project Manager tree and select **Paste**
    - Rename the newly created design to **"8_Matching_Negative"**
- Modify Slave Boundary
  - **To Modify Slave Boundary**
    - Expand the Project Manager tree to view **Boundaries**
    - Double click on the tab **Slave1**
    - In Slave Boundary window,
      - Change the relation to **Bs = -Bm**
      - Be sure that U vector is pointing positive Y direction and change the relation if needed
      - Press **OK**

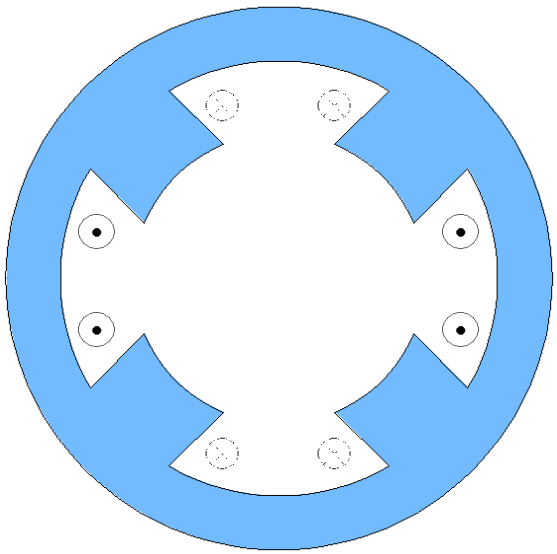


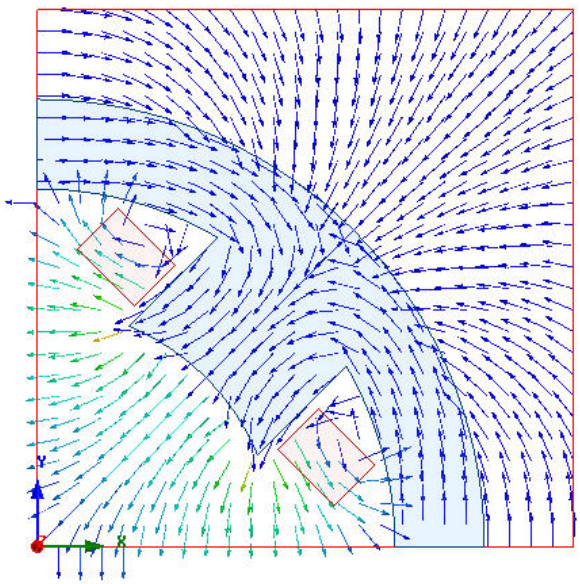
- Analyze

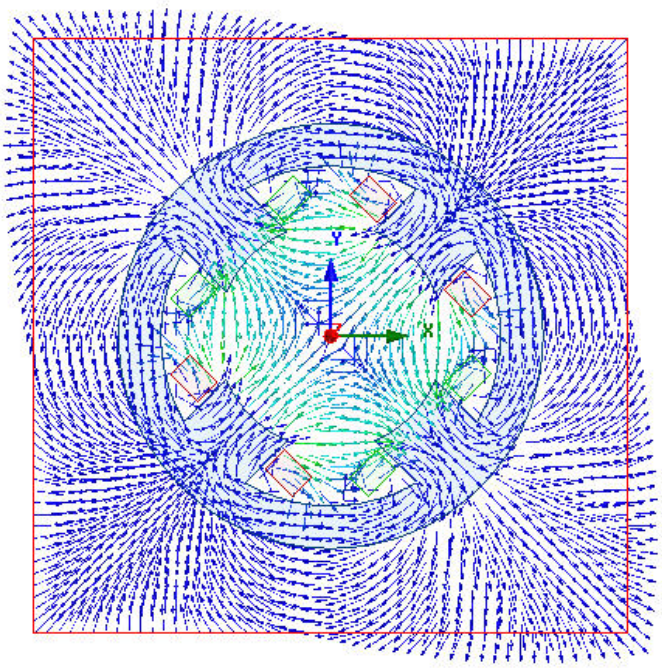
Results



=







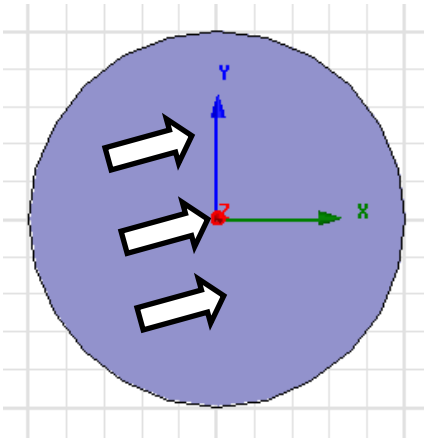
Basic Exercises – Permanent Magnet Assignment

Permanent Magnets Assignment

- This exercise will discuss how to set up a permanent magnet (PM) material to an object. This procedure is applicable for Magnetostatic and Transient Solvers.

Problem definition

- We are interested to solve the magnetic field of a Circular PM placed in vacuum. The material of PM is NdFeB35 and the magnet is magnetized in the direction 30 degrees relative to the Global X direction.
- This can be achieved by two methods
  - By defining components for material
  - By Defining coordinate system in required direction



Create Design

- **To Create Design**
  - Select the menu item *Project > Insert Maxwell 2D Design*, or click on the icon

Set Solution Type

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

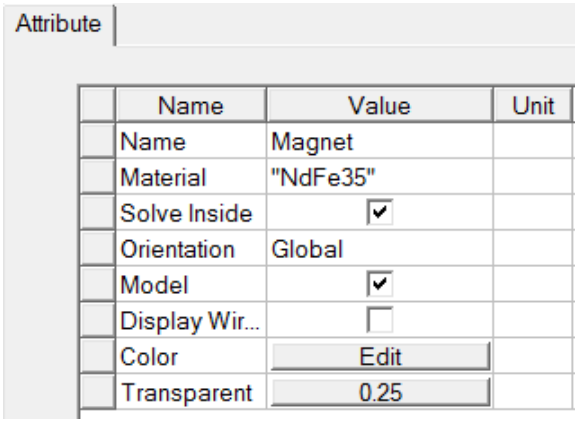
Basic Exercises – Permanent Magnet Assignment

Create Model

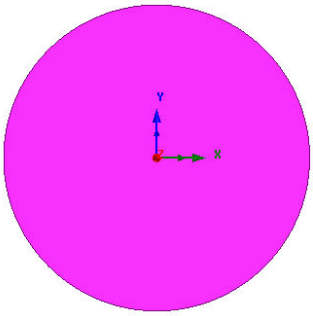
- **To Create Model**
  - Select the menu item *Draw > Circle*
    1. Using the coordinate entry fields, enter the center of circle
       - X: **0**, Y: **0**, Z: **0**, Press the **Enter** key
    2. Using the coordinate entry fields, enter the radius of circle
       - dX: **1**, dY: **0**, dZ: **0**, Press the **Enter** key
- **Change Attributes**
  - Change the name of the resulting sheet to **Magnet**
  - Change Material to **NeFe35**

Object Orientation

- Each object in Maxwell is associated with certain coordinate system. This is called *Orientation* and it is specified under attributes for each object.
- **To Check the Orientation of the Sheet**
  - Double click on the sheet **Magnet** from history tree to open Properties window
  - In Properties window,
    - Orientation is specified as **Global**
    - This means that our object **magnet** is currently associated with the Global coordinate system. Global Coordinate system exists by default in a newly inserted Maxwell Design. Left-clicking on Global allows changing the *Orientation* to other coordinate system, provided, of course, that some other coordinate system exists.
    - Press **OK** to close the window

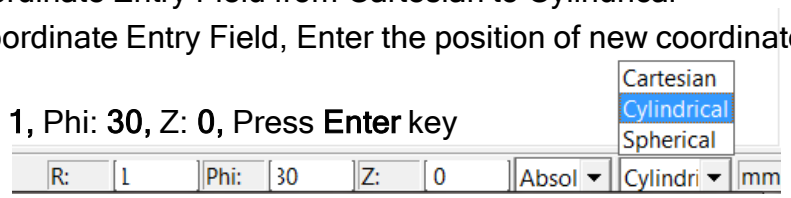


- Orientation can also be viewed graphically. First, make sure that this feature is enabled:
- **To view Orientation Graphically**
  - Select the menu item *Tools > Options > Modeler Options*
  - In the window,
    - **Display** tab
      - Enable the option **Show orientation of selected object**
  - After the object is selected from Graphic window, orientation coordinate system can be seen in small associated with each object.



Create Relative CS

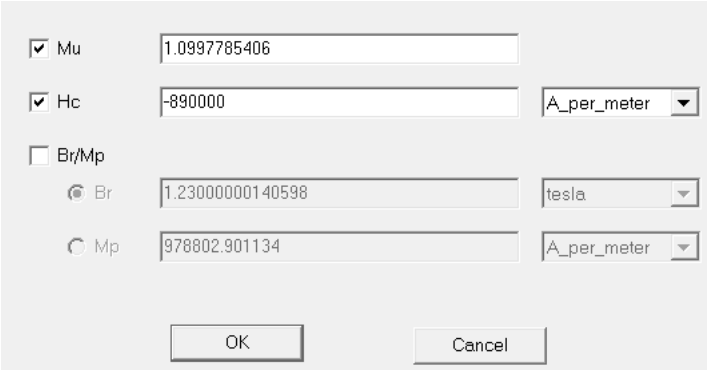
- All the magnetic materials in Material library of Maxwell are magnetized in X direction. Hence once a magnetic material is applied to any object, it is magnetized in X direction of its orientation coordinate system
- Alternately, magnetic material can be edited to change its direction of magnetization.
- **To Create Relative CS**
  - Select the menu item *Modeler > Coordinate System > Create > Relative CS > Rotated*
  - Change the Coordinate Entry Field from Cartesian to Cylindrical
    - Using Coordinate Entry Field, Enter the position of new coordinate system
      - R: **1,** Phi: **30,** Z: **0,** Press **Enter** key



    - Note :If you want to specify in Cartesian coordinates, enter the tangent of the angle instead. as X: 1, Y: 0.57735, Z: 0
  - The newly created CS automatically becomes a working CS (small **w** sign is displayed next to the icon of a Working CS).
  - Select the menu item *Modeler > Coordinate System > Set Work CS*
    - Select **Global**

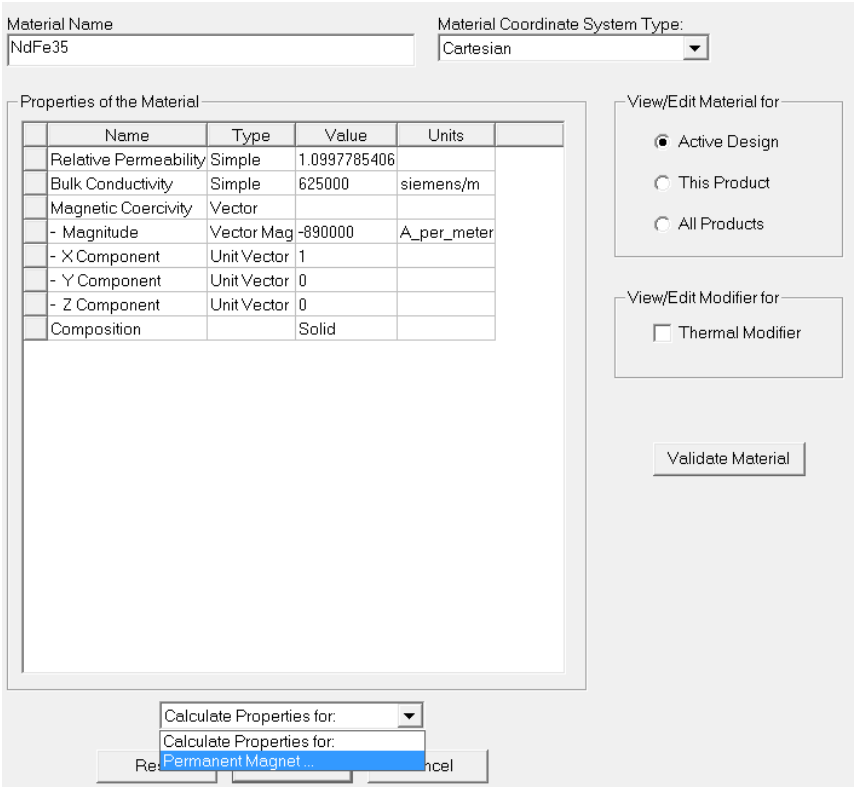
Assign Material

- **To Assign Magnetic Material to the Sheet**
  - Select sheet **Magnet** from history tree, right click and select **Assign Material**
  - In Select Definition window,
    - Select material **NdFe35** from the list and select **View/Edit Material**
  - In View/Edit Material window,
    - A Permanent Magnet (PM) with linear characteristic is uniquely defined by specifying two of the following: Coercive Field, Remanent Flux Density, Relative Permeability. Coercive Field and Relative Permeability are chosen by default to be specified. If any other combination of quantities is known instead, select **Calculate Properties for PM** and specify the two known quantities. The remaining quantity will be determined automatically.



    - Direction of magnetization is specified by the unit vector relative to the object orientation. If the object is oriented with respect to **Global** coordinate system, unit vector is specified relative to **Global.**
      - As X component of unit Vector is **1,** and Y,Z components are **0,** unit vector is in X direction of orientation coordinate system
      - Hence the object **Magnet** is now oriented in X direction of **Global** coordinate system . We will change the orientation of the **Magnet** on the next page to alter the direction.
    - **Material Coordinate System** defines if Magnetization is in Cartesian or cylindrical coordinate system
      - If Cylindrical Coordinate system is used, unit vectors will change to **R, Phi** and **Z**
    - Select **OK** to assign material

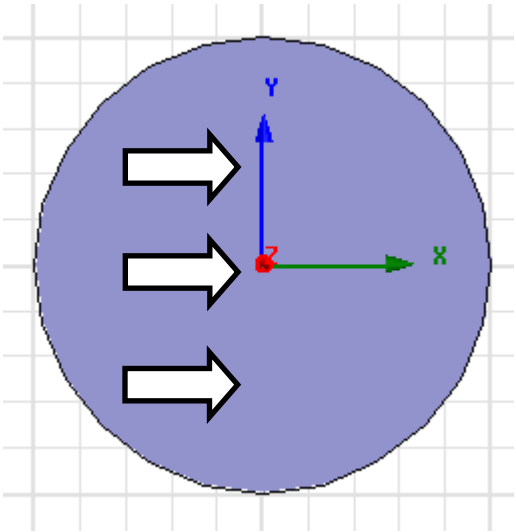
Basic Exercises - Permanent Magnet Assignment



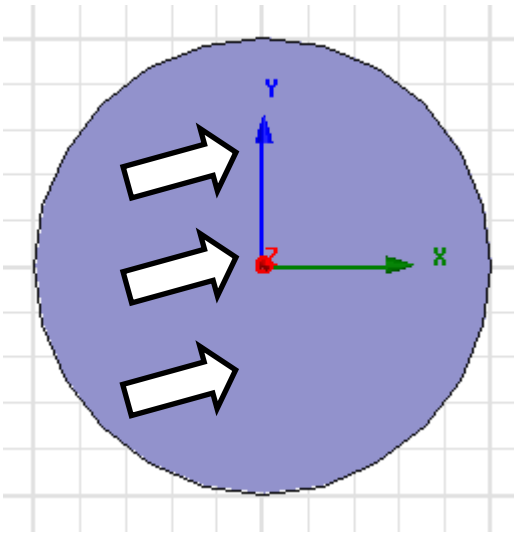
Magnetization Directions with various Options

Effect of Orientation Coordinate System

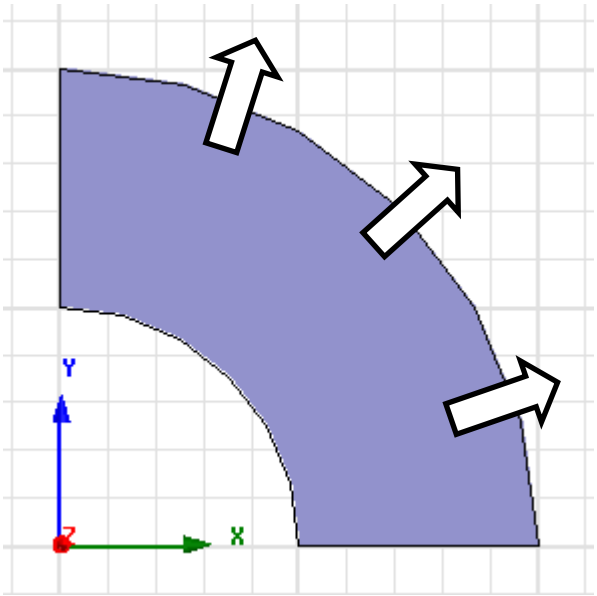
- Orientation: **Global,** Material Coordinate System: **Cartesian,** Unit Vectors (X,Y,Z) : **1, 0,0**



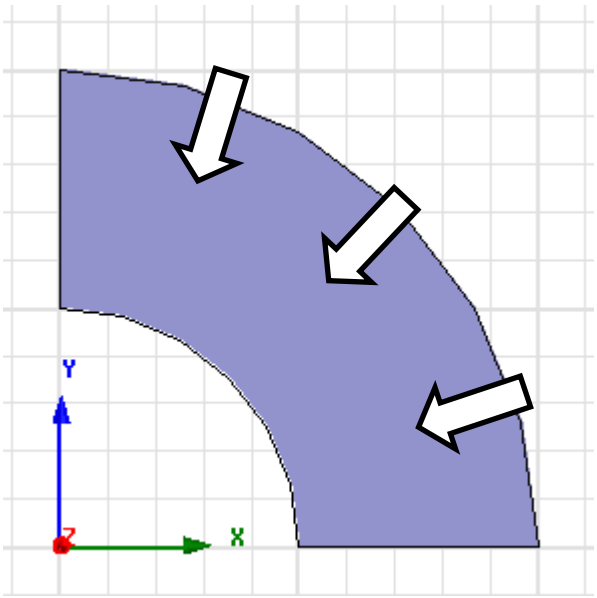
- Orientation: **RelativeCS1,** Material Coordinate System: **Cartesian,** Unit Vectors (X,Y,Z) : **1, 0,0**



- **Effect of Material Coordinate System**
  - Orientation: **Global,** Material Coordinate System: **Cylindrical,** Unit Vectors (R,Phi,Z) : **1, 0,0**

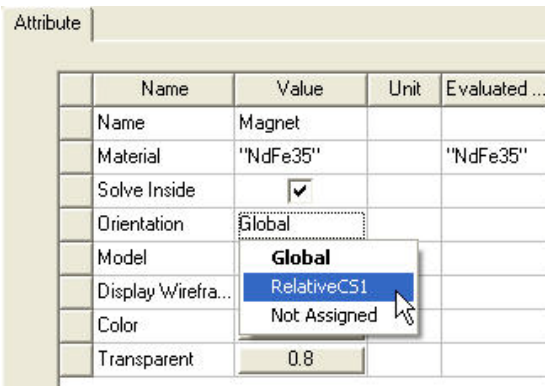


- Orientation: **Global,** Material Coordinate System: **Cylindrical,** Unit Vectors (R,Phi,Z) : **-1, 0,0**



Change Magnet Orientation

- **To Change the Orientation of the Magnet**
  - Double click on the sheet **Magnet** from history tree to open Properties window
  - Left click in the Value column for **Orientation** and select **RelativeCS1**
  - Now Magnet is oriented with respect to RelativeCS1. Once the magnetic material is applied to the sheet, the object will be Magnetized in X direction of RelativeCS1



Basic Exercises – Permanent Magnet Assignment

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: **500**
    4. Press **OK**

Assign Boundary

- **To Assign Boundary to Region Edges**
  - Select the object **Region** from history tree
  - Select the menu item *Edit > Select > All Object Edges*
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Vector Potential*
  - In Vector Potential Boundary window,
    - Leave the Value to **0**
    - Press **OK**

Analysis Setup

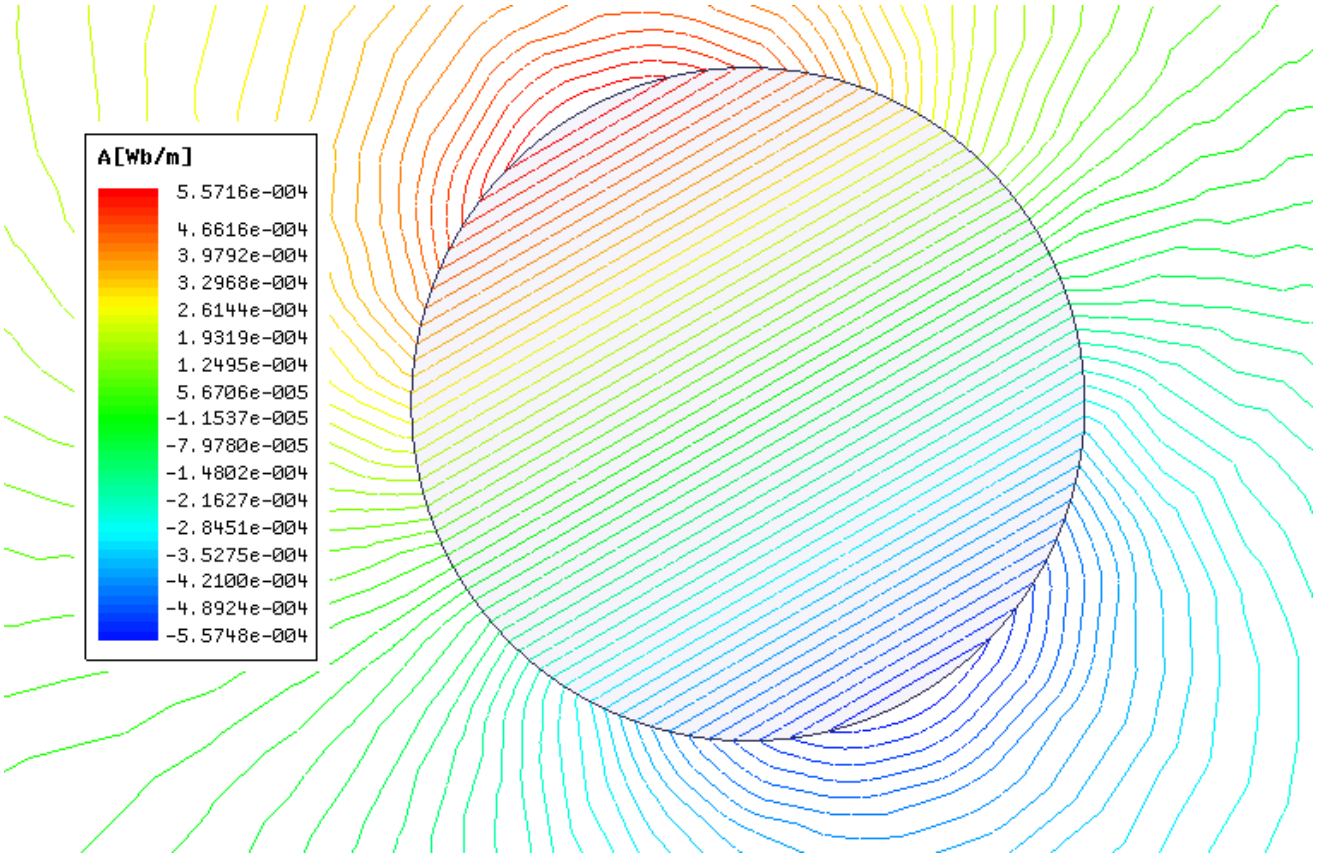
- **To Create Analysis Setup**
  - Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
  - In Solve Setup window,
    - Press **OK**

Analyze

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

Plot Results

- **To Plot Flux Lines**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window,
    - Press **Done**
- **Modify Attributes**
  - Double click on the legend to modify plot attributes
  - In the window,
    - **Scale** tab
      - Change Num. Division to **50**
    - Press **OK**



ANSYS, Inc. Electromagnetic Products

Maxwell® 15.0



ANSYS

Realize Your Product Promise™

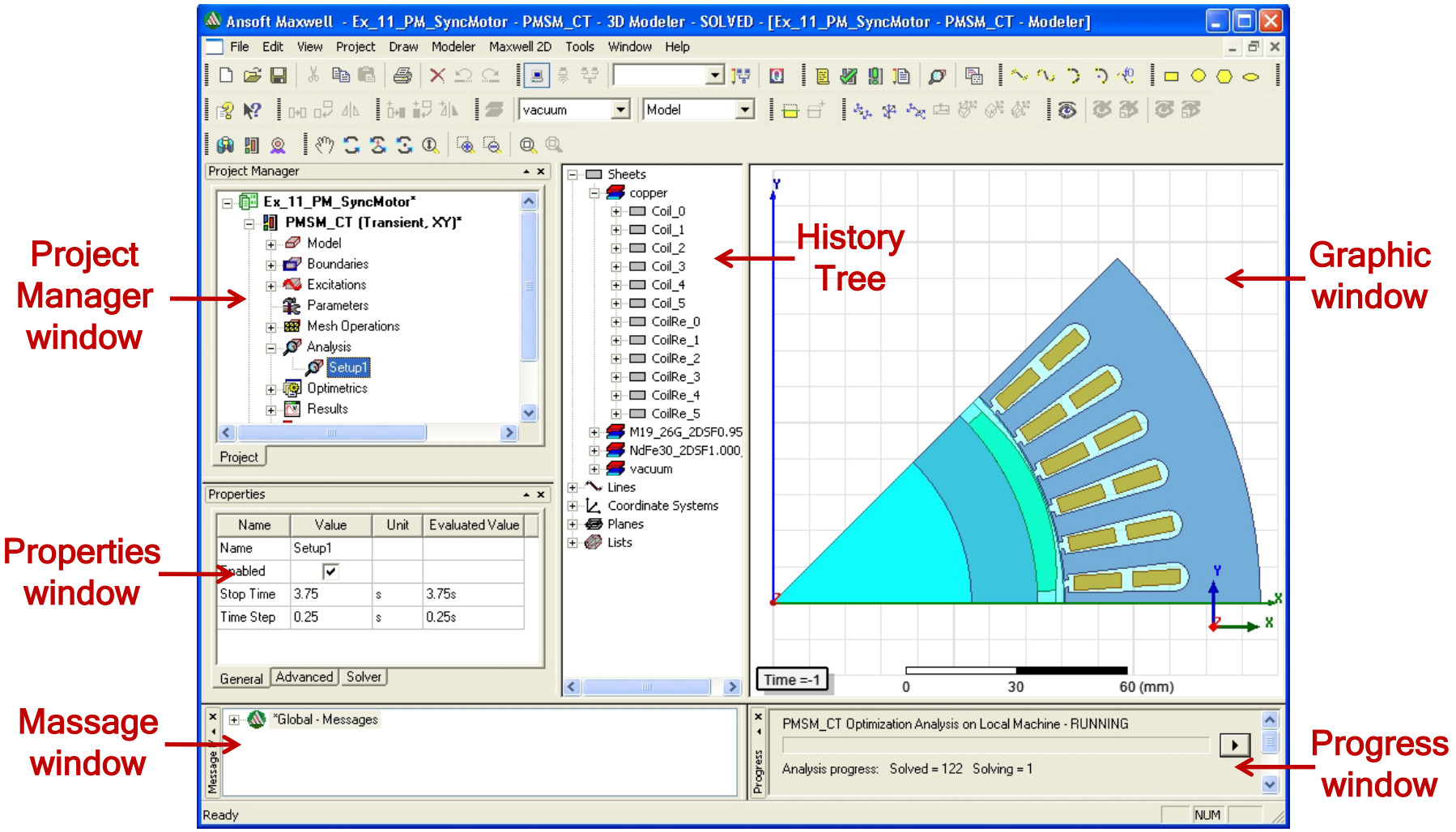
Electrical Machine Design Reference

Content

- General Tips - 3
- RMxprt → Maxwell - 8
- Geometry Creation - 11
- Material Assignment - 14
- Excitation Setup - 20
- Boundary Conditions - 24
- Mesh Operations - 27
- Solution Setup - 31
- Force / Torque Calculation - 35
- Inductance Calculation - 36
- Loss Calculation - 39
- View Results - 41
- Parameter Sweep and Optimization - 44
- Increasing Simulation Speed - 47

General Tips

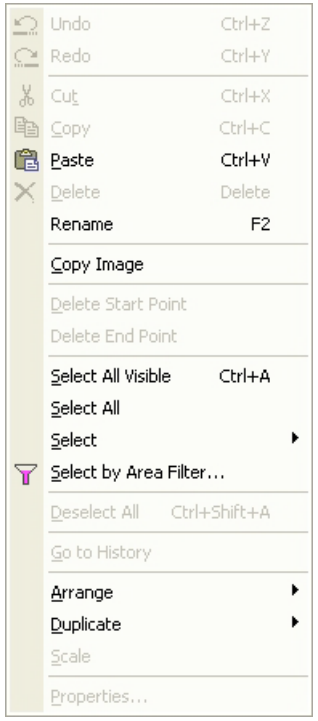
1. Project Interface



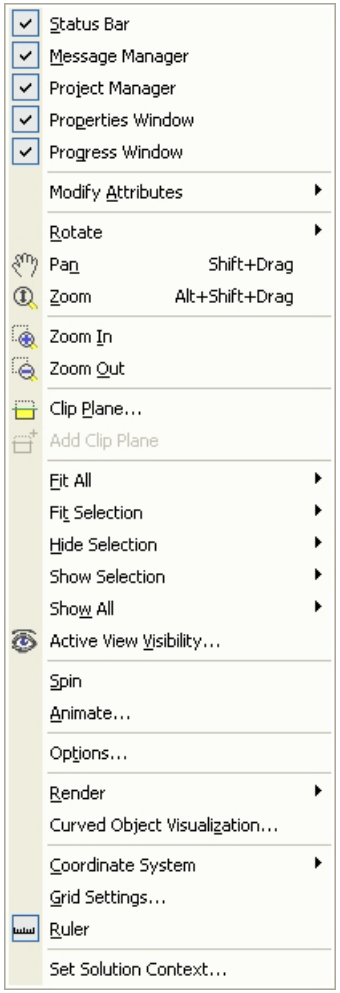
General Tips

2. Executing Commands from Menu Bar

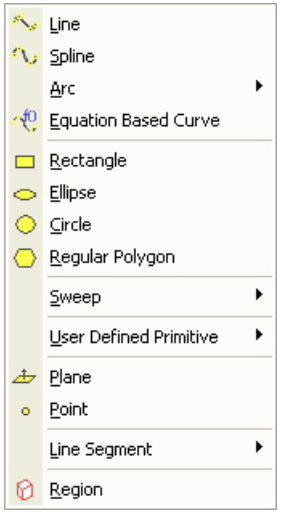
Edit



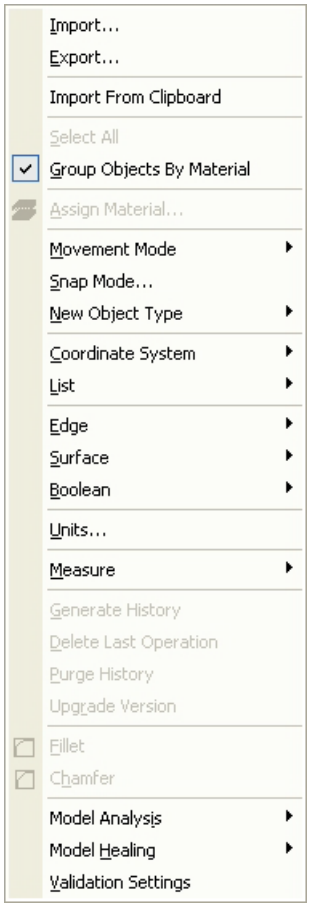
View



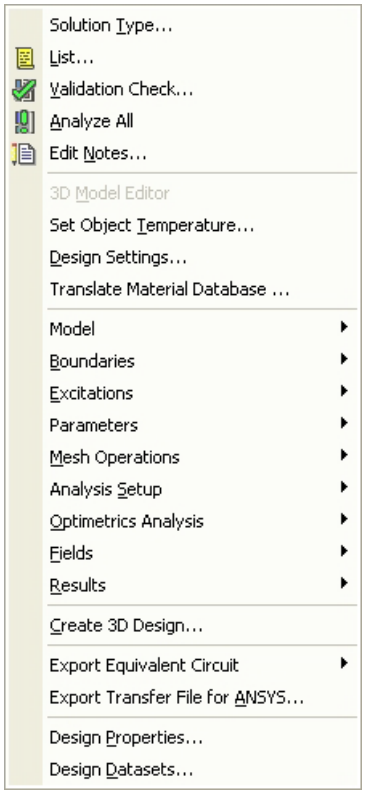
Draw



Modeler



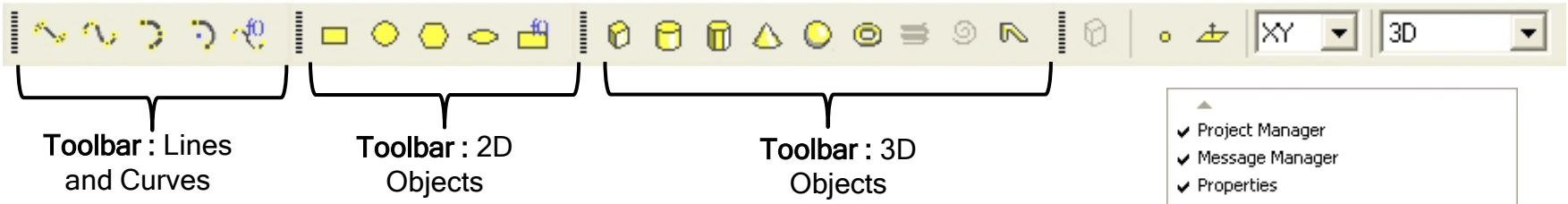
Maxwell 2D/3D



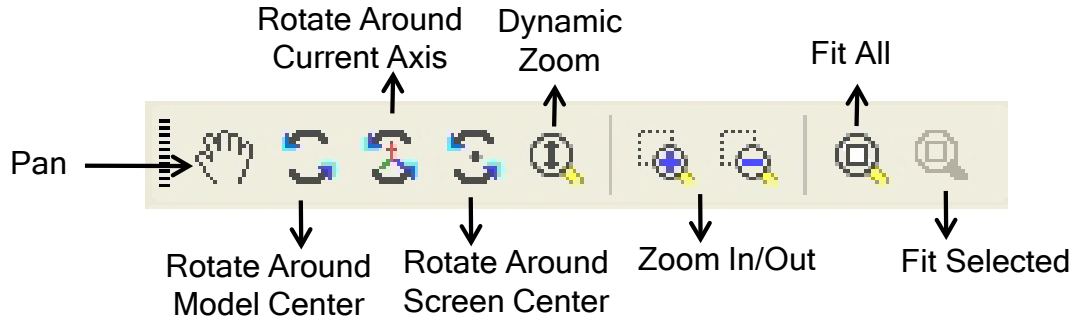
General Tips

3. Executing Commands from Toolbar Icons

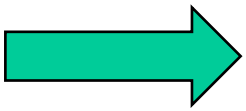
Modeler Toolbar

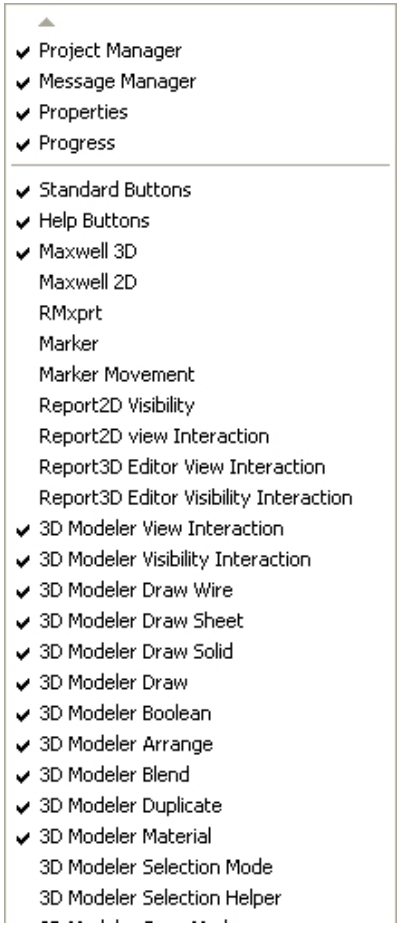


Geometry Manipulation Toolbar



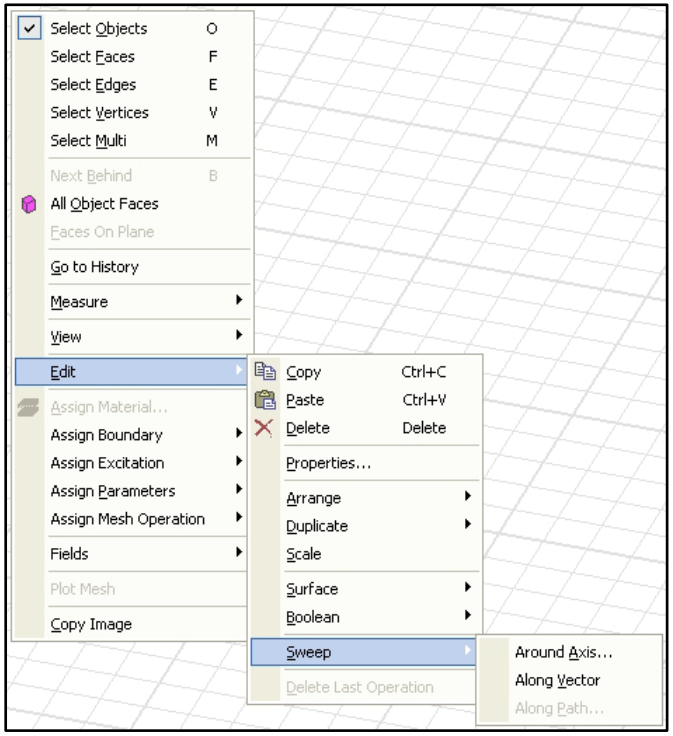
- **Displaying or Hiding Individual Toolbars**
  - Right Click on Maxwell window frame
    - A list of all Toolbars is displayed
    - The toolbars with a checkmark besides them are visible while others are invisible
    - Click the Toolbar name to turn its display On or Off

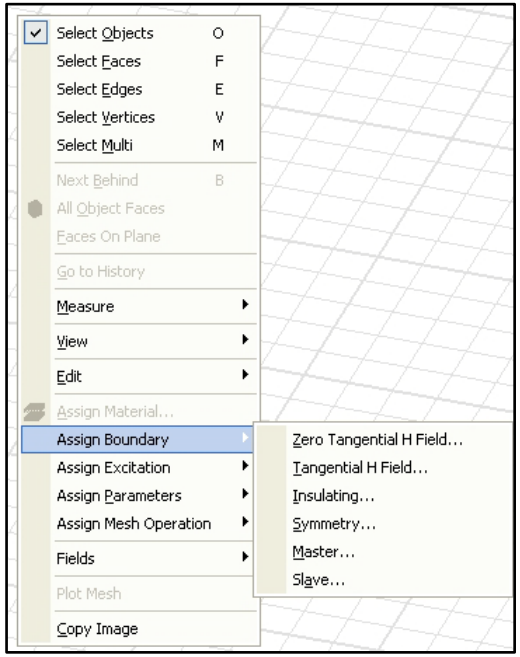


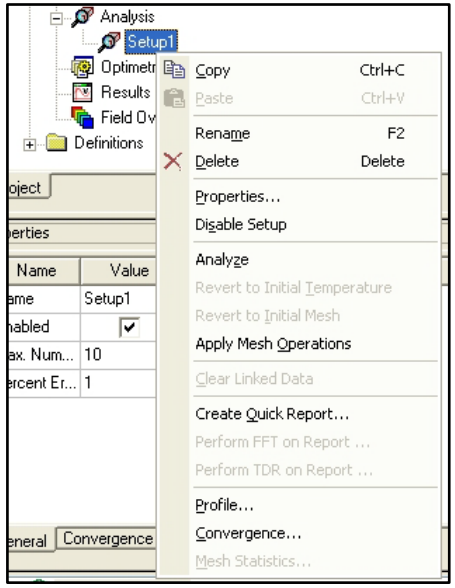


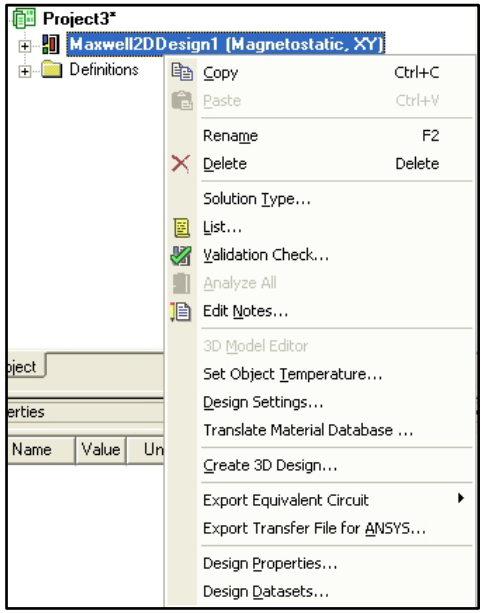
General Tips

4. Executing Commands By Right Mouse Click









- If Don't Know where to find a command
  - Try a Right mouse click

General Tips

5. Executing Commands using Keyboard Shortcuts

General Shortcuts

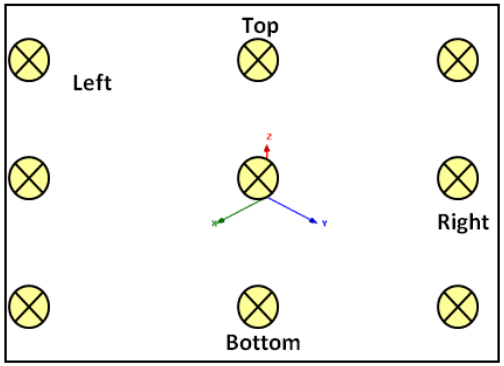
- Help: **F1**
- Context help: **Shift + F1**
- Close program: **CTRL + F4**
- Copy: **CTRL + C**
- New project: **CTRL + N**
- Open: **CTRL + O**
- Save: **CTRL + S**
- Print: **CTRL + P**
- Paste: **CTRL + V**
- Cut: **CTRL + X**
- Redo: **CTRL + Y**
- Undo: **CTRL + Z**
- Cascade windows: **CTRL +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

Modeler Shortcuts

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

- Set model projection to standard isometric projections: Alt + Double Click Left Mouse Button at points on screen
- ALT + Right Mouse Button + Double Click Left Mouse Button at points on screen: give the nine opposite projections

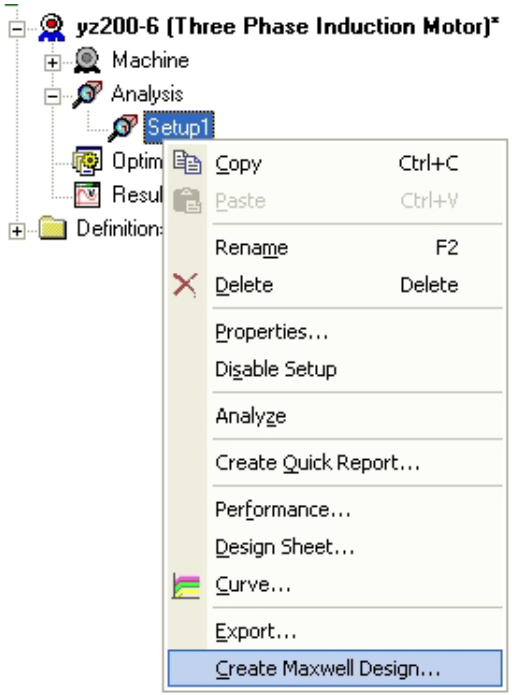
Predefined View Angles

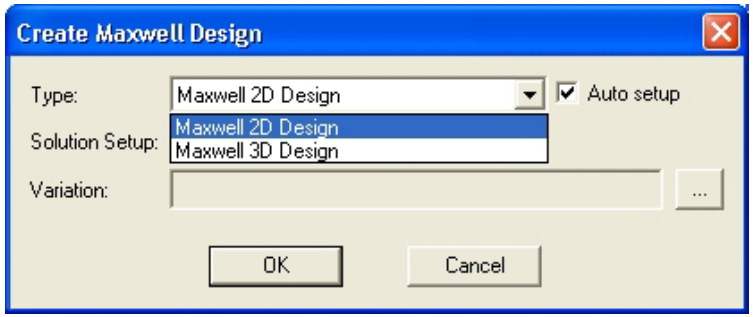


RMxprt → Maxwell

Full Integration of RMxprt and Maxwell

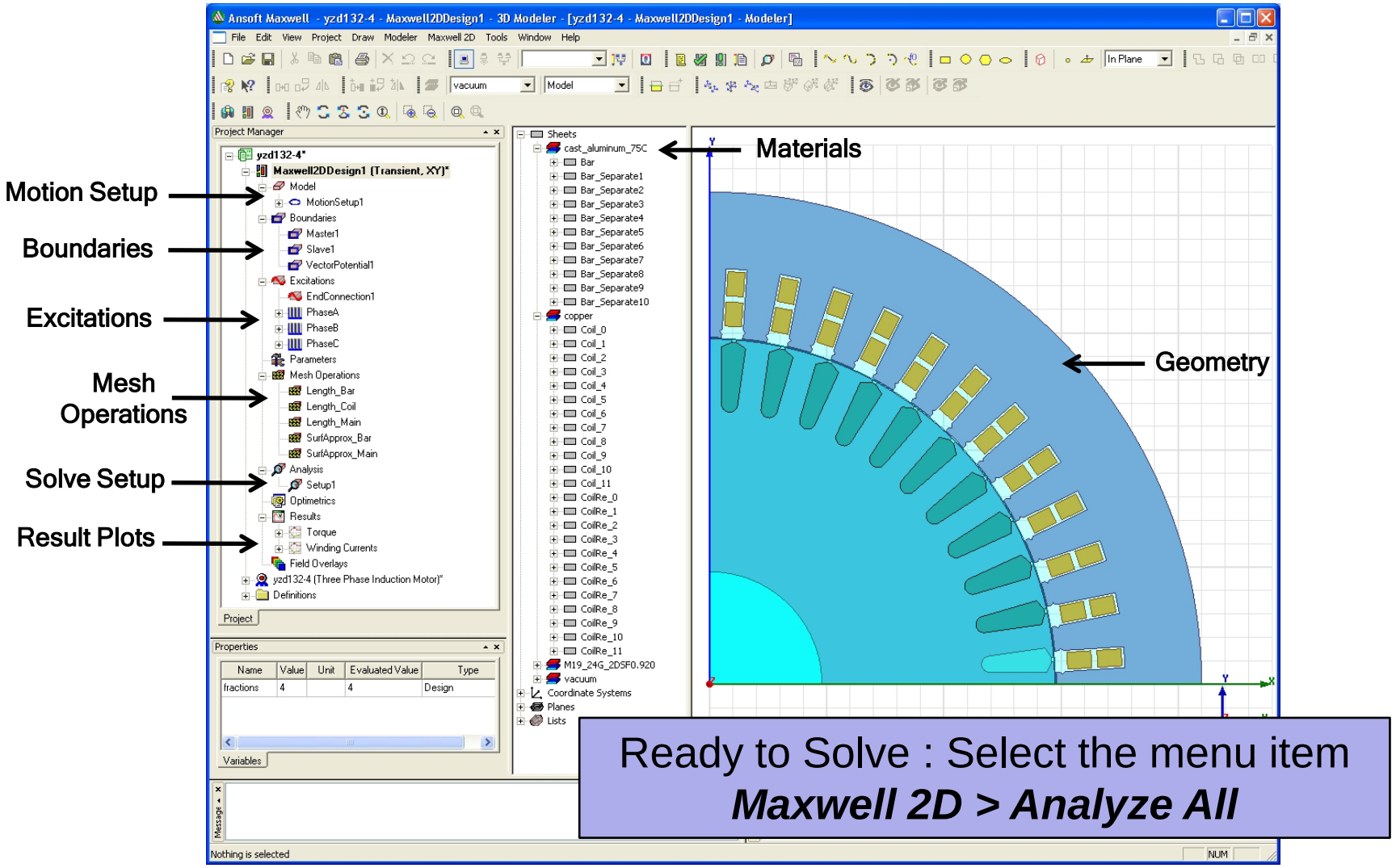
- RMxprt and Maxwell are now Fully Integrated
- All RMxprt data is exported to Maxwell 2D or 3D
- This includes Geometry, Materials, Boundaries, Excitations, Motion Setup, Mesh Operations, Solution Setups, Results, Plots etc.
- With Maxwell V15, even Maxwell 3D Project imports RMxprt Boundaries, Excitations, Mesh Operations, Solution Setup, Results, Plots etc





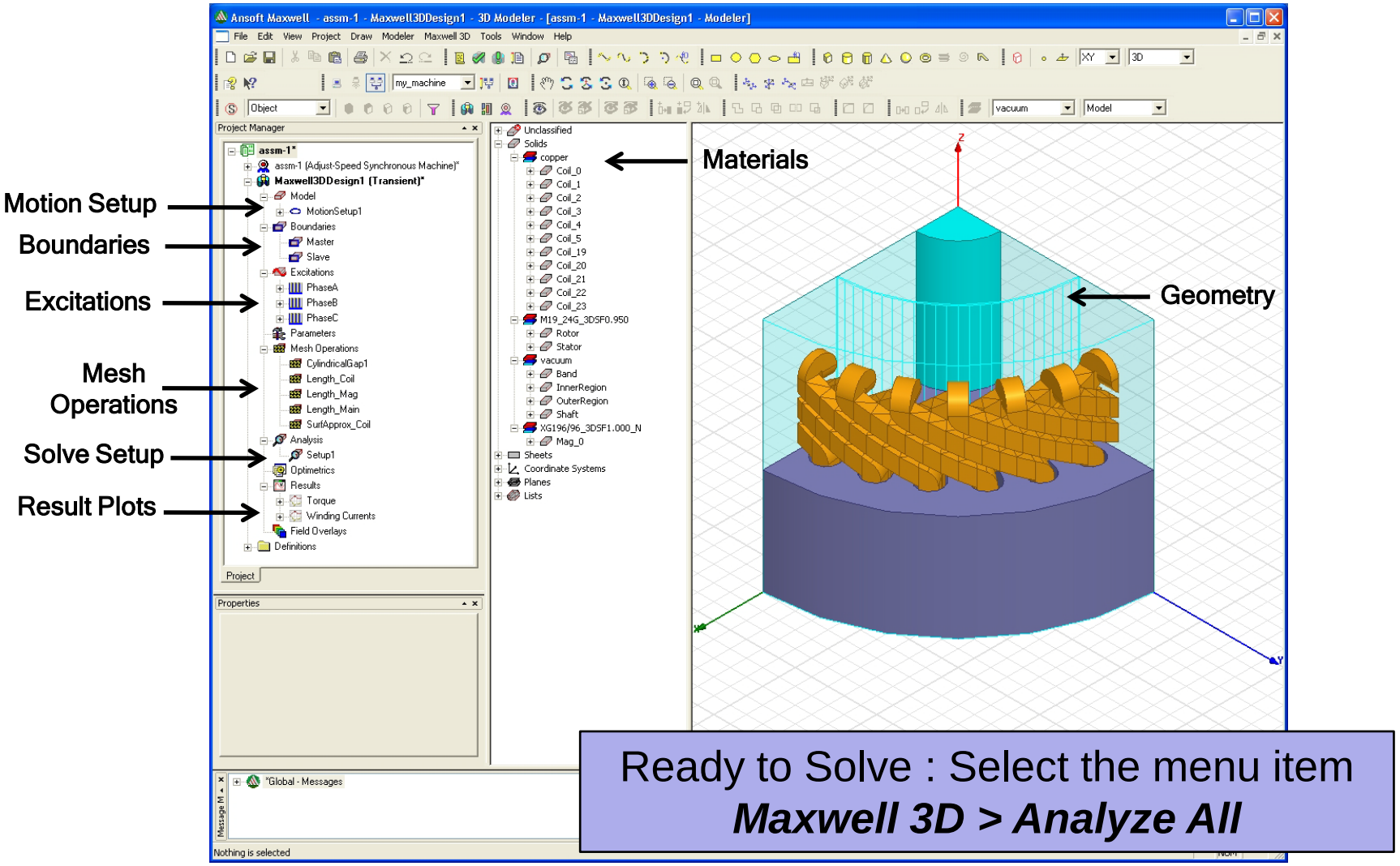
RMxprt → Maxwell

Full Integration of RMxprt and Maxwell 2D



RMxprt → Maxwell

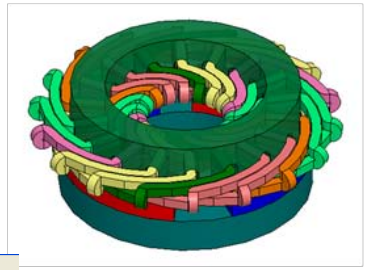
Full Integration of RMxprt and Maxwell 2D

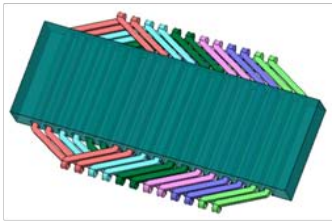


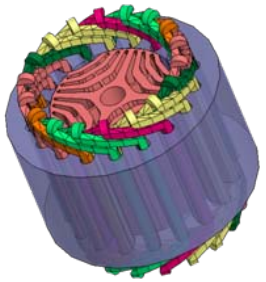
Ready to Solve : Select the menu item
Maxwell 3D > Analyze All

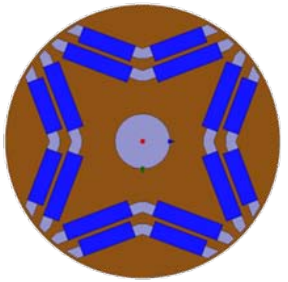
Geometry Creation

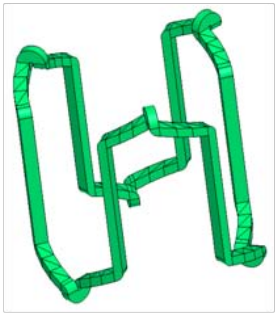
1. **Use RMxprt**
   - Create Model in RMxprt and export to Maxwell
2. **Use User Defined Primitives**
   - Create Model in Maxwell using UDPs

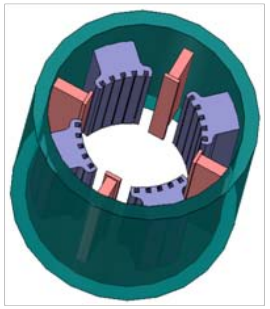


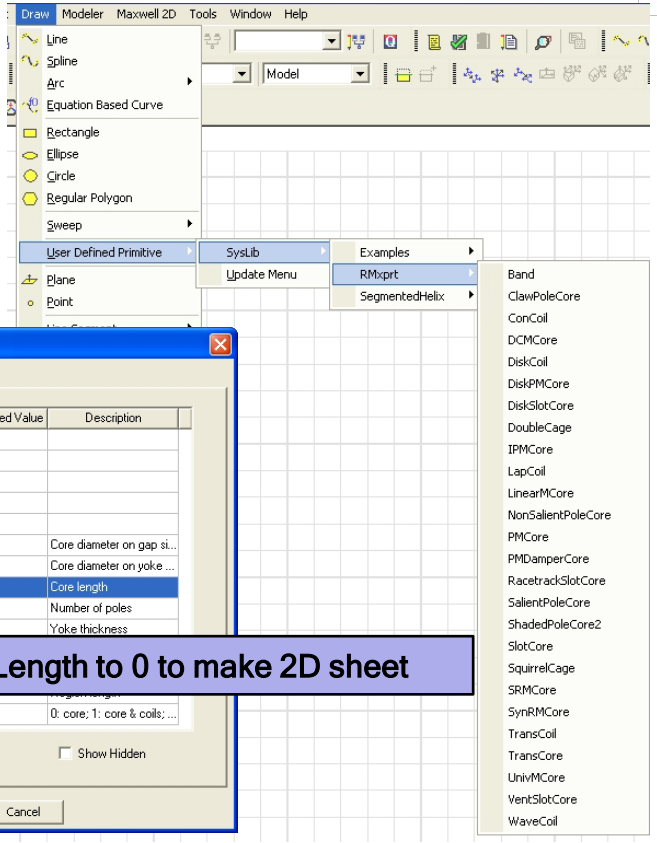


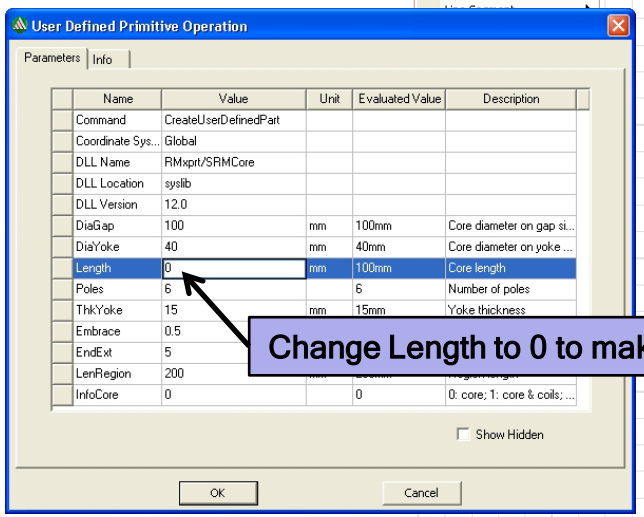








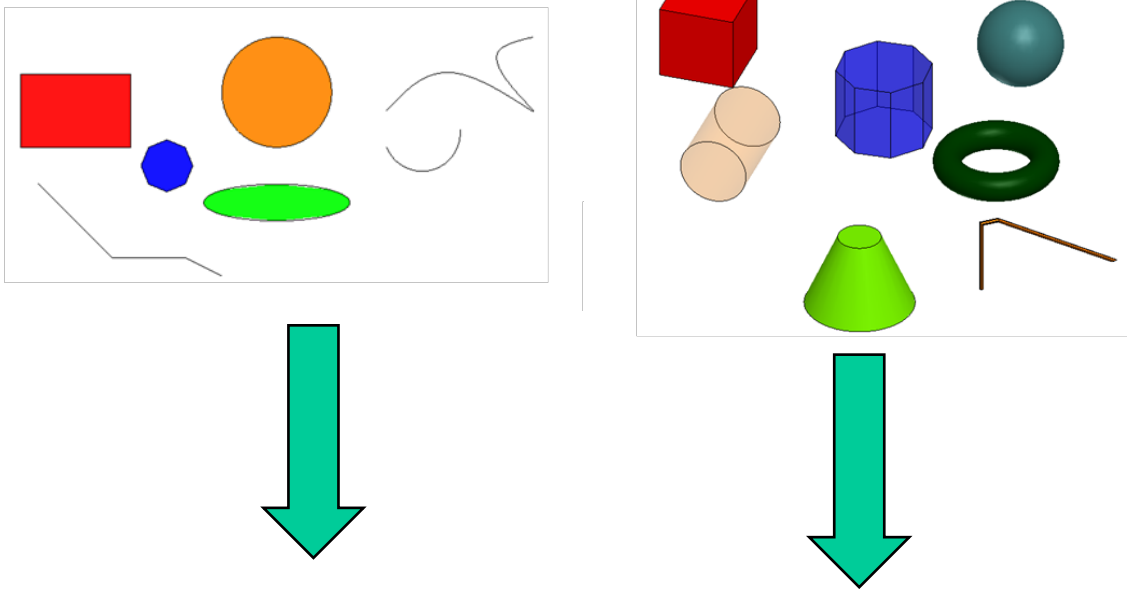


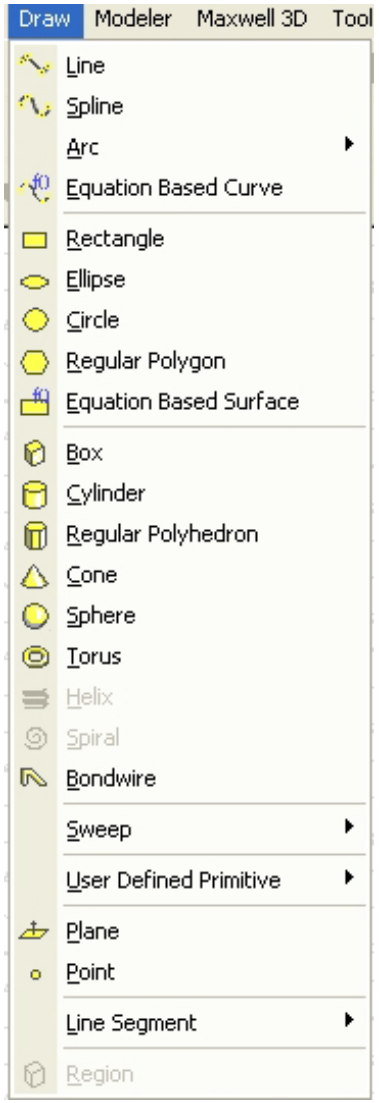


Geometry Creation

3. Use Maxwell Geometry Utilities

- Create Model Maxwell directly using Maxwell Geometry creation utilities

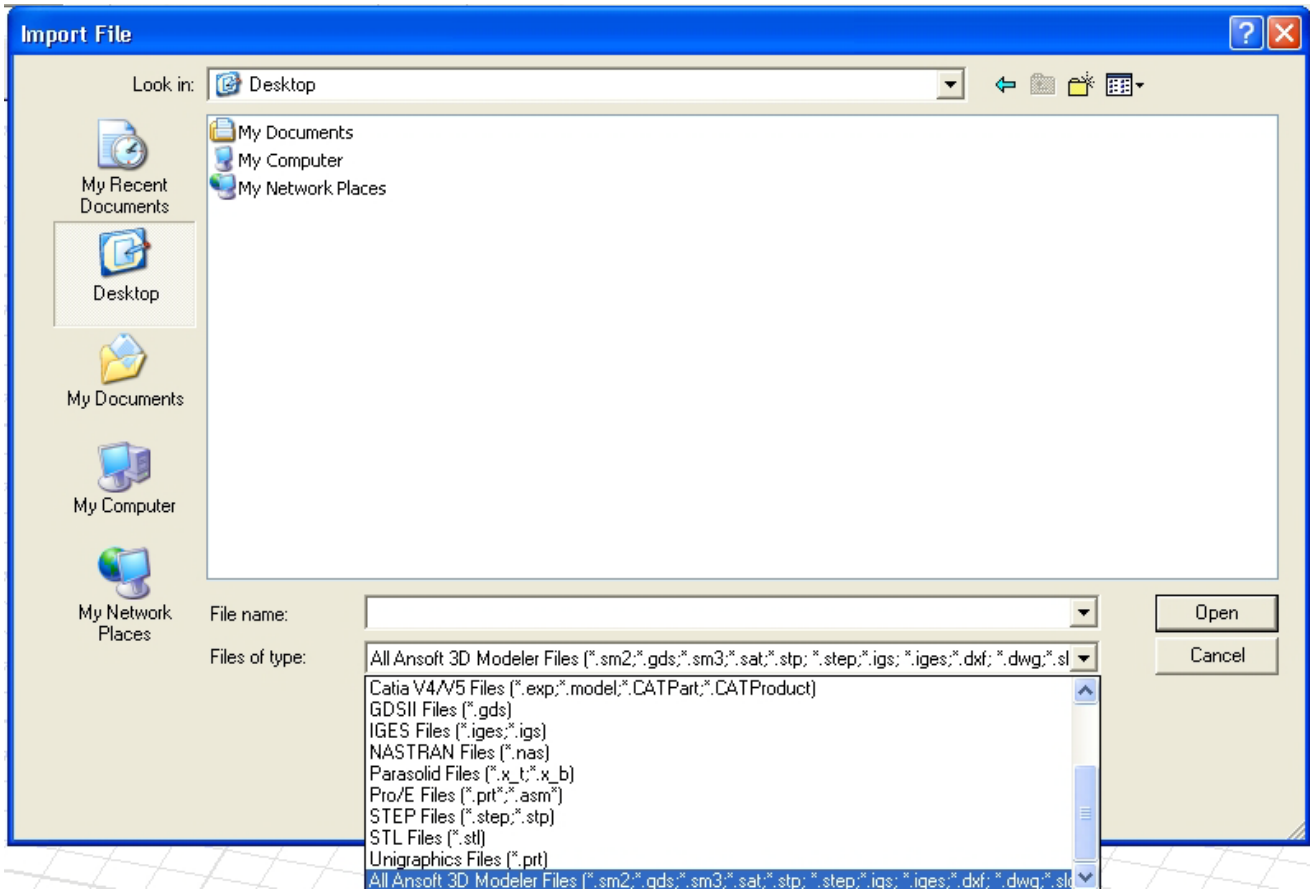




Geometry Creation

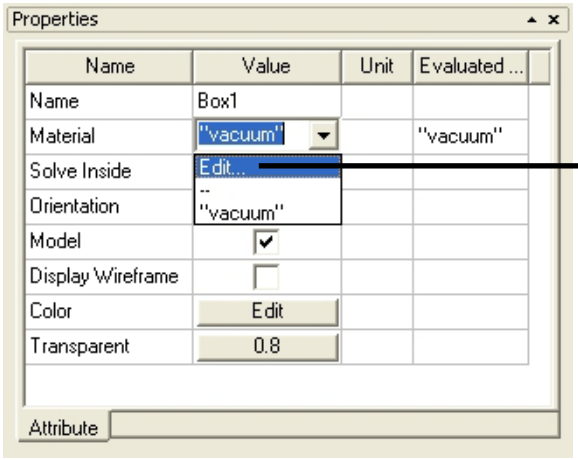
4. Import Geometry

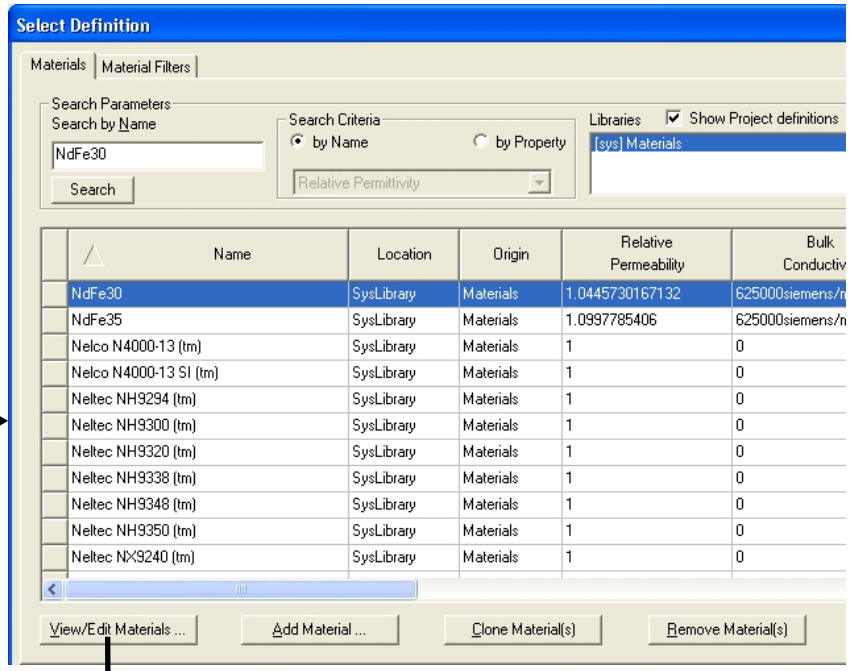
- Geometry can also be imported in Maxwell from External source
- Maxwell can import geometry files from various sources
- With Maxwell V15, external CAD files can also be imported in Maxwell directly

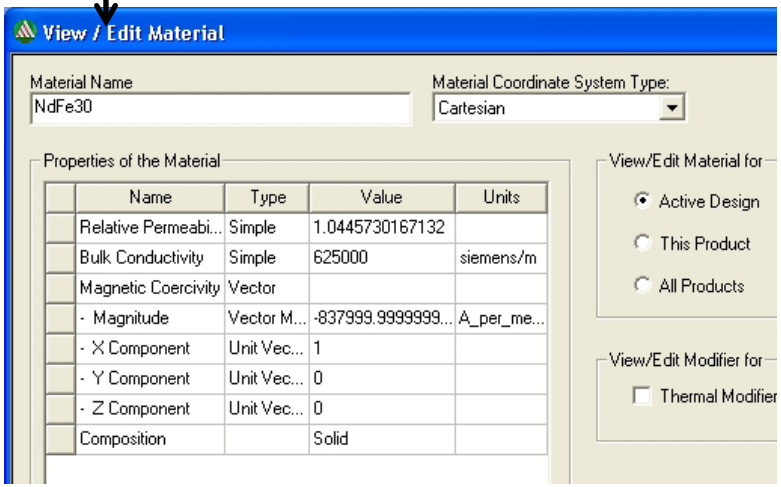


Material Assignment

1. How to Assign Material



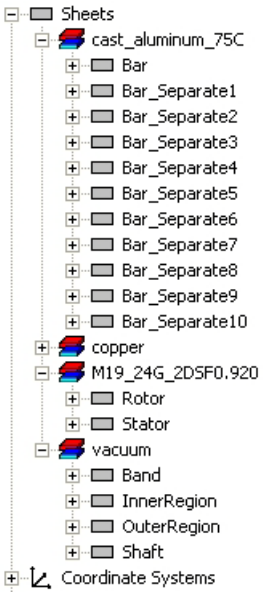




Material Assignment

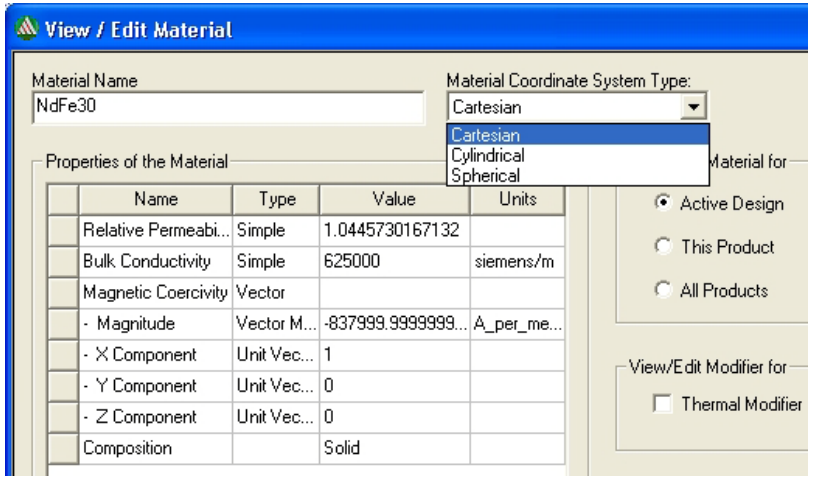
2. Objects can be Grouped by Material





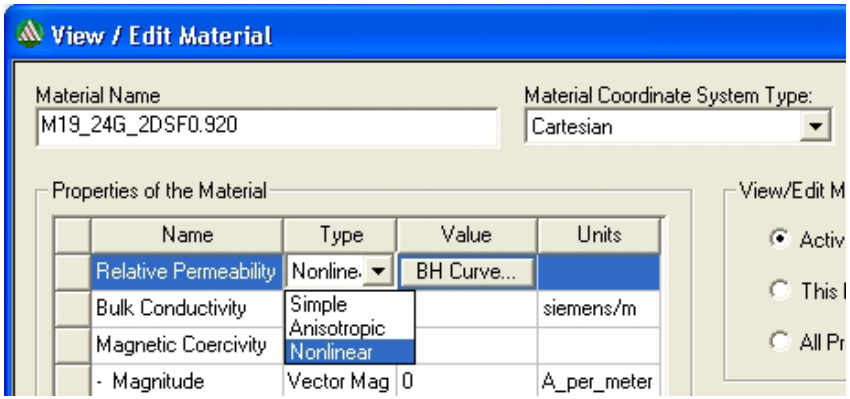
3. Material Coordinate Systems

- Materials can be defined in Cartesian, Cylindrical or Spherical Coordinate Systems



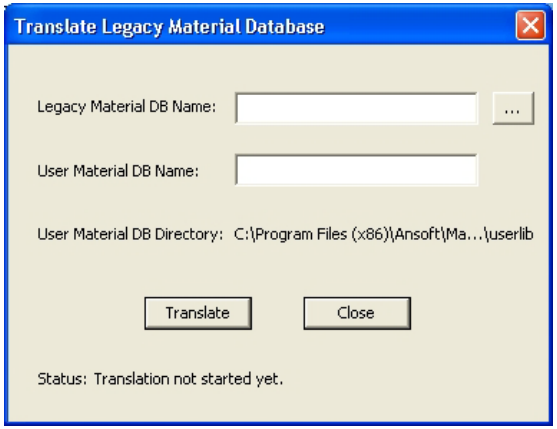
Material Assignment

4. Linear, Nonlinear and Anisotropic Materials



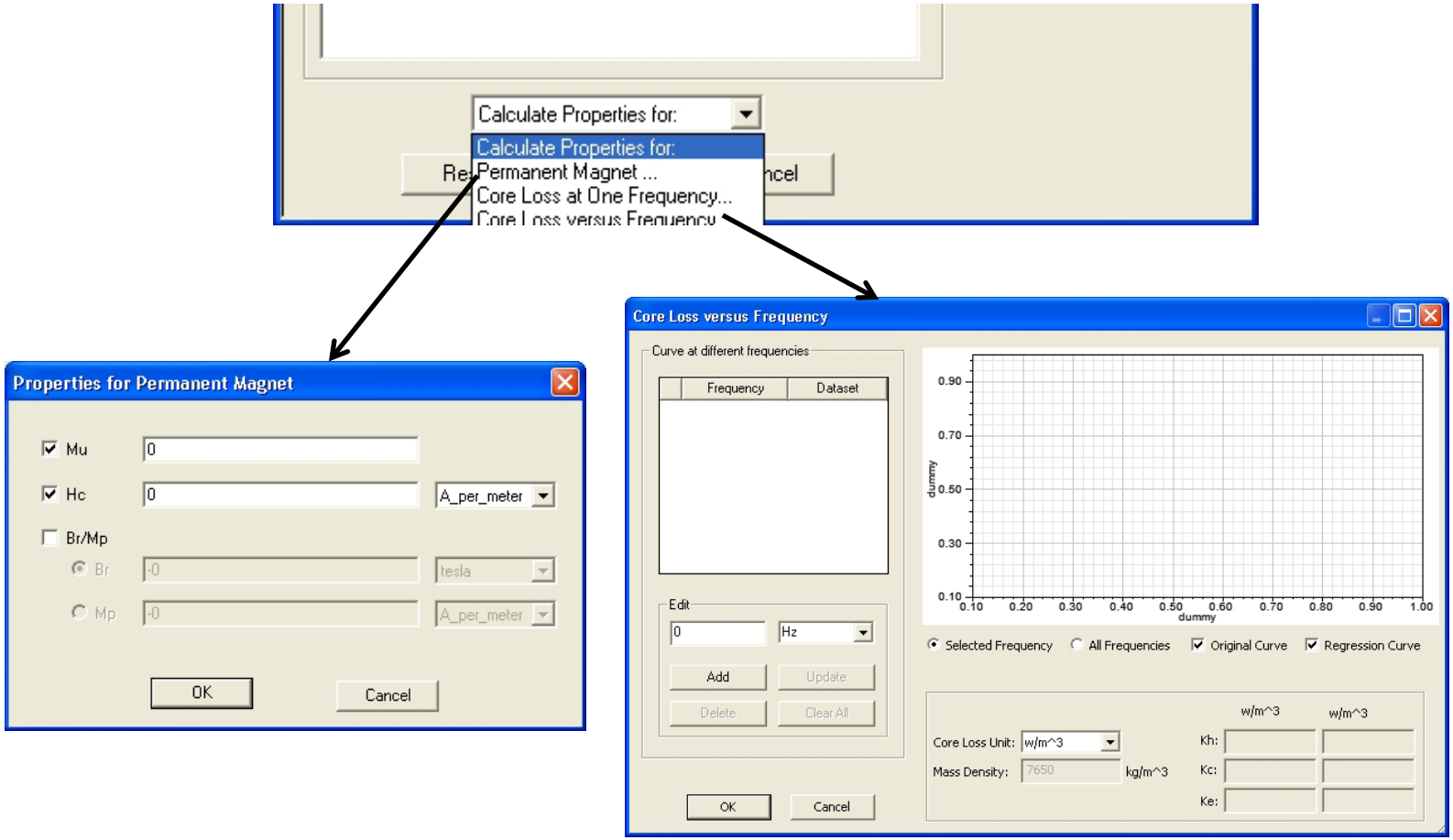
5. Import Old Material Libraries (from Older Versions of Maxwell)

- From the menu item *Maxwell 2D > Translate Material Database*



Material Assignment

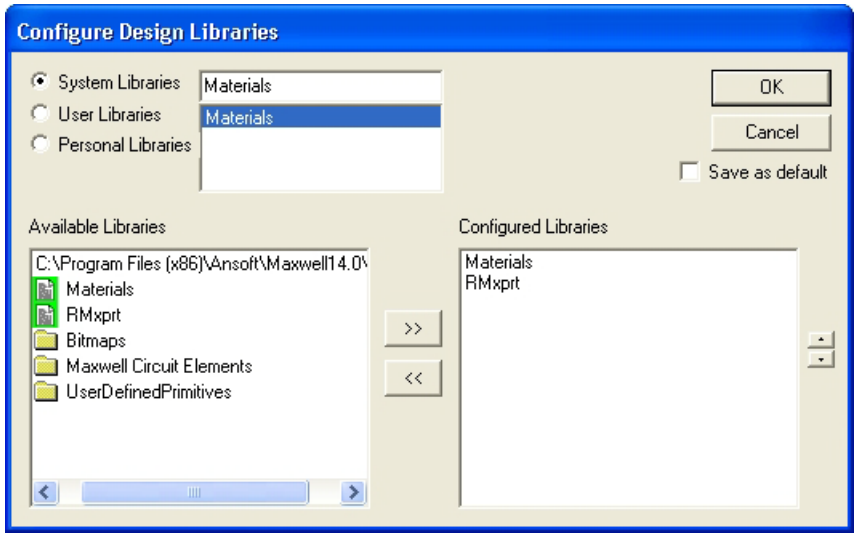
6. Options to calculate Permanent magnet or Core Loss Properties

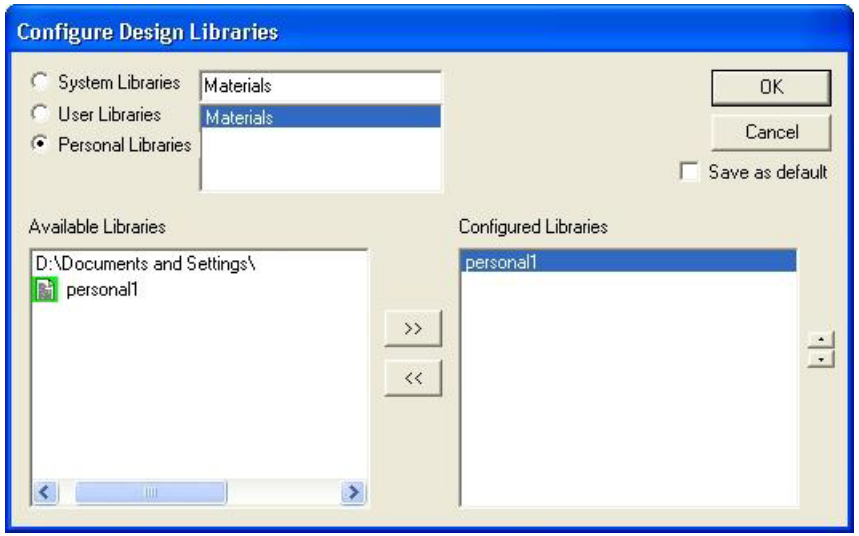


Material Assignment

7. Material Library Management

- 3- Tier Library Structure
  - **System (Global) Level:** Predefined with Maxwell and ships with new upgrades, users cannot modify this
  - **User Library:** To be shared among the several users at a company (can be encrypted)
  - **Personal Libraries:** To be used by single user (can be encrypted)
- Add New materials: ***Tools > Edit Configured Libraries > Materials***
- New Interface for Material setting shared with RMxprt
- Click **"Add Material…"** . Material is only available in current project.
- To Add Materials to User or Personal Library
  - Click on **"Export Library"** and save it in the desired library

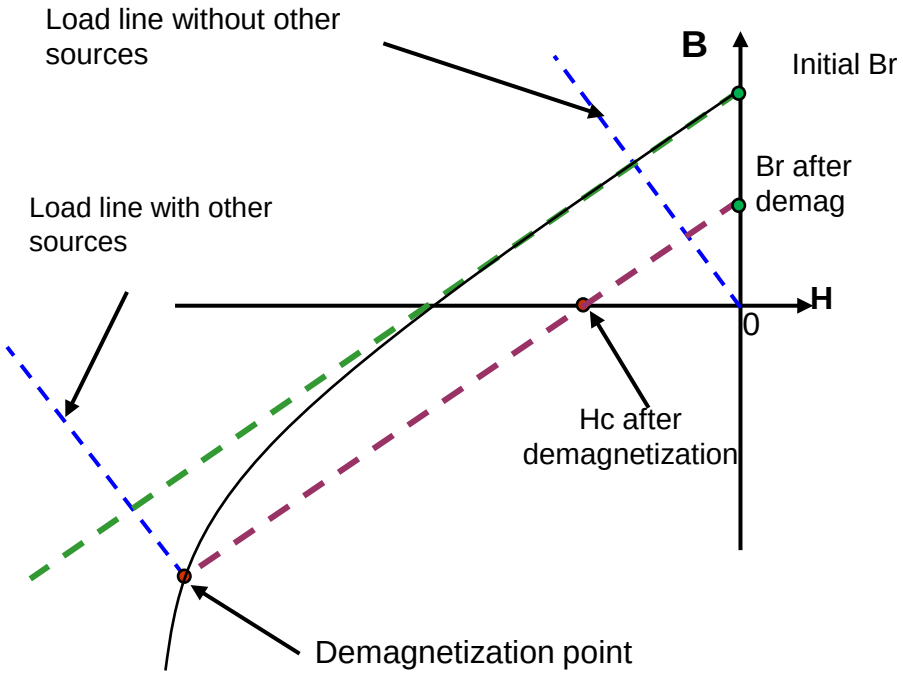




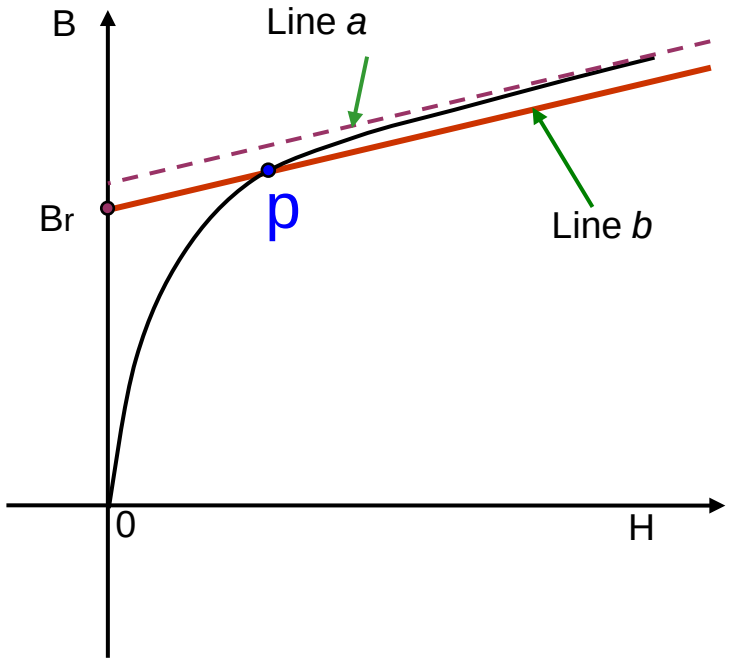
Material Assignment

8. Magnetization and Demagnetization

3rd Quadrant Demagnetization



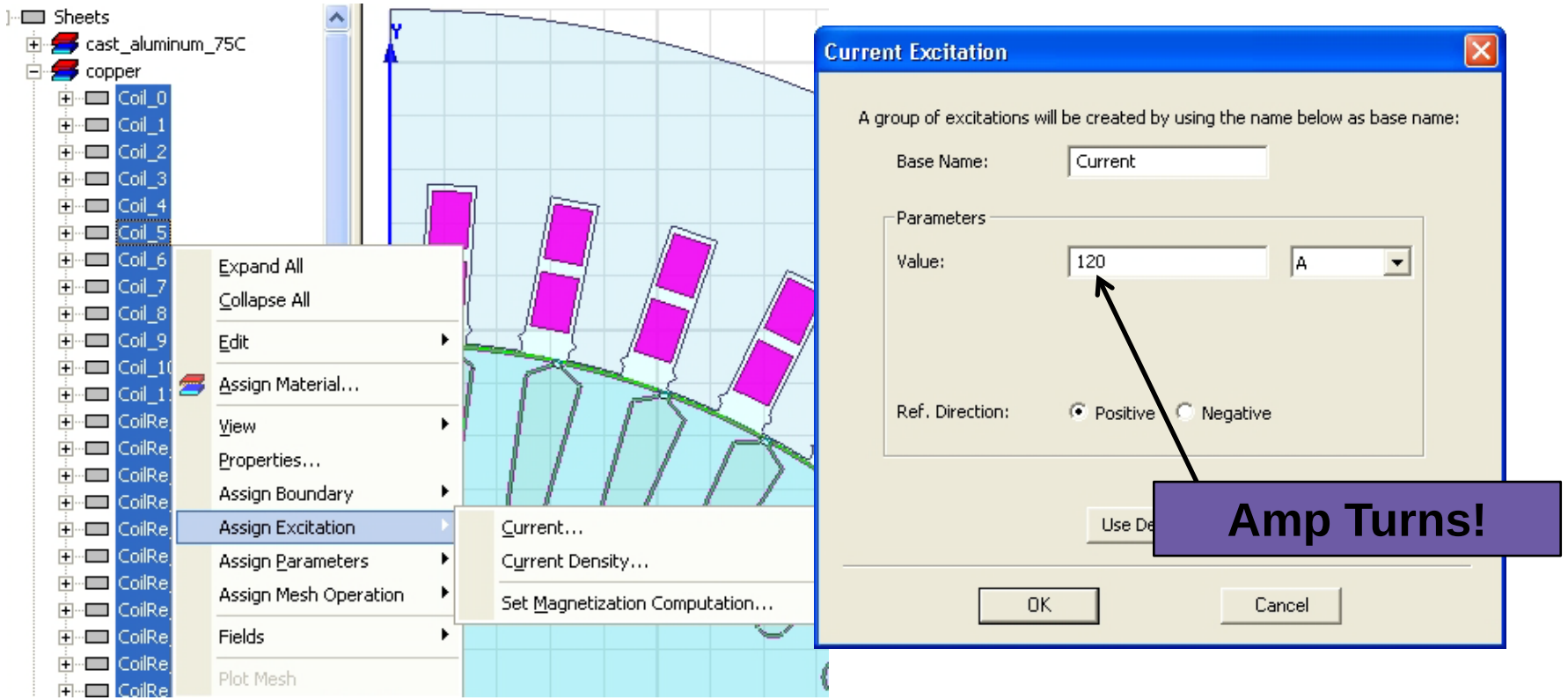
1st Quadrant Magnetization



Excitation Setup

Magnetostatic Solver

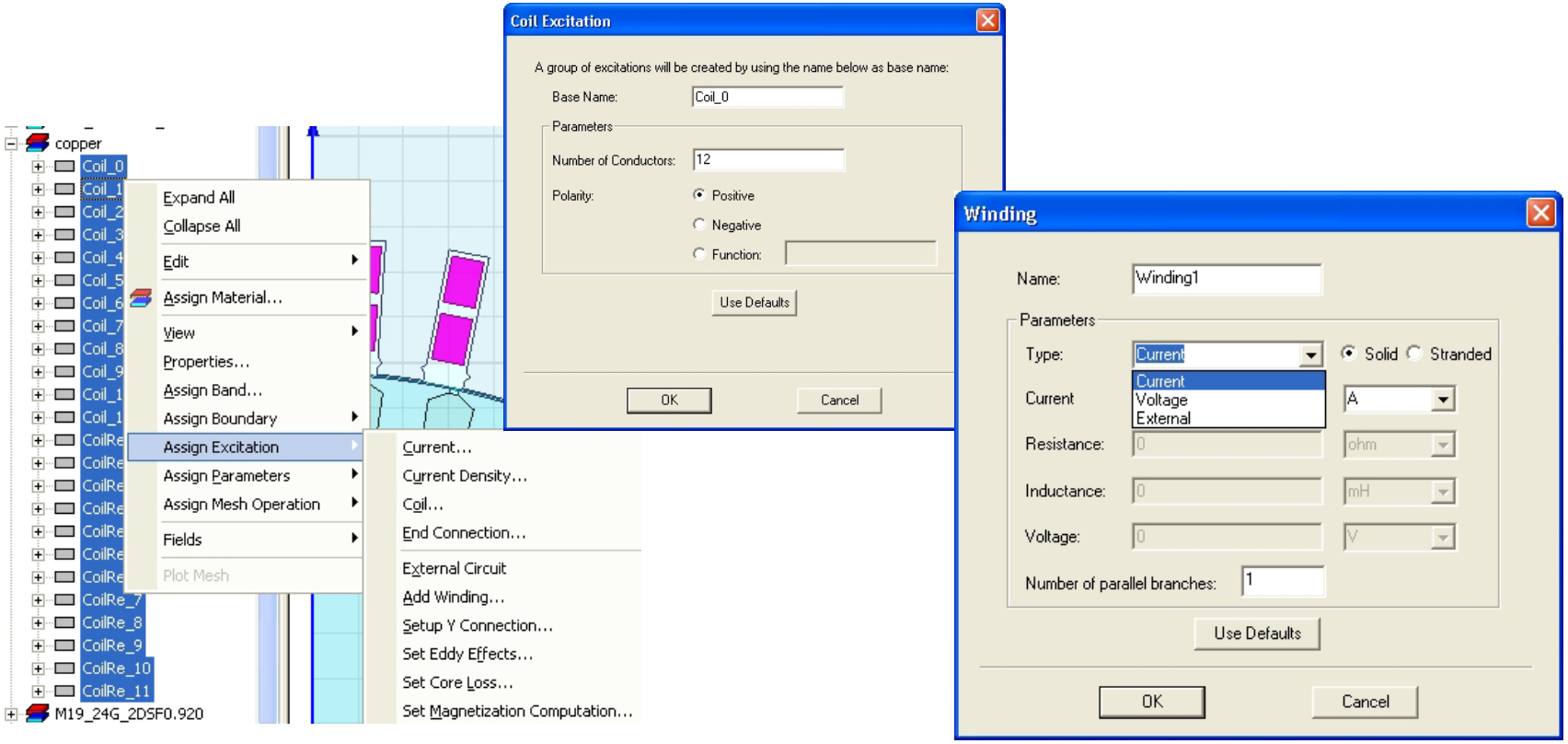
- To Assign Excitations in Magnetostatic Solver, select coils, right click and select **Assign Excitations**



Excitation Setup

Transient Solver

- To Assign Excitations in Magnetostatic Solver
  - Select coils, right click and select **Assign Excitations**
  - Add Winding and choose excitation type: **Current, Voltage** or **External**

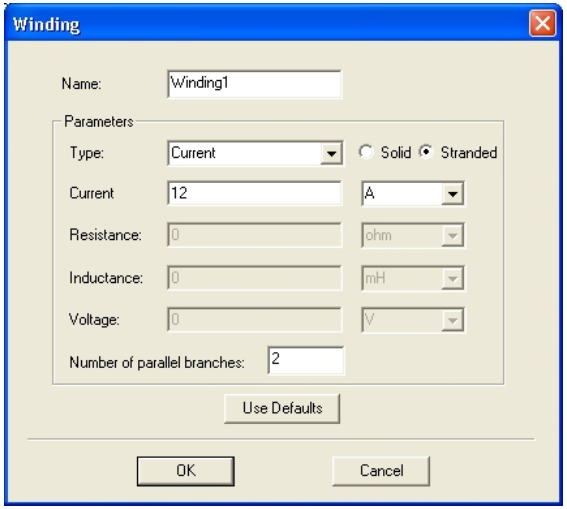


Excitation Setup

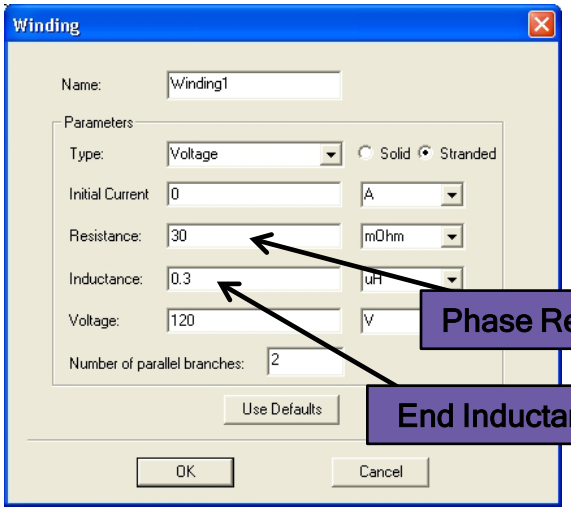
Transient Solver

Winding Types

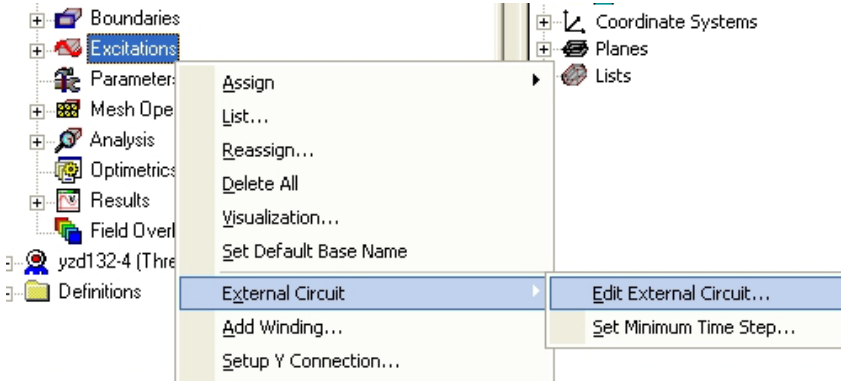
Current Excitation



Voltage Excitation



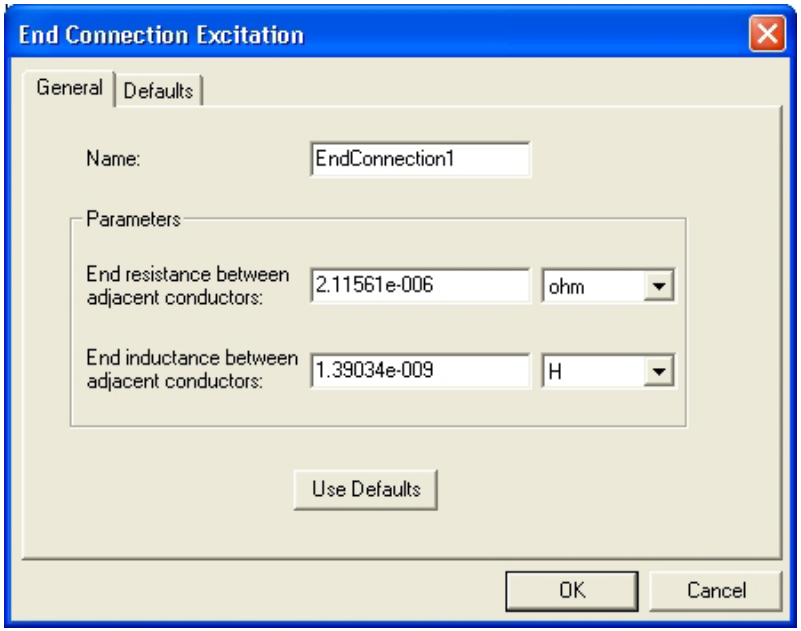
External Circuit

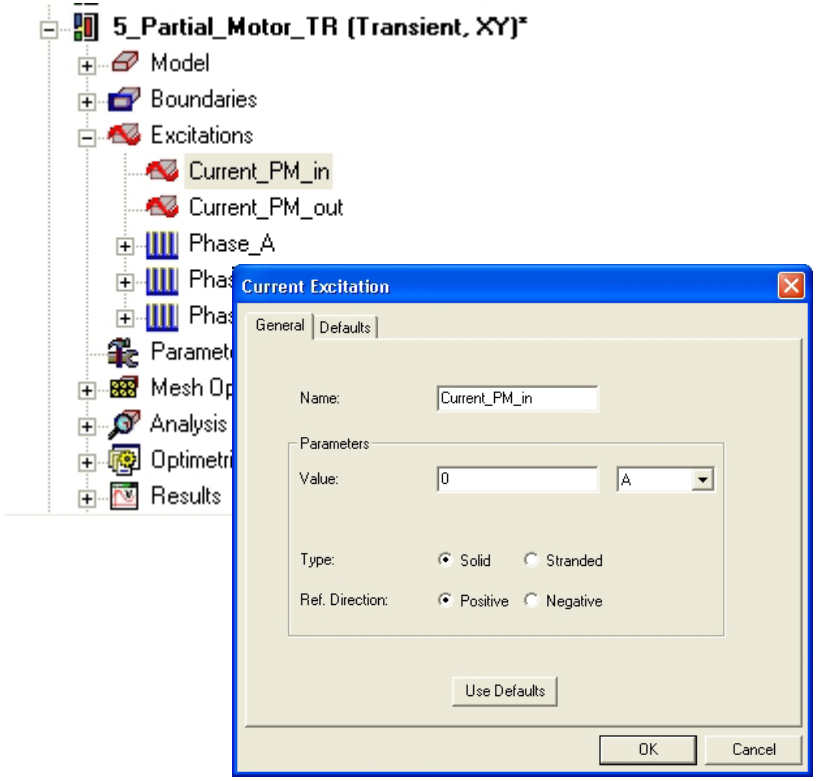


Excitation Setup

Transient Solver

- Excitations for Electrical Machines
  - **End Connection**
    - For Induction Machine Squirrel Cage end-rings
  - **Zero Current**
    - For Permanent Magnets / Solid Rotor when calculating eddy current losses

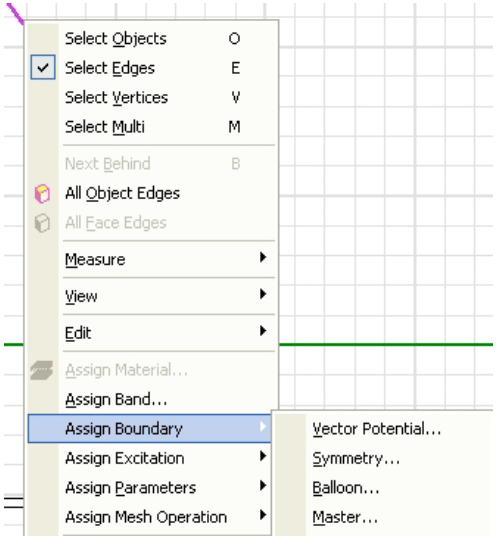




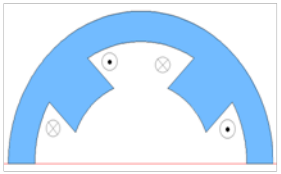
Boundary Conditions

1. To Assign Boundary Conditions

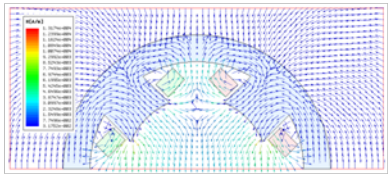
- Select the edges to be assigned the boundary condition
- Right click and select **Assign Boundary**



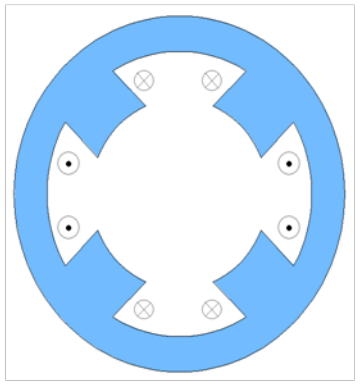
2. Symmetry Boundary: Even & Odd

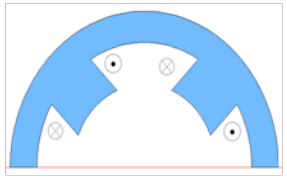


Symmetry: Even (Flux Normal)

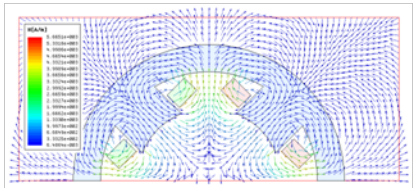


=

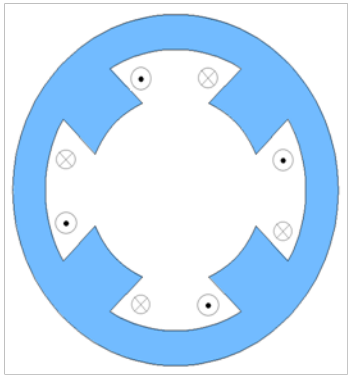




Symmetry: Odd (Flux Tangential)

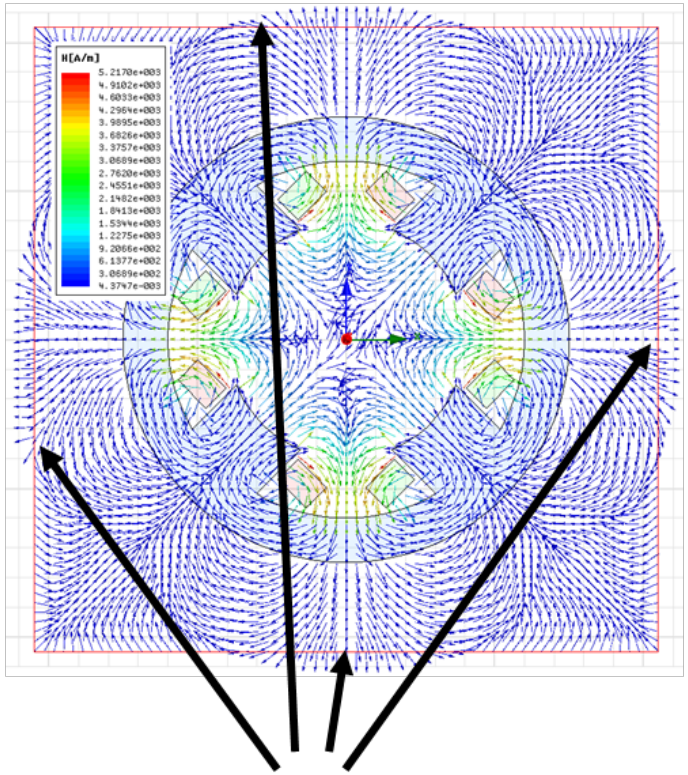


=



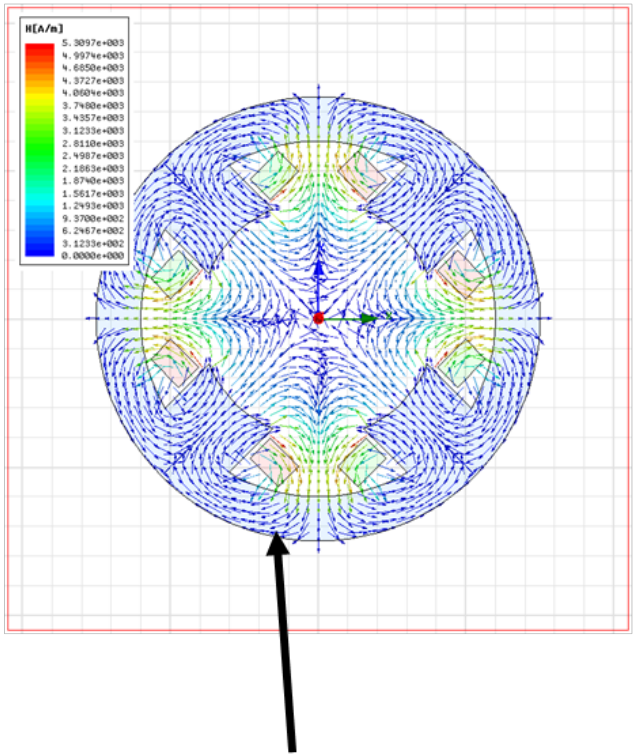
Boundary Conditions

3. Vector Potential = 0 and Balloon



Balloon on all Edges of Region

Flux Can Exit and Re-enter the Boundary

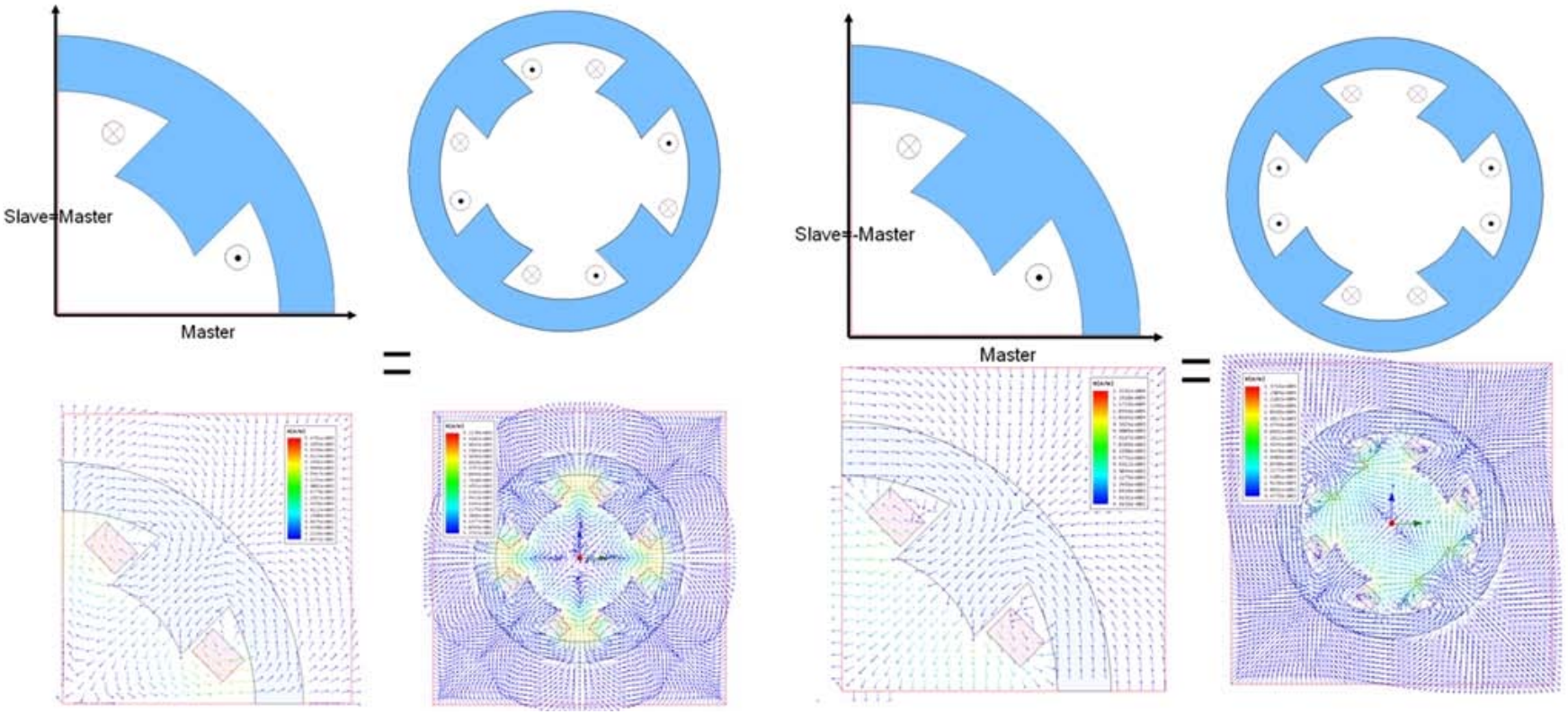


Zero Vector Potential on outer Edges of Stator

Flux Can NOT Escape the Boundary

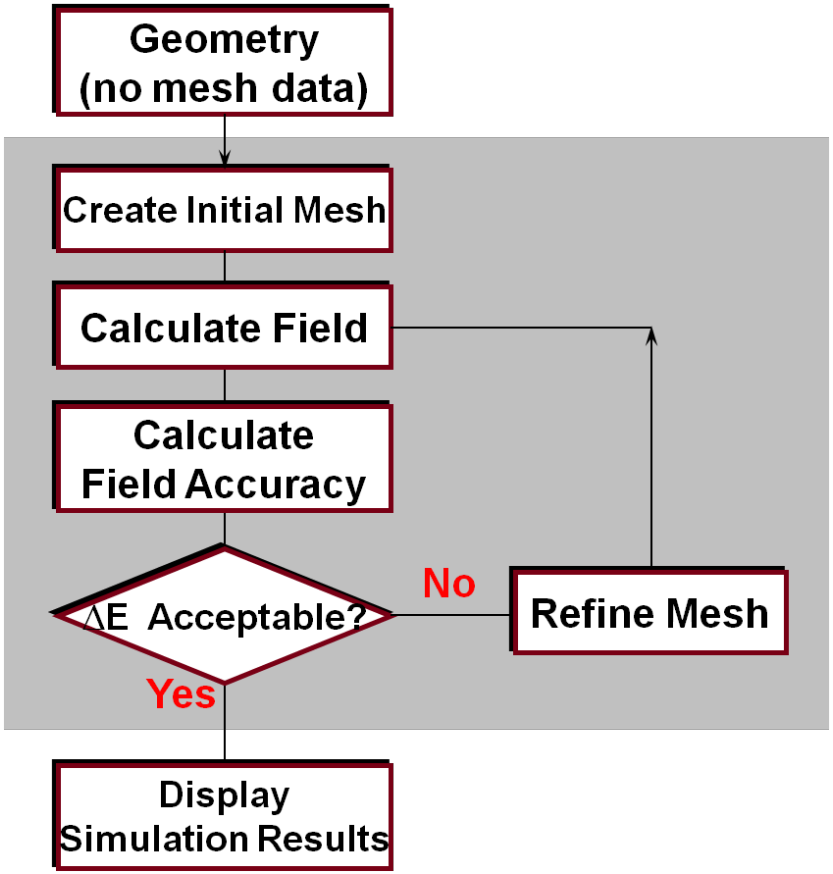
Boundary Conditions

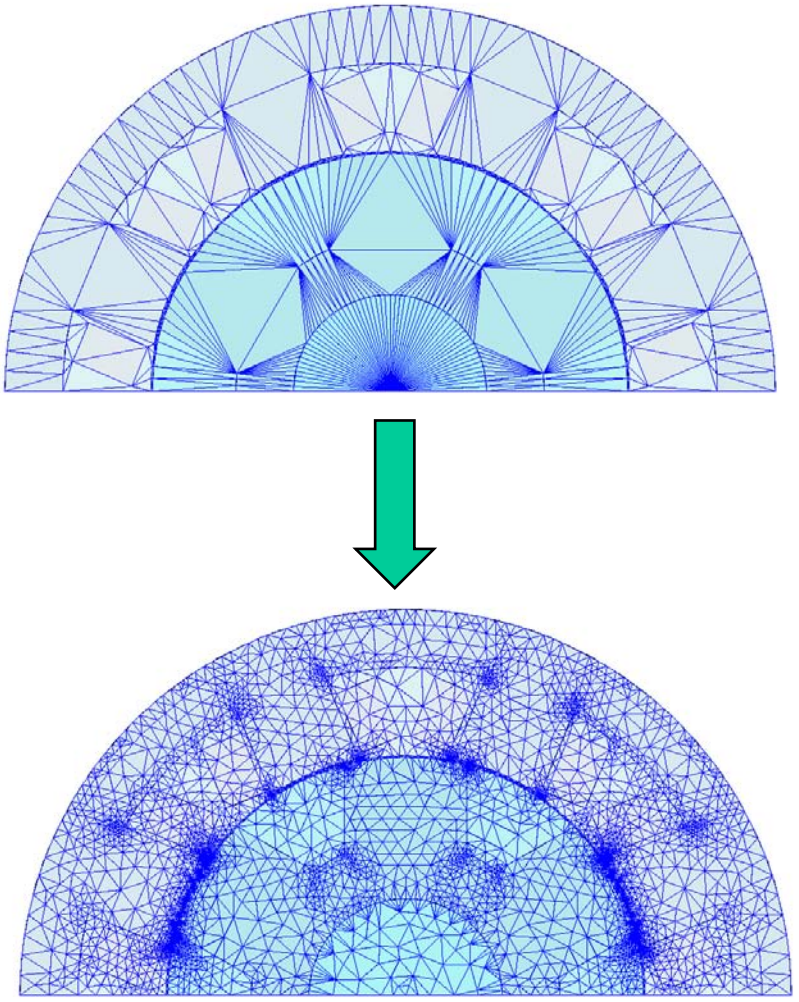
4. Master / Slave (Matching Boundary, Periodic Boundary)



Mesh Operation

1. Automatic and Adaptive Meshing

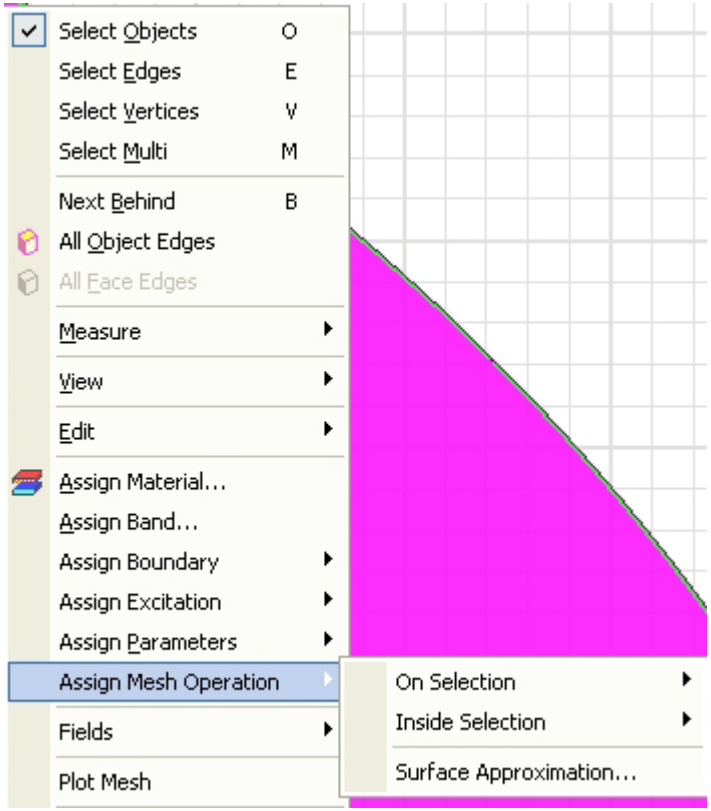


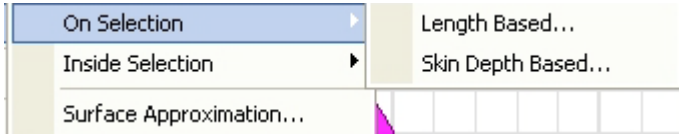


Mesh Operation

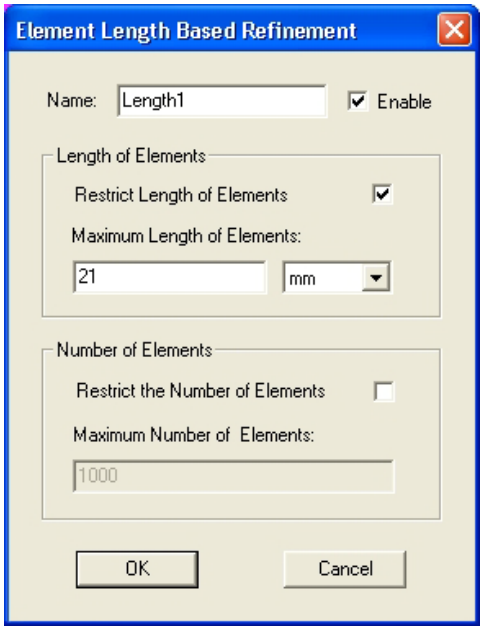
2. To Assign Mesh Operations

- Select the geometry to be assigned the mesh operation
- Right click and select **Assign Mesh operation**
  - **On Selection**
  - **Inside Selection**





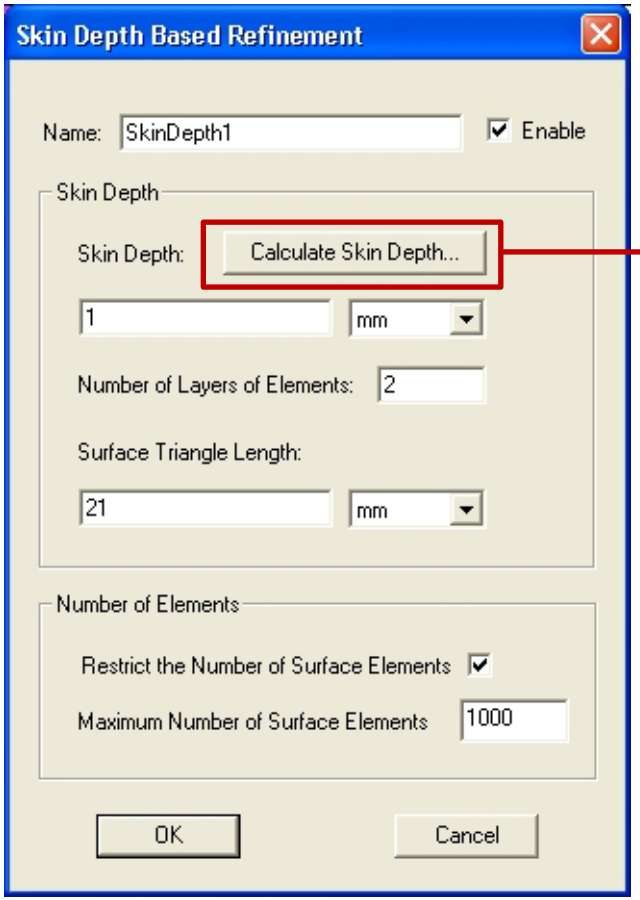


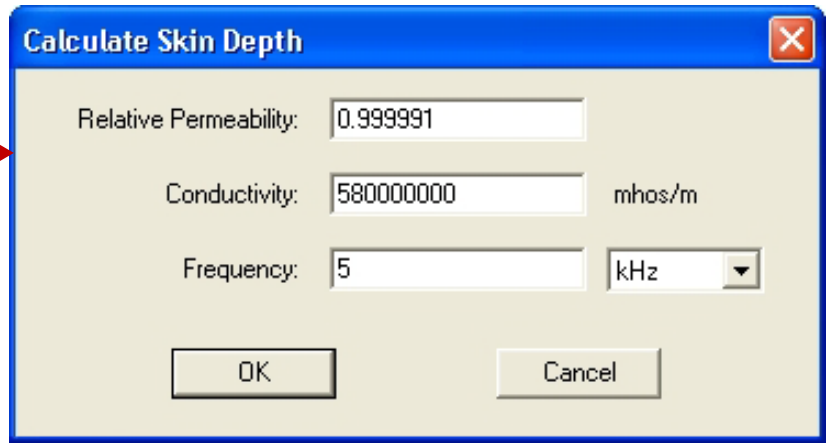


Mesh Operation

3. Skin Depth Based Mesh



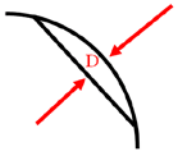




Mesh Operation

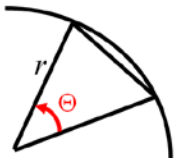
3. Surface Approximation Mesh

- **Maximum Surface Deviation**
  - Maximum deviation that triangle surfaces may have from true curved geometry surface



D = Maximum Surface Deviation

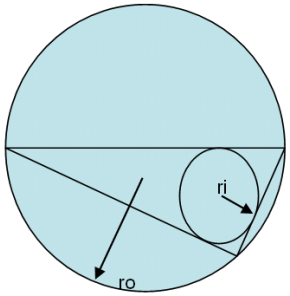
- **Maximum Surface Normal Deviation**
  - Maximum angular difference, in degrees, that a triangle face's normal can have from the surface normal for the true geometry which it is meant to represent



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

Θ = Maximum Surface Normal Deviation

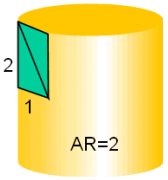
- **Maximum Aspect Ratio:**
  - Maximum allowed Aspect Ratio of all the faces



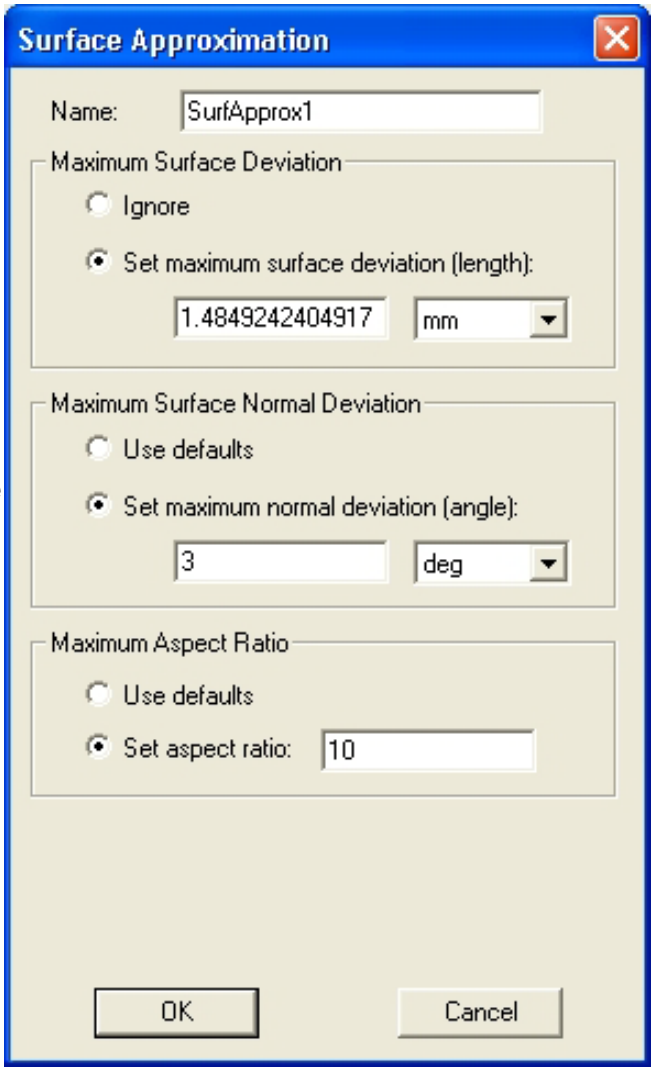
$$\frac{2*ri}{ro} = ShapeFactor\,(2D)$$

$$\frac{3*ri}{ro} = SF\,(3D)$$

$$AspectRatio = \frac{1}{SF}$$



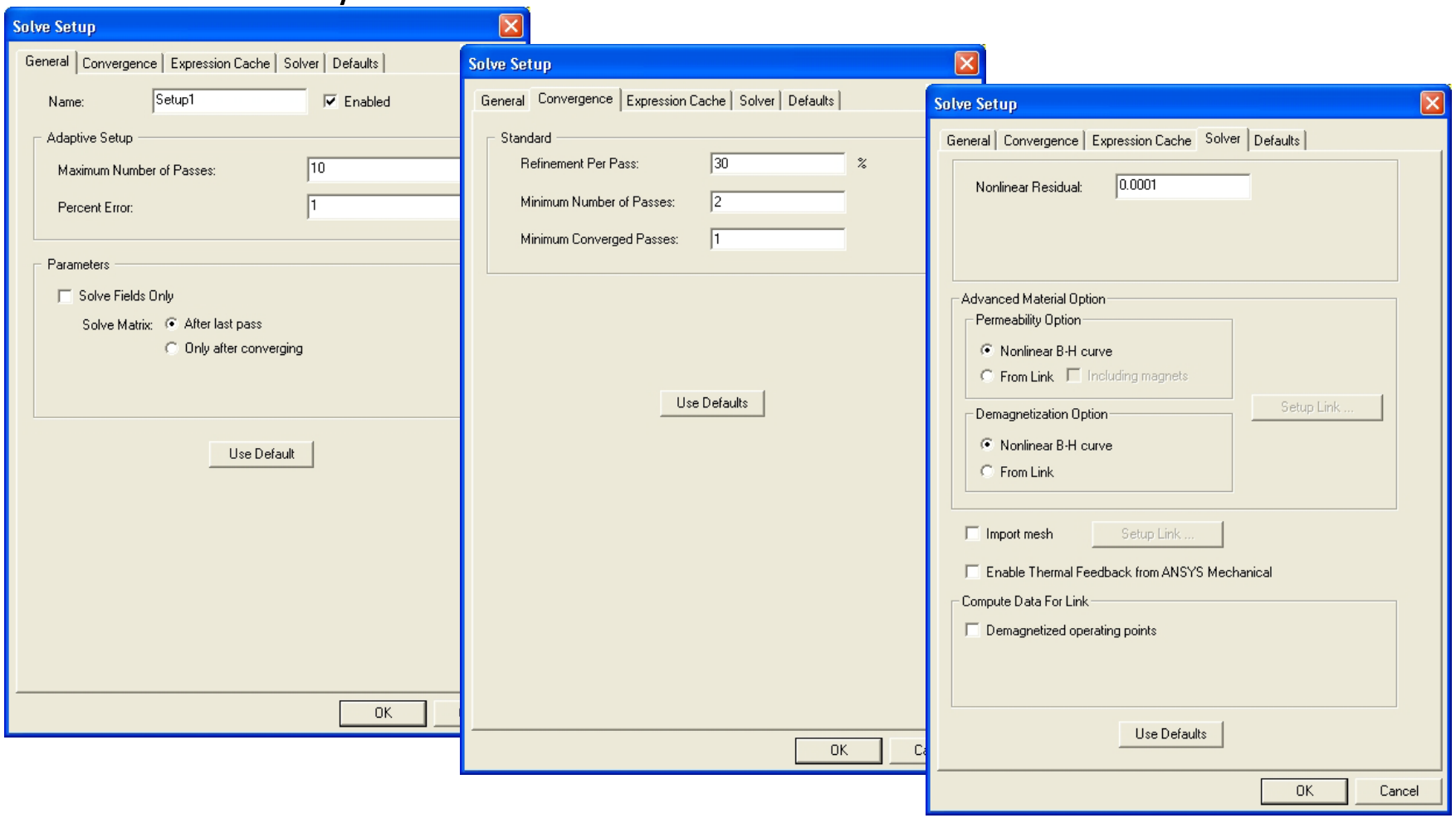
Aspect Ratio of Cells, not of triangles



Solution Setup

1. Magnetostatic Solver

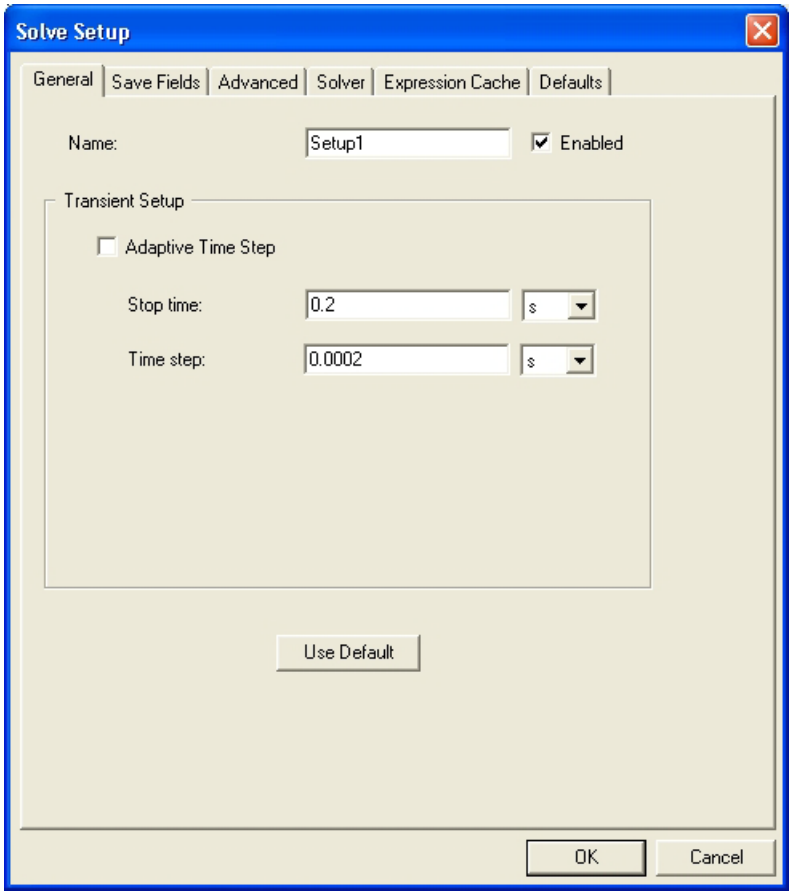
- Solution Setup can be added from the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*

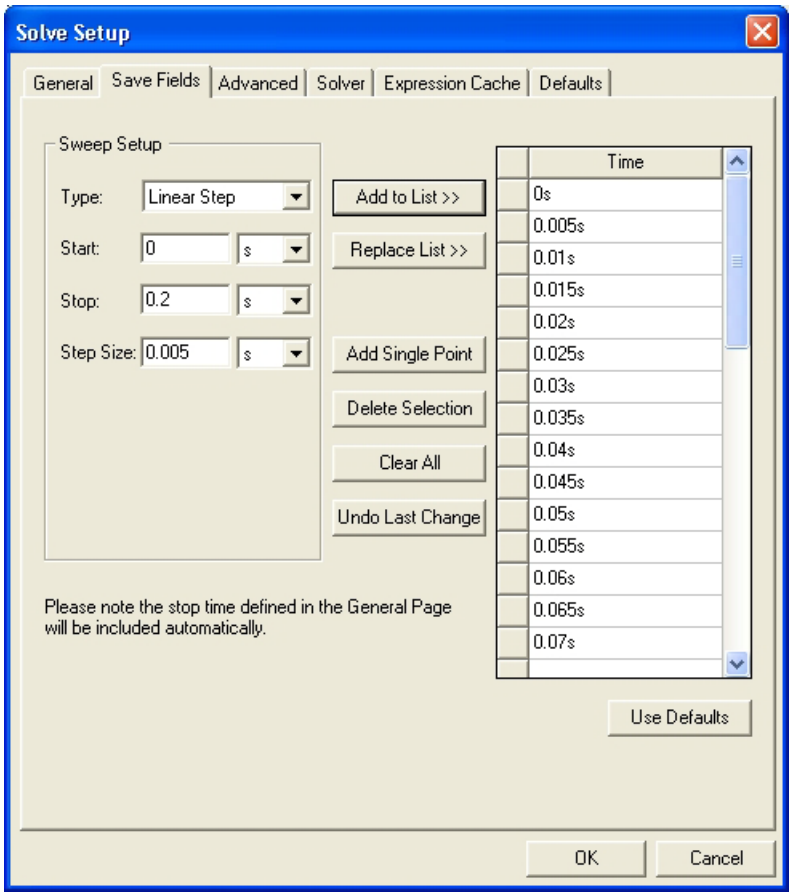


Solution Setup

2. Transient Solver

- Solution Setup can be added from the menu item ***Maxwell 2D > Analysis Setup > Add Solution Setup***

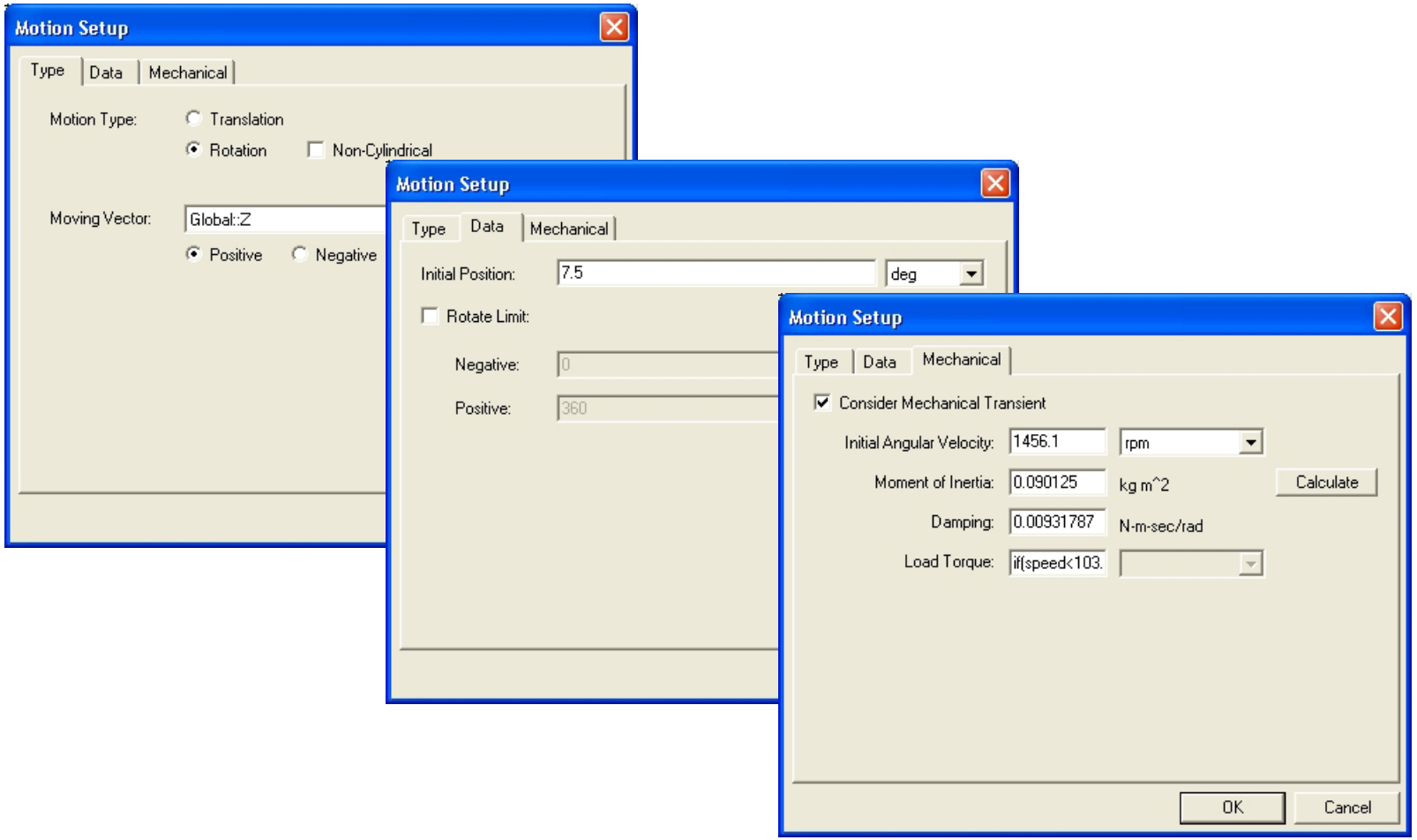




Solution Setup

3. Transient Solver

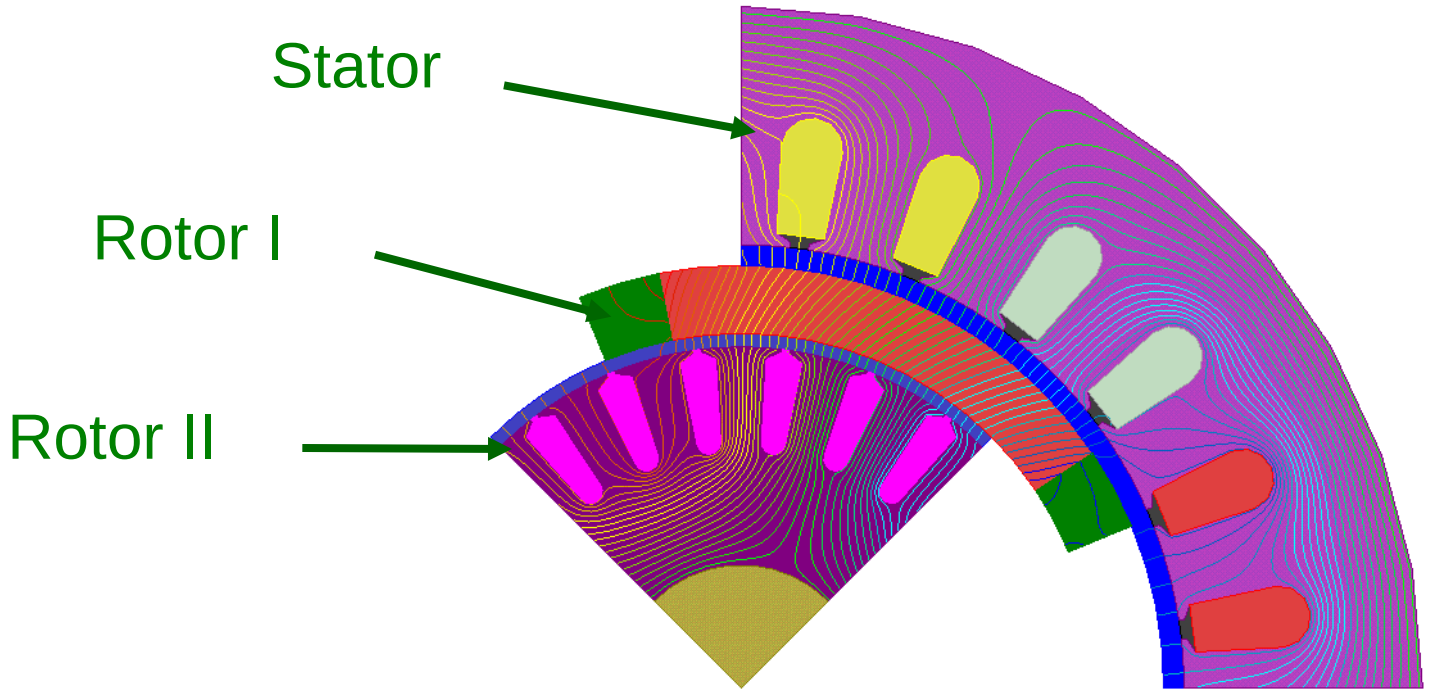
- **Motion Setup**
  - Select the band, right click and select **Assign Band**



Solution Setup

4. Transient Solver

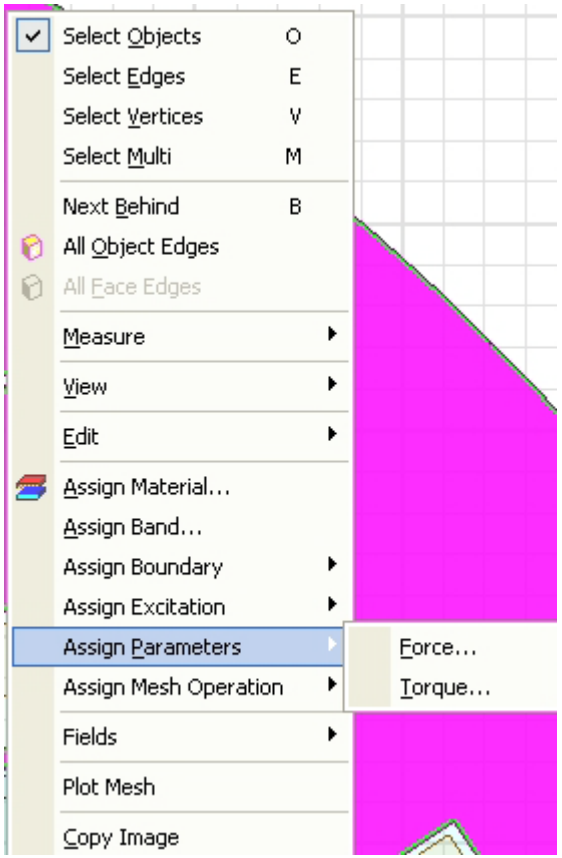
- **Two Bands**
  - For transient motion solver, two bands with two independently moving objects can be specified
  - Both rotational and translational solvers can be specified with two bands



Torque / Force Calculation

1. For Magnetostatic Solver

- Select the object to be assign the calculation
- Right click and select **Assign Parameter**
- A Band object is recommended in Magnetostatic simulations, but not required.
- The band is a dummy air/vacuum object between moving and stationary parts.
- For Magnetostatic Solver, User have to manually specify the Force / Torque Calculations

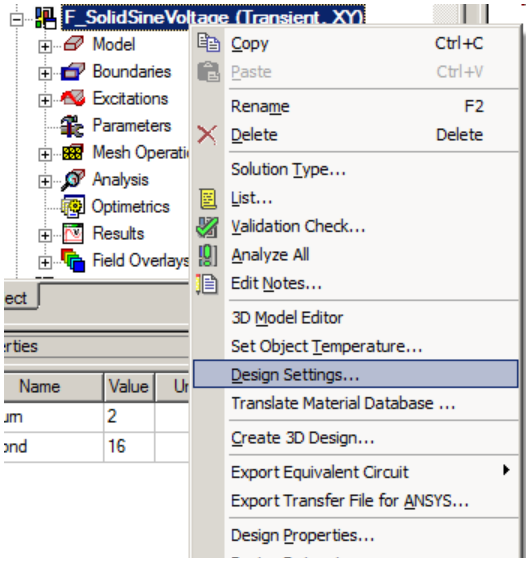


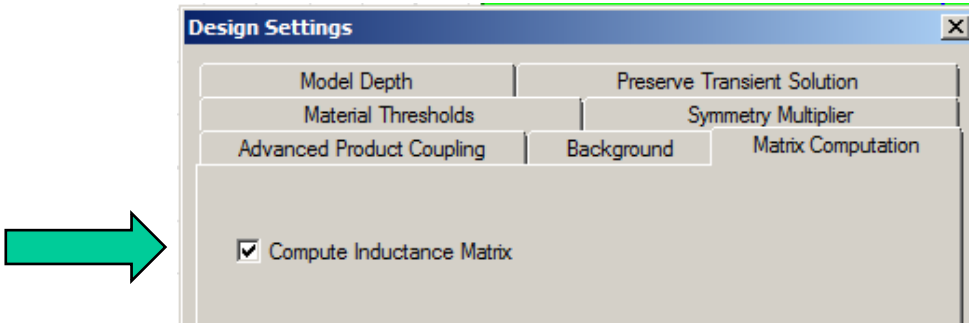
2. For Transient Solver

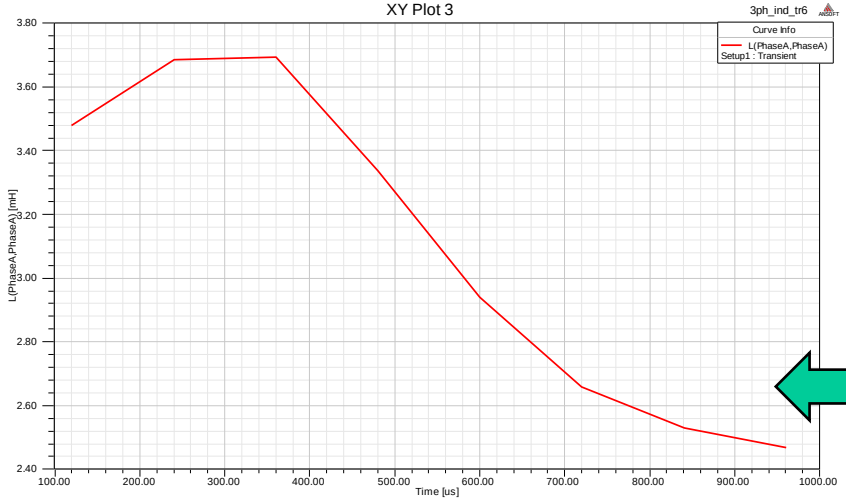
- A Band object is required for Transient simulations with motion.
- In a Transient simulation, moving torque is assigned by the solver automatically.
- The moving torque calculated in a Transient simulation is the torque on all objects INSIDE the band (not including the band).

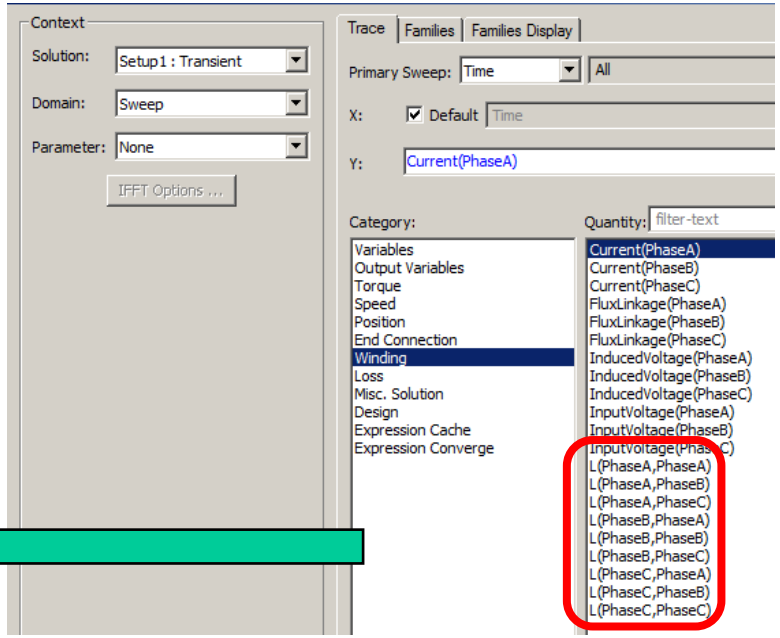
Inductance Calculation

1. Inductance Calculations with Transient Solver



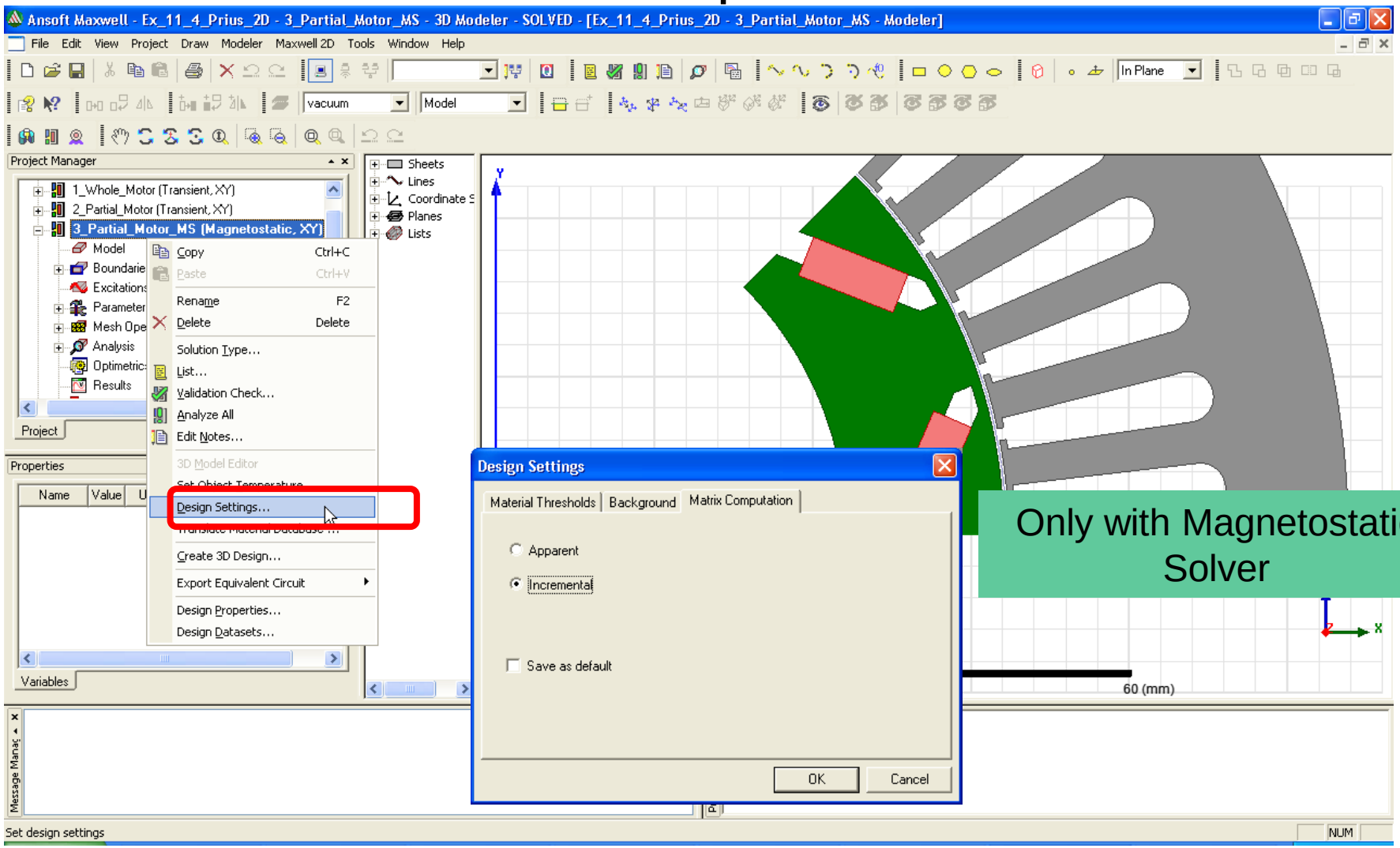






Inductance Calculation

2. Incremental Inductance Computation



Inductance Calculation

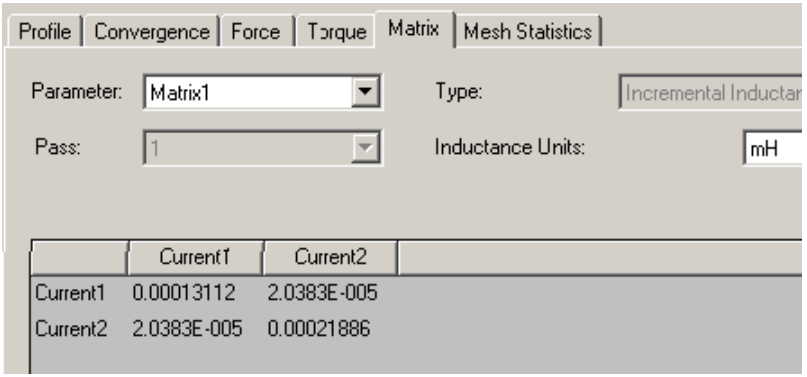
3. Apparent Inductance and Incremental Inductance

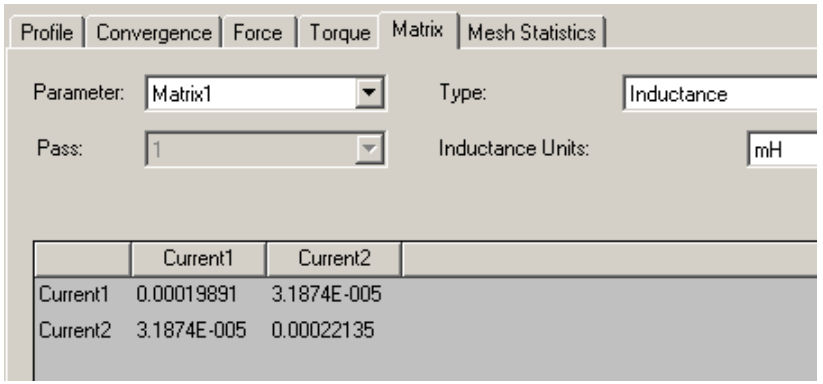
- Apparent Inductance

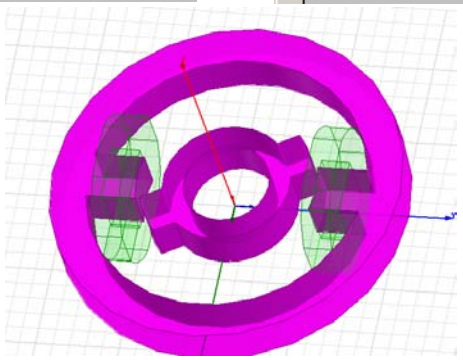
$$L^a_{jk} = \frac{\lambda_j}{i_k}$$

- Incremental Inductance

$$L^d_{jk} = \frac{\partial \lambda_j}{\partial i_k}$$





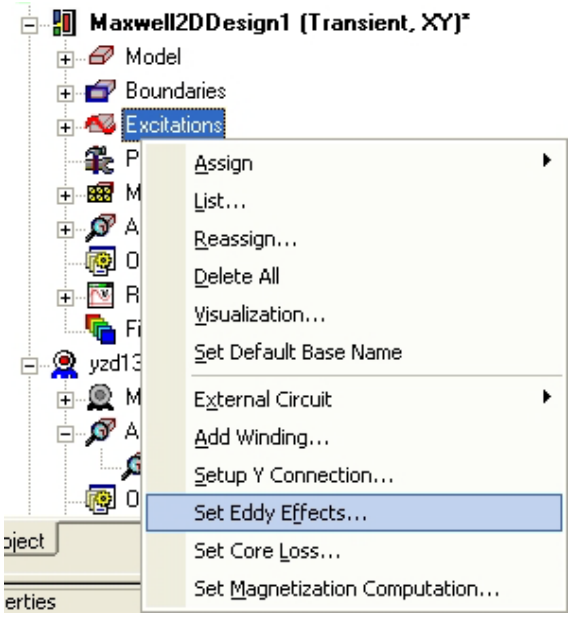


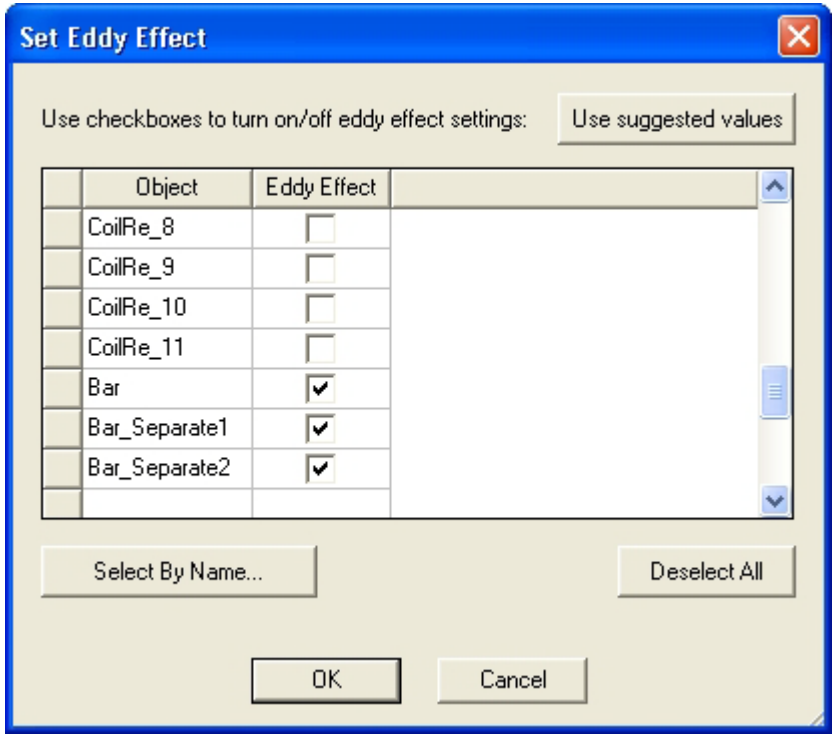
Loss Calculations

1. To Enable Loss Calculations

- Right click on **Excitations** tab from Project Manager window

2. Eddy Loss Calculation



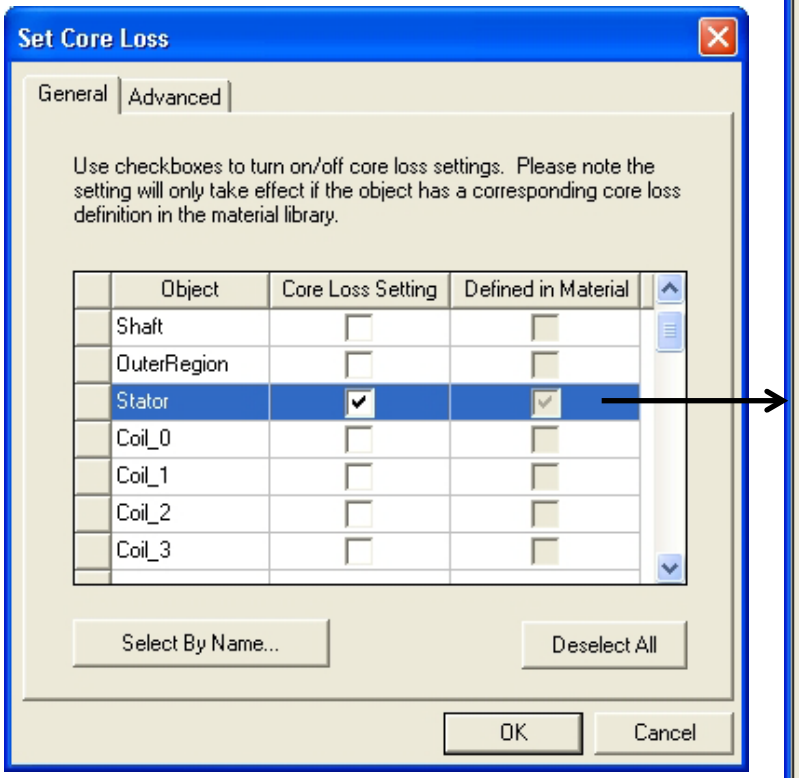


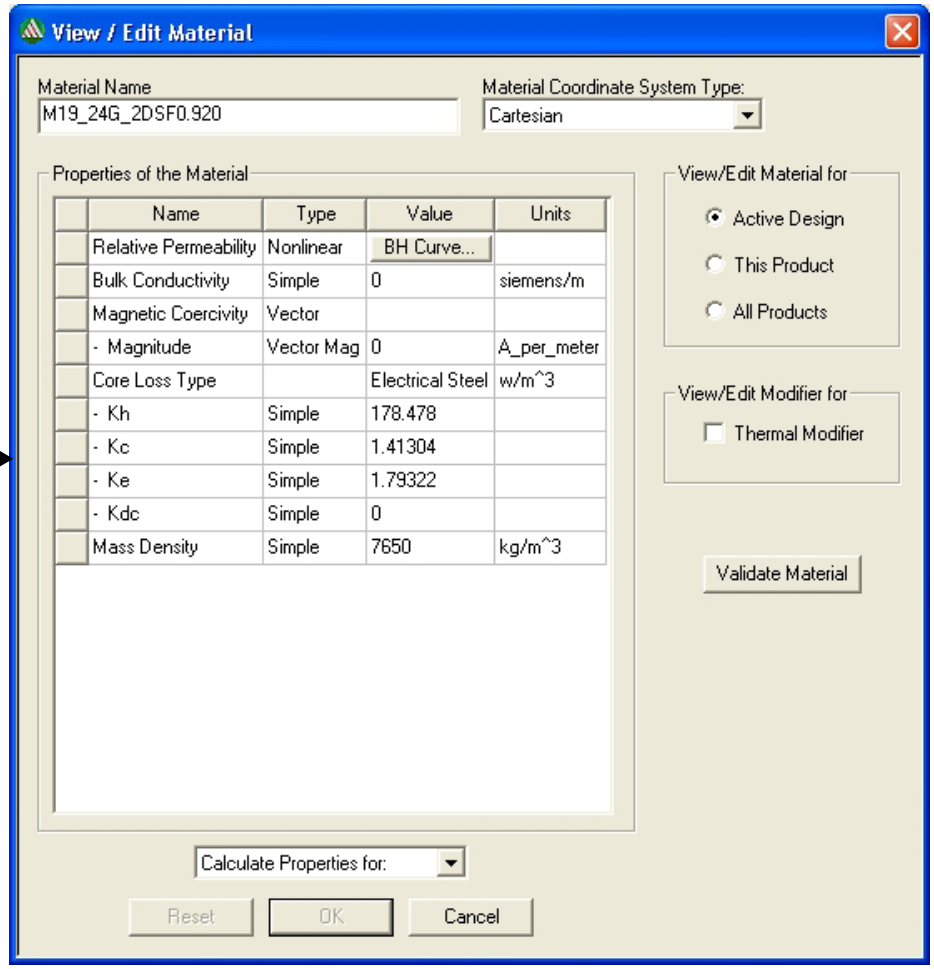
- If Eddy Loss is to be calculated, it is recommended to calculate the skin depth first, then have at least two layers of the elements fro the first skin depth

Loss Calculations

3. Core Loss Calculations

- Core Loss properties needs to be set to the material for which losses are to be calculated

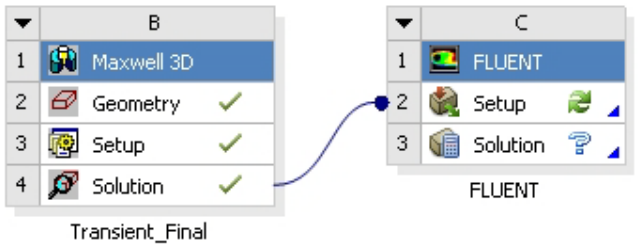


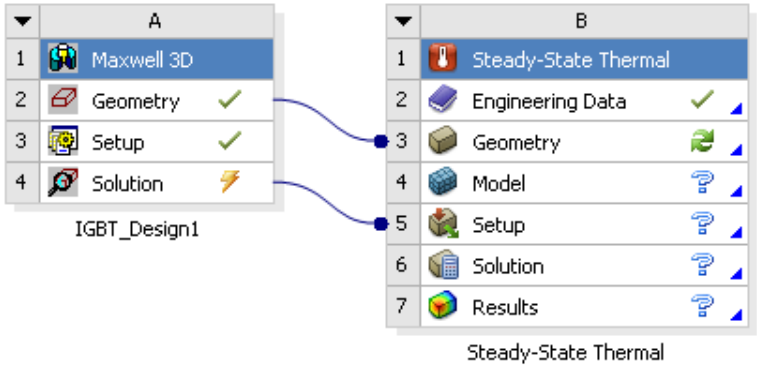


Loss Calculations

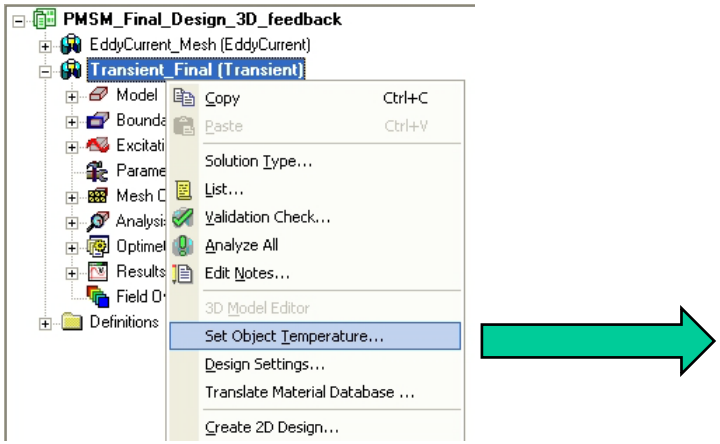
4. Loss Mapping to Thermal Solvers

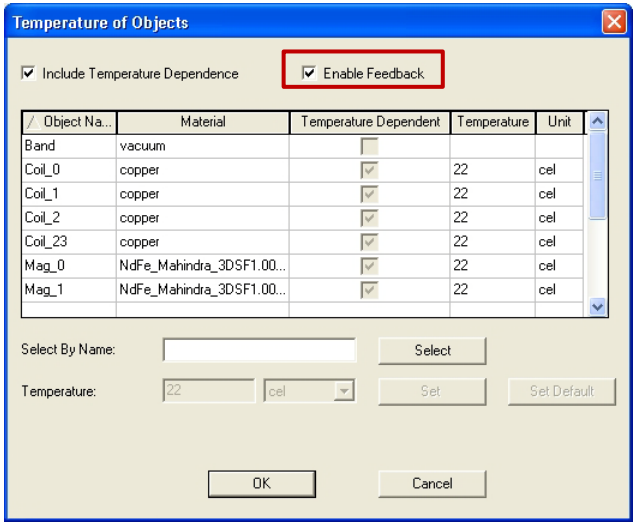
- Losses calculated in Maxwell can be mapped to ANSYS FLEUNT or ANSYS Mechanical for thermal solution





- Thermal Feedback to Maxwell

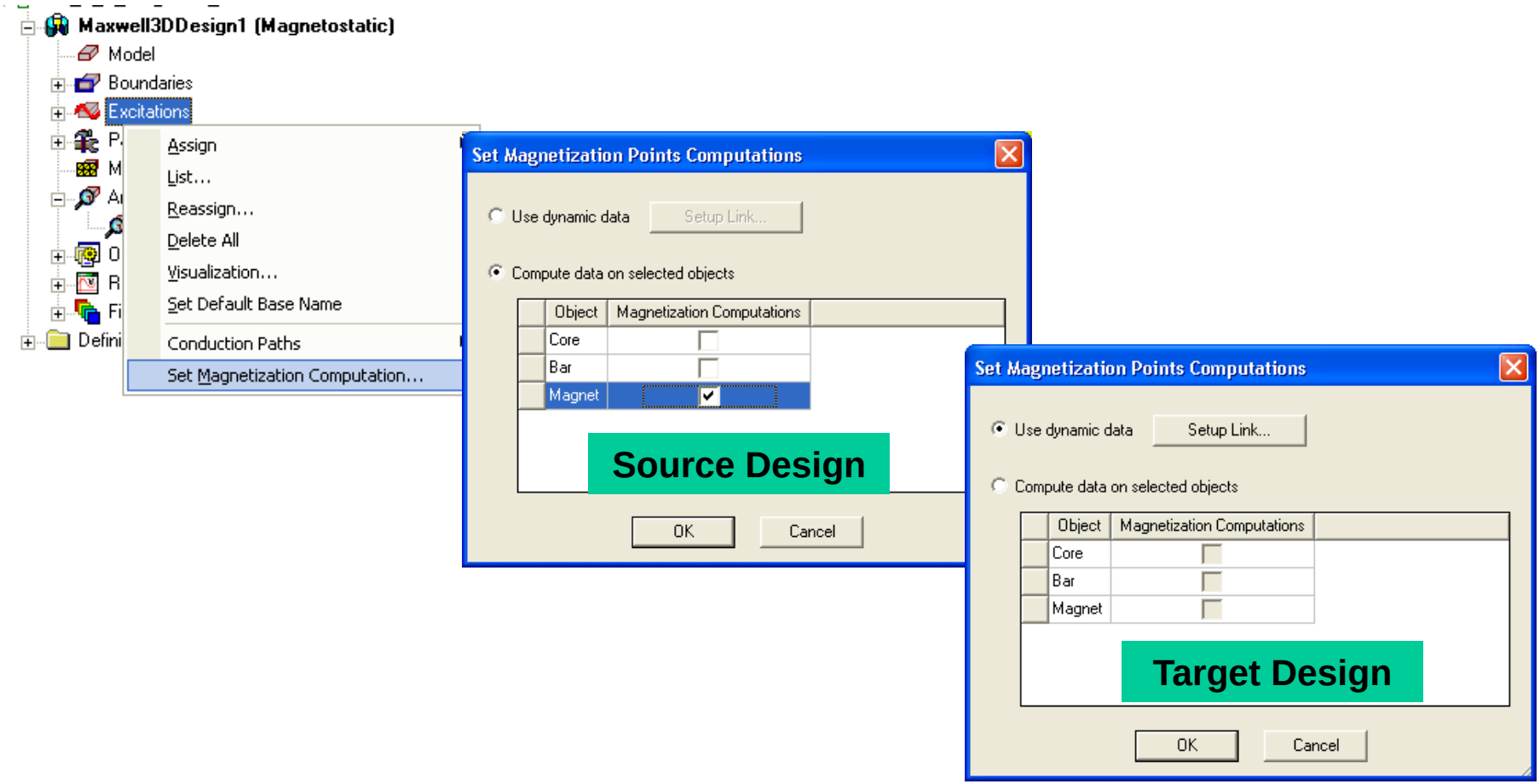




Magnetization and Demagnetization

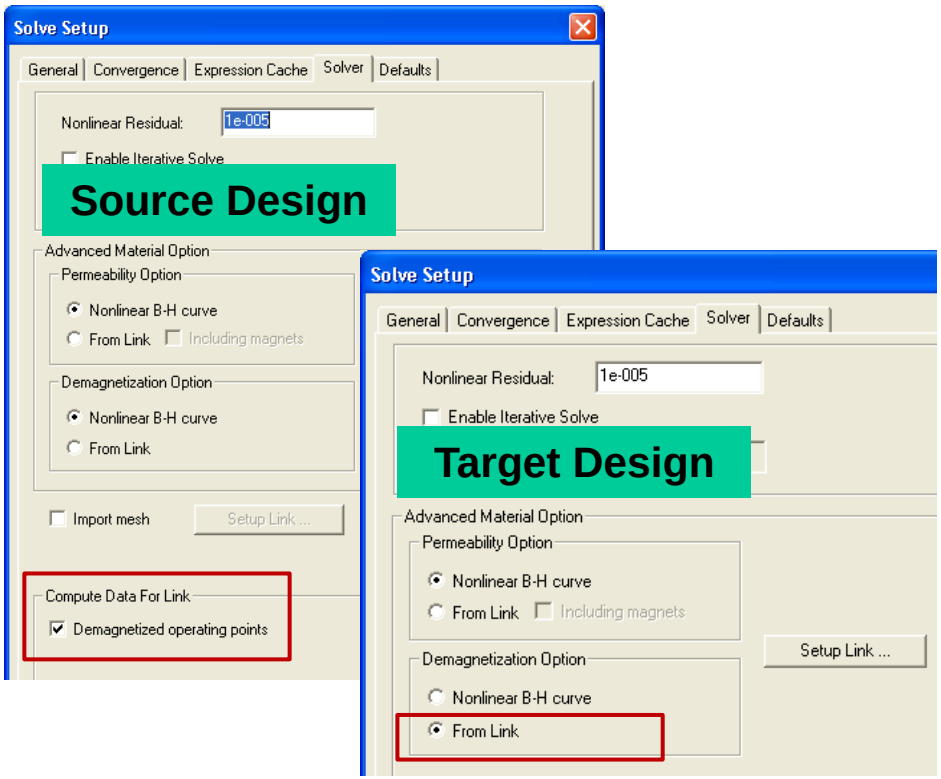
Magnetization

- Mesh must be linked between Source and Target design
- Source and Target designs can be Magnetostatic or Transient or combination of both

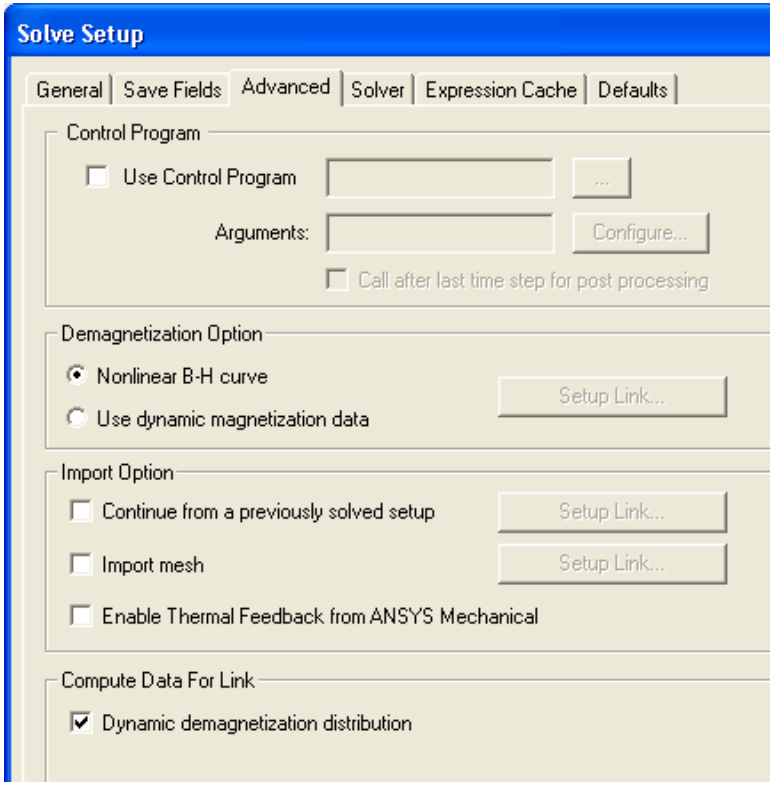


Magnetization and Demagnetization

Demagnetization



Magnetostatic Solver



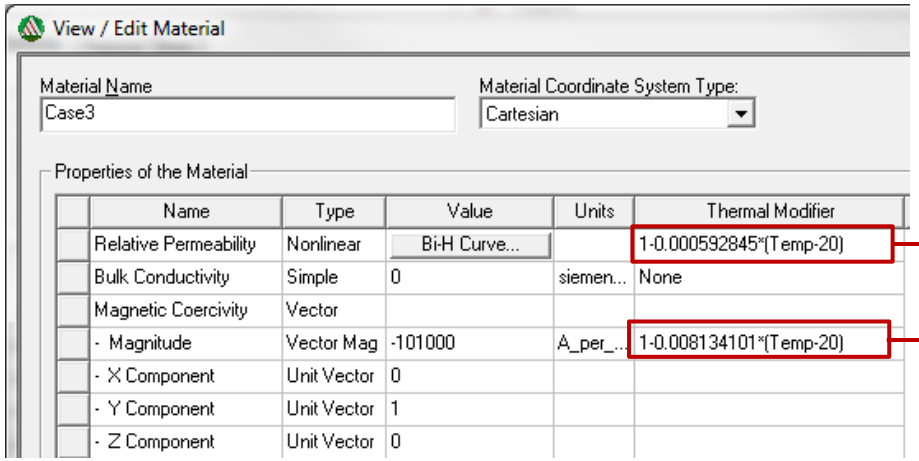
Transient Solver

Magnetization and Demagnetization

Temperature Dependent Demagnetization

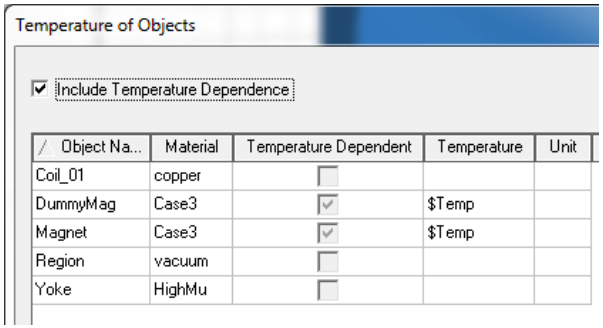
- Works with intrinsic BH Curve

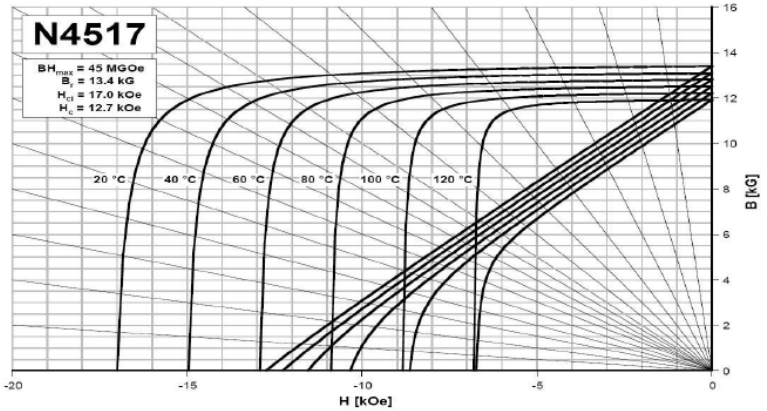
$$\begin{cases} B_r(T) = B_r(T_0)\cdot[1+\alpha(T-T_0)] = B_r \cdot p(T) \\ H_{ci}(T) = H_{ci}(T_0)\cdot[1+\beta(T-T_0)] = H_{ci} \cdot q(T) \end{cases}$$



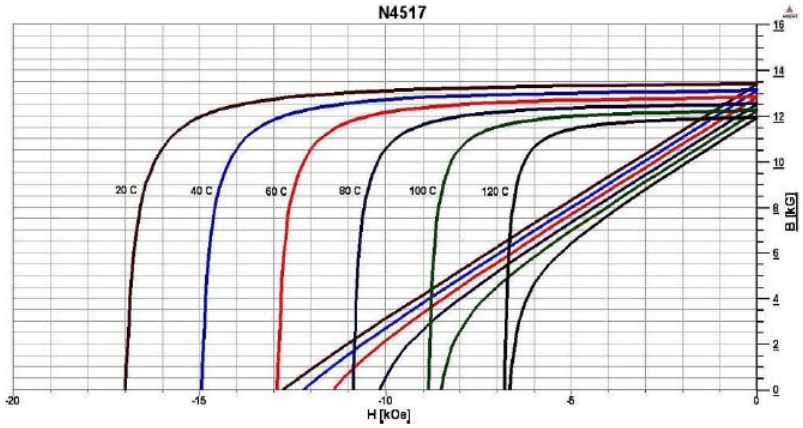
$p(T)$

$q(T)$





From Supplier Data



Derived from Maxwell

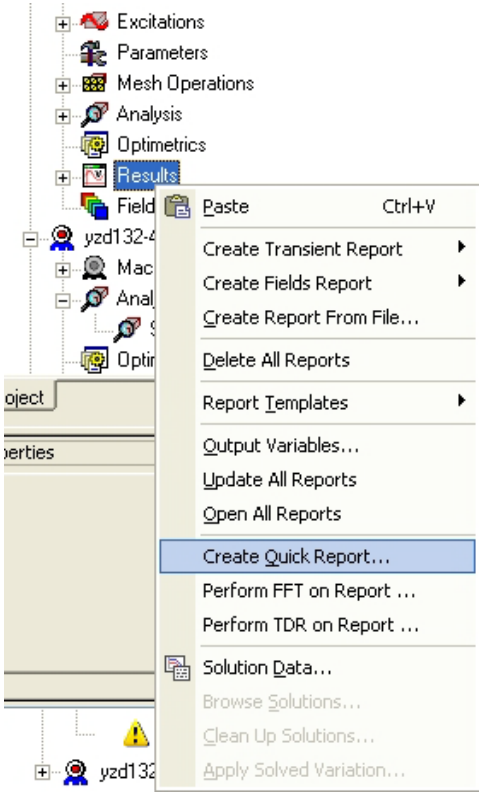
View Results

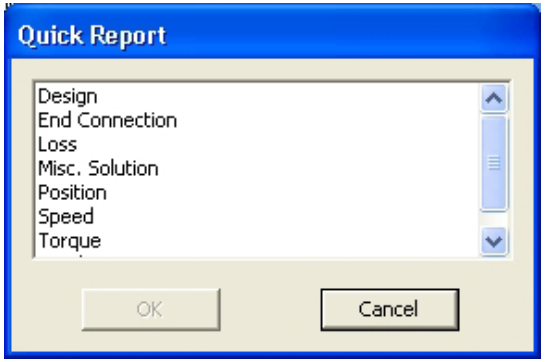
1. Create 2D Plots

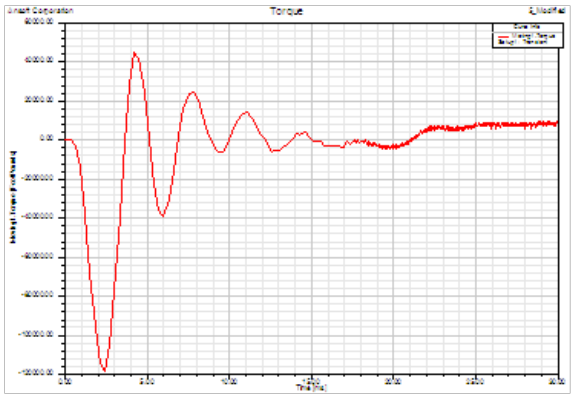
- Right mouse click on the tab **Results** in Project Manager window

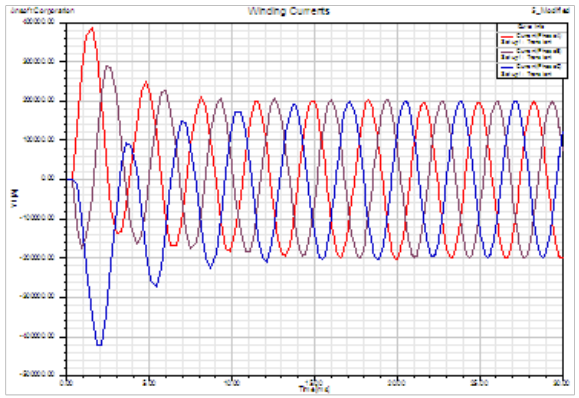
2. Create Quick Reports

- Available with Transient Solver
- Most key plots are included: **Torque, Phase Currents, Back EMF, Losses etc.**





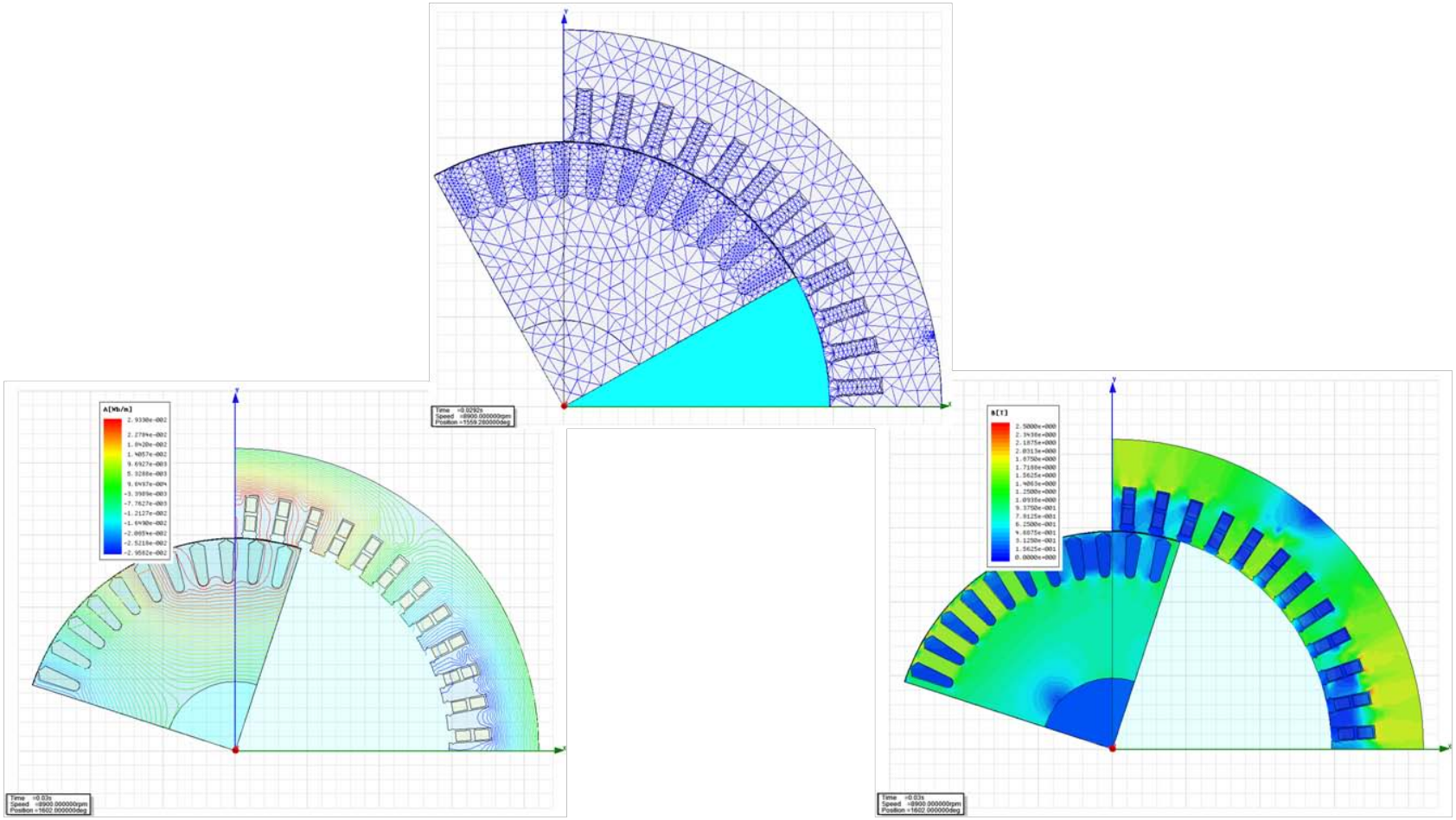




View Results

3. To View Mesh / Field Plots

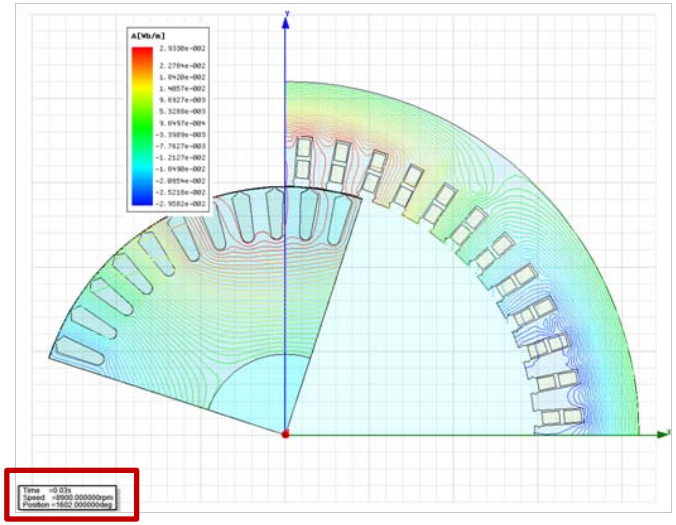
- Right mouse click on the tab **Field Overlays** in Project Manager window

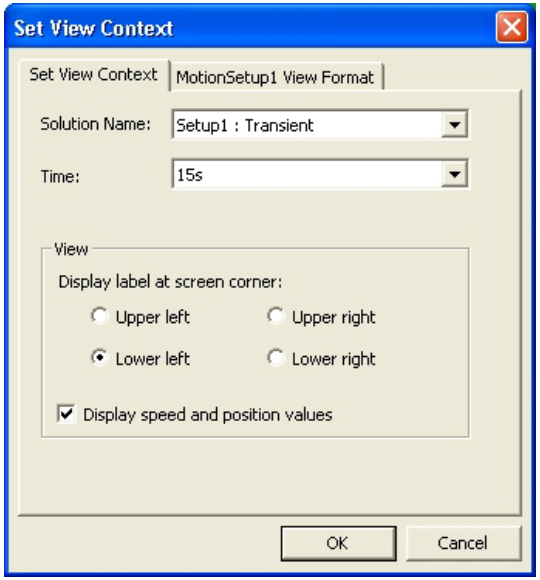


View Results

4. To View Mesh / Field Plots for Transient Solver

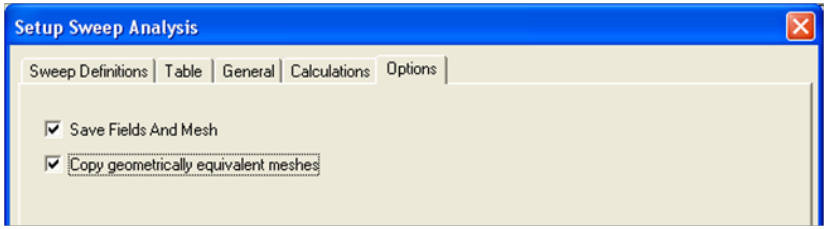
- To View Mesh or Field plots at various time intervals of Transient solver
  - Select the menu item *View > Set Solution Context* or double click on Time indication at the bottom of Graphic window





5. To View Mesh / Field Plots in Magnetostatic Solver

- To View Mesh or Field plots at different variations in Magnetostatic Solver, fields of those variations must be saved



Parametric Sweep and Optimization

1. Optimetrics Module

- **Parametrics:**
  - Define one or more variable sweep definitions, each specifying a series of variable values within a range. Easily view and compare the results using plot or table to determine how each design variation affects the performance of the design.
- **Optimization:**
  - Identify the cost function and the optimization goal. Optimetrics automatically changes the design parameter(s) to meet the goal. The cost function can be based on any solution quantity that can be computed, such as field values, R,L,C force, torque, volume or weight.
- **Sensitivity:**
  - Determine the sensitivity of the design to small changes in variables in the vicinity of a design point. Outputs include: Regression value at the current variable value, first derivative of the regression, second derivative of the regression.
- **Tuning:**
  - Variable values are changed interactively and the performance of the design is monitored. Useful after performing an optimization in Optimetrics to fine tune the optimal variable value and see how the design results are affected.
- **Statistical:**
  - Shows the distribution (Histogram) of a design output like force, torque or loss caused by a statistical variation (Monte Carlo) of input variables.

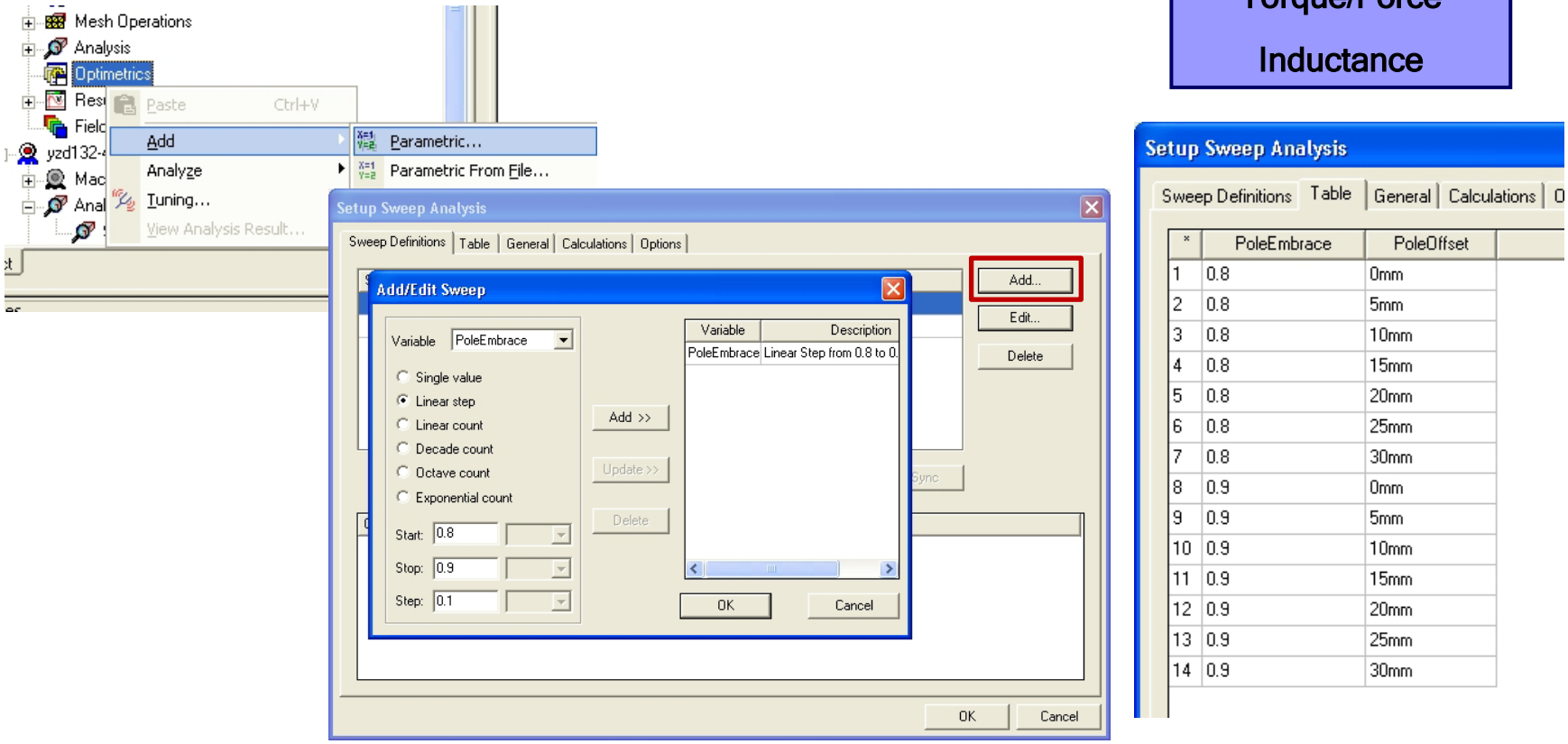
Parametric Sweep and Optimization

2. Parameters for Optimetrics

- Every item that has a value can be set as a variable and can be swept or optimized

Geometry
Excitation
Materials
Frequency
Speed
Torque/Force
Inductance

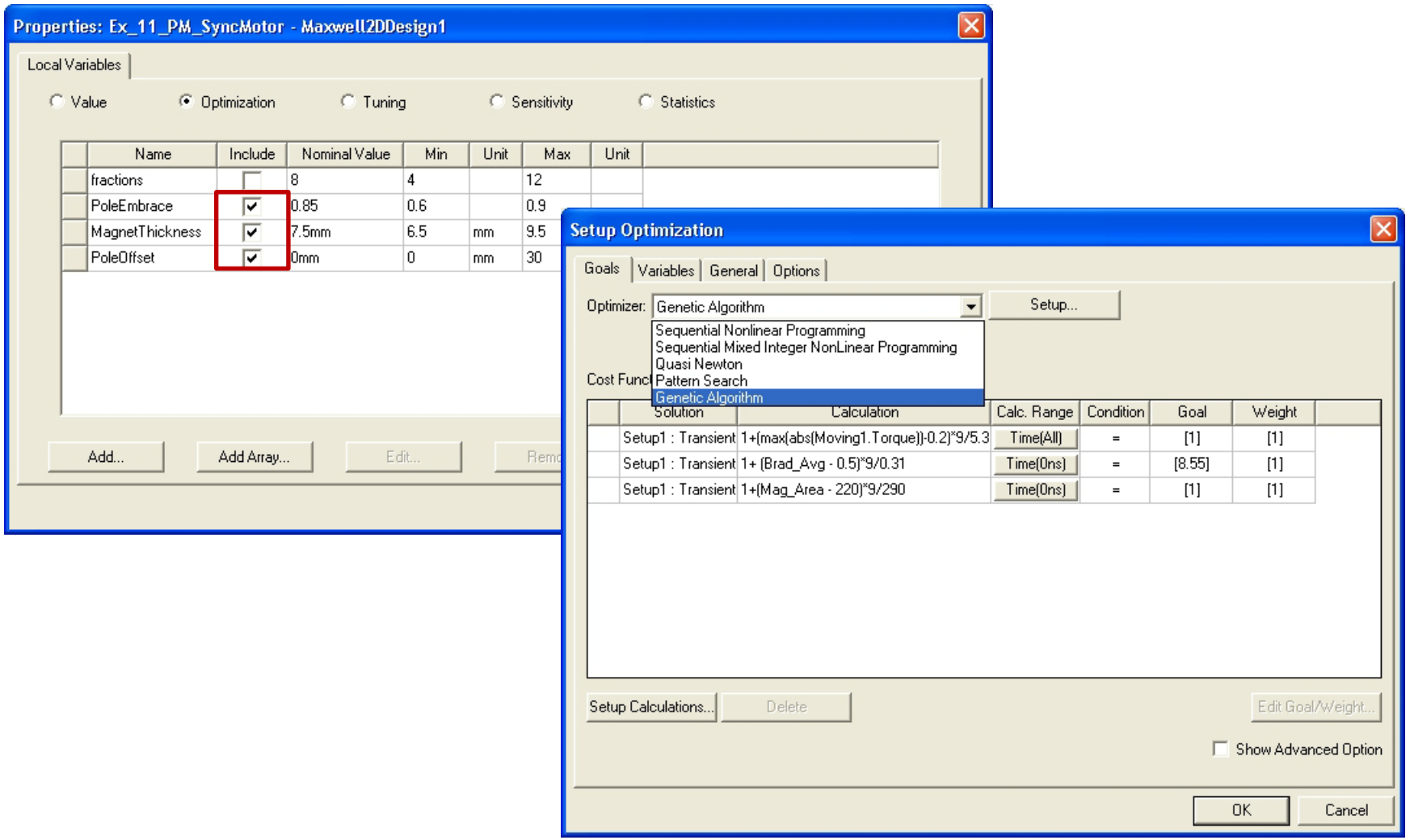
3. Setup Parametric Sweep



*	PoleEmbrace	PoleOffset
1	0.8	0mm
2	0.8	5mm
3	0.8	10mm
4	0.8	15mm
5	0.8	20mm
6	0.8	25mm
7	0.8	30mm
8	0.9	0mm
9	0.9	5mm
10	0.9	10mm
11	0.9	15mm
12	0.9	20mm
13	0.9	25mm
14	0.9	30mm

Parametric Sweep and Optimization

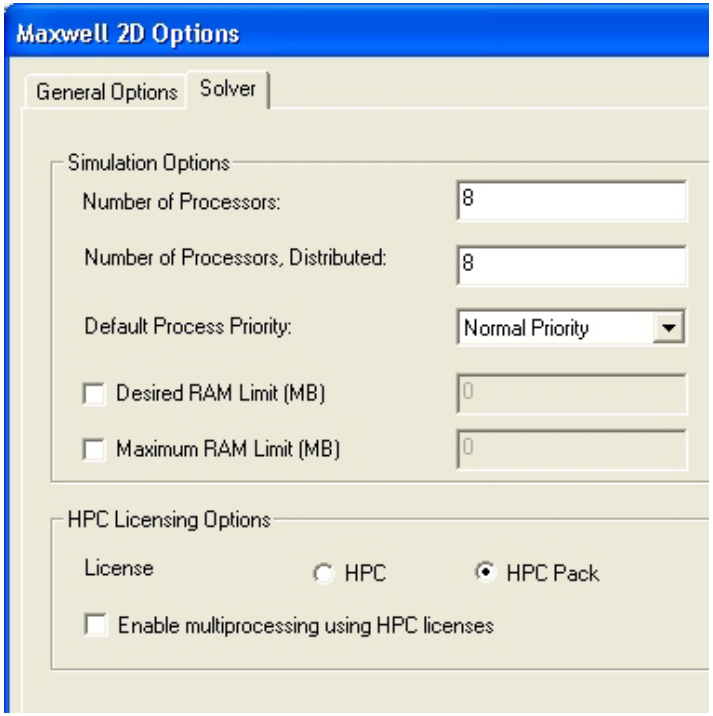
4. Setup Optimization



Increasing Simulation Speed

1. To Enable Multi-processor option

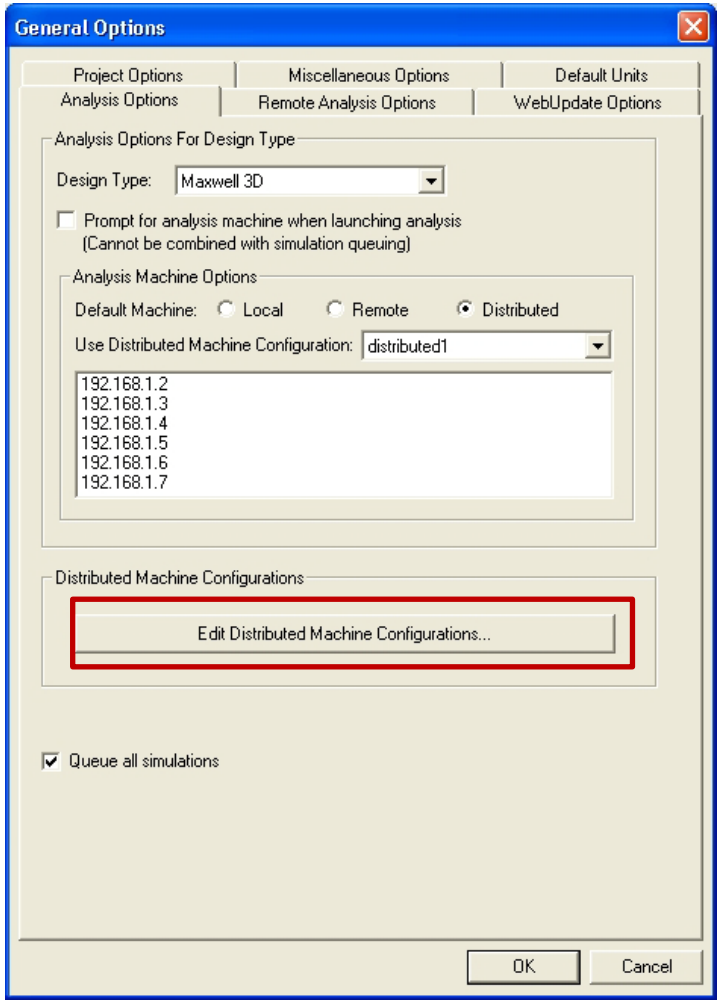
- Select the menu item *Tools > Options > Maxwell 2D Options*
- Maxwell will split a large matrix into smaller ones and solve simultaneously on different processors
- Each additional processor will contribute to solution speed improvements
- Speed increase will vary based on design and size of the model
- The benefit of multiprocessor option can be best seen in large 3D designs

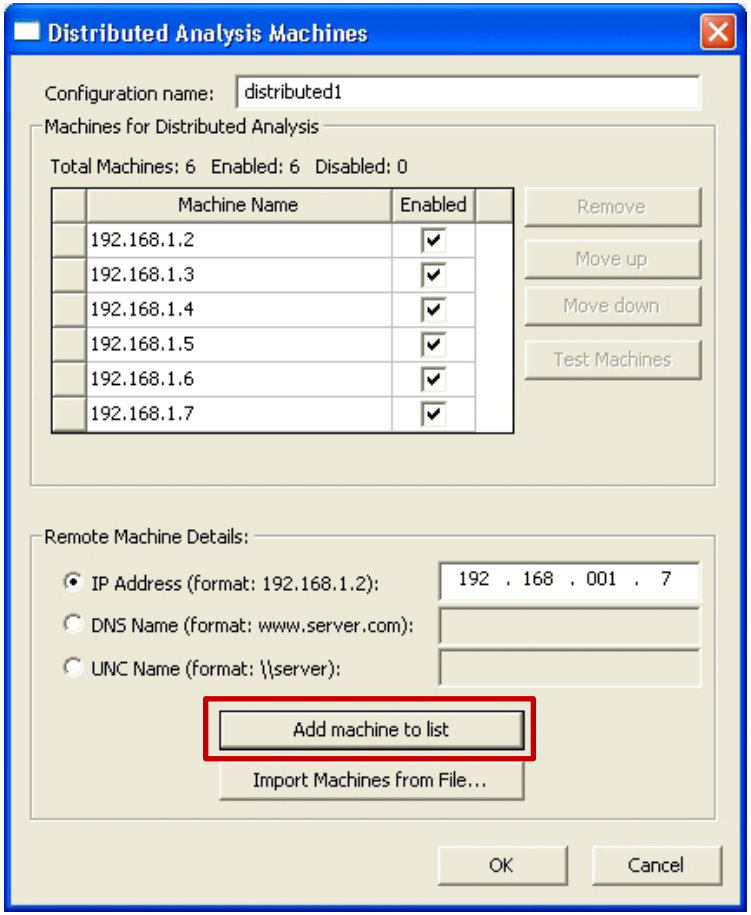


Increasing Simulation Speed

2. To Enable Distributed Analysis to all Parametric Designs

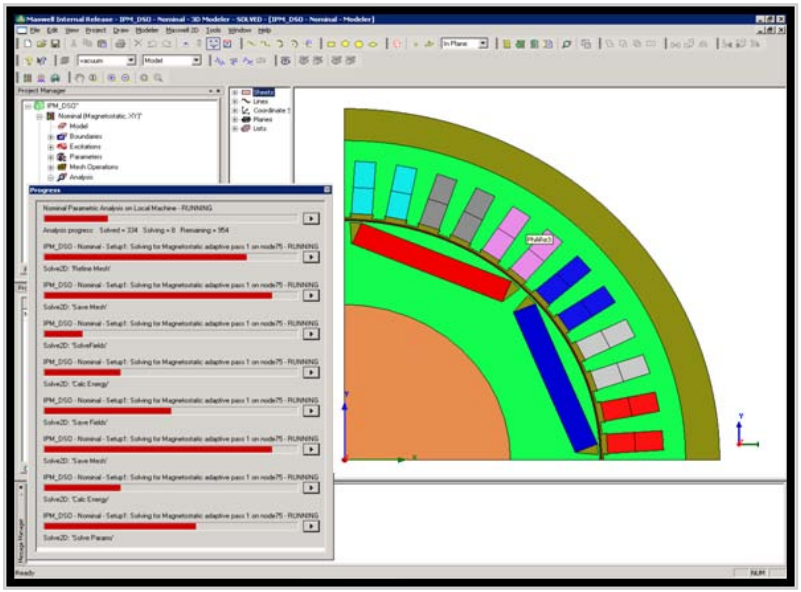
- Select the menu item *Tools > Options > General Options*

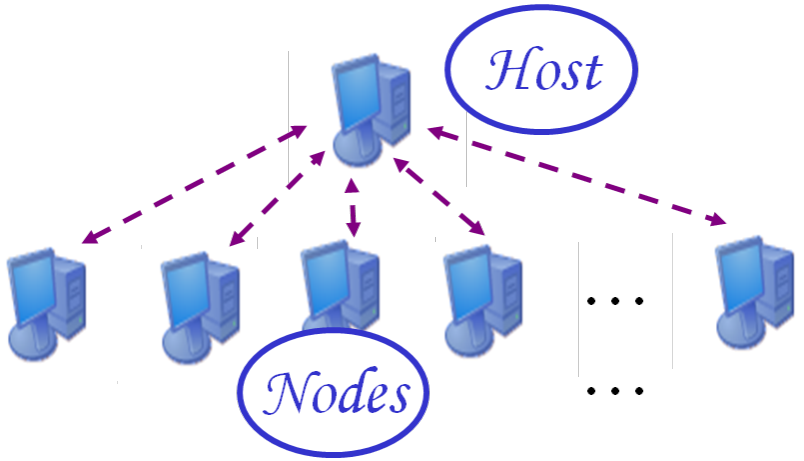




Increasing Simulation Speed

3. Substantial Speed Increase





RED HAT ENTERPRISE LINUX
:: The corporate Linux standard

solaris

Microsoft Windows

Microsoft Windows xp Professional x64 Edition

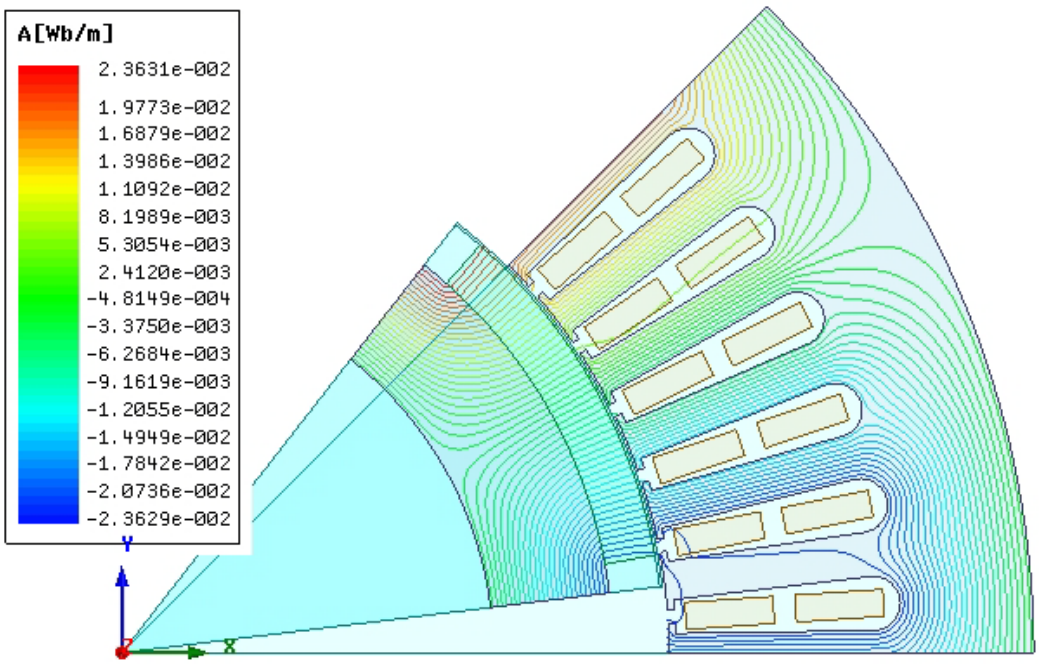
Parametric variations: 1296

Solve time, one processor: 13 hr, 15 min

Solve time, distributed, 8 CPUs: 2 hr, 30 min

Speed improvement: 5.3X

Permanent Magnet Synchronous Machine



Motors - Permanent Magnet Synchronous Machine

Content

- **RMxprt (pg 3-14)**
  - Basic Theory
  - Review Example
  - Add Unique Winding Arrangement
  - Setup Parametric Problem
  - Export Design to Maxwell 2D
- **Maxwell: Cogging Torque (pg 15-35)**
  - Review Maxwell Setup
  - Apply Mesh Operations
  - Solve Nominal Problem
  - Create Variables
  - Setup Optimization Problem
  - Review pre-solved Optimization Results
- **Maxwell: Open Circuit Back EMF (pg 36-42)**
  - Define Material Core Loss Characteristics
  - Consider Power Loss in Magnets
  - Solve Problem and Review Results
- **Maxwell: Rated Condition (pg 43-53)**
  - Functional Voltage Source
  - Modify Rotor Geometry using UDP's
  - Winding Setup Definitions and Variable Definition
  - Choosing Optimal Time Step
  - Solve Problem and Review pre-solved Results
- **Simplorer: Drive Design (pg 54-57)**
  - Create a Machine Model
  - Use the Model in Circuit Simulation

Notes:

- RMxprt/Maxwell V14 or higher is required
- Basic knowledge of electric machine is required
- Basic understanding of Finite Element Analysis is required

1. RMxprt

ASSM: Adjustable-Speed Synchronous Machine

- Rotor speed is controlled by adjusting the frequency of the input voltage
- Unlike brushless PMDC motors, ASSM does not utilize the position sensors.
- Rotor can be either inner or outer type
- Can operate as a generator or as a motor
  - **Motor** Mode:
    - Sinusoidal AC source
    - DC source via a DC to AC inverter
  - **Generator** Mode:
    - Supplies an AC source for electric loads

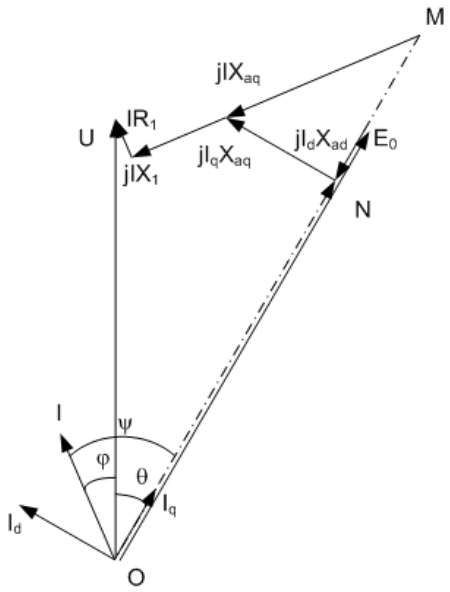
- **Background**
  - Input voltage *U* is the reference phasor, let the angle *I* lags *U* be φ, the power factor angle

$$\mathbf{I} = I\angle -\varphi$$

  - Let the angle *I* lags *E0* be ψ. The d- and the q-axis currents can be obtained respectively as follows

$$\mathbf{I} = \begin{bmatrix} I_d \\ I_q \end{bmatrix} = I\begin{bmatrix} \sin\psi \\ \cos\psi \end{bmatrix}$$

$$\psi = \tan^{-1}\frac{I_d}{I_q}$$



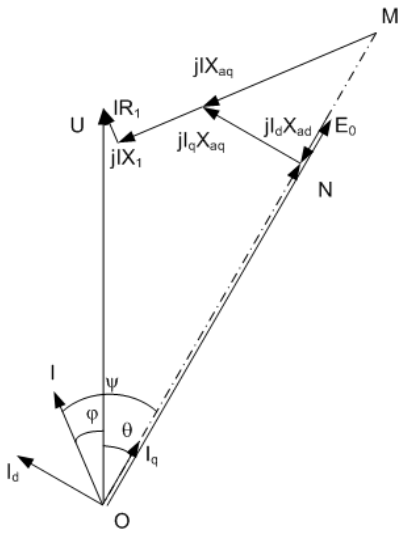
  - *OM* can be used to determine the direction of *E*0

$$\overline{OM} = \mathbf{U} - \mathbf{I}(R_1 + \mathrm{j}X_1 + \mathrm{j}X_{aq})$$

Motors - Permanent Magnet Synchronous Machine

- Let the angle *E0* lags *U* be θ, which is called the torque angle for the motor, then the angle is

$$\psi = \varphi - \theta$$



- For a given torque angle θ:

$$\begin{bmatrix} X_d & R_1 \\ -R_1 & X_q \end{bmatrix} \begin{bmatrix} I_d \\ I_q \end{bmatrix} = \begin{bmatrix} U\cos\theta - E_0 \\ -U\sin\theta \end{bmatrix}$$

- Solving for *Id* and *Iq* yields:

$$\begin{bmatrix} I_d \\ I_q \end{bmatrix} = \frac{1}{R_1^2 + X_d X_q} \begin{bmatrix} X_q(U\cos\theta - E_0) + R_1 U\sin\theta \\ R_1(U\cos\theta - E_0) - X_d U\sin\theta \end{bmatrix}$$

- The power factor angle φ is

$$\varphi = \psi + \theta$$

- The Input electric power is

$$P_1 = 3UI\cos\varphi$$

- The Output mechanical power is

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_{Fe})$$

  - Pf_w: Frictional and Wind Loss
  - $P_{Cua:}$ Armature Copper Loss
  - P_{Fe}: Iron-core Loss

- Torque:

$$T_2 = \frac{P_2}{\omega}$$

- Efficiency:

$$\eta = \frac{P_2}{P_1} \times 100\%$$

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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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 Input File

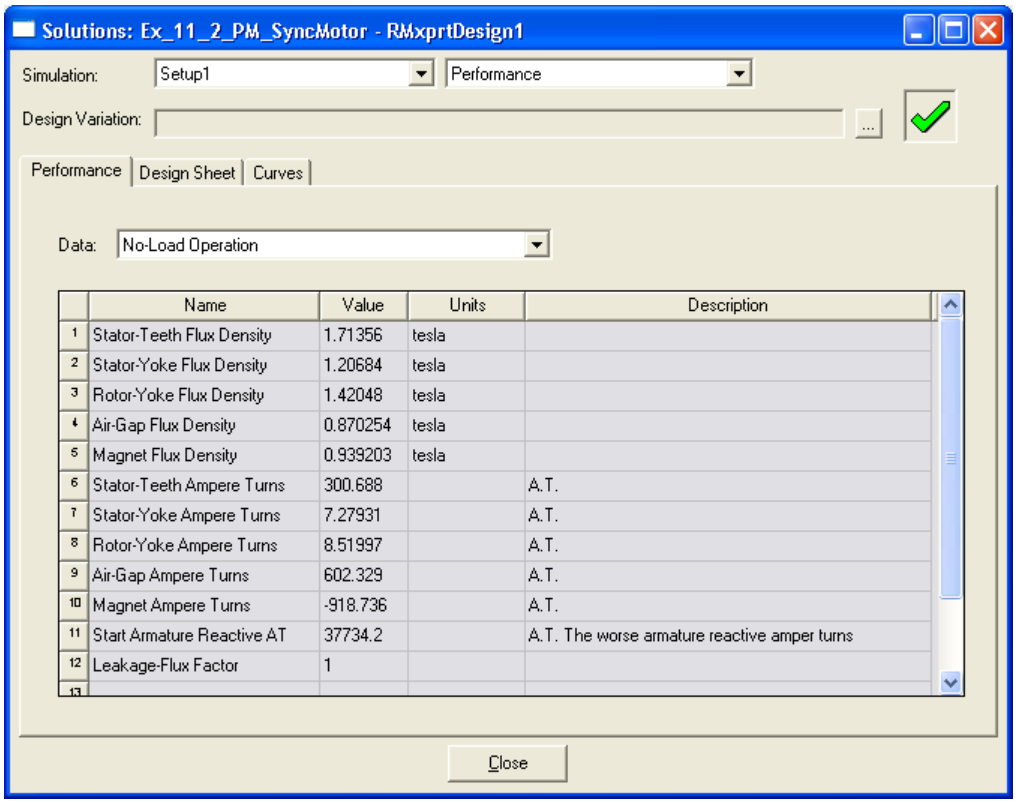
- To Open File
  - Select the menu item *File > Open*
  - Locate the file **"Ex_11_2_PM_SyncMotor.mxwl"** and **Open** it

Analyze Default Setup

- To Analyze Default Setup
  - Select the menu item *RMxprt > Analyze All*

Results

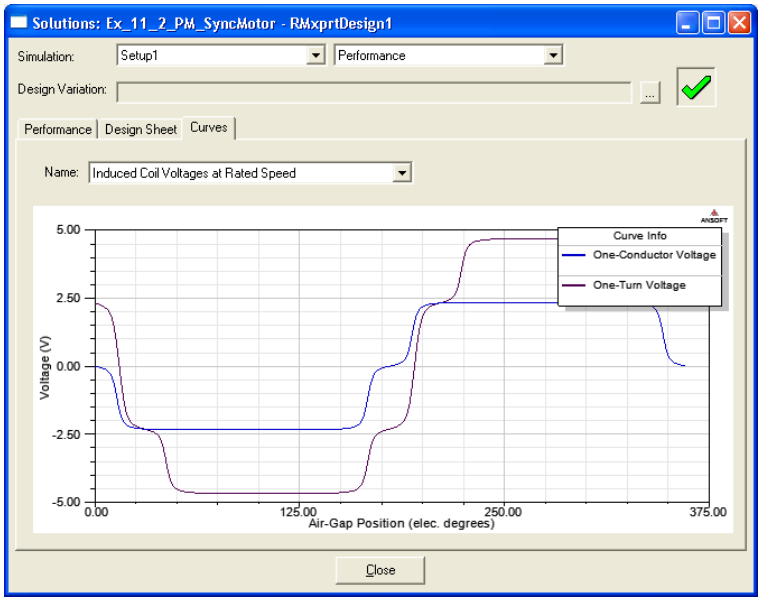
- To View Results
  - Select the menu item *RMxprt > Results > Solution Data*
  - In Solutions window,
    - **Performance** tab
      - Set Data to **No-load Operation** to view corresponding results



	Name	Value	Units	Description
1	Stator-Teeth Flux Density	1.71356	tesla	
2	Stator-Yoke Flux Density	1.20684	tesla	
3	Rotor-Yoke Flux Density	1.42048	tesla	
4	Air-Gap Flux Density	0.870254	tesla	
5	Magnet Flux Density	0.939203	tesla	
6	Stator-Teeth Ampere Turns	300.688		A.T.
7	Stator-Yoke Ampere Turns	7.27931		A.T.
8	Rotor-Yoke Ampere Turns	8.51997		A.T.
9	Air-Gap Ampere Turns	602.329		A.T.
10	Magnet Ampere Turns	-918.736		A.T.
11	Start Armature Reactive AT	37734.2		A.T. The worse armature reactive amper turns
12	Leakage-Flux Factor	1		
13				

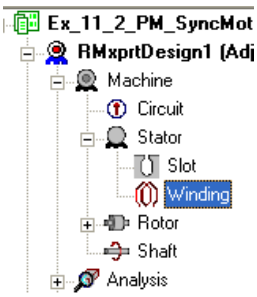
Motors - Permanent Magnet Synchronous Machine

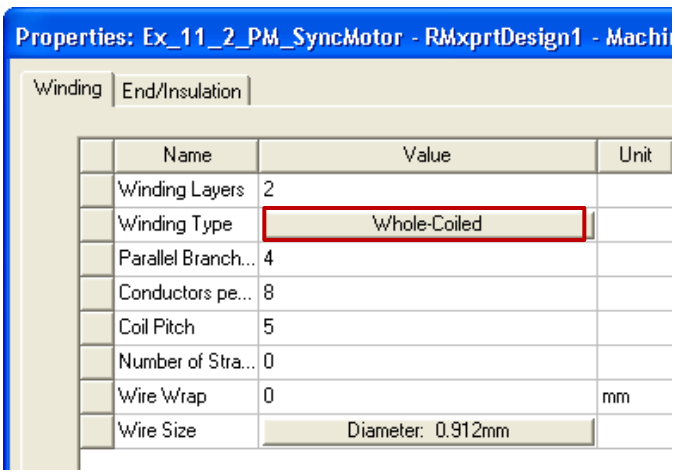
- **Curves** tab
  - Select the Performance curve that needs to be analyzed

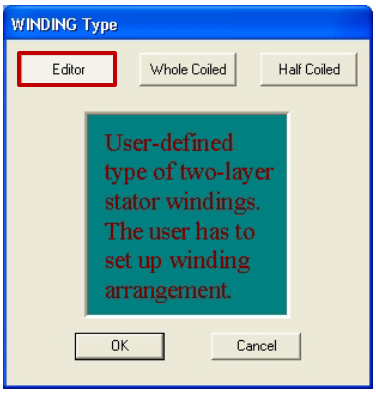


Add New Winding Arrangement

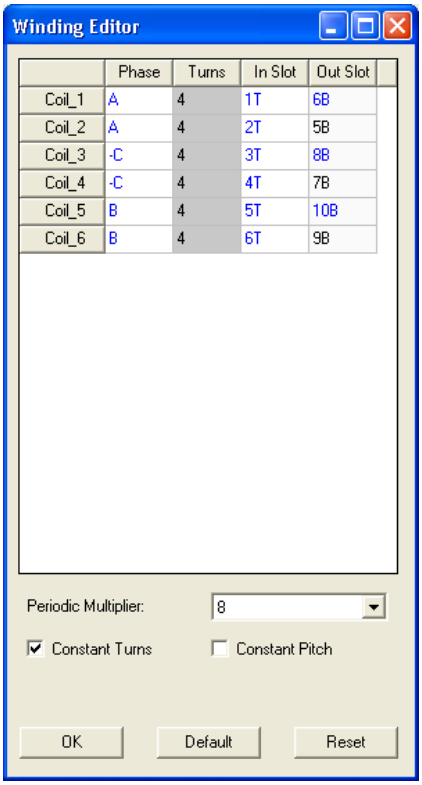
- **To Set User Defined Winding**
  - Expand the Project Manager tree to view **Stator**
  - Double click on the tab **Winding** to modify its parameters
  - In Properties window,
    - Select the tab **Whole-Coiled** to modify its parameters
    - In Winding window,
      - Select the tab **Editor**
    - Press **OK** to close winding window



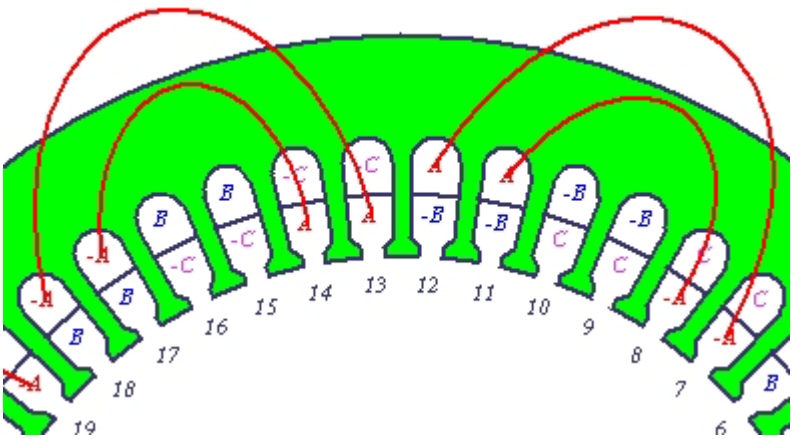




- **To Modify Winding Arrangement**
  - Right click on the **Winding Editor** page and select **Edit Layout**
  - In Winding Editor window,
    - Constant Pitch : ☐ **Unchecked**
    - Change Outer Slot for
      - Coil_2: **5B**
      - Coil_4: **7B**
      - Coil_6: **9B**
    - Press **OK**



- **To View New winding Arrangement**
  - Place the mouse on any of the A Phase Coils, right click and select **Connect one-phase coils**

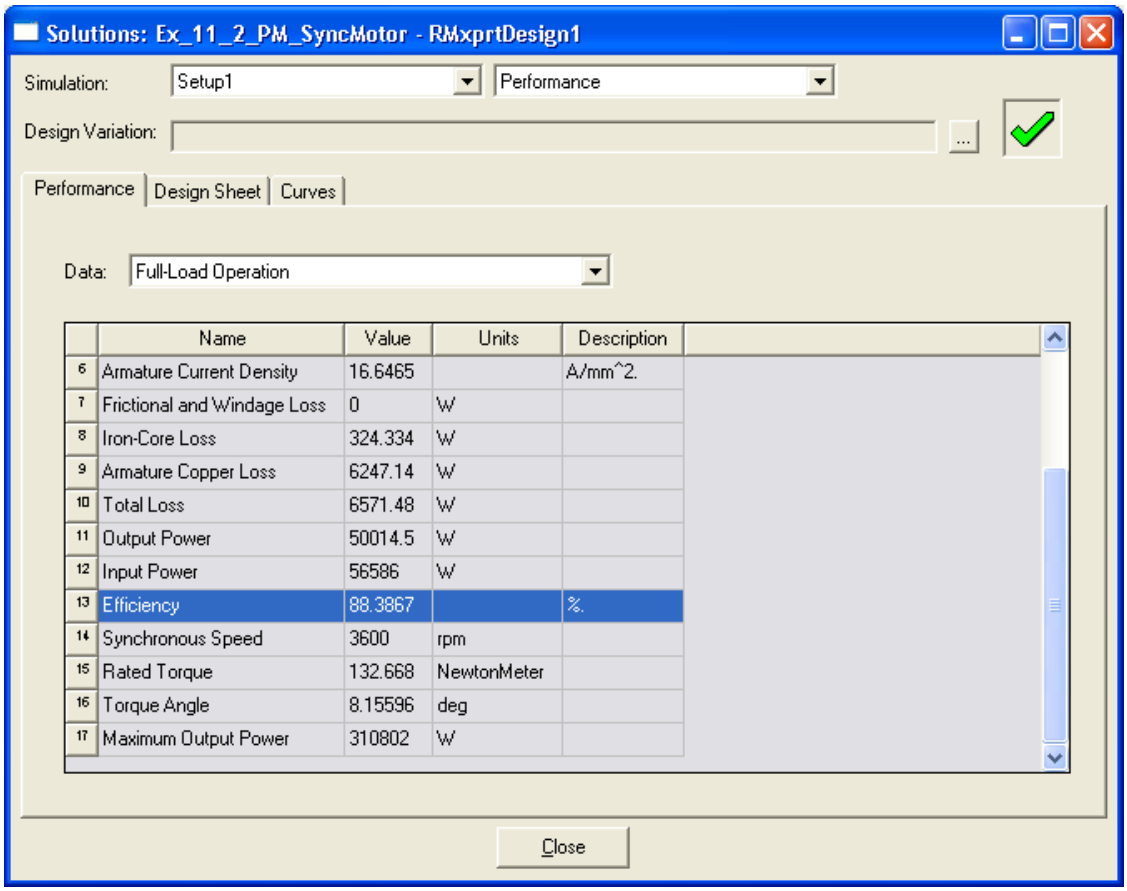


Analyze

- To Analyze Setup
  - Select the menu item *RMxprt > Analyze All*

Results

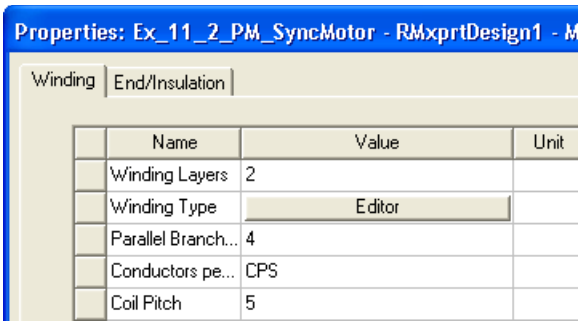
- To View Results
  - Select the menu item *RMxprt > Results > Solution Data*
  - In Solutions window,
    - **Performance** tab
      - Set Data to **Full-Load Operation** to view corresponding results

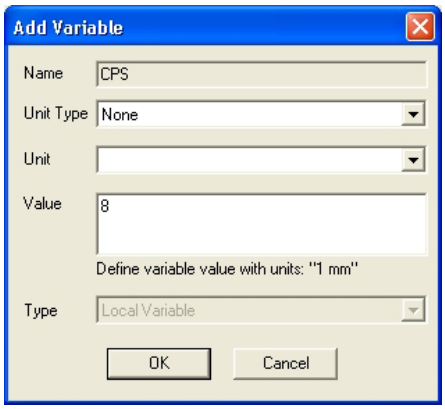


	Name	Value	Units	Description
6	Armature Current Density	16.6465		A/mm^2.
7	Frictional and Windage Loss	0	W	
8	Iron-Core Loss	324.334	W	
9	Armature Copper Loss	6247.14	W	
10	Total Loss	6571.48	W	
11	Output Power	50014.5	W	
12	Input Power	56586	W	
13	Efficiency	88.3867		%
14	Synchronous Speed	3600	rpm	
15	Rated Torque	132.668	NewtonMeter	
16	Torque Angle	8.15596	deg	
17	Maximum Output Power	310802	W	

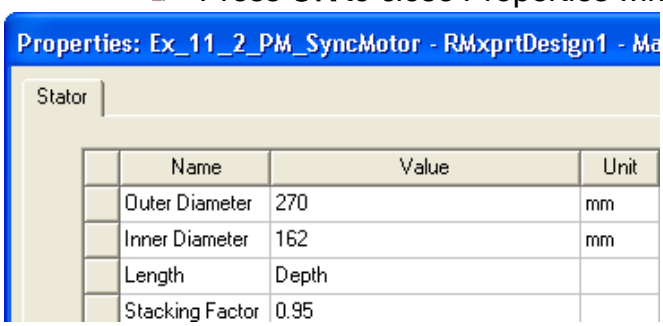
Setup Parametric Analysis

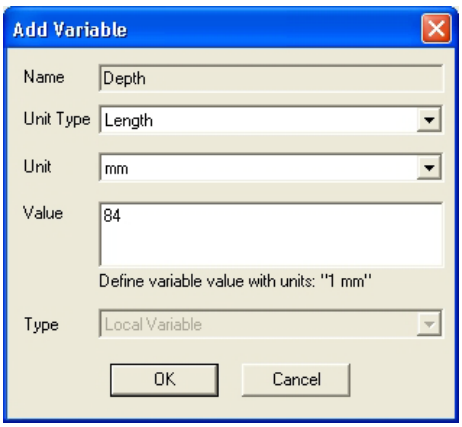
- **Add variable for No. of Conductors**
  - Expand the Project Manager tree to view **Stator**
  - Double click on the tab **Winding** to modify its Properties
  - In Winding window,
    - Conductors Per Slot: **CPS**
    - Press **Tab** key
    - In Add Variable window,
      - Value: **8**
      - Press **OK**
    - Press **OK** to close Properties window





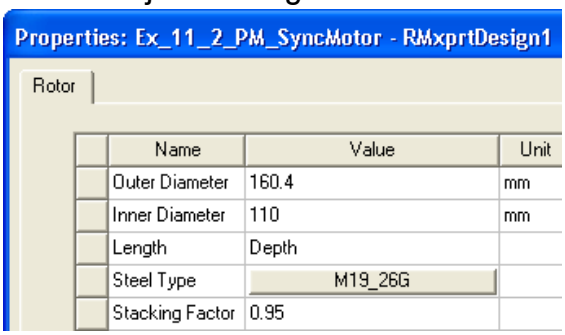
- **Set Depth Parameter for Stator**
  - Double click on the tab **Stator** from Project Manager tree
  - In Properties window,
    - Length: **Depth**
    - Press **Tab** key
    - In Add Variable window,
      - Value: **84**
      - Press **OK**
    - Press **OK** to close Properties window



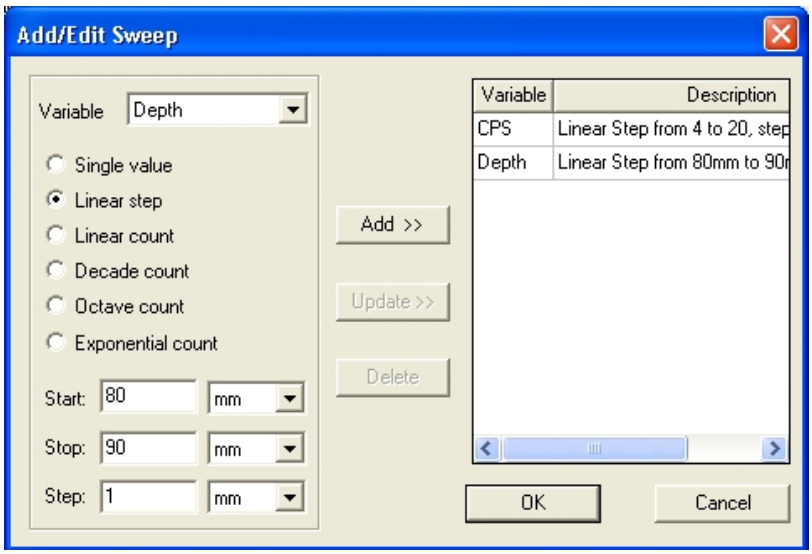


Motors - Permanent Magnet Synchronous Machine

- **Set Depth Parameter for Rotor**
  - Double click on the tab **Rotor** from Project Manager tree
  - In Properties window,
    - Length: **Depth**
    - Press **OK**

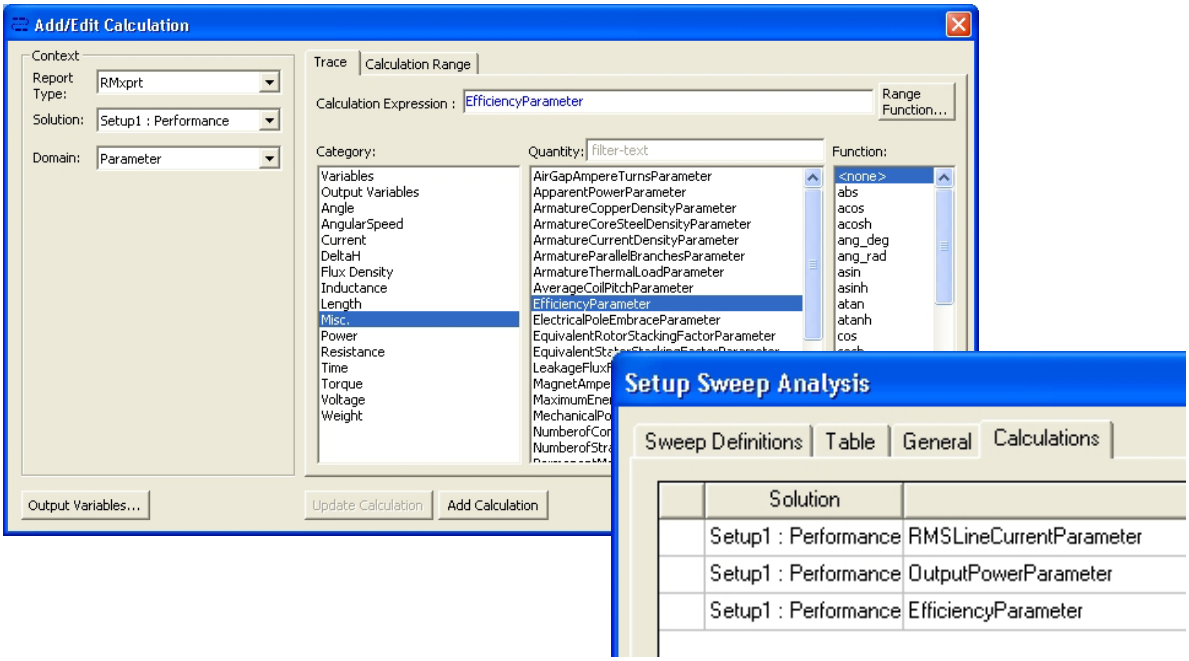


- **Add Parametric Setup**
  - Select the menu item *RMxprt > Optimetrics Analysis > Add Parametric*
  - In Setup Sweep Analysis window, select **Add**
  - In Add/Edit Sweep window,
    - Variable: **CPS**
    - Select **Linear step**
    - Start: **4**
    - Stop: **20**
    - Step: **1**
    - Select **Add>>**
    - Change Variable to **Depth**
    - Select **Linear step**
    - Start: **80**
    - Stop: **90**
    - Step: **1**
    - Select **Add>>**
    - Press **OK**



Motors - Permanent Magnet Synchronous Machine

- In Set Sweep Analysis window, **Calculations** tab select **Setup Calculations**
- In Add/Edit Calculations window,
  - Category: **Current**
  - Quantity: **RMSLineCurrentParameter**
  - Select **Add Calculation**
  - Change Category to **Power**
  - Quantity: **OutputPowerParameter**
  - Select **Add Calculation**
  - Change Category to **Misc.**
  - Quantity: **EfficiencyParameter**
  - Select **Add Calculation**
  - Select **Done**
- Press **OK** to close Setup Sweep Analysis window



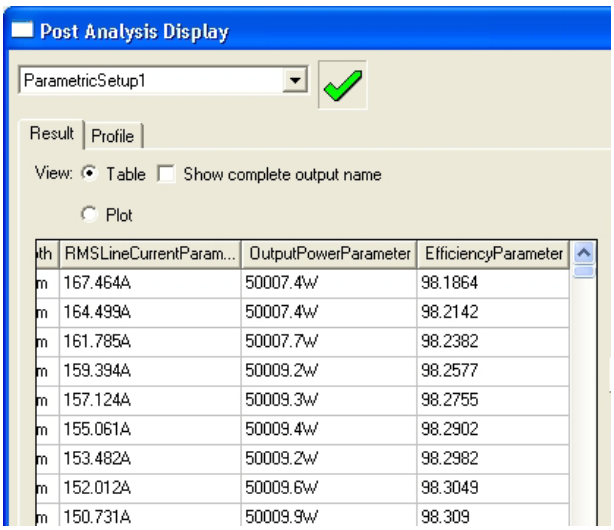
Analyze Parametric Setup

To Run Parametric Analysis

- Expand the Project Manager tree to view **Optimetrics**
- Right Click on the tab **ParametricSetup1** and select **Analyze**

View Results

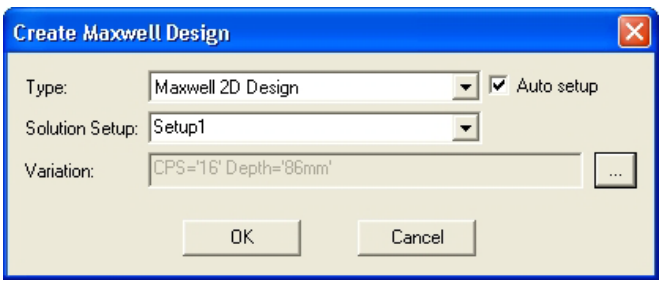
- To View Results
  - Select the menu item *RMxprt > Optimetrics Analysis > Optimetrics Results*
  - In Post Analysis Display window,
    - Set view to **Table**
    - It can be seen that Efficiency increases from 87 to 98

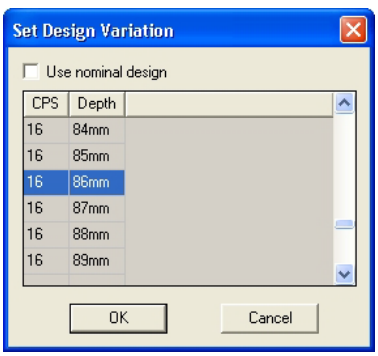


ith	RMSLineCurrentParam...	OutputPowerParameter	EfficiencyParameter
m	167.464A	50007.4W	98.1864
m	164.499A	50007.4W	98.2142
m	161.785A	50007.7W	98.2382
m	159.394A	50009.2W	98.2577
m	157.124A	50009.3W	98.2755
m	155.061A	50009.4W	98.2902
m	153.482A	50009.2W	98.2982
m	152.012A	50009.6W	98.3049
m	150.731A	50009.9W	98.309

Create Maxwell Design

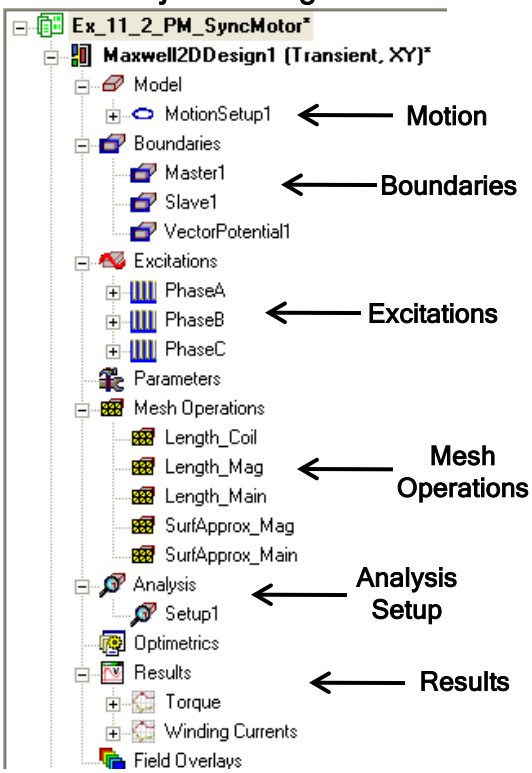
- To Create Maxwell Design
  - Expand the Project Manager tree to view **Analysis**
  - Right click on the tab **Setup1** and select **Create Maxwell Design**
  - In Create Maxwell Design window,
    - Select the button for the **variations**
    - In Set Design Variations window,
      - Use Nominal Design: ☐ **Unchecked**
      - Select the variation with CPS **16** and Depth of **86mm**
      - Press **OK**
    - Press **OK** to create Maxwell design



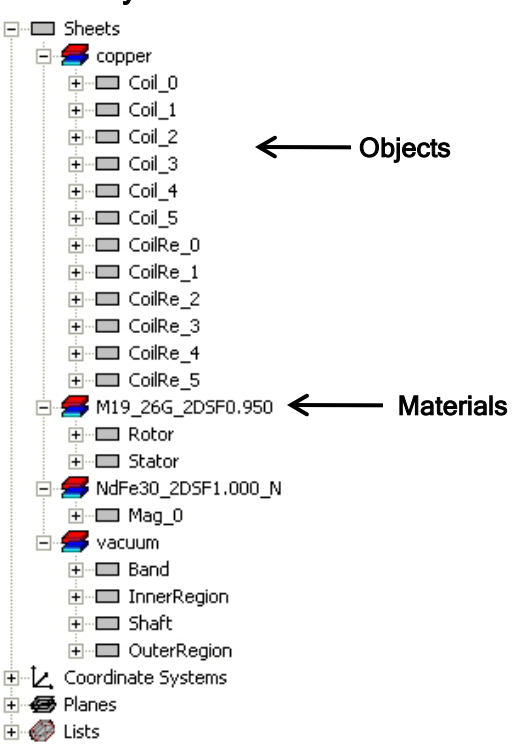


Maxwell 2D : Base Design

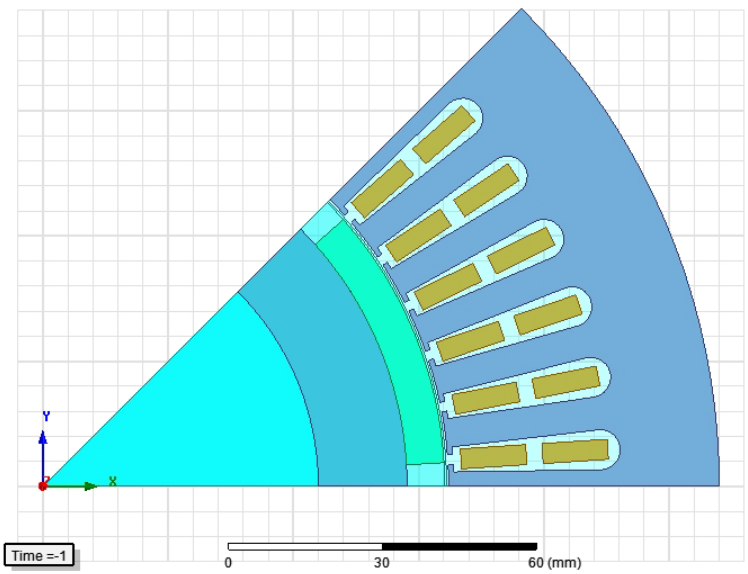
Project Manager Tree



History Tree



Graphics Window



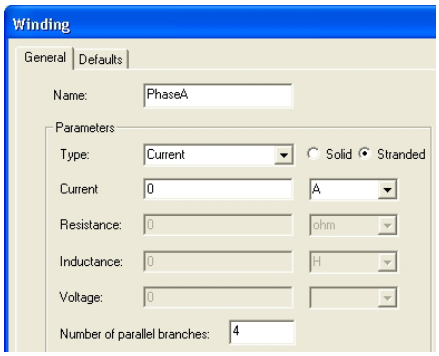
2. Maxwell Cogging Torque

Change Design Name

- **To Change the Name of Design**
  - Right click on the Maxwell2D Design from Project Manager tree and select **Rename**
  - Change the name of the design to **PMSM_CT**

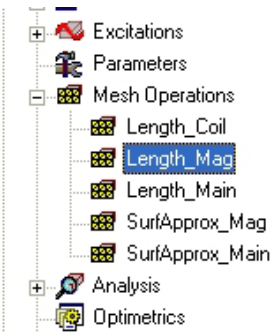
Modify Excitations

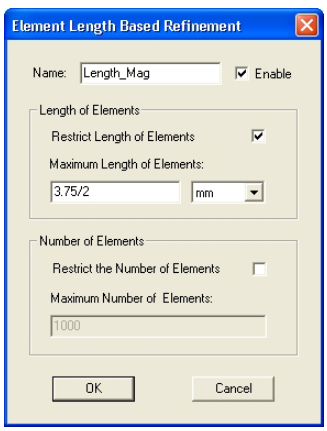
- **To Modify Excitations**
  - Expand the Project Manager tree to view **Excitations**
  - Double click on the tab **PhaseA** to modify its parameters
  - In Winding window,
    - Change Type to **Current**
    - Press **OK**
  - Repeat the same for **PhaseB** and **PhaseC**



Modify Mesh Parameters

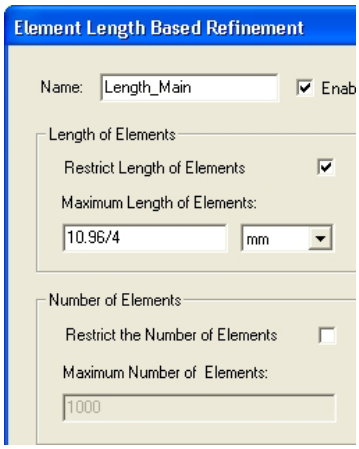
- **To Modify Mesh Operation for Magnets**
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on the tab **Length_Mag** to modify its parameters
  - In Element Length Based Refinement window,
    - Maximum Length of Elements: Change to **3.75/2 mm**
    - Press **OK**



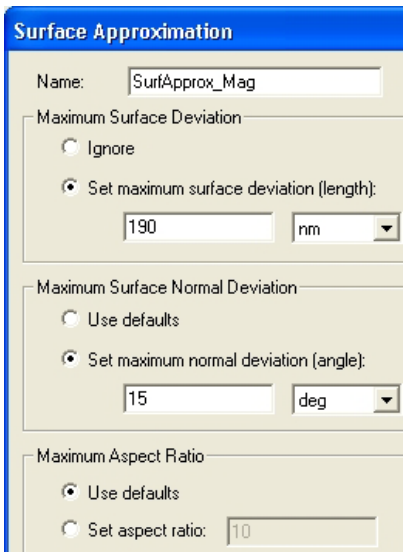


Motors - Permanent Magnet Synchronous Machine

- **Modify Mesh operation Length_Main**
  - Double click on mesh operation **Length_Main**
  - In Element Length Based Refinement window,
    - Maximum Length of Elements: Change to **10.96/4 mm**
    - Press **OK**



- **Modify Surface Approximation for Magnet**
  - Double click on mesh operation **SurfApprox_Mag**
  - In Element Length Based Refinement window,
    - Set maximum surface deviation (length): **190 nm**
    - Press **OK**

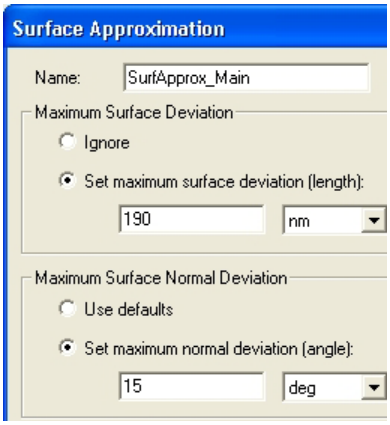


  - This yields an angular segmentation of **θ** = 0.25 deg.

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

  - r is the inside radius of the stator which is 81mm

- **Modify Surface Approximation for Main**
  - Double click on mesh operation **SurfApprox_Main**
  - In Element Length Based Refinement window,
    - Set maximum surface deviation (length): **190 nm**
    - Press **OK**



Modify Band

- **To Modify Band Objects**
  - Expand the history tree for the sheet **Band**
  - Double click on the command **CreateUserDefinedPart**
  - In Properties window,
    - Change SegAngle to **0.25 deg**
    - Press **OK**

Command

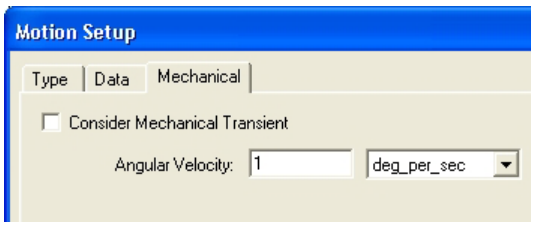
Name	Value	Unit	Evaluated...	Description
Coordinate System	Global			
DLL Name	RMxprt/B...			
DLL Location	syslib			
DLL Version	12.1			
DiaGap	161.2	mm	161.2mm	Band diameter in gap center, DiaGap<DiaYoke for
DiaYoke	110	mm	110mm	Band diameter on yoke side, DiaYoke<DiaGap for
Length	0	mm	0mm	Band length
SegAngle	0.25	deg	0.25deg	Deviation angle for band (0.1~5 degrees).
Fractions	1		1	Number of circumferential fractions, 1 for circular re
HalfAxial	0		0	0: full model; 1: half model in axial direction.
InfoCore	0		0	0: band; 1: tool; 2: master; 3: slave; 100: region.

  - **NOTE**: This small value for angular segmentation, 0.25deg, is needed only for very sensitive calculations such as Cogging Torque

Motors - Permanent Magnet Synchronous Machine

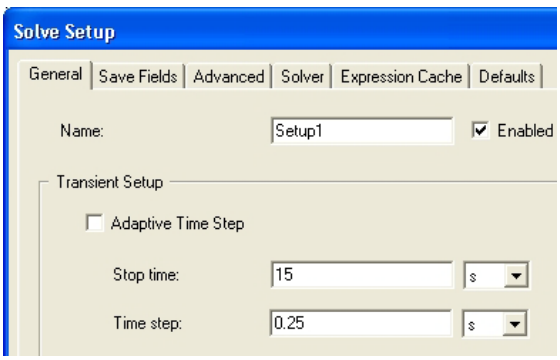
Modify Motion Setup

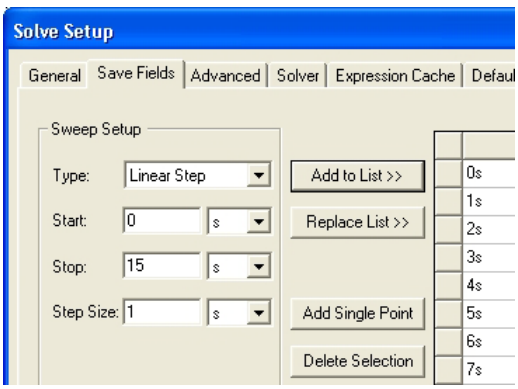
- To Modify Motion
  - Expand the Project Manager tree to view **Model**
  - Double click on the tab **MotionSetup1**
  - In Motion Setup window,
    - **Mechanical** tab
      - Angular Velocity: **1 deg_per_sec**
      - Press **OK**



Modify Analysis Setup

- To Modify Analysis Setup
  - Expand the Project Manager tree to view **Analysis**
  - Double Click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **15 s**
      - Time Step: **0.25 s**
    - **Save Fields** tab
      - Type: **Linear Step**
      - Start: **0 s**
      - Stop: **15 s**
      - Step: **1 s**
      - Select the button **Add to List**
    - Press **OK**





Create Output Parameters

- **Create Geometry for Output Parameters**
  - Select the menu item *Draw > Arc > Center Point*
    - Change the Coordinate Entry Field from Cartesian to **Cylindrical**
    - Using Coordinate Entry Field, enter the axis of the arc
      - R: **0,** Phi: **0,** Z: **0,** Press the **Enter** key
    - Using Coordinate Entry Field, enter the first radial point
      - R: **80.8,** Phi: **0,** Z: **0,** Press the **Enter** key
    - Using Coordinate Entry Field, enter the sweep arc length
      - R: **80.8,** Phi: **45,** Z: **0,** Press the **Enter** key
    - Press **Enter** again to exit drawing
  - Change the name of the resulting polyline to **AG_Arc**
- **Create Calculator Parameter Brad**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Calculator window,
    - Select **Input > Quantity > B**
    - Select **Vector > Scal? > ScalarX**
    - Select **Input > Function**
      - Select the quantity **PHI**
      - Press **OK**
    - Select **Scalar > Trig > Cos**
    - Select **General > ***
    - Select **Input > Quantity > B**
    - Select **Vector > Scal? > ScalarY**
    - Select **Input > Function**
      - Select the quantity **PHI**
      - Press **OK**
    - Select **Scalar > Trig > Sin**
    - Select **General > ***
    - Select **General > +**
    - Select the button **Add**
      - Name: **Brad** and Press **OK**

- **Create Calculator Parameter Brad_Avg**
  - In calculator window,
    - Select **Brad** from Named Expressions list and Select **Copy to stack**
    - Select **Input > Geometry**
      - Set the radio button to **Line**
      - Select the entity **AG_Arc**
      - Press **OK**
    - Select **Scalar > Integrate**
    - Select **Input > Number**
      - Value: **1**
      - Press **OK**
    - Select **Input > Geometry**
      - Set the radio button to **Line**
      - Select the entity **AG_Arc**
      - Press **OK**
    - Select **Scalar > Integrate**
    - Select **General > /**
    - Select **Add**
      - Name: **Brad_Avg** and Press **OK**
- **Create Parameter Mag_Area**
    - Select **Input > Number**
      - Value: **1**
      - Press **OK**
    - Select **Input > Geometry**
      - Set the radio button to **Surface**
      - Select the entity **Mag_0**
      - Press **OK**
    - Select **Scalar > Integrate**
    - Select **Input > Number**
      - Value: **1e6** ….(To Convert m^2 to mm^2)
      - Press **OK**
    - Select **General > ***
    - Select **Add**
      - Name: **Mag_Area** and Press **OK**

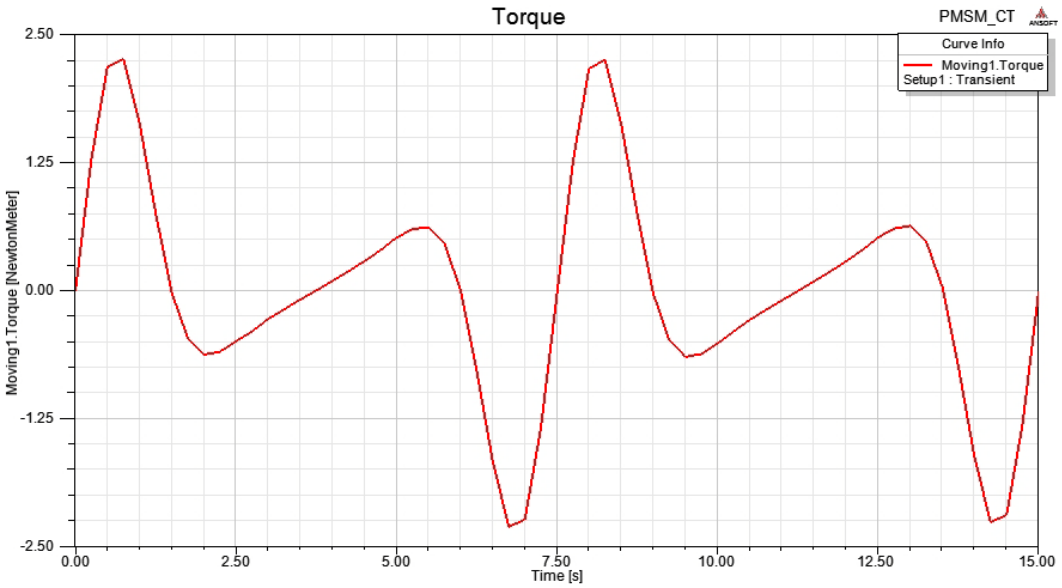
Motors - Permanent Magnet Synchronous Machine

Analyze

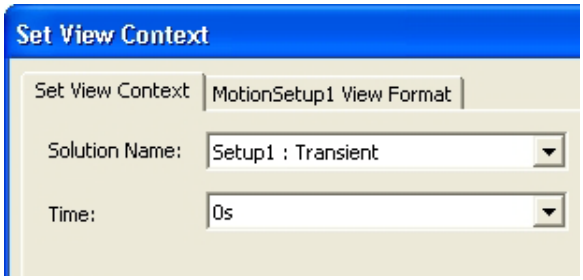
- **To Run the Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

Results

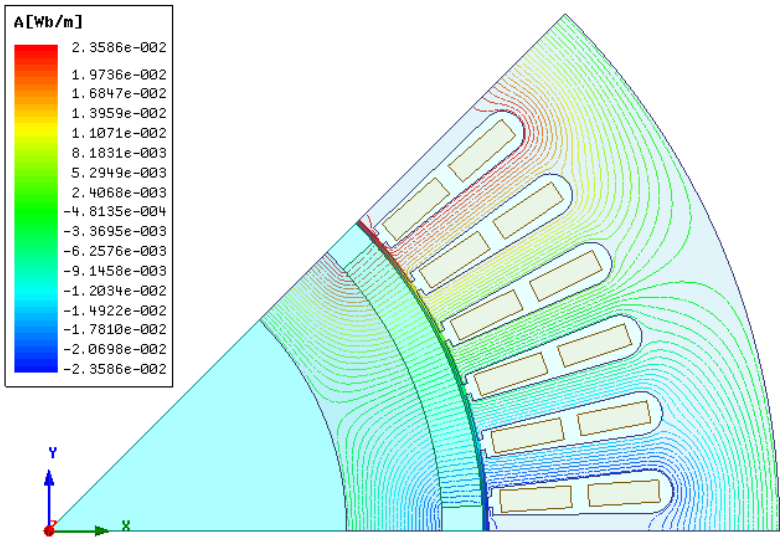
- **View Torque Vs Time**
  - Expand the Project Manager tree to view **Results**
  - Double click on XY Plot **Torque > Moving1.Torque** to view torque results



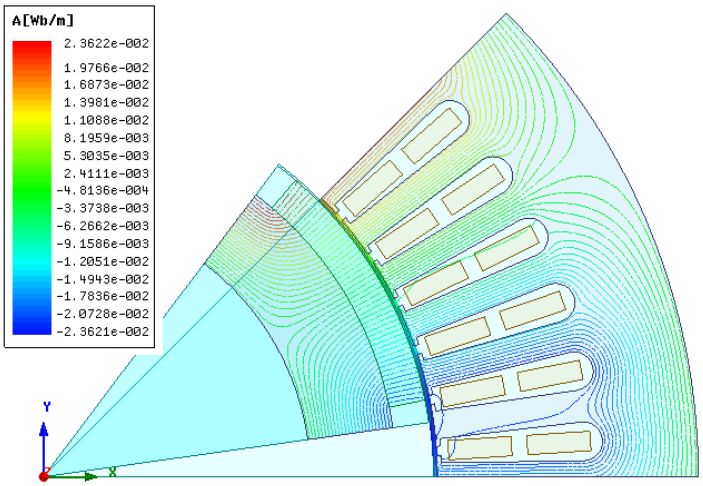
  - Right click on the plot and select **Export**
    - Save the plot which we will use later
  - Since the speed is held constant at 1.0 deg/sec, the X-Axis represents both time and position, i.e. 10 sec = 10 deg
- **Set Solution Context**
  - Select the menu item *View > Set Solution Context*
  - In Set View Context window,
    - Set the Time to **0 s**
    - Press **OK**

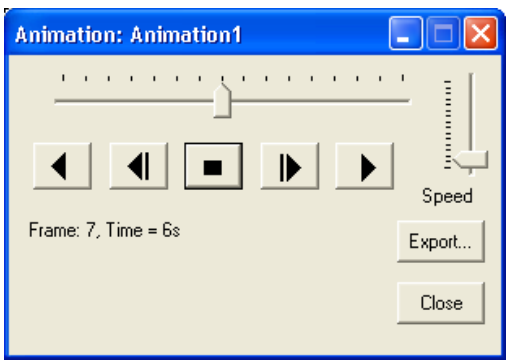


- **Plot Flux Lines**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window,
    - Press **Done**
  - Double click on the Legend to modify its attributes
  - In the window,
    - **Scale** tab
      - Num. Division: **50**



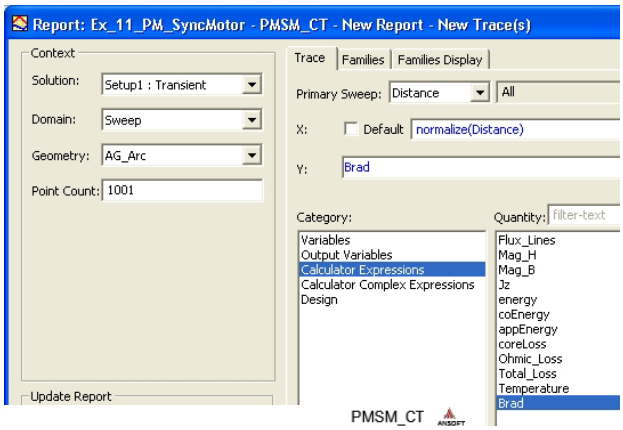
  - Select create Field Plot from Project manager tree, right click and select **Animate**
  - In Setup Animation window, press **OK**

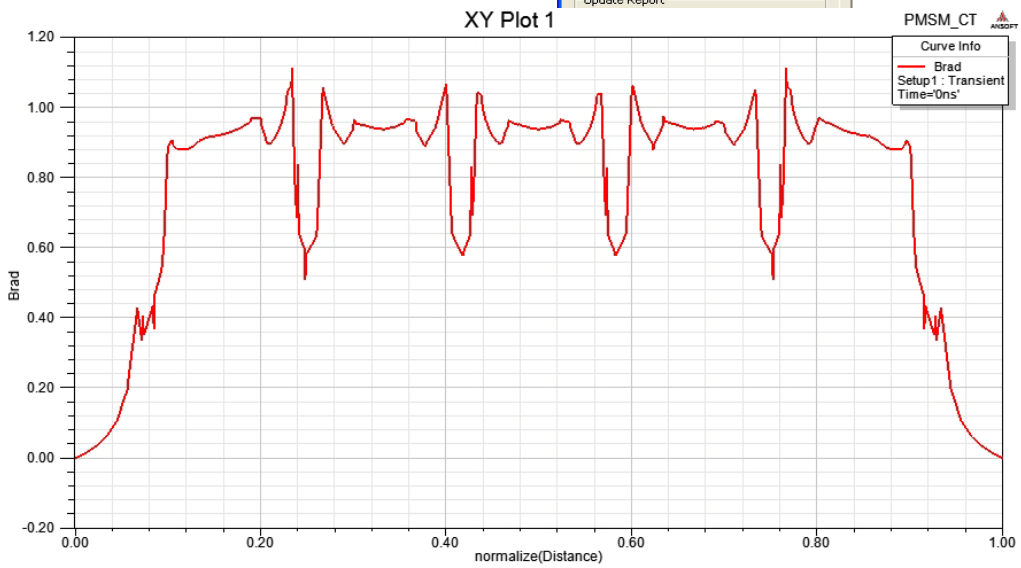




Plot Brad in Airgap

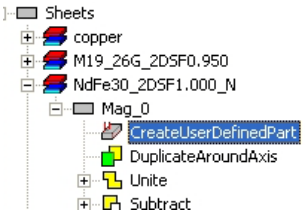
- **To Plot Brad**
  - Select the menu item *Maxwell 2D > Results > Create Field Report > Rectangular Plot*
  - In Report window,
    - **Trace** tab
      - Geometry: **AG_Arc**
      - X:
        - Default: ☐**Unchecked**
        - Set X to **normalize(Distance)**
      - Y:
        - Category: **Calculator Expressions**
        - Quantity: **Brad**
    - **Families** tab
      - Time: Set to **0 s**
    - Select **New Report**





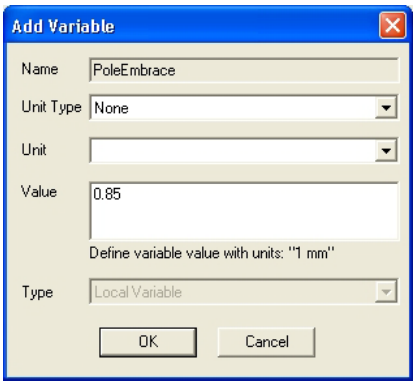
Add Variables

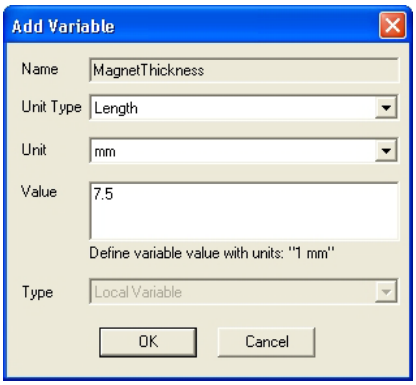
- **To Add variables to Magnet**
  - Expand the history tree for the sheet **Mag_0**
  - Double click on the command **CreateUserDefinedPart** to modify its Properties
  - In Properties window,
    - Embrace: Set to **PoleEmbrace ,** Press **Tab** key
    - In Add Variable window,
      - Value: **0.85**
      - Press **OK**
    - ThickMag: Set to **MagnetThickness,** Press **Tab** Key
    - In Add Variable window,
      - Value: **7.5 mm**
      - Press **OK**
    - Offset: **PoleOffset ,** Press **Tab** Key
    - In Add Variable window,
      - Value: **0 mm**
      - Press **OK**
    - Press **OK** to close Properties window

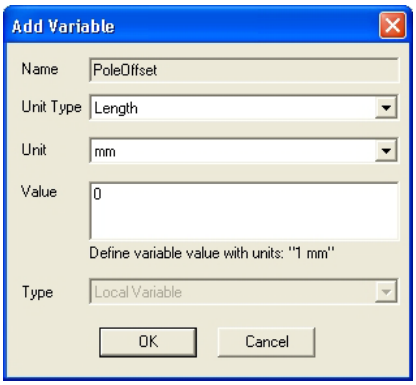


Command

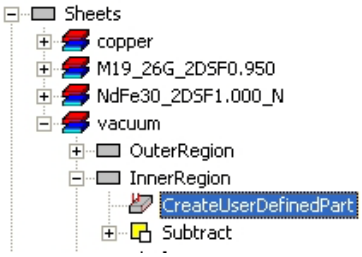
	Name	Value	Unit	Evaluated...	Descriptio
	Poles	8		8	Number of poles
	PoleType	1		1	Pole type: 1 to 5
	Embrace	PoleEmbrace		0.85	Pole embrace (not for type 4)
	ThickMag	MagnetThickness		7.5mm	Max thickness of magnets
	WidthMag	0	mm	0mm	Magnet width (for types 4 & 5)
	Offset	PoleOffset		0mm	Pole arc offset (for types 1, 2 & 3)
	Bridge	0	mm	0mm	Bridge thickness (for type 5 only)







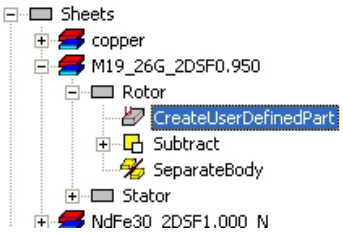
- **Add Variables to InnerRegion**
  - Expand the history tree for the sheet **InnerRegion**
  - Double click on the command **CreateUserDefinedPart** to modify its Properties
  - In Properties window,
    - Embrace: **PoleEmbrace**
    - ThickMag: **MagnetThickness**
    - Offset: **PoleOffset**
    - Press **OK** to close Properties window



Command

	Name	Value	Unit	Evaluated...	Description
	Poles	8		8	Number of poles
	PoleType	1		1	Pole type: 1 to 5
	Embrace	PoleEmbrace		0.85	Pole embrace (not for type 4)
	ThickMag	MagnetThickness		7.5mm	Max thickness of magnets
	WidthMag	0	mm	0mm	Magnet width (for types 4 & 5)
	Offset	PoleOffset		0mm	Pole arc offset (for types 1, 2 & 3)
	Bridge	0	mm	0mm	Bridge thickness (for type 5 only)

- **Add Variables to Rotor**
  - Expand the history tree for the sheet **Rotor**
  - Double click on the command **CreateUserDefinedPart** to modify its Properties
  - In Properties window,
    - Embrace: **PoleEmbrace**
    - ThickMag: **MagnetThickness**
    - Offset: **PoleOffset**
    - Press **OK** to close Properties window



Command

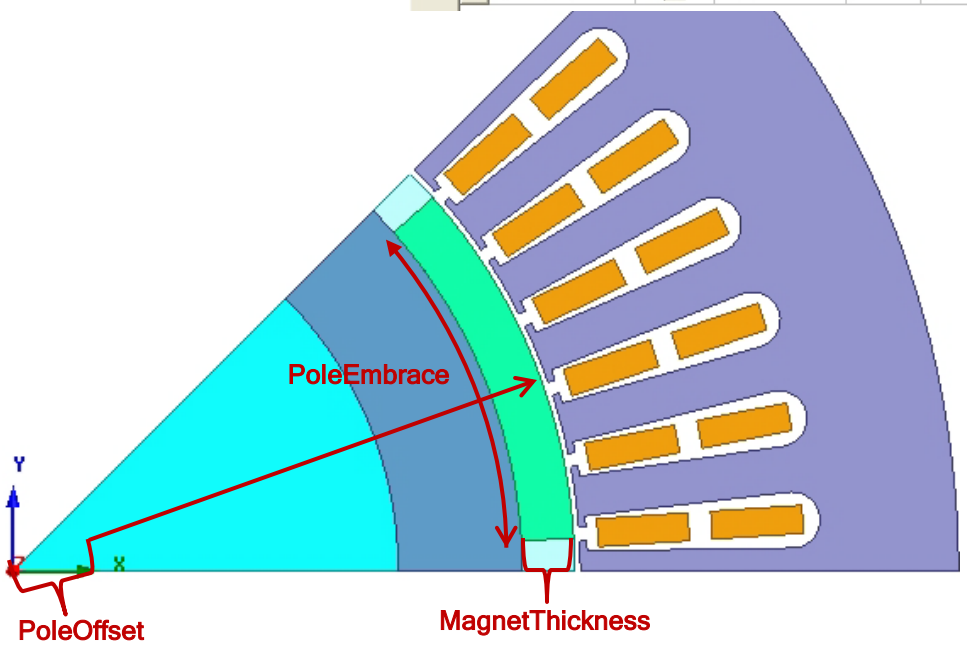
	Name	Value	Unit	Evaluated...	Description
	Poles	8		8	Number of poles
	PoleType	1		1	Pole type: 1 to 5
	Embrace	PoleEmbrace		0.85	Pole embrace (not for type 4)
	ThickMag	MagnetThickness		7.5mm	Max thickness of magnets
	WidthMag	0	mm	0mm	Magnet width (for types 4 & 5)
	Offset	PoleOffset		0mm	Pole arc offset (for types 1, 2 & 3)
	Bridge	0	mm	0mm	Bridge thickness (for type 5 only)

- **Modify Variable Values**
  - Select the menu item *Maxwell 2D > Design Properties*
  - In Properties window,
    - Set the radio button to **Optimization**
    - For the Parameter **PoleEmbrace**
      - Include: ☑ **Checked**
      - Min: **0.6**
      - Max: **0.9**
    - For the Parameter **MagnetThickness**
      - Include: ☑ **Checked**
      - Min: **6.5mm**
      - Max: **9.5mm**
    - For the Parameter **PoleOffset**
      - Include: ☑ **Checked**
      - Min: **0mm**
      - Max: **30mm**

Local Variables

Value | Optimization | Tuning | Sensitivity

	Name	Include	Nominal Value	Min	Unit	Max	Unit
	fractions	☐	8	4		12	
	PoleEmbrace	☑	0.85	0.6		0.9	
	MagnetThickness	☑	7.5mm	6.5	mm	9.5	mm
	PoleOffset	☑	0mm	0	mm	30	mm

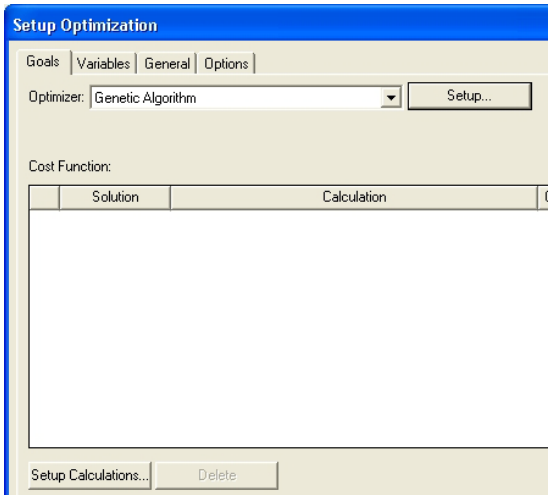


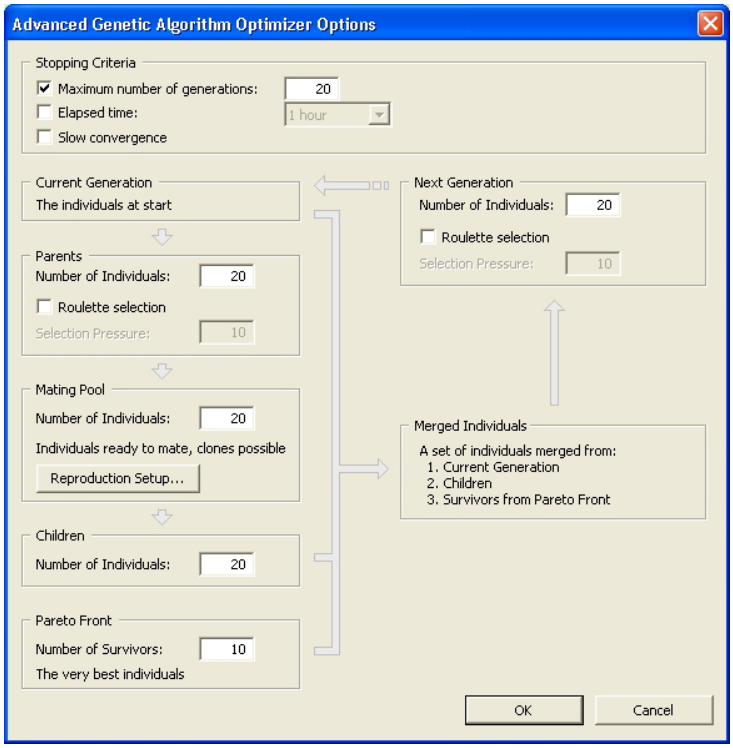
About Optimization Requirements

- **Cogging Torque**
  - Peak value of Cogging Torque for Nominal Setup = **2.2 N-m**
    - The above value was calculated with
      - PoleEmbrace = 0.85, MagnetThickness = 7.5mm, PoleOffset = 0mm
  - Maximum Cogging Torque = **5.5N-m**
    - The above value is considered for maximum values of parameters
      - PoleEmbrace = 0.9, MagnetThickness = 9.5mm, PoleOffset = 0mm
    - **Note:** The maximum value of cogging torque may lay outside these parameter values, i.e somewhere else in the solution domain. These values are used just to define a range for the objective.
  - Optimization Goal is to achieve Cogging Torque of **0.2N-m**
    - Subjectively chosen to reduce Cogging Torque by more than 10X
  - Normalize Solution Range: 1 to 10
    - **G1 = 1 + (max(abs(Torque)) - 0.2) * (9 / 5.3)**
    - Objective: **G1 = 1.0**
    - **Note:** G1 is set such that G1 =10 at 5.5N-m and 1 at 0.2N-m
- **B_{avg}**
  - Nominal value of **B_{avg} = 0.76 Tesla**
  - Range: **0.50 < B_{avg} < 0.81 Tesla**
  - Normalize Range: 1 to 10
    - **G2 = 1 + (Brad_Avg - 0.5) * 9 / 0.31**
    - Objective: **G2 = 8.55** (To achieve **B_{avg}** value of 0.76 Tesla)
- **Magnet Area**
  - Range: 220 < Mag_area < 510 mm^2
  - Normalize Range: 1 to 10
    - **G3 = 1 + (Mag_area - 220) * 9 / 290**
    - Objective: **G3 = 1.0**
- **Note:**
  - The Range for B_{avg} and Magnet Area was calculated by simulation the minimum and maximum values:
    - MagnetThickness : 6.5 mm and 9.5 mm, PoleEmbrace: 0.6 and 0.9, PoleOffset : 0 mm and 30 mm

Setup Optimization

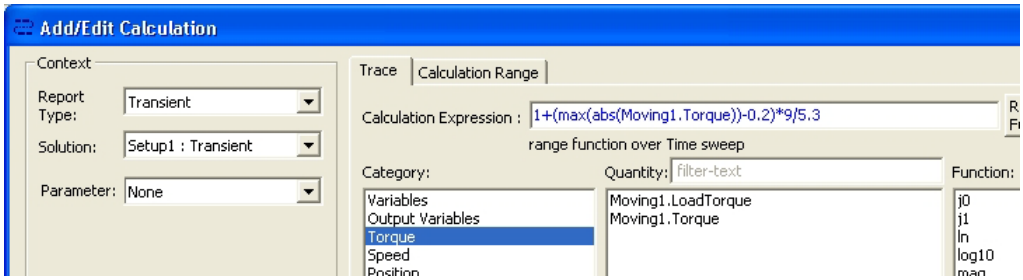
- **To Setup Optimization Analysis**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Add Optimization*
  - In Setup Optimization window,
    - Optimizer: **Genetic Algorithm**
    - Select the button **Setup** adjacent to Optimizer option
      - Maximum Number of Generations: **20**
      - Parents
        - Number of Individuals: **20**
      - Mating Pool
        - Number of Individuals: **20**
      - Children
        - Number of Individuals: **20**
      - Next Generation
        - Number of Individuals: **20**
        - Roulette Selection: ☐ **Unchecked**
      - Press **OK**



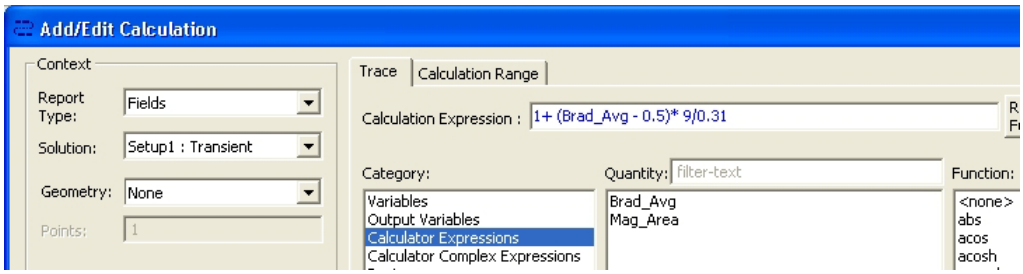


Motors - Permanent Magnet Synchronous Machine

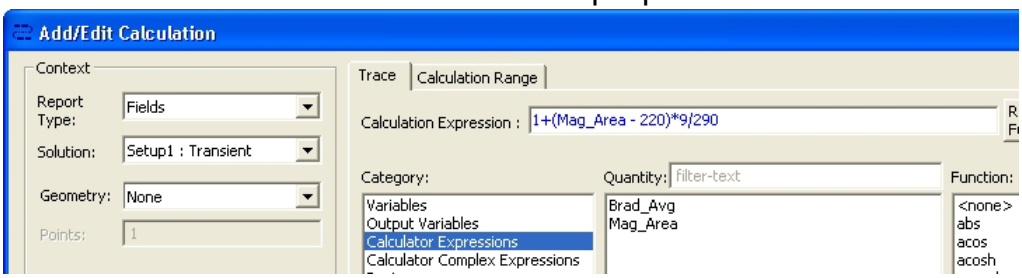
- In Setup Optimization window, Select **Setup Calculations**
- In Add/Edit Calculations window,
  - Category: **Torque**
  - Quantity: **Moving1.Torque**
  - Function: **abs**
  - Change Calculation Expression to: **1+(max(abs(Moving1.Torque))-0.2)*9/5.3**
  - Select **Add Calculation**



  - Change Report Type to **Fields**
  - Category: **Calculator Expressions**
  - Quantity: **Brad_Avg**
  - Change Calculator Expression to: **1+ (Brad_Avg - 0.5)*9/0.31**
  - Select **Add Calculation**

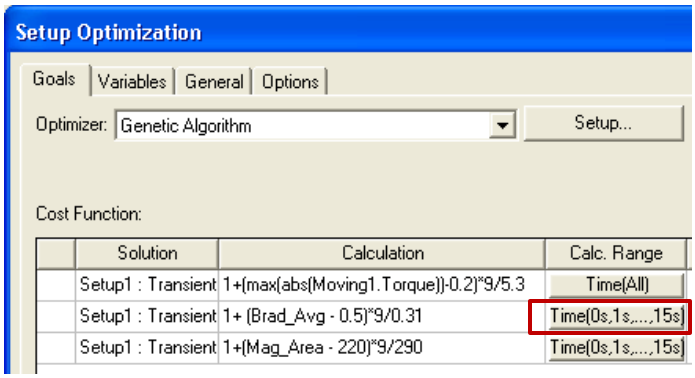


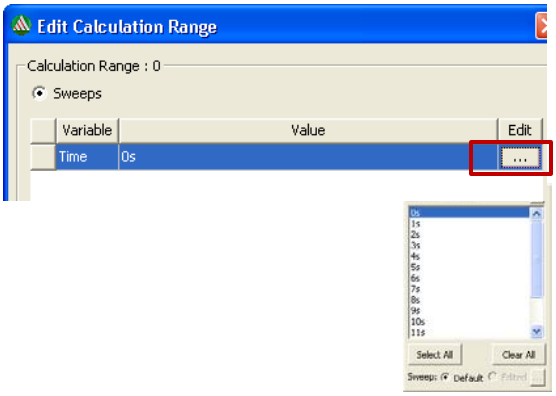
  - Change Quantity to **Mag_Area**
  - Change Calculator Expression to: **1+(Mag_Area - 220)*9/290**
  - Select **Add Calculation**
  - Press **Close** to return to Setup Optimization window



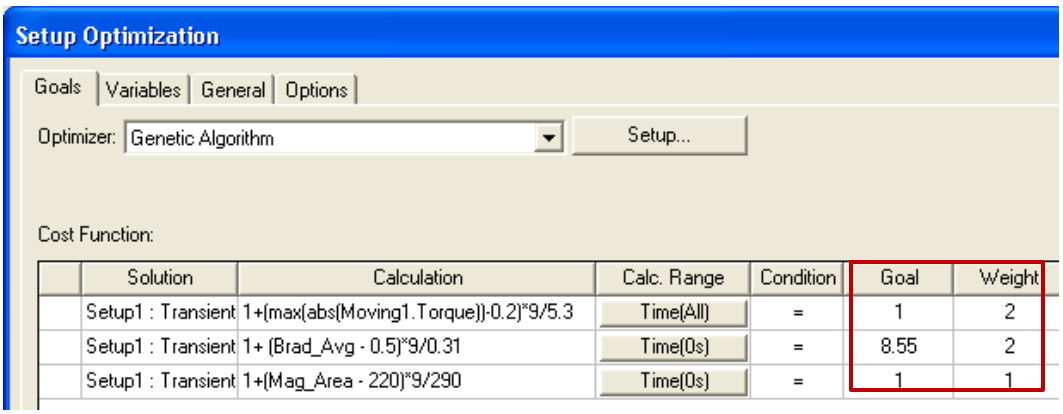
Motors - Permanent Magnet Synchronous Machine

- In Setup Optimization window,
  - Change Calculation range for the expressions with **Brad_Avg** and **Mag_Area** to **0s**



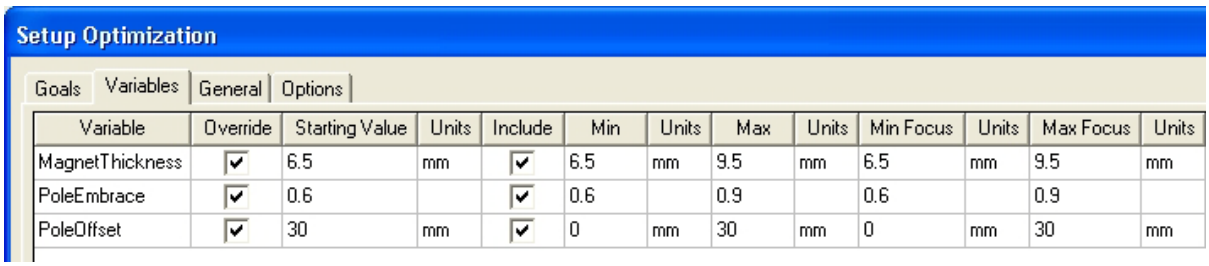


  - Set the Goal and Weight values as shown in below image
    - Weight values for Cogging Torque and Air Gap Flux Density are set to same value which is twice the Magnet Area weight



- **Note:**
  - Cost1 = $(G1 - 1)^2 * W1$
    - where G1 = 1+(max(abs(Torque))- 0.2)* 9 / 5.3
  - Cost2 = $(G2 - 8.55)^2 * W2$
    - where G2 = 1 + (Brad_Avg - 0.5) * 9 / 0.31
  - Cost3 = $(G3 - 1)^2 * W3$
    - where G3 = 1 + (Mag_area - 220) * 9 / 290
  - Cost = Cost1 + Cost2 + Cost3

- In Setup Optimization window, select **Variables** tab
  - Change Starting values for all parameters as shown in below image



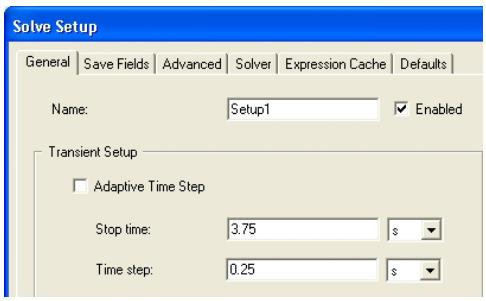
Variable	Override	Starting Value	Units	Include	Min	Units	Max	Units	Min Focus	Units	Max Focus	Units
MagnetThickness	☑	6.5	mm	☑	6.5	mm	9.5	mm	6.5	mm	9.5	mm
PoleEmbrace	☑	0.6		☑	0.6		0.9		0.6		0.9	
PoleOffset	☑	30	mm	☑	0	mm	30	mm	0	mm	30	mm

  - Press **OK** to close Setup optimization window

Modify Analysis Settings

To Modify Analysis Setup

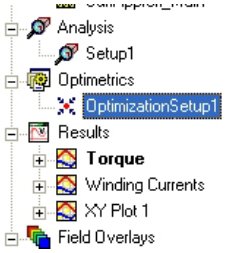
- Expand the Project Manager tree to view **Analysis**
- Double click on the tab **Setup1** to modify its parameters
- In Solve Setup window,
  - Change Stop Time to **3.75s**
  - Press **OK**



Run Optimization

To Run Optimization Analysis

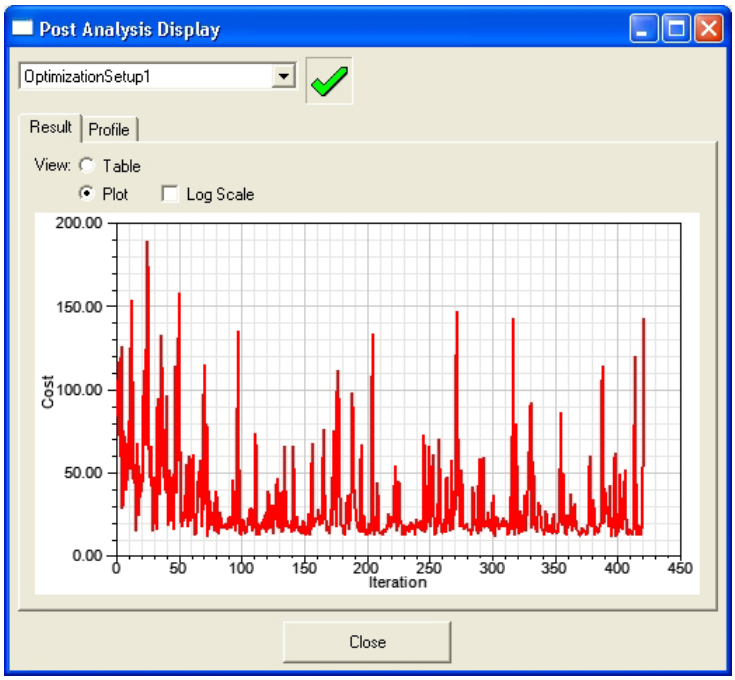
- Expand the Project Manager tree to view **Optimetrics**
- Right click on the tab **OptimizationSetup1** and select **Analyze**



Optimization Results

To View Optimization results

- Right click on the tab **OptimizationSetup1** and select **View Analysis Results**



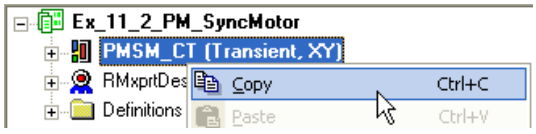
- Select the **Table** option to view results in tabular form

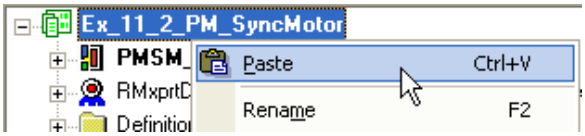
Iteration	MagnetThickness	PoleEmbrace	PoleOffset	Cost
301	6.53724150712635mm	0.879893237846424	19.6086345482546mm	12.036
73	6.50308567020767mm	0.859548230731297	16.7280590198902mm	12.038
394	6.51464592815262mm	0.869951744346114	19.6282504887929mm	12.107
374	6.55986942174642mm	0.882703976370301	18.0638775606513mm	12.366
411	6.54931105413028mm	0.883291417423796	21.1034326758612mm	12.383
199	6.5577713046569 6mm	0.861684119692056	17.4966143088534mm	12.392
398	6.56742410013486mm	0.880352050136734	20.4560178088968mm	12.414
381	6.56856663259477mm	0.858190099239542	14.8758346117964mm	12.465
99	6.55747934709094mm	0.857249924708023	10.8452245418037mm	12.525
369	6.54839327462128mm	0.85580085596214	10.8666161113414mm	12.555
417	6.56482813591384mm	0.893094697477481	21.7142427233121mm	12.595

Motors - Permanent Magnet Synchronous Machine

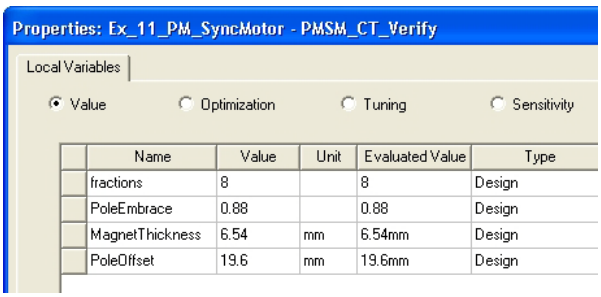
Verify Optimized Results

- Since the field solution was not saved for each variation in the optimization solution, create a second Maxwell 2D design and solve the problem with the optimized design variable values.
- **Copy Design**
  - Select the Maxwell 2D Design **"PMSM_CT"** from Project Manager tree, right click and select **Copy**
  - Right click on Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"PMSM_CT_Verify"**



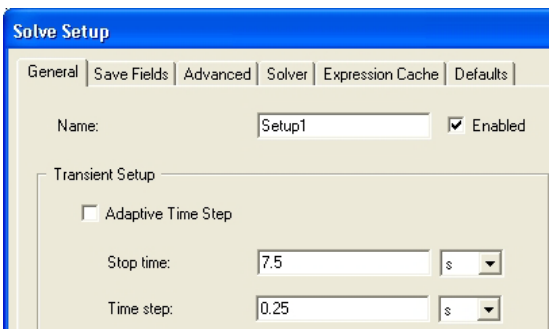


- **Modify Design Variables**
  - Select the menu item *Maxwell 2D > Design Properties*
  - In Properties window,
    - Change the value for **PoleEmbrace** to **0.88**
    - Change the value of **MagnetThickness** to **6.54mm**
    - Change the value of **PoleOffset** to **19.6mm**
    - Press **OK**



Modify Analysis Settings

- **To Modify Analysis Setup**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - Change Stop Time to **7.5s**
    - Press **OK**



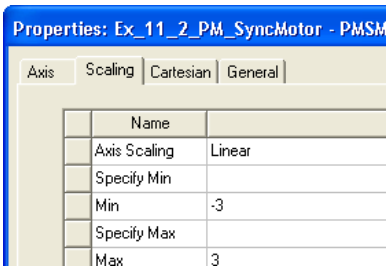
Motors - Permanent Magnet Synchronous Machine

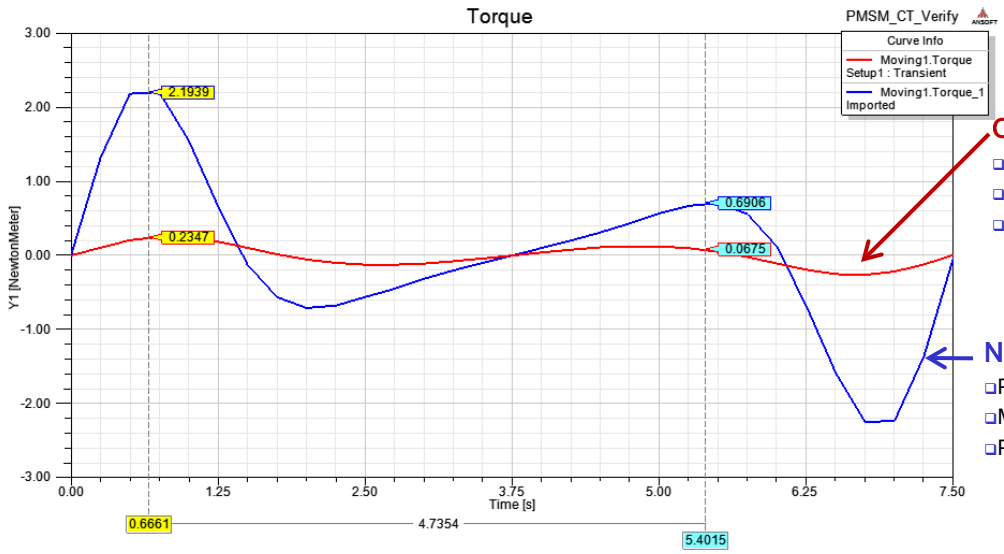
Analyze

- To Run the Solution
  - Expand the Project Manager tree to view **Analysis**
  - Right click on the tab **Setup1** and select **Analyze**

Results

- Cogging Torque
  - Double click on the Torque Plot to view Cogging Torque Results
  - Right click on created plot and select **Import**
    - Select the **Torque** plot that we exported previously and open it
  - Double click on the Y Axis of the plot to modify its attributes
    - Change Min to **-3** and Max to **3**
    - Press **OK**
  - Right click on the plot and select **Marker > Add X Marker**
  - Move the add X marker along X axis and check the Y values a corresponding locations

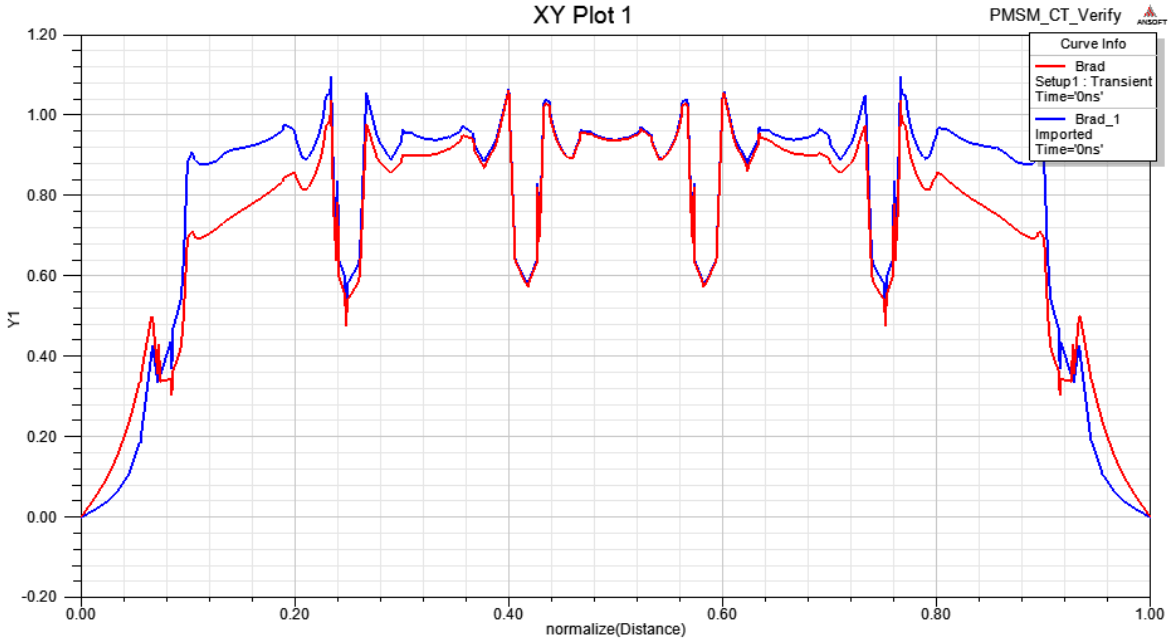




Optimized Design
- PoleEmbrace= 0.88
- MagnetThickness = 6.54 mm
- PoleOffset = 19.6 mm

Nominal Design
- PoleEmbrace= 0.85
- MagnetThickness = 7.5 mm
- PoleOffset = 0 mm

- **Air Gap Flux Density**
  - Double click on the XY plot corresponding to Brad that already exists
  - From the Project manager tree, select the XY plot for Flux Density from Nominal design **PMSM_CT**, right click and select **Copy data**
  - Goto the new design **PMSM_CT_Verify,** Select the XY Plot corresponding to Flux Density, right click and select **Paste**



- **Calculate Average Airgap Flux Density and Magnet Area**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Brad_Avg** from Named Expressions list
    - Select the Button **Copy to stack**
    - Select **Output > Eval**
      - Result value is 0.72 Tesla which close to targeted value (0.76 Tesla)
    - Similarly add the expression **Mag_Area**
    - Select **Output > Eval**
      - Result value is 319 mm^2 which close to targeted value (383 mm^2)

Scl : 319.557970752817
Scl : Mag_Area
Scl : 0.721311854821372
Scl : Brad_Avg

3. Open Circuit Back EMF

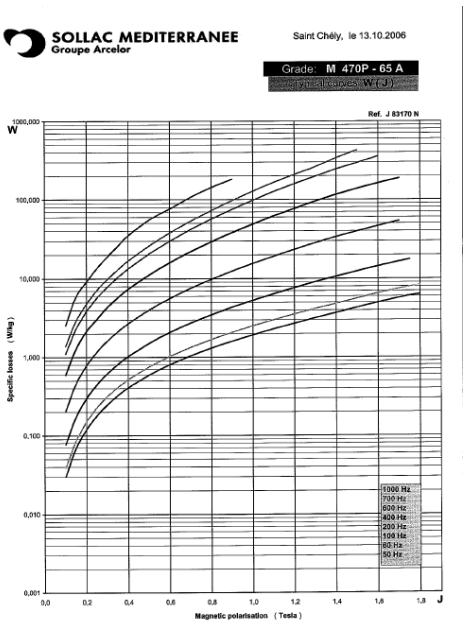
Copy Design

- **To Copy Design**
  - Select the Maxwell 2D Design **"PMSM_CT_Verify"** from Project Manager tree, right click and select **Copy**
  - Right click on Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"PMSM_OC_EMF"**

Modify Motion

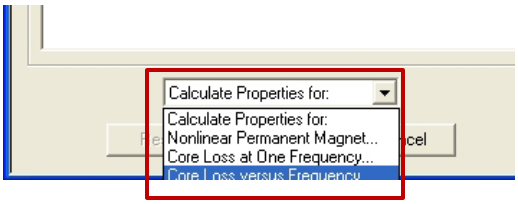
- **To Modify Motion Parameters**
  - Expand the Project Manager tree to view **Model**
  - Double click on the tab **MotionSetup1** to modify its parameters
  - In Motion Setup window,
    - **Mechanical** tab
      - Change Angular Velocity to **3600 rpm**
      - Press **OK**

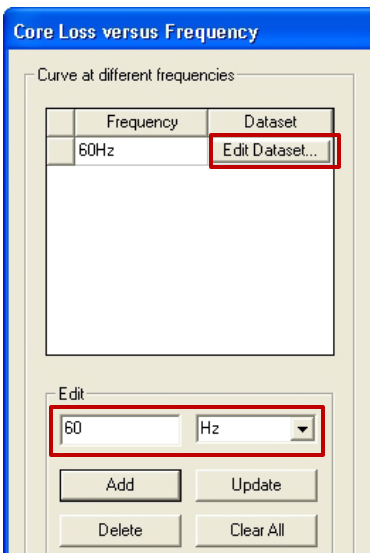
Calculate Core Loss Coefficients from Multiple Core Loss Curves

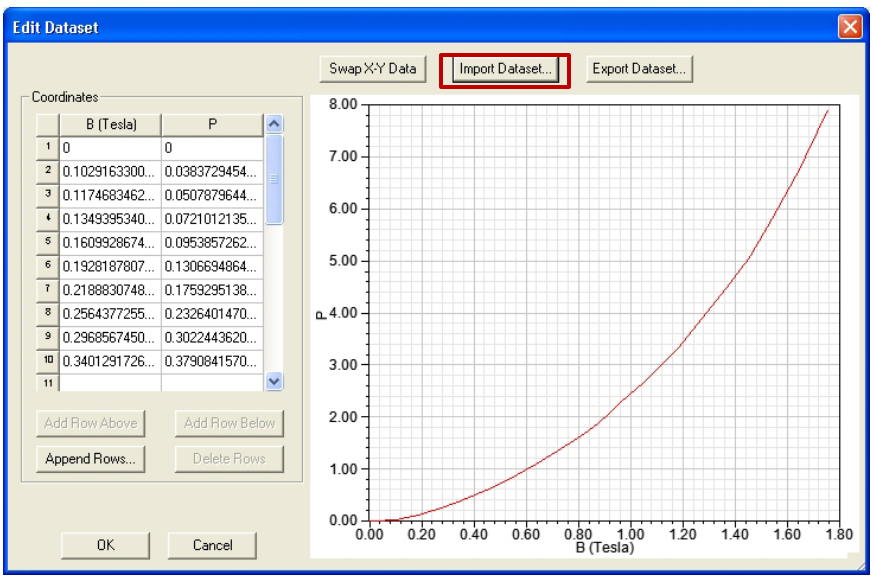


Motors - Permanent Magnet Synchronous Machine

- **To Set Core Loss Parameters**
  - Select the material **M19_26G_2DSF0.950** from History tree, right click and select **Properties**
  - In Select Definition window, select the button **View/Edit Materials**
  - In View/Edit Materials window,
    - Select the option at the bottom of the window: Calculate Properties for **Core Loss versus Frequency**
    - In Core Loss Versus Frequency window,
      - In Edit field, set value to **60 Hz**
      - Select **Add**
      - The Frequency value gets added to the list
      - Select the button **Edit Dataset** adjacent to added frequency
      - In Edit Dataset window,
        - Select **Import Dataset**
        - Locate the file **M470-65A-60Hz.tab** and **Open** it
        - Press **OK**



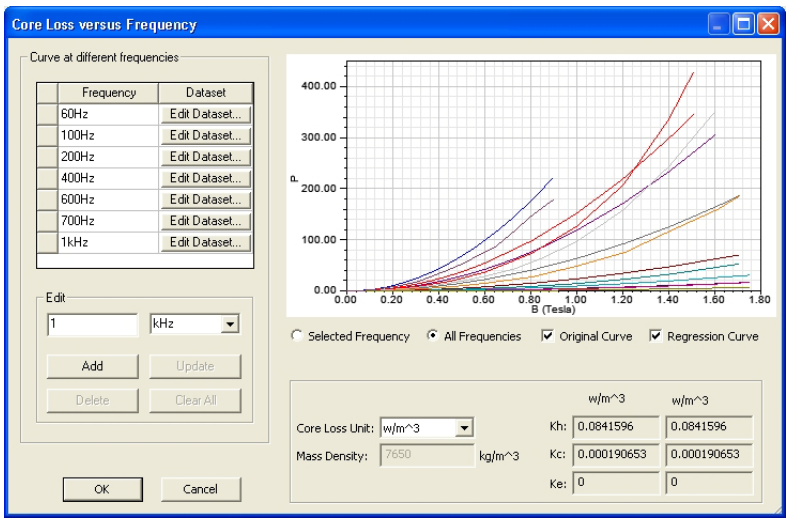


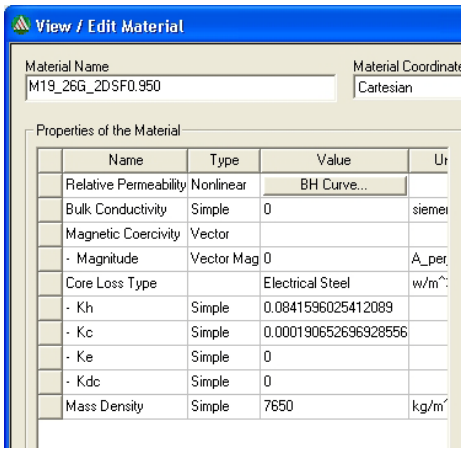


Motors - Permanent Magnet Synchronous Machine

- In Edit field, set frequency to **100 Hz**
- Select **Add**
- Select the button **Edit Dataset** adjacent to added frequency
- In Edit Dataset window,
  - Select **Import Dataset**
  - Locate the file **M470-65A-100Hz.tab** and **Open** it
  - Press **OK**
- In Similar way continue adding datasets for more frequencies using files mentioned below
  - **200 Hz**
    - M470-65A-200Hz.tab
  - **400 Hz**
    - M470-65A-400Hz.tab
  - **600 Hz**
    - M470-65A-600Hz.tab
  - **700 Hz**
    - M470-65A-700Hz.tab
  - **1 kHz**
    - M470-65A-1kHz.tab
- Select **OK** to close Core Loss Versus Frequency window

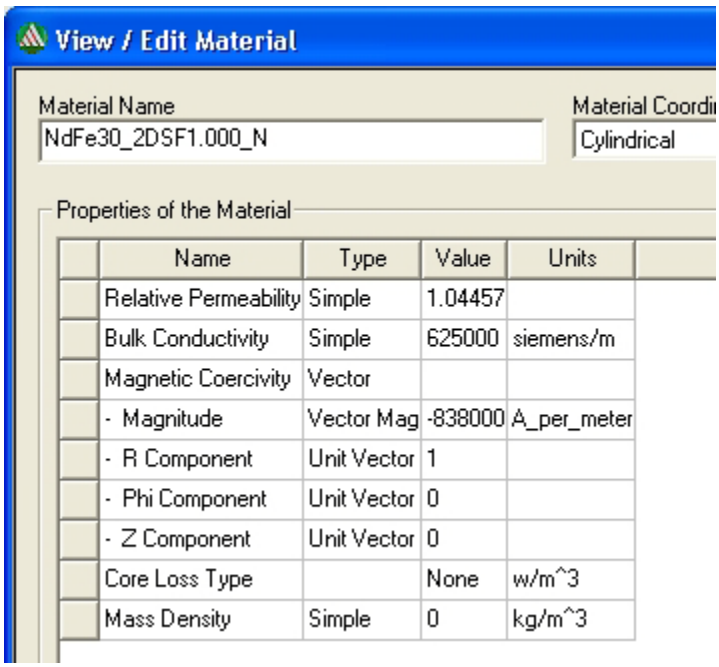
- Select **OK** to close View/Edit materials window





Modify Magnet Material

- **To Modify Magnet Properties**
  - Select the material **NdFe30_2DSF1.000_N** from history tree, right click and select **Properties**
  - In Select Definition window, select the button **View/Edit Materials**
  - In View/Edit Materials window,
    - Change Bulk Conductivity to **625000 s/m**
    - Press **OK**



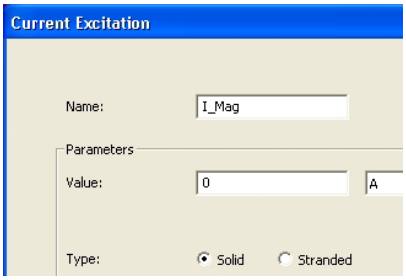
Name	Type	Value	Units
Relative Permeability	Simple	1.04457	
Bulk Conductivity	Simple	625000	siemens/m
Magnetic Coercivity	Vector		
- Magnitude	Vector Mag	-838000	A_per_meter
- R Component	Unit Vector	1	
- Phi Component	Unit Vector	0	
- Z Component	Unit Vector	0	
Core Loss Type		None	w/m^3
Mass Density	Simple	0	kg/m^3

- **Note:** Material properties are global quantities, the affect all designs. Thus when modifying materials that are common to various designs, the solutions to the designs become invalid.

Assign Excitation

Set Excitation for Magnet

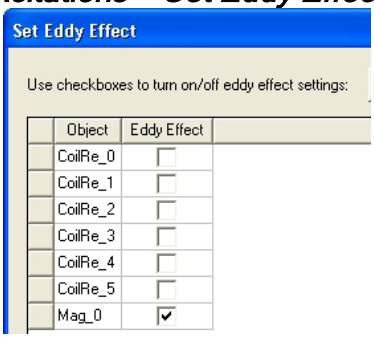
- Select the sheet **Mag_0** from history tree
- Select the menu item *Maxwell 2D > Excitations > Assign > Current*
- In Current Excitations window,
  - Name: **I_Mag**
  - Value: **0 A**
  - Press **OK**



- **Note:** By assigning zero current to the magnet it is assured that total current into and out of this magnet is zero. If there were more than one magnet, each one should have a separate excitation of zero amps.

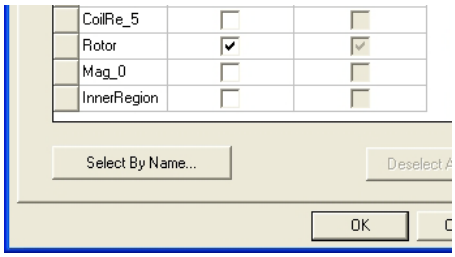
Set Eddy Effects

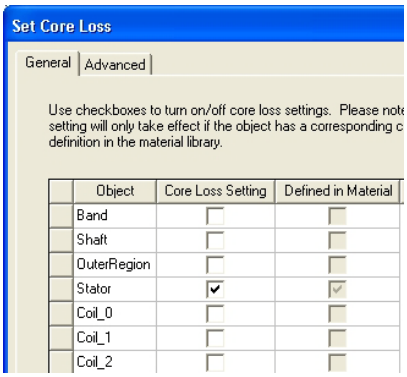
- Select the menu item *Maxwell 2D > Excitations > Set Eddy Effects*
- In Set Eddy Effect window
  - For the object **Mag_0**
    - Include: ☑ **Checked**
    - Press **OK**



Set Core Loss Calculation

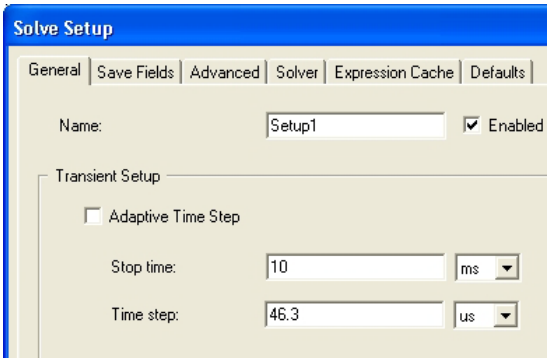
- Select the menu item *Maxwell 2D > Excitations > Set Core Loss*
- In Set Core Loss window,
  - For the objects **Stator** and **Rotor**
    - Core Loss Settings: ☑ **Checked**
    - Press **OK**

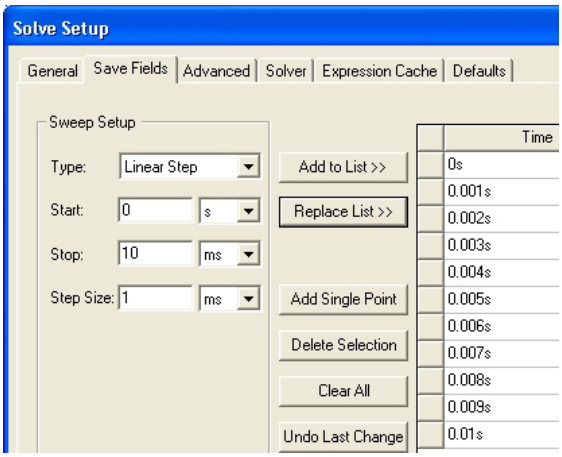




Modify Analysis Settings

- **To Modify Analysis Settings**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **10 ms**
      - Time Step: **46.3 us**
    - **Save Fields** tab
      - Type: **Linear Step**
      - Start: **0s**
      - Stop: **10ms**
      - Step: **1ms**
      - Select **Replace List>>**
    - Press **OK**





- **Note:** The time step is determined by:

$$\frac{3600 rev}{\min} * \frac{360 \deg}{rev} * \frac{1 \min}{60 \sec} = \frac{21600 \deg}{\sec} = \frac{1 \deg}{46.3u \sec}$$

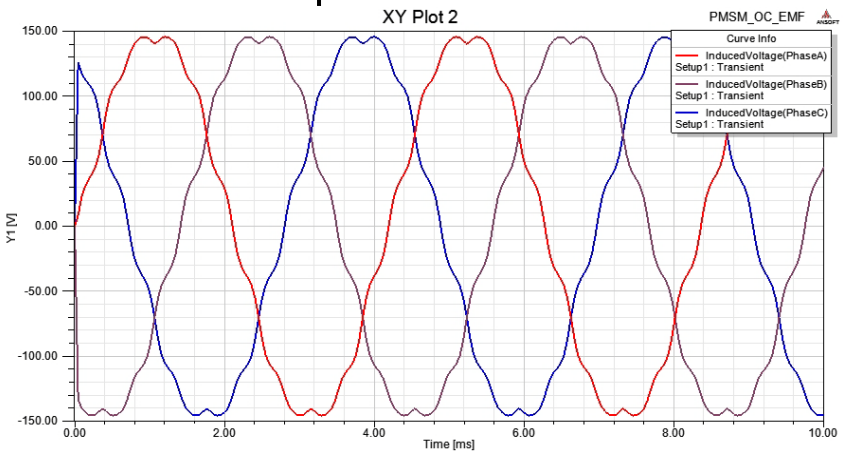
  - The frequency is 240 Hz, which gives a period of 4.2 msec. Thus 10 msec is ~2.5 cycles

Analyze

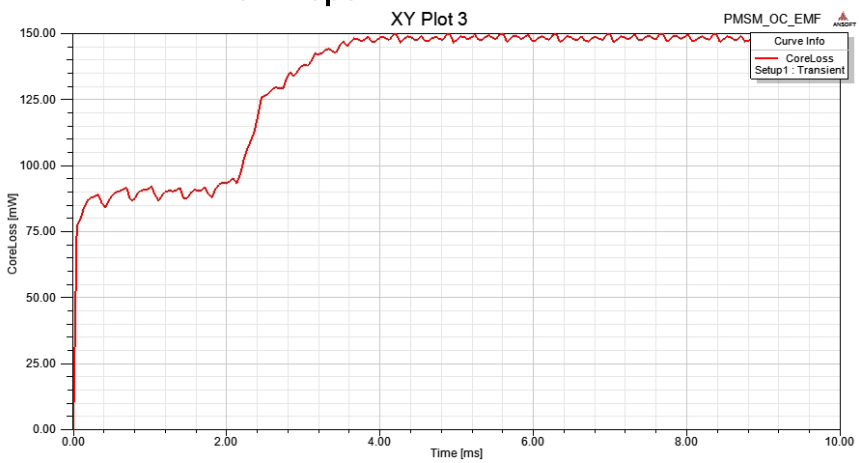
- **To Run the calculations**
  - Expand the Project Manager tree to view **Analysis**
  - Right click on the tab **Setup1** and select **Analyze**

Results

- **Plot Induced Voltages vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In Report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **InducedVoltage(PhaseA)**, **InducedVoltage(PhaseB)** and **InducedVoltage(PhaseC)**
    - Select **New Report**



- **Plot Core Loss Vs Time**
  - In Report window,
    - Change Category to **Loss**
    - Change Quantity to **Core Loss**
    - Select **New Report**



4. Rated Condition

Copy Design

- **To Copy Design**
  - Select the Maxwell 2D Design **"PMSM_OC_EMF"** from Project Manager tree, right click and select **Copy**
  - Right click on Project name in Project Manager tree and select **Paste**
  - Rename the newly created design to **"PMSM_Rated"**

Modify Geometry

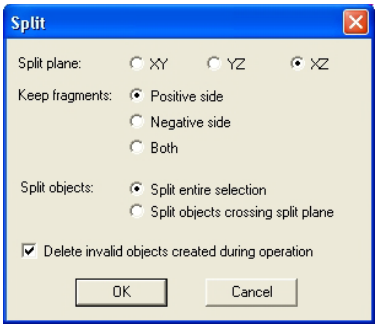
- **Delete Objects**
  - Press **Ctrl** and select the sheets **InnerRegion, Mag_0** and **Rotor** from history tree
  - Select the menu item *Edit > Delete*
- **Create New Rotor**
  - Select the menu item *Draw > User Defined Primitive > SysLib > RMxprt > IPMCore*
  - In User Defined Primitive Operation window,
    - Change Parameters as shown in below image

Parameters | Info

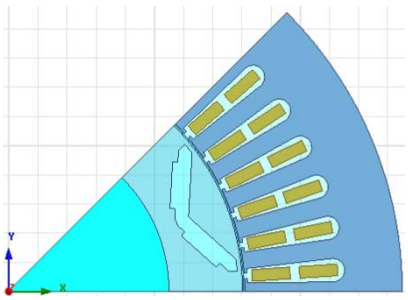
	Name	Value	Unit	Evaluated Value	Description
	DiaGap	160.4	mm	160.4mm	Core diameter on gap si...
	DiaYoke	110	mm	110mm	Core diameter on yoke ...
	Length	0	mm	0mm	Core length
	Poles	8		8	Number of poles
	PoleType	3		3	Pole type: 1 to 6.
	D1	157.44	mm	157.44mm	Limited diameter of PM ...
	O1	0	mm	0mm	Bottom 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	6.5	mm	6.5mm	Magnet thickness
	WidthMag	32	mm	32mm	Total width of all magn...
	LenRegion	0	mm	0mm	Region length
	InfoCore	0		0	0: core; 1: magnets; 2: ...

Show Hidden

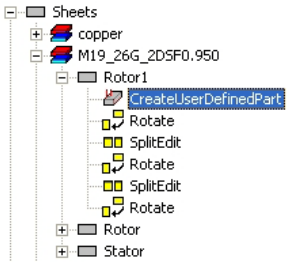
- **Change Attributes**
  - Select the newly created sheet and goto the Properties window
    - Change the name of the sheet to **Rotor**
    - Change the Material of the sheet to **M19_26G_2DSF0.950**
- **Arrange Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item *Edit > Arrange > Rotate*
    - Axis: **Z**
    - Angle: **22.5 deg**
    - Press **OK**
- **Split Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item *Modeler > Boolean > Split*
    - Split plane: **XZ**
    - Keep fragments: **Positive side**
    - Press **OK**



- **Rotate the sheet Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item *Edit > Arrange > Rotate*
    - Axis: **Z**
    - Angle: **-45 deg**
    - Press **OK**
- **Split Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item *Modeler > Boolean > Split*
    - Split plane: **XZ**
    - Keep fragments: **Negative side**
    - Press **OK**
- **Rotate the sheet Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item *Edit > Arrange > Rotate*
    - Axis: **Z**
    - Angle: **45 deg**
    - Press **OK**

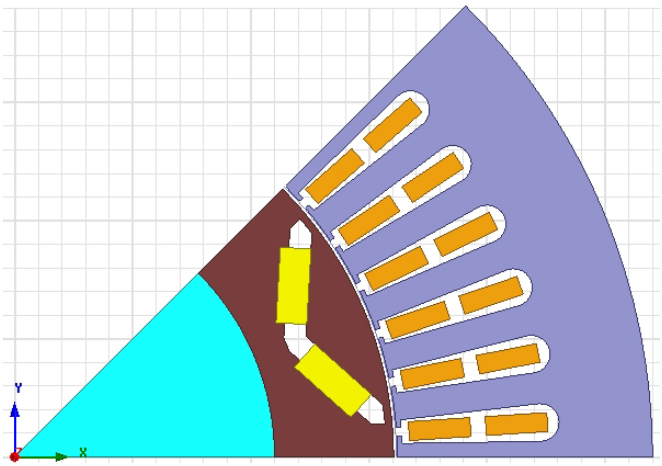


- **Create Magnets**
  - Select the sheet **Rotor** from History tree
  - Select the menu item ***Edit > Copy*** or press **Ctrl+C** from keyboard
  - Select the menu item ***Edit > Paste*** or press **Ctrl+V** from keyboard
  - New Sheet named **Rotor1** is created
  - Expand the history tree for **Rotor1**
  - Double click on the command **CreateUserDefinedPart** to modify its parameters
  - In the window,
    - Change InforCore to **1**
    - Press **OK**

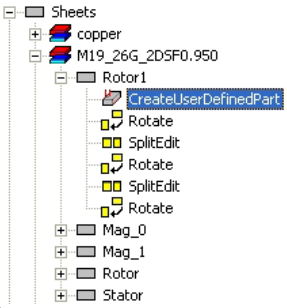


	HRib	3	mm	3mm	Rib height
	DminMag	4.5	mm	4.5mm	Minimum distance between side magnets
	ThickMag	6.5	mm	6.5mm	Magnet thickness
	WidthMag	32	mm	32mm	Total width of all magnet per pole
	LenRegion	0	mm	0mm	Region length
	InfoCore	1		1	0: core; 1: magnets; 2: ducts; 100: region.

- **Separate Magnets**
  - Select the sheet **Rotor1** from history tree
  - Select the menu item ***Modeler > Boolean > Separate Bodies***
- **Change Attributes**
  - Change the name of sheets to **Mag_0** and **Mag_1**
  - Change the color of the sheets to **Yellow**

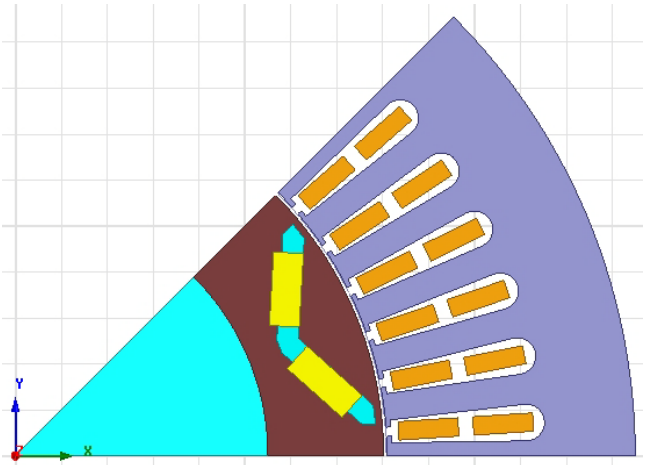


- **Create Duct**
  - Select the sheet **Rotor** from History tree
  - Select the menu item ***Edit > Copy*** or press **Ctrl+C** from keyboard
  - Select the menu item ***Edit > Paste*** or press **Ctrl+V** from keyboard
  - New Sheet named **Rotor1** is created
  - Expand the history tree for **Rotor1**
  - Double click on the command **CreateUserDefinedPart** to modify its parameters
  - In the window,
    - Change InforCore to **2**
    - Press **OK**



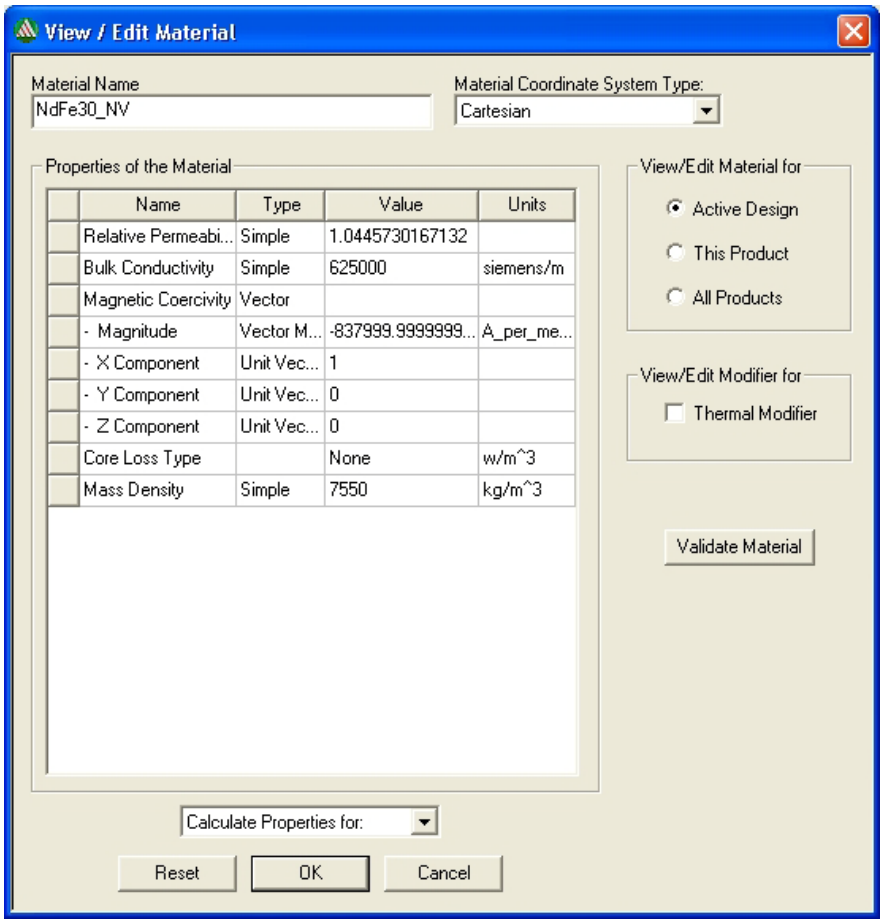
	HRib	3	mm	3mm	Rib height
	DminMag	4.5	mm	4.5mm	Minimum distance between side magnets
	ThickMag	6.5	mm	6.5mm	Magnet thickness
	WidthMag	32	mm	32mm	Total width of all magnet per pole
	LenRegion	0	mm	0mm	Region length
	InfoCore	2		2	0: core; 1: magnets; 2: ducts; 100: region.

- **Change Attributes**
  - Change the name of the Rotor1 to **Duct**
  - Change the color of the sheet to **Light Blue**
  - Change the material to **Vacuum**



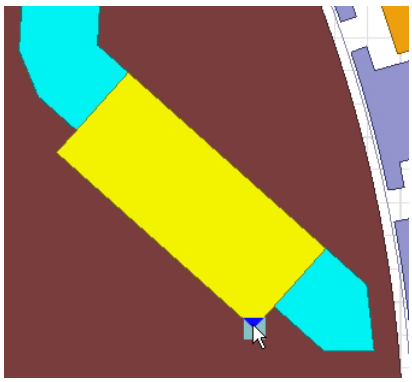
Assign Material to Magnets

- **To Assign magnetic Material**
  - Press **Ctrl** and select the sheets **Mag_0** and **Mag_1,** right click and select **Assign Material**
  - In Select Definition window,
    - Type **NdFe30** in Search by Name field
    - Select the button **Clone Material**
    - In View/Edit Material window,
      - Name: **NdFe30_NV**
      - Ensure that X component is set to **1**
      - Ensure Y and Z components are set to **0**
      - Press **OK**
    - Press **OK** to close Select Definition window

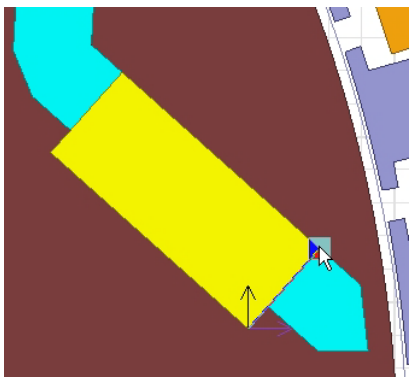


Set Magnetization Direction for Magnets

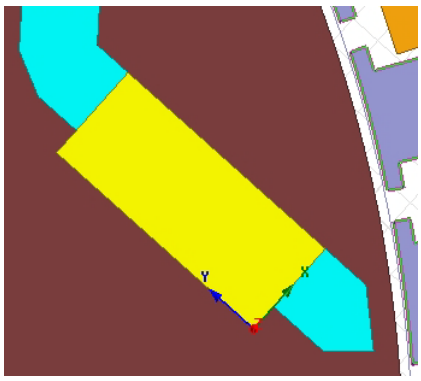
- As Material defined has magnetization X direction of the Orientation coordinate system, we need to create relative CS with X axis in direction of required magnetization direction.
- **Create Relative CS for Mag_0**
  - Select the menu item *Modeler > Coordinate System > Create > Relative CS > Both*
  - Using Graphic window, select the origin new coordinate system as shown in first image below
  - Using Graphic window, select the vertex to define X axis as shown in second image below



Origin

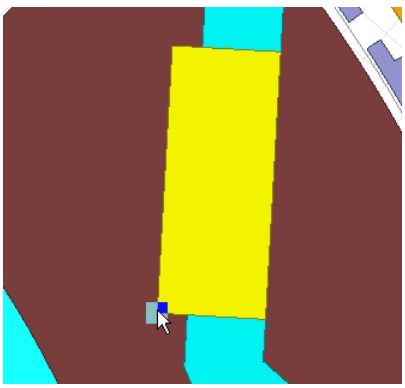


X Axis

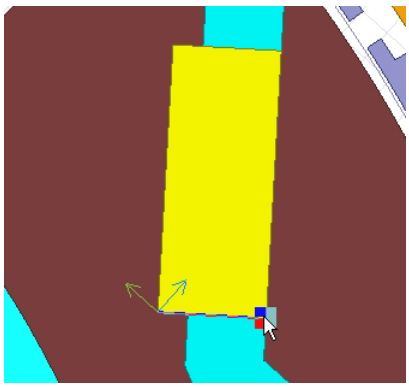


Created Coordinate System

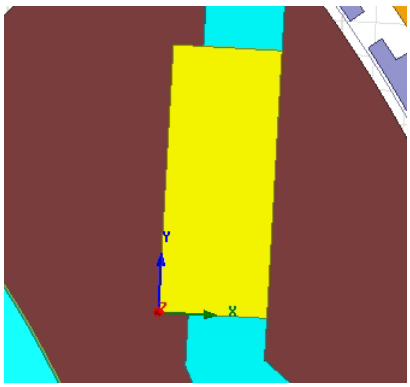
- **Create Relative CS for Mag_1**
  - Using Similar steps as above, create Relative CS for **Mag_1**



Origin

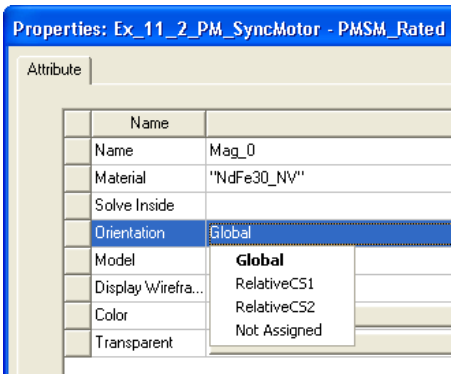


X Axis



Created Coordinate System

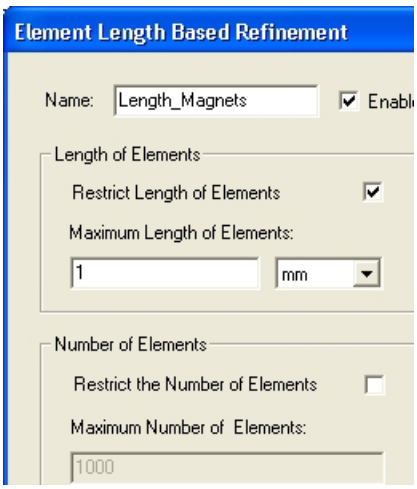
- **Assign Relative CS for Magnets**
  - Double click on the sheet **Mag_0** from history tree to open Properties window
  - In Properties window,
    - Change Orientation to **RelativeCS1**
    - Press **OK**



  - Similarly change the orientation of the sheet **Mag_1** to **RelativeCS2**

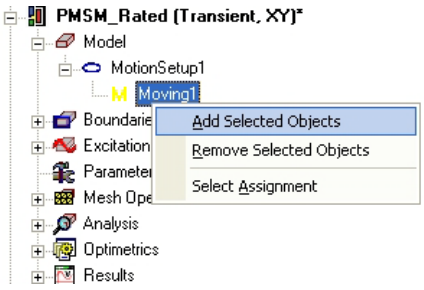
Assign Mesh Operation to Magnets

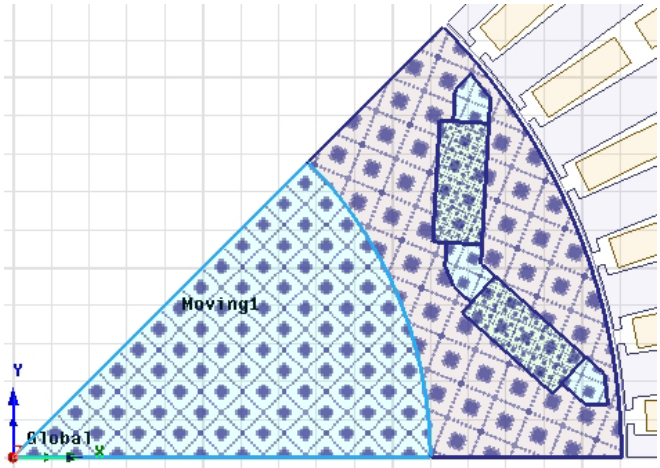
- **To Assign Mesh operation**
  - Press **Ctrl** and select the sheets **Mag_0** and **Mag_1**
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length Based Refinement window,
    - Name: **Length_Magnet**
    - Restrict Length of Elements: ☑ **Checked**
    - Maximum Length of Elements: **1mm**
    - Press **OK**



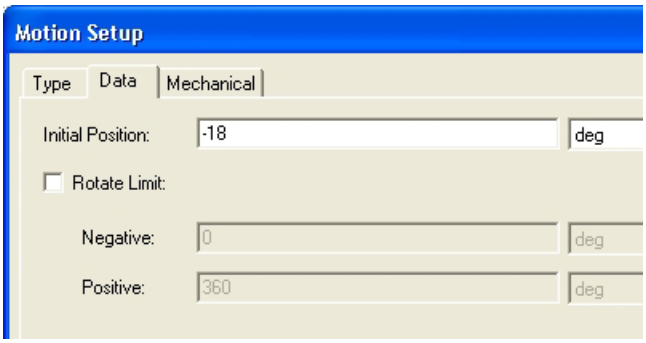
Modify motion

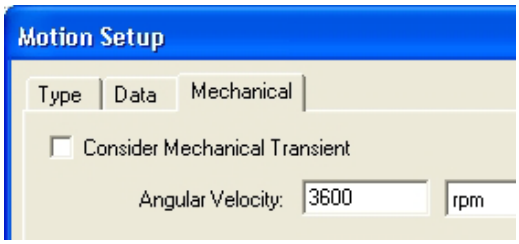
- **Add New Objects to Motion**
  - Press **Ctrl** and select the sheets **Rotor, Mag_0, Mag_1** and **Duct**
  - Expand the Project Manager tree to view **Model > MotionSetup1**
  - Right click on the tab **Moving1** and select **Add Selected objects**





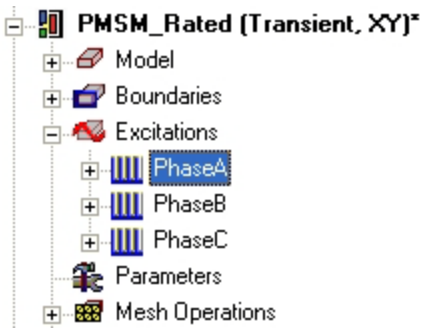
- **Modify Motion Parameters**
  - Double click on the tab **MotionSetup1** to modify parameters
  - In Motion Setup window,
    - **Data** tab
      - Change Initial Position to **-18 deg**
    - **Mechanical** tab
      - Change Angular Velocity to **3600 rpm**
    - Press **OK**

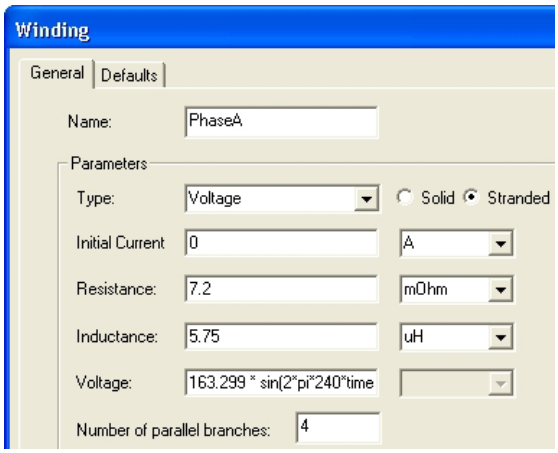




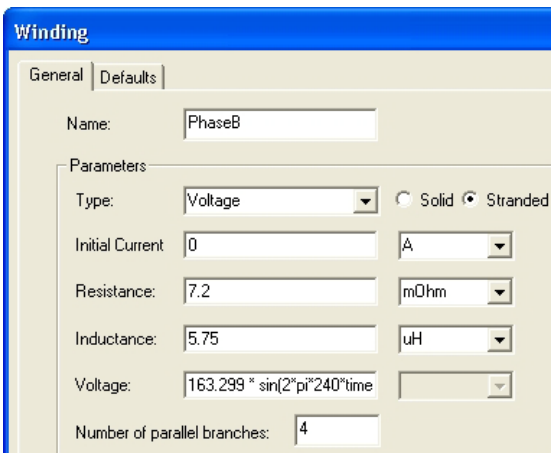
Modify Excitations

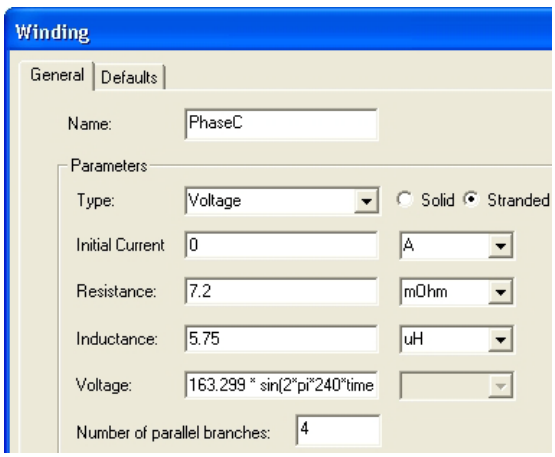
- **To Modify Excitations for PhaseA**
  - Expand the Project Manager tree to view **Excitations**
  - Double click on the tab **PhaseA** to modify its parameters
  - In Winding window,
    - Change Type to **Voltage**
    - Resistance: **7.2 mOhm**
    - Inductance: **5.75 uH**
    - Voltage: **163.299 * sin(2*pi*240*time+18.2635*pi/180)**
    - Press **OK**



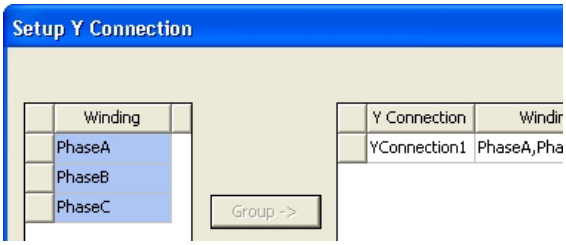


- **Modify Excitations for PhaseB and Phase C**
  - Similarly change parameters for **PhaseB** and **PhaseC** as below
  - Voltages
    - PhaseB: **163.299 * sin(2*pi*240*time+18.2635*pi/180-2*pi/3)**
    - PhaseC: **163.299 * sin(2*pi*240*time+18.2635*pi/180-4*pi/3)**



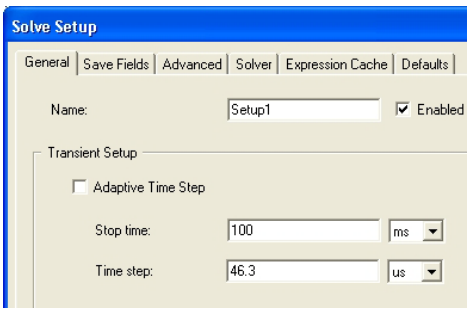


- **Set Y Connection**
  - Select the menu item *Maxwell 2D > Excitations > Set Y Connection*
  - In Set Y Connection window,
    - Press **Ctrl** and select all three windings
    - Select the button **Group ->**
    - Press **OK**



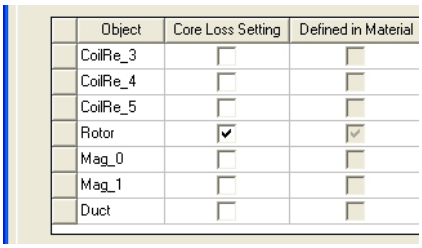
Modify Analysis Setup

- **To Modify Analysis Setup**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Change Stop Time to **100 ms**
      - Change Time Step to **46.3 us**
    - Press **OK**



Set Core Loss

- **To Set Core Loss for Rotor**
  - Select the menu item *Maxwell 2D > Excitations > Set Core Loss*
  - In Core Loss window,
    - For the object **Rotor**
      - Core Loss Settings: ☑ **Checked**
    - Press **OK**



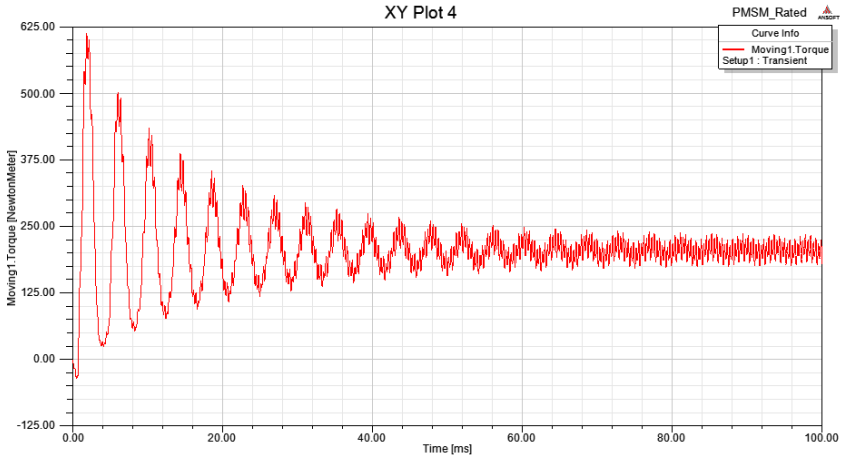
Analyze

- **To Run the calculations**
  - Expand the Project Manager tree to view **Analysis**
  - Right click on the tab **Setup1** and select **Analyze**

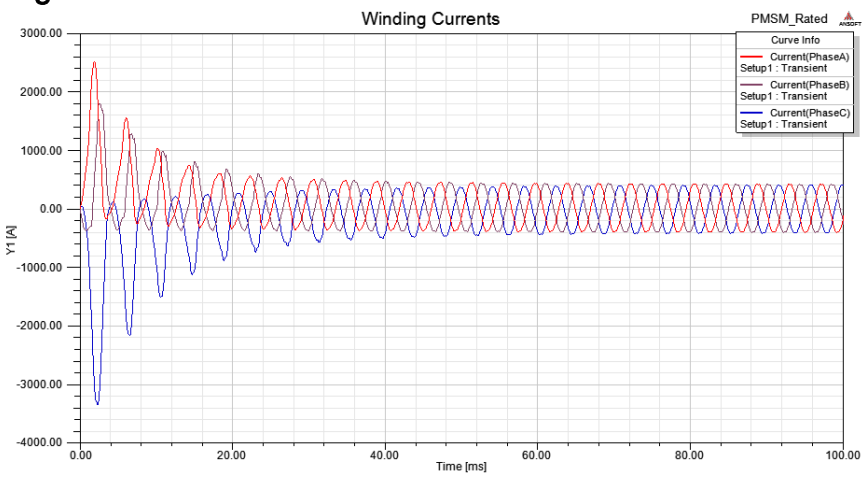
- **Note:** Problem takes about 15min to solve

Results

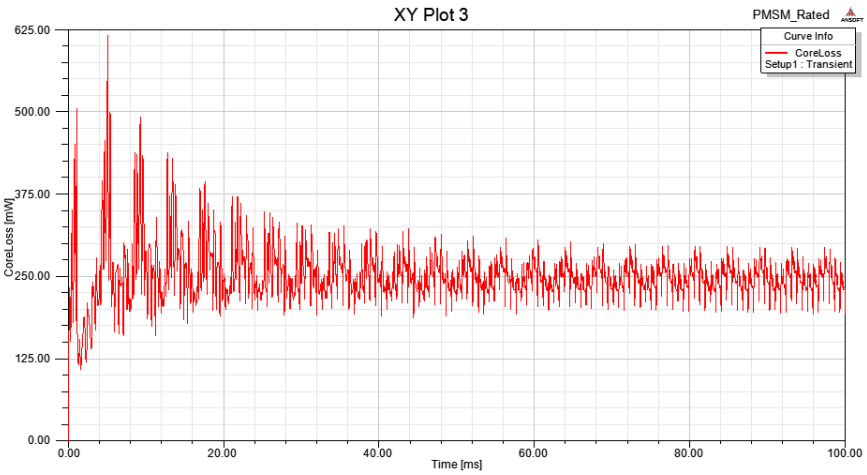
Torque Vs Time



Winding Currents Vs Time



Core Loss Vs Time



5. Simplorer Drive Design

Save Maxwell Project

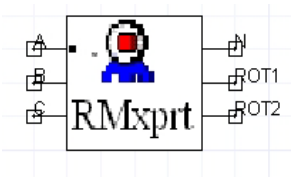
- **To Save Maxwell Project**
  - Select the menu item ***File > Save***
  - Save the file with appropriate name at desired location

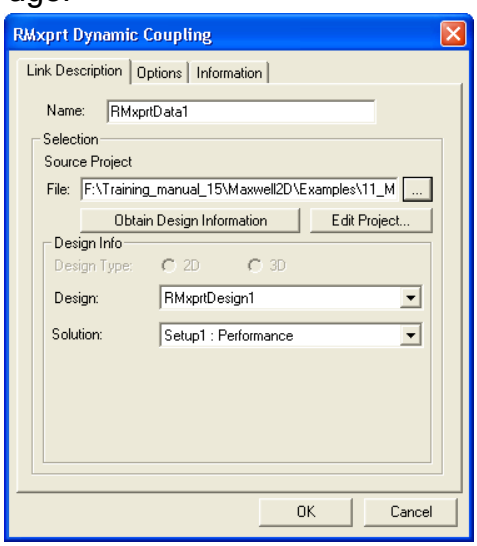
Launch Simplorer

- **To Launch Simplorer**
  - Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Simplorer 10.0** and select **Simplorer 10.0**

Add RMxprt Component

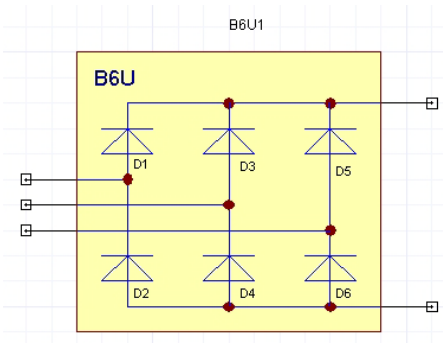
- **To Add RMxprt Component**
  - Select the menu item ***Simplorer Circuit > SubCircuit > Add RMxprt Dynamic Component***
  - In RMxprt Dynamic Coupling window,
    - Source Project: Browse to the Maxwell Project file which was saved in last step and select it
    - RMxprt Design will be automatically selected
    - Press **OK**
  - Place the RMxprt component on Project Page.
  - Press **Esc** to leave component addition

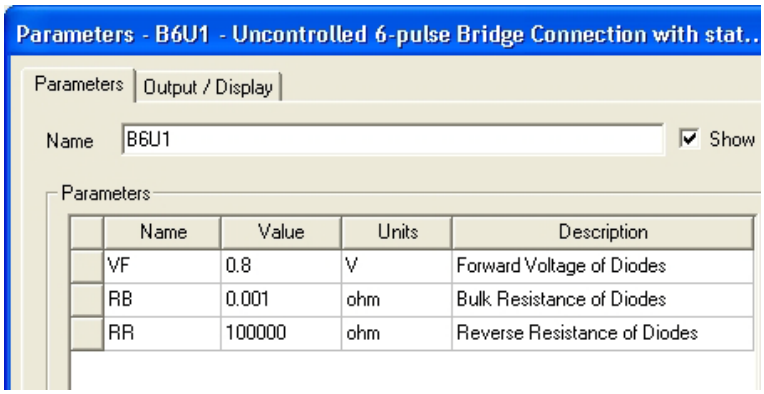




Add Components

- **Add Line Commuted Converter**
  - Select the button **Components** at the bottom of Project Manager window
  - Select the component from **Simplorer Elements > Multiphysics > Power System > Power Distribution > Power Conversion > Converter > B6U:Uncontrolled 6-pulse Bridge Connection with static Models for Diodes**
  - Drag and drop it on the Project page
  - Press **Esc** to exit component addition
  - Double click on added component to modify its parameters
    - Set the parameters as shown in below image
    - Press **OK**

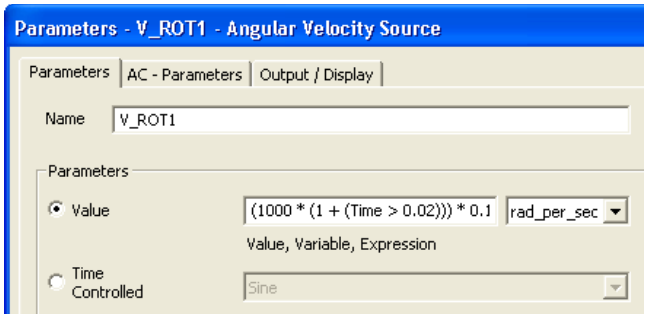




- **Add Other Components**
  - Follow the same steps as above and add rest of components
  - **Resistor**
    - From **Simplorer Elements > Basic Elements > Circuit > Passive Elements > R: Resistor**
    - Parameters
      - Resistance: **1 kOhm**
  - **Capacitor**
    - From **Simplorer Elements > Basic Elements > Circuit > Passive Elements > C: Capacitor**
    - Parameters
      - Capacitance: **1 uF**

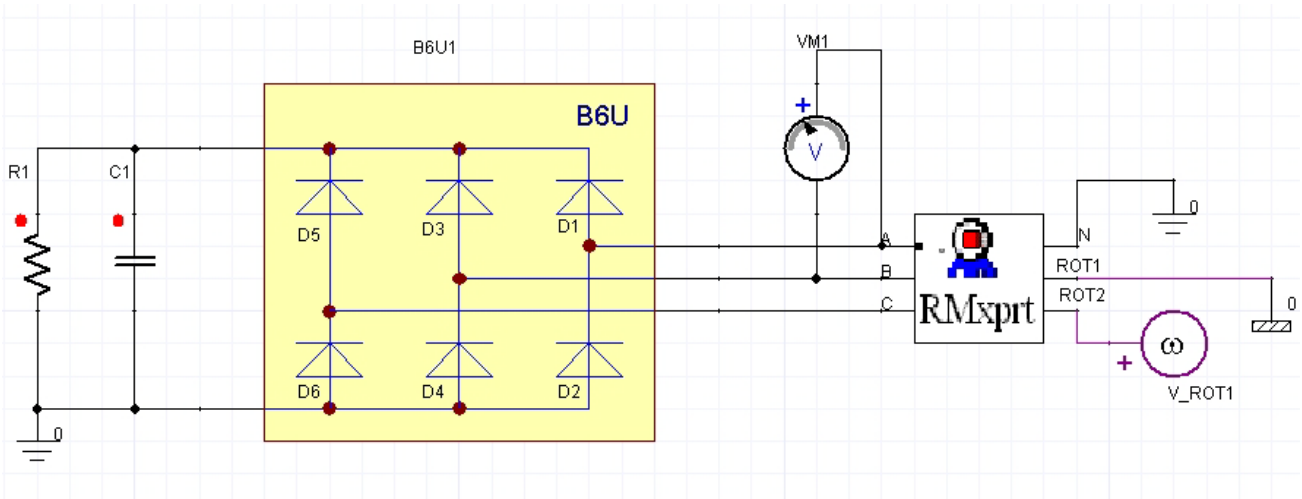
Motors - Permanent Magnet Synchronous Machine

- **Voltmeter**
  - From **Simplorer Elements > Basic Elements > Measurement > Electrical > VM: Voltmeter**
- **Angular Velocity Source**
  - From **Simplorer Elements > Basic Elements > Physical Domain > Mechanical > Velocity-Force-Representation > Rotational_V > V_Rot: Angular Velocity Source**
  - Parameters
    - OMEGA1: **(1000 * (1 + (Time > 0.02))) * 0.10471975511966**
    - Press **OK**



Build Circuit

- **To Build the Circuit**
  - Position all components as shown in below image
  - Connect the components by Wires
  - Add Ground from ***Draw > Ground***
  - Build the circuit as shown in below image



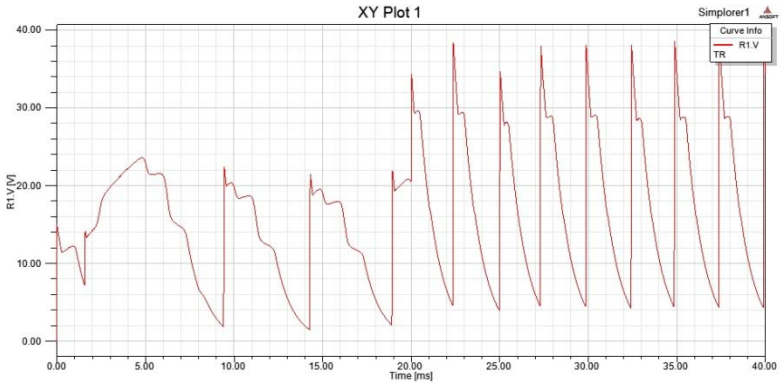
Analyze

To Run the Solution

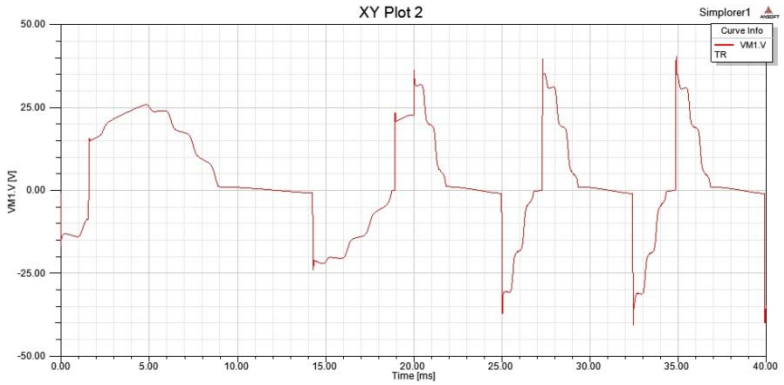
- Select the menu item *Simplorer Circuit > Analyze*

Results

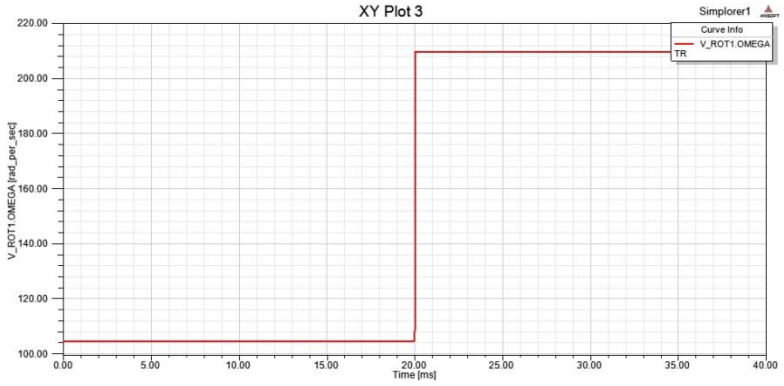
DC Link Voltage



Back EMF



Input Speed from Shaft



Motors - Three Phase Induction Machine

Content

I. RMxprt
- Basic Theory
- Main Features
- Open and Run an Example and View Results
- Modify the Design and View New Results
- Create a Variable-Voltage Variable-Frequency Design

II. RMxprt -> Maxwell 2D
- Create an Maxwell 2D Design from RMxprt
- Review Maxwell 2D Transient Setup

III. Nominal Maxwell 2D Design
- Create an Maxwell 2D Design from RMxprt
- Change Simulation Time Step
- Set up Save Solution Fields
- Set Eddy Effects for All Bars
- Run Simulation and Review Quick Report
- Measure Position Change of Each Time Step
- Create Mesh/Field Plots and Animation

IV. Modifying Maxwell 2D Design
- Set up Variable Simulation Time Step
- Change Angular Velocity
- Create output Variables for Stator Teeth and Stator Yoke Flux Densities
- Set up Force Calculation
- Multi-frequency Core Loss
- Run Simulation and View New Results

Content *(Contd..)*

V. Locked Rotor Simulation
- Change Initial Position and Angular Velocity
- Run Simulation and View Results

VI. Design Sweep
- Add Design Variables and Change Variable Values
- Set up Parametric Sweep
- Run Design Sweep and View Results

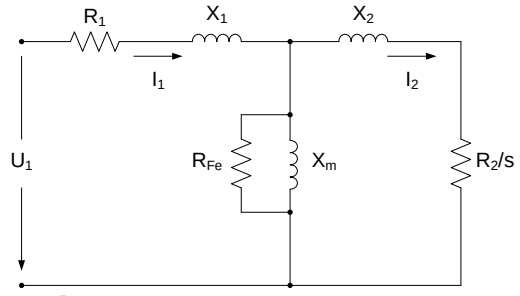
VII. Drive Design
- Create Induction Machine Model
- Drive Simulation

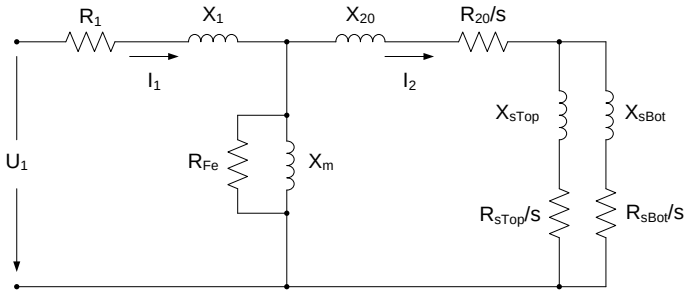
- **Note:**
  1. RMxprt/Maxwell V13 or higher is required
  2. Basic knowledge of electric machine is required
  3. Basic understanding of Finite Element is required

Motors - Three Phase Induction Machine

1- RMxprt

Basic Theory





$$P_m = 3I_2^2 \frac{R_2}{s} \qquad T_2 = T_m - T_{fw} \qquad P_2 = T_2 \omega_2 \qquad \cos\varphi = \frac{P_1}{mU_1 I_1}$$

$$T_m = \frac{P_m}{\omega} \qquad P_1 = P_2 + P_{fw} + P_{Cu2} + P_{Fe} + P_{Cu1} + P_s \qquad \eta = \frac{P_2}{P_1} \times 100\%$$

Main Features

- Optimization of Winding and Coil Arrangement
- Winding Editor Supporting Any Single- and Double-Layer Windings
- More than 20 Slot Shapes for Single and Double Squirrel-Cage Rotor
  - Pear type with round shoulder
  - Pear type with tapered shoulder
  - Trapezoid type with tapered shoulder
  - Trapezoid type with round shoulder
- Nonlinear and Distributed Parameters at Any Operation Condition

Launching Maxwell

- **To access Maxwell:**
  - 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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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 File

- To Open File
  - Select the menu item *File > Open Examples*
  - Browse to the directory *RMxprt* | *indm3* , locate the file **"yzd132-4.mxwl"** and **Open** it.
  - Change the name of the design to **"1_Original"**

Save File

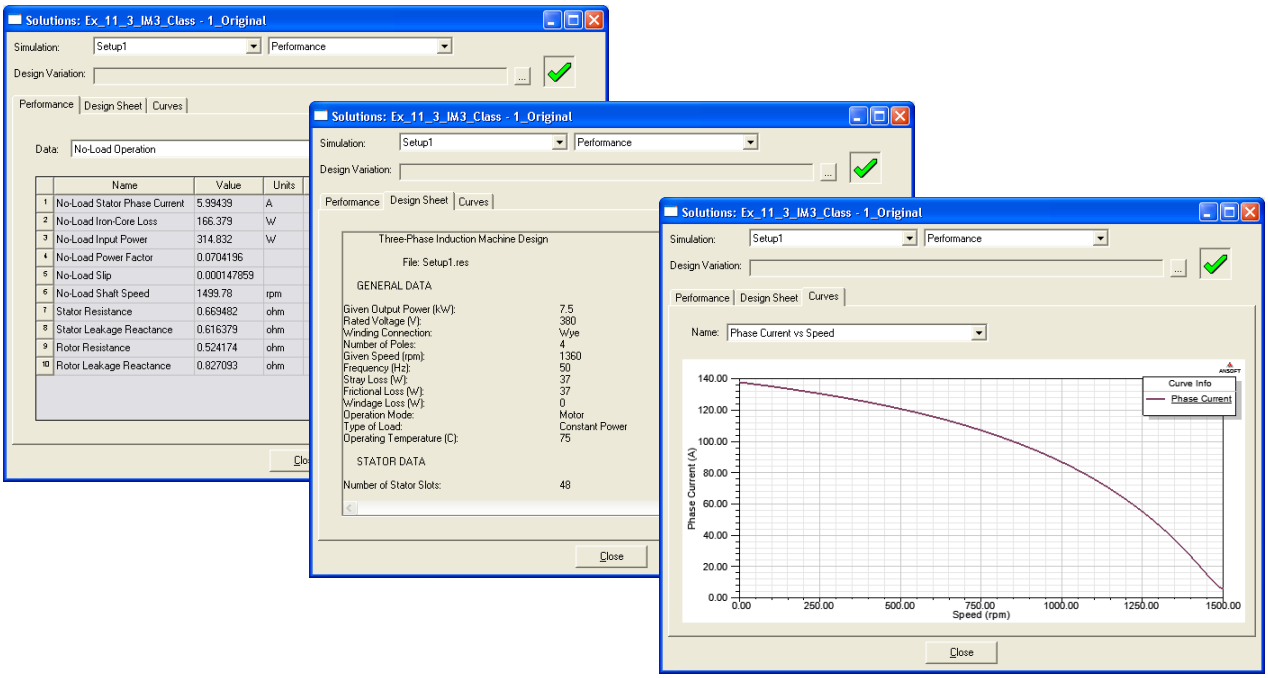
- To Save File
  - Save the file with the name **"Ex_11_3_IM3_Class.mxwl"**

Run Solution

- To Run Solution
  - Select the menu item *RMxprt > Analyze All*

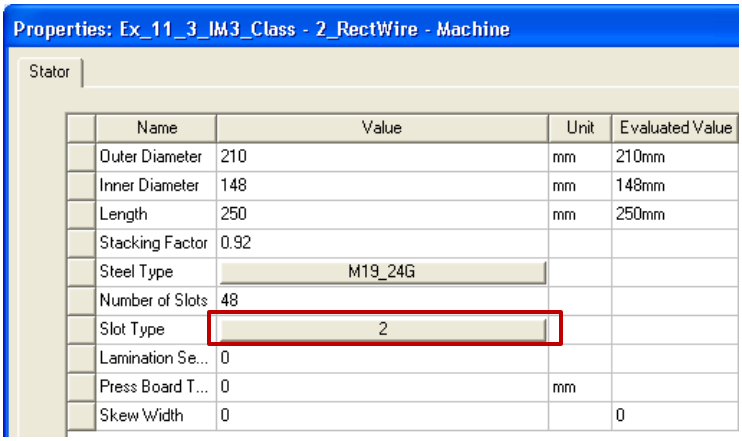
View Results

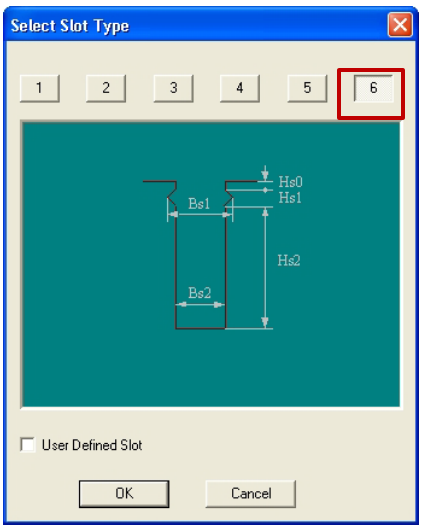
- To View Results
  - Select the menu item *RMxprt > Results > Solution Data*

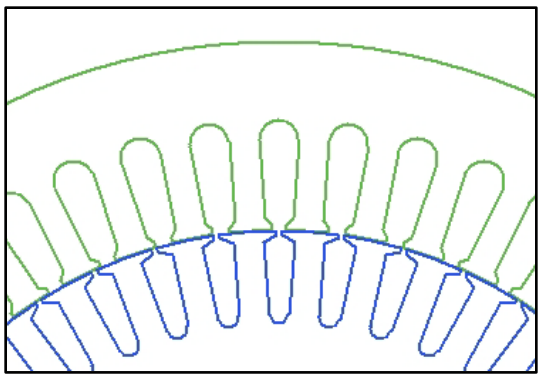


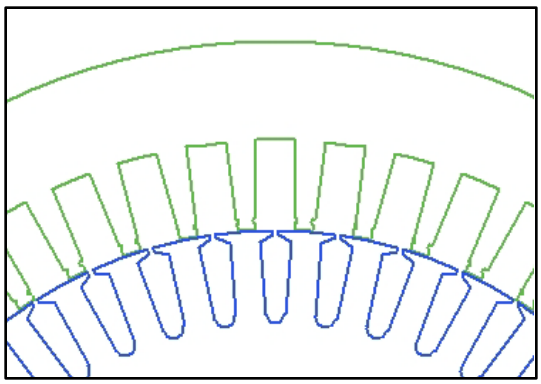
Modify Design

- **Copy Design**
  - Select the design **"1_Original"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Change the name of the design to **"2_RectWire"**
- **Modify Slot Type**
  - Expand the Project Manager tree to view **Machine**
  - Double click on the tab **Stator** to edit its parameters
  - In Properties window,
    - Click in Value field for **Slot Type**
    - In Select Slot Type window,
      - Change Type to **6**
      - Press **OK**
    - Press **OK** to close Properties window



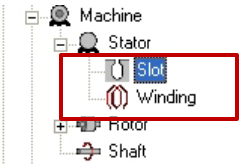


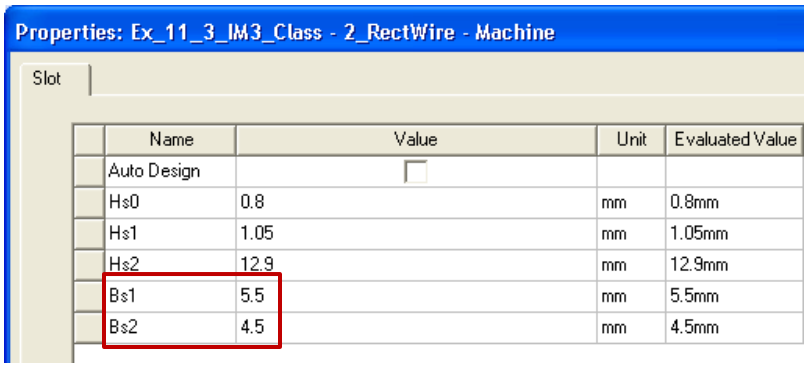




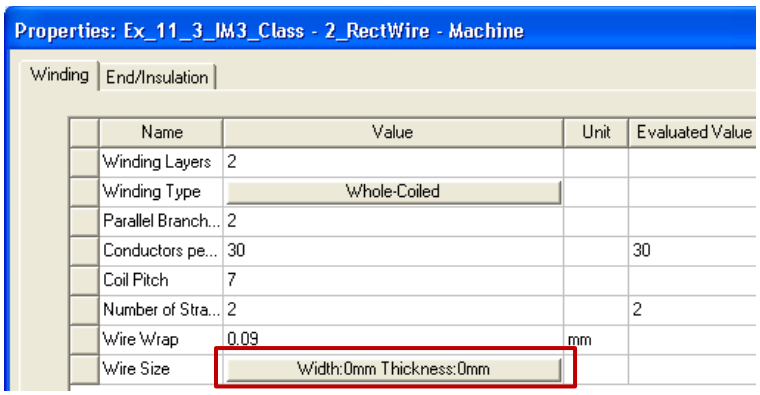
Motors - Three Phase Induction Machine

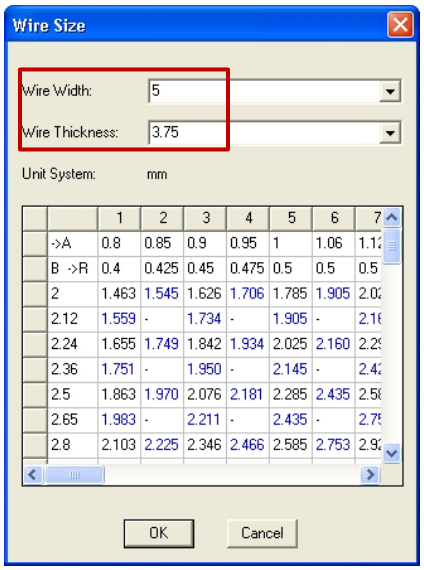
- **Modify Slot Parameters**
  - Expand the Project Manager tree to view **Stator**
  - Double click on the tab **Slot** to modify its parameters
  - In Properties window,
    - Change Bs1 to **5.5**
    - Change Bs2 to **4.5**
    - Press **OK**





- **Modify Winding**
  - Double click on the tab **Winding**
  - In Properties window,
    - Select the button in the value field for Wire Size
    - In Wire Size window,
      - Set Wire Width to **5**
      - Set Wire Thickness to **3.75**
      - Press **OK**
    - Press **OK** to close Properties window

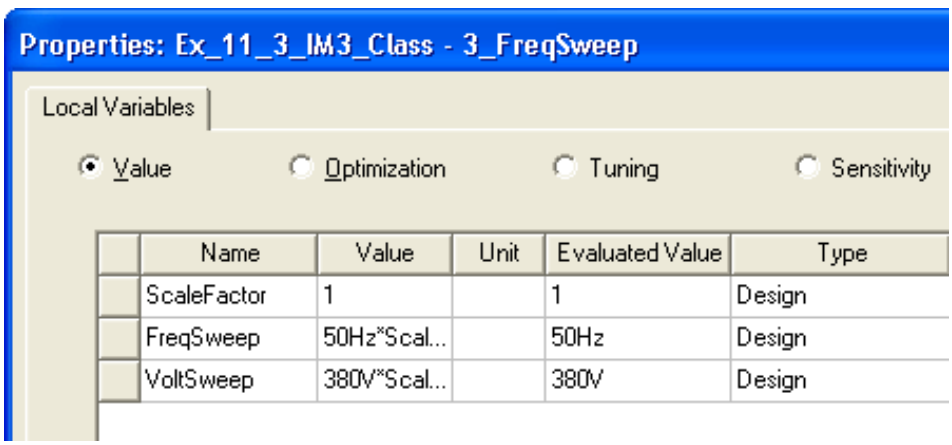




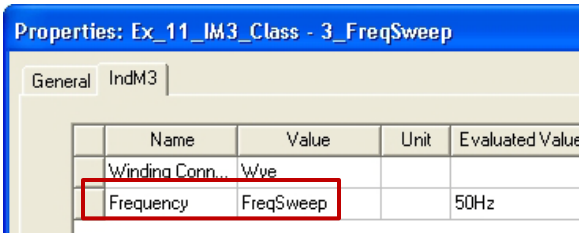
- **Run Solution and View Results**

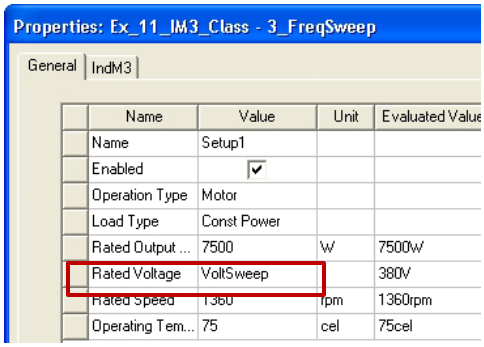
Create a Variable-Voltage Variable-Frequency Design

- **Copy Design**
  - Select the design **"2_RectWire"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Change the name of the design to **"3_FreqSweep"**
- **Add Design Variables**
  - Select the menu item *RMxprt > Design Properties*
  - In Properties window,
    - Select **Add**
    - In Add Property window,
      - Name: **ScaleFactor**
      - Value: **1**
      - Press **OK**
    - Similarly add two more parameters
    - Frequency
      - Name: **FreqSweep**
      - Value: **50Hz*ScaleFactor**
    - Voltage
      - Name: **VoltSweep**
      - Value: **380V*ScaleFactor**
    - Press **OK**

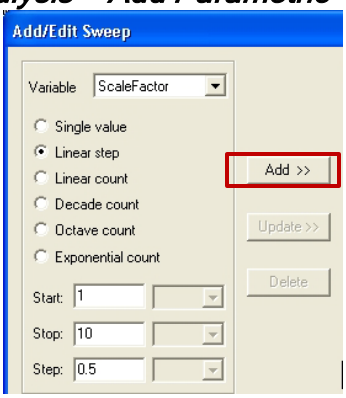


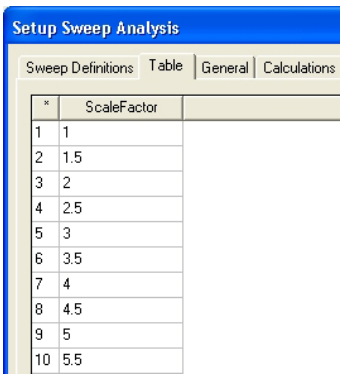
- **Assign Sweep Parameters**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Properties window,
    - **General** tab
      - Change Rated Voltage to **VoltSweep**
    - **IndM3** tab
      - Change Frequency to **FreqSweep**
    - Press **OK**





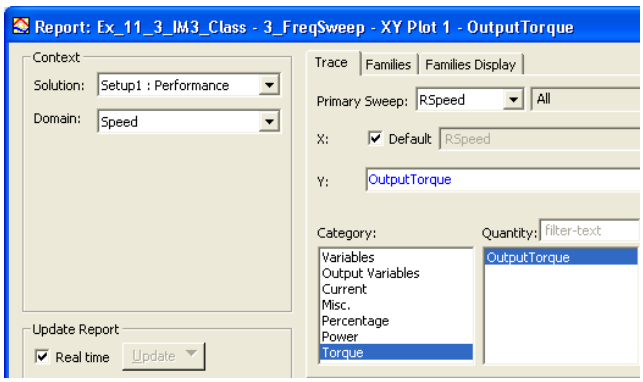
- **Create Parametric Sweep**
  - Select the menu item *RMxprt > Optimetrics Analysis > Add Parametric*
  - In Setup Sweep Analysis window, select **Add**
    - In Add/Edit Sweep window,
      - Variable: **ScaleFactor**
      - Select **Linear Step**
      - Start: **1**
      - Stop: **10**
      - Step: **0.5**
      - Select **Add>>**
      - Press **OK**
    - Select **Table** tab
      - To view Sweep Variations
    - Press **OK**

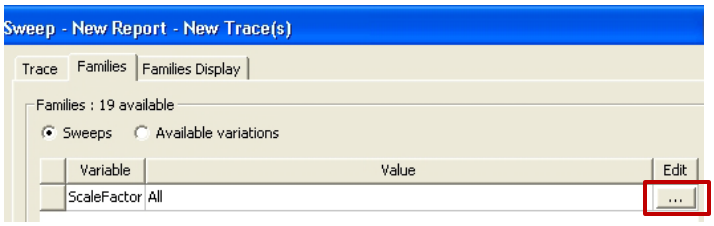


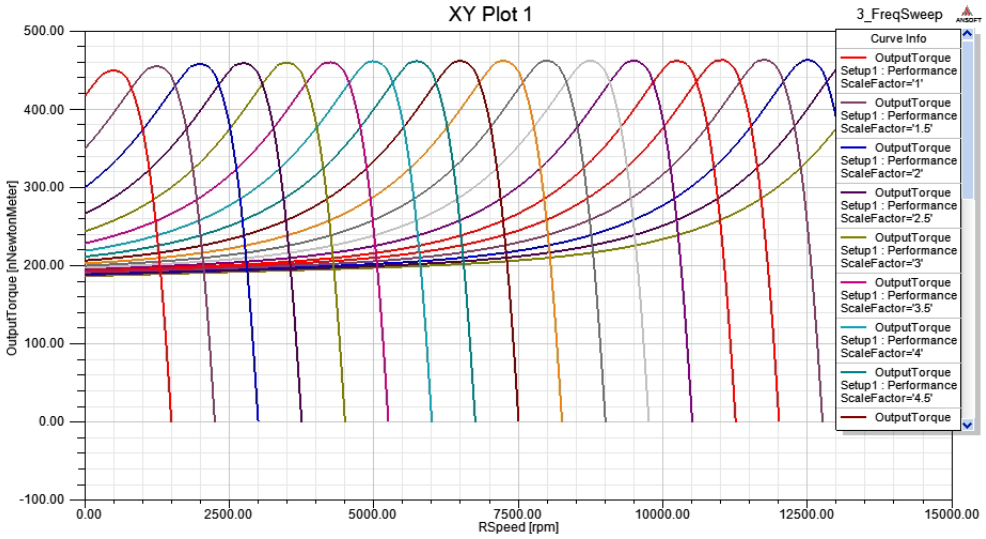


Motors - Three Phase Induction Machine

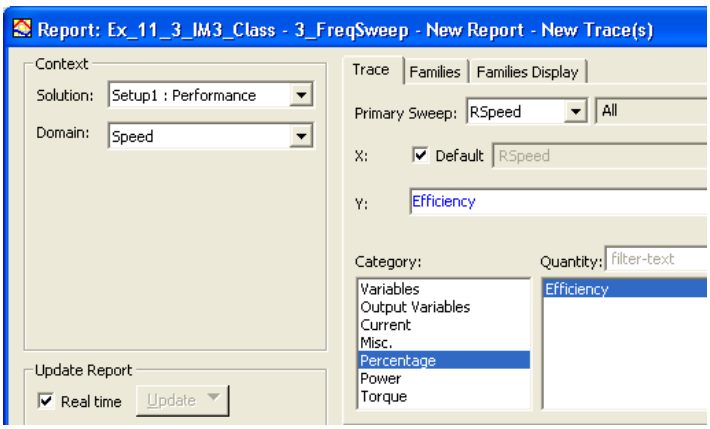
- **Run Parametric Sweep**
  - Expand the Project Manager tree to view **Optimetrics**
  - Right click on the tab **ParametricSetup1** and select **Analyze**
- **Plot Torque Vs Speed**
  - Select the menu item *RMxprt > Results > Create RMxprt Report > Rectangular Plot*
  - In Report window,
    - **Trace** tab
      - Primary Sweep: **RSpeed**
      - Category: **Torque**
      - Quantity: **OutputTorque**
    - **Families** tab
      - For the variable ScaleFactor, select the button in edit column
      - Check the button **Use All Values**
    - Select **New Report**

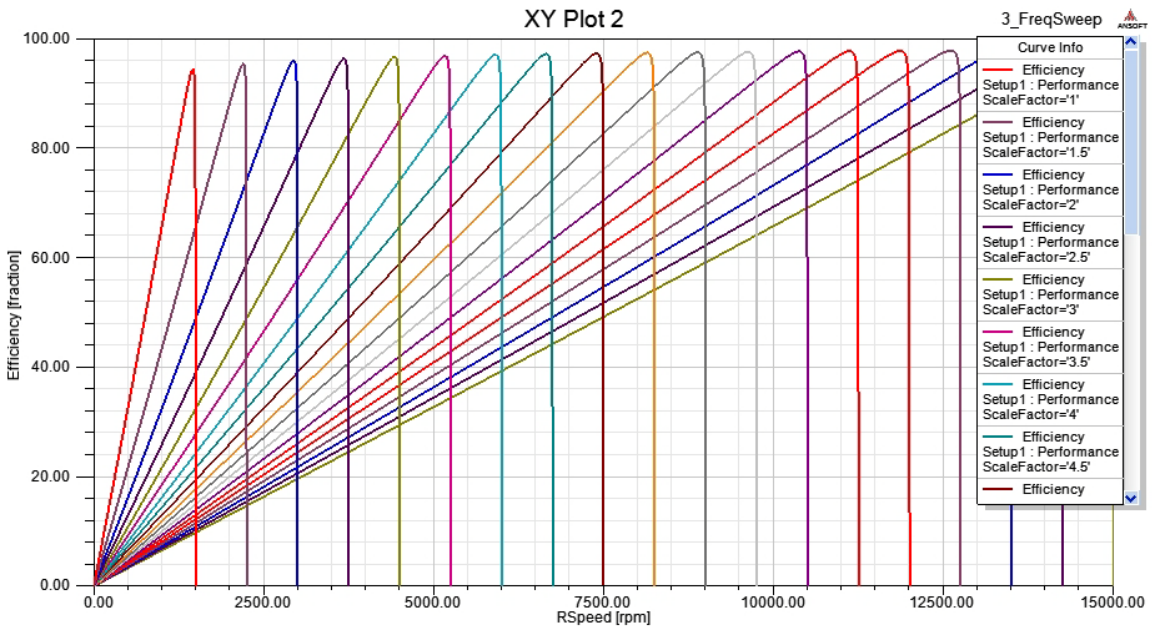






- **Plot Efficiency Vs Speed**
  - Select the menu item *RMxprt > Results > Create RMxprt Report > Rectangular Plot*
  - In Report window,
    - **Trace** tab
      - Primary Sweep: **RSpeed**
      - Category: **Percentage**
      - Quantity: **Efficiency**
    - **Families** tab
      - For the variable ScaleFactor, select the button in edit column
      - Check the button **Use All Values**
    - Select **New Report**



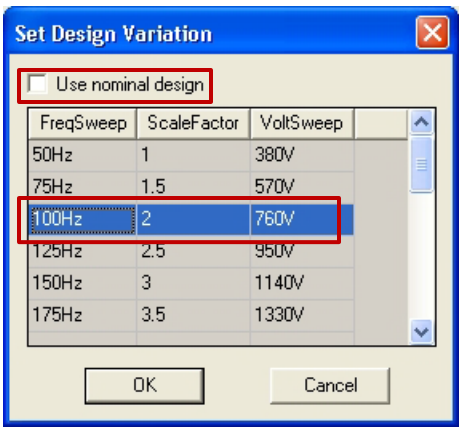


2. RMxprt → Maxwell 2D

Create Maxwell Design

- **To Create Maxwell Design**
  - Select the menu item ***RMXprt > Analysis Setup > Create Maxwell Design***
  - In Create Maxwell Design window,
    - Auto Setup: ☑ **Checked**
    - Select the button in Variation filed
    - In Design Variation window,
      - Use nominal design: ☐ **Unchecked**
      - Select the variation of FreqSweep **100Hz**
      - Press **OK**
    - Press **OK**



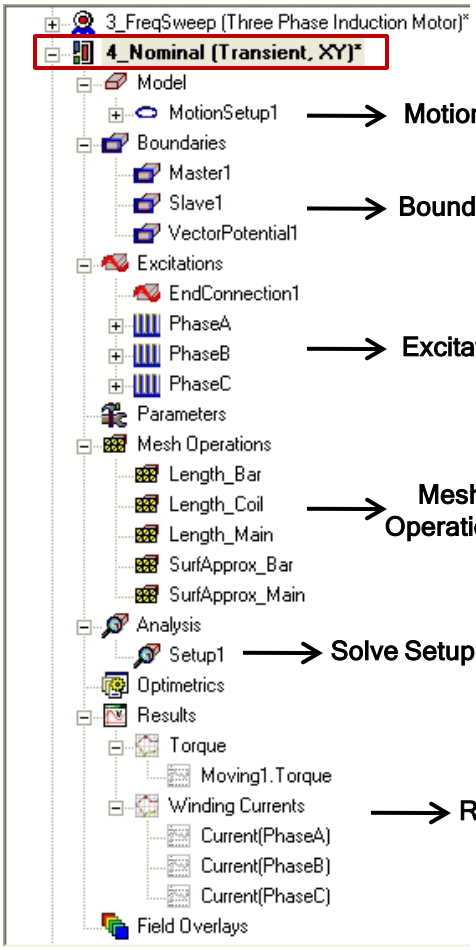


  - Change the name of the design to **"4_Nominal"**

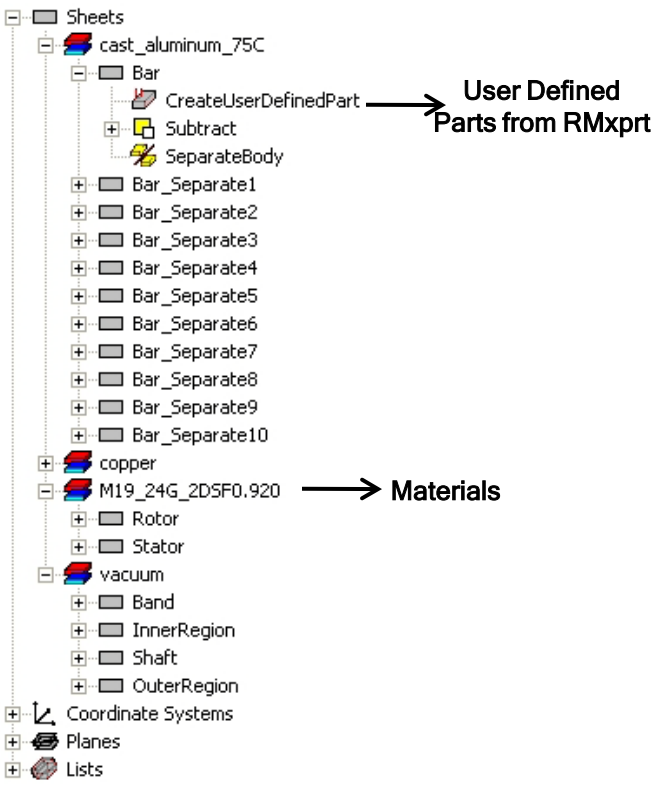
Motors - Three Phase Induction Machine

Review Maxwell 2D Transient Setup

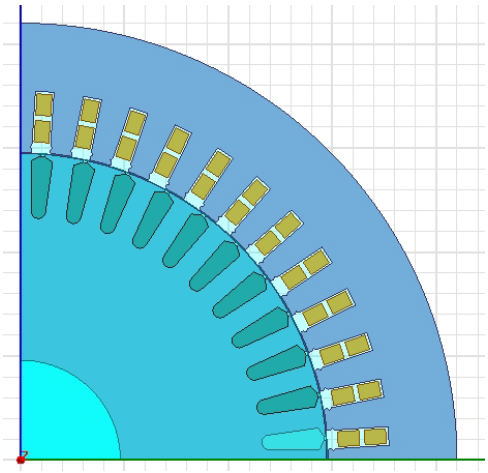
Project Manager Tree



History Tree



Geometry



Analyze

- Design is ready to solve
- Select the menu item *Maxwell 2D > Analyze All*

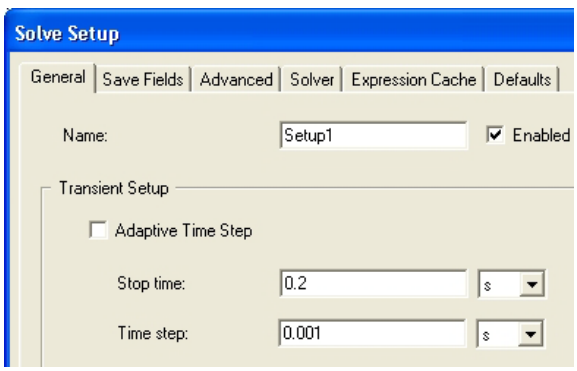
3. Nominal Maxwell 2D Design

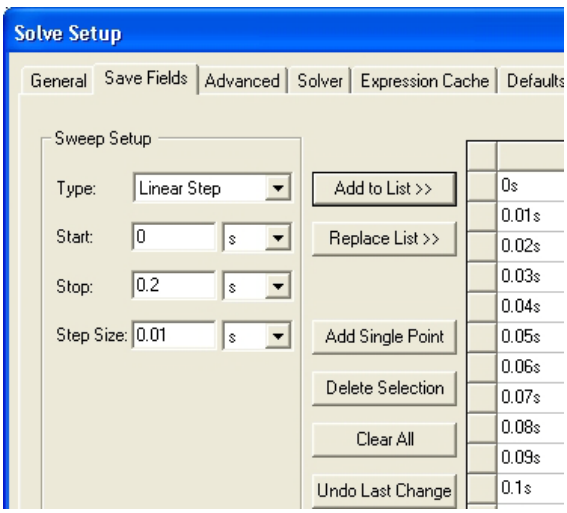
Create Maxwell 2D Design

- **To Create a 2D Design**
  - Expand the Project manager tree for the design **1_Original** to view **Analysis**
  - Right click on the tab **Setup1** and select **Create Maxwell Design**
  - In Create Maxwell Design window,
    - Press **OK**
  - Change the name of the resulting Maxwell design to **"5_Nominal2"**

Modify Solution Setup

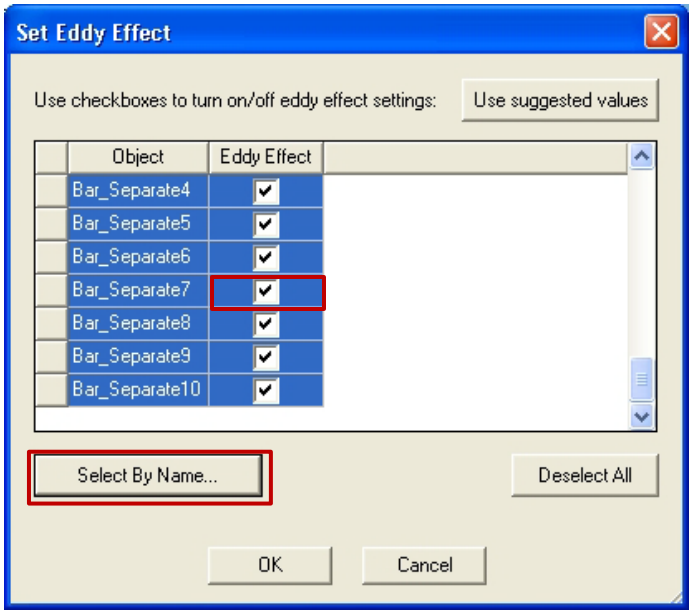
- **To Modify Solution Setup**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Time Step: **0.001s**
    - **Save Fields** tab
      - Type: **Linear Step**
      - Start: **0 s**
      - Stop: **0.2s**
      - Step: **0.01s**
      - Select **Add to List >>**
    - Press **OK**





Set Eddy Effects

- **To Set Eddy Effects for all Bars**
  - Select the menu item *Maxwell 2D > Excitations > Set Eddy Effects*
  - In Set Eddy Effect window,
    - Select the button **Select By Name**
      - Type the name **Bar*** (Search is case sensitive)
      - Press **OK**
    - All objects with name starting with **Bar** will be selected
    - Check the Box in **Eddy Effects** column for any of the selected objects
    - All other objects will be automatically selected
    - Press **OK**

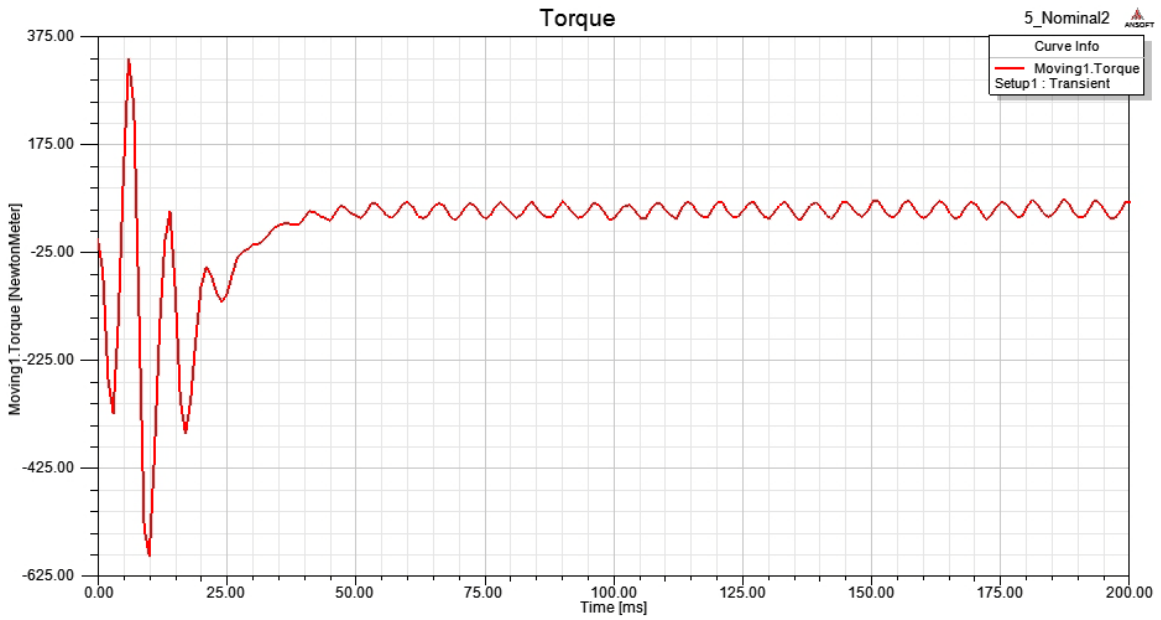


Analyze

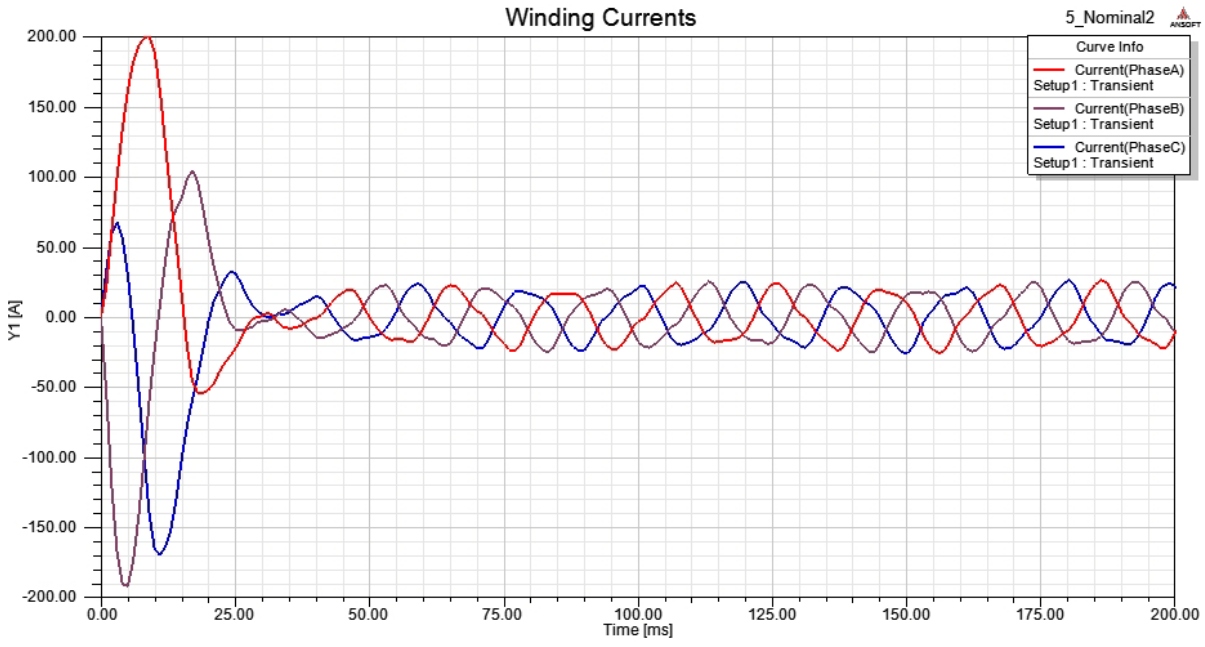
- **To Run the Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

Results

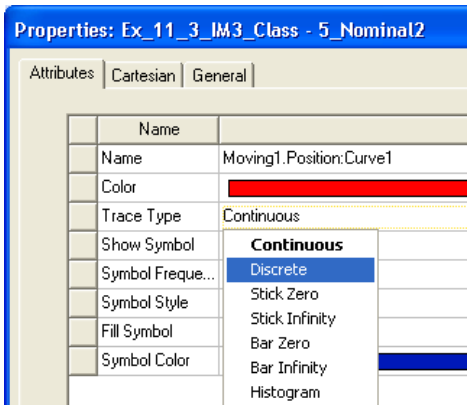
Torque Vs Time



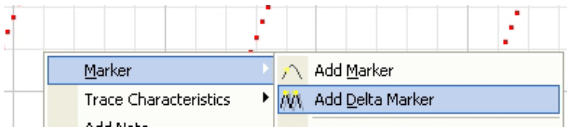
Phase Currents Vs Time

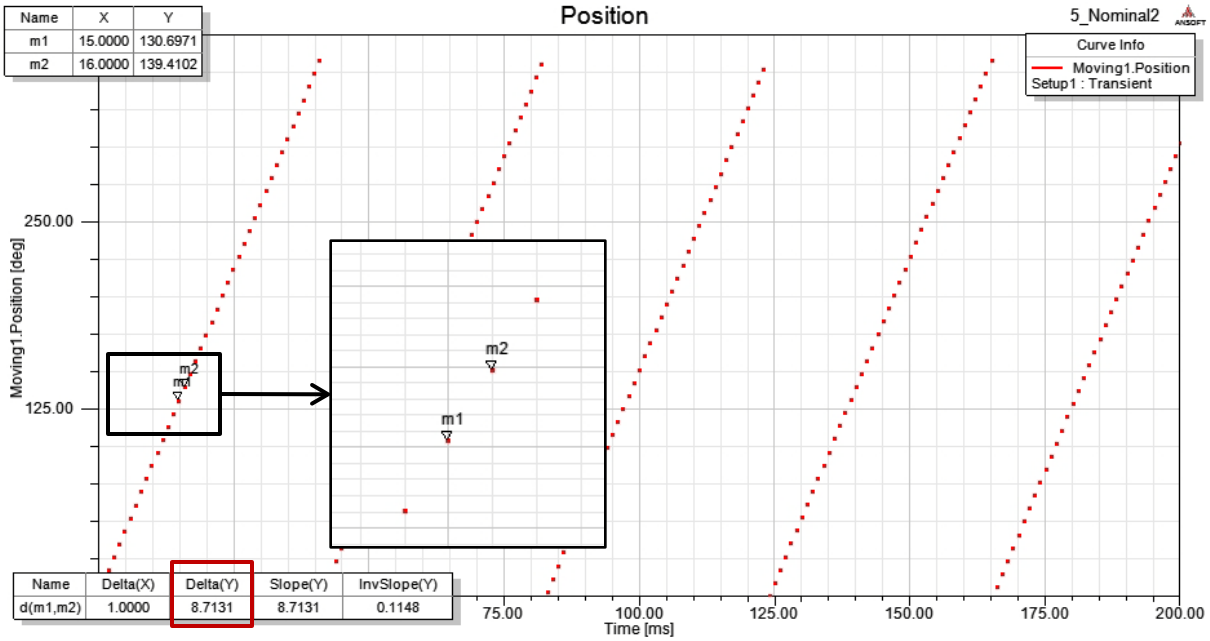


- **Plot Position Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Quick Report*
  - In Quick Report window,
    - Select the quantity **Position**
    - Press **OK**
  - Double click on the trace to modify its properties
  - In Properties window,
    - Change Trace Type to **Discrete**
    - Press **OK**

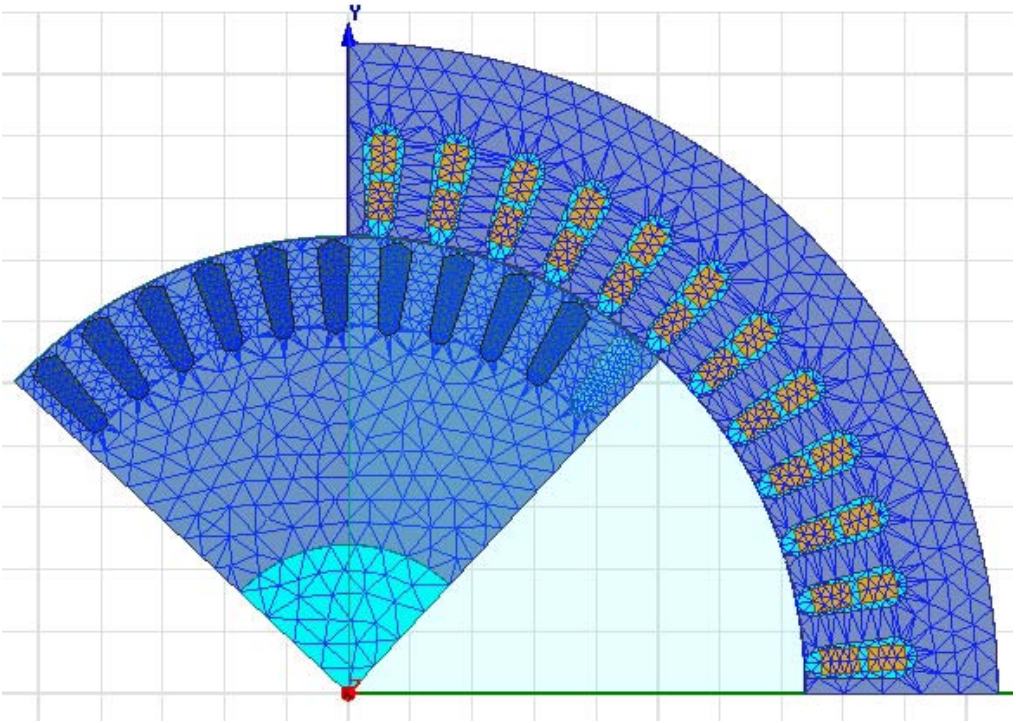


- **Calculate Position Change Per Time Step**
  - Right click on Position plot and select **Marker > Add Delta Marker**
  - Select the two adjacent points from the trace (Use Zoom In button for selection)
  - Press **Esc** after done
  - Reported value is around **8.71 deg**

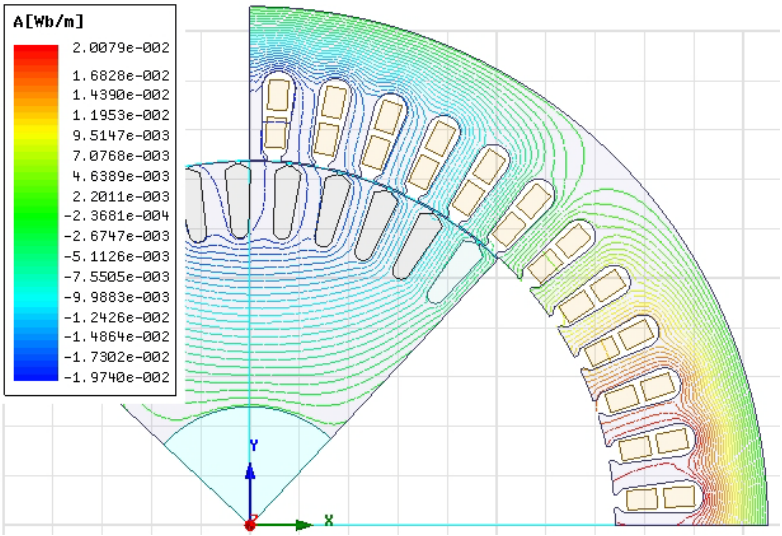




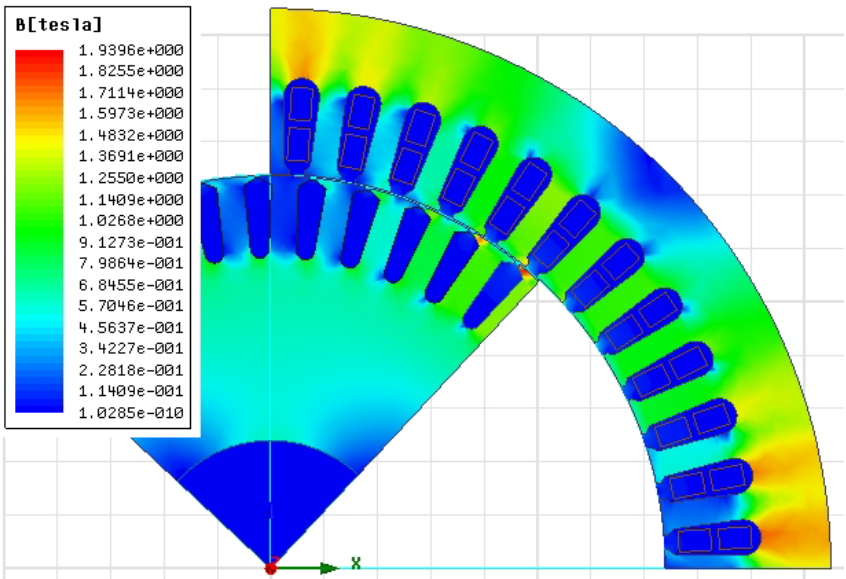
- **Set Solution Context**
  - Select the menu item *View > Set Solution Context*
  - In Set view Context window,
    - Set the Time to **0.15s**
    - Press **OK**
- **Plot Mesh**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  - In Create Mesh Plot window,
    - Press **Done**



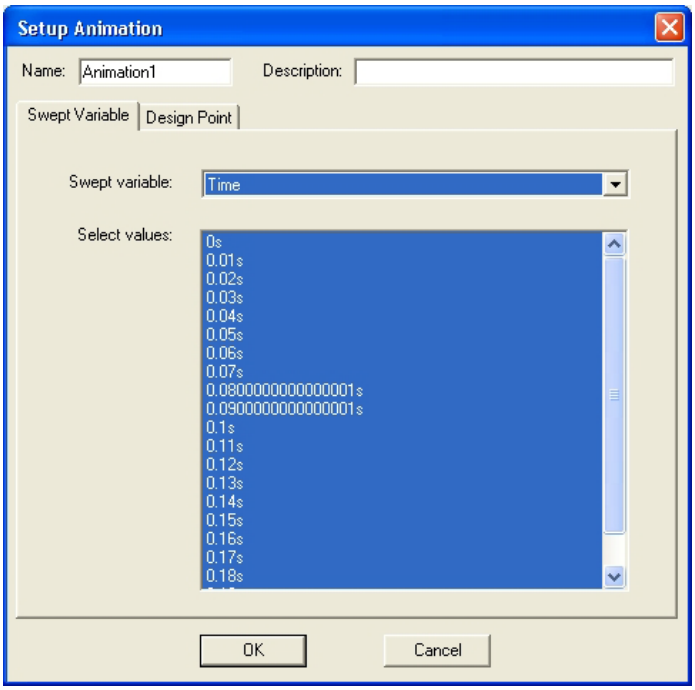
- **Plot Flux Lines**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > A > Flux_Lines*
  - In Create Field Plot window,
    - Press **Done**
  - Double click on the legend and modify its attributes as needed

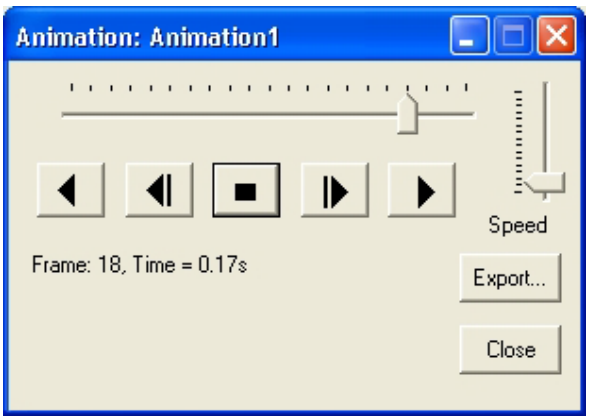


- **Plot B Field**
  - **Similarly Plot B field**



- **Animate B Plot**
  - Right click on the **Mag_B1** plot and select **Animate**
  - In Setup Animation window,
    - Press **OK**
  - Animation window will appear which can be used to Start/Stop/Pause animation
    - Select **Export** button in Animation window to write animation in AVI or GIF file

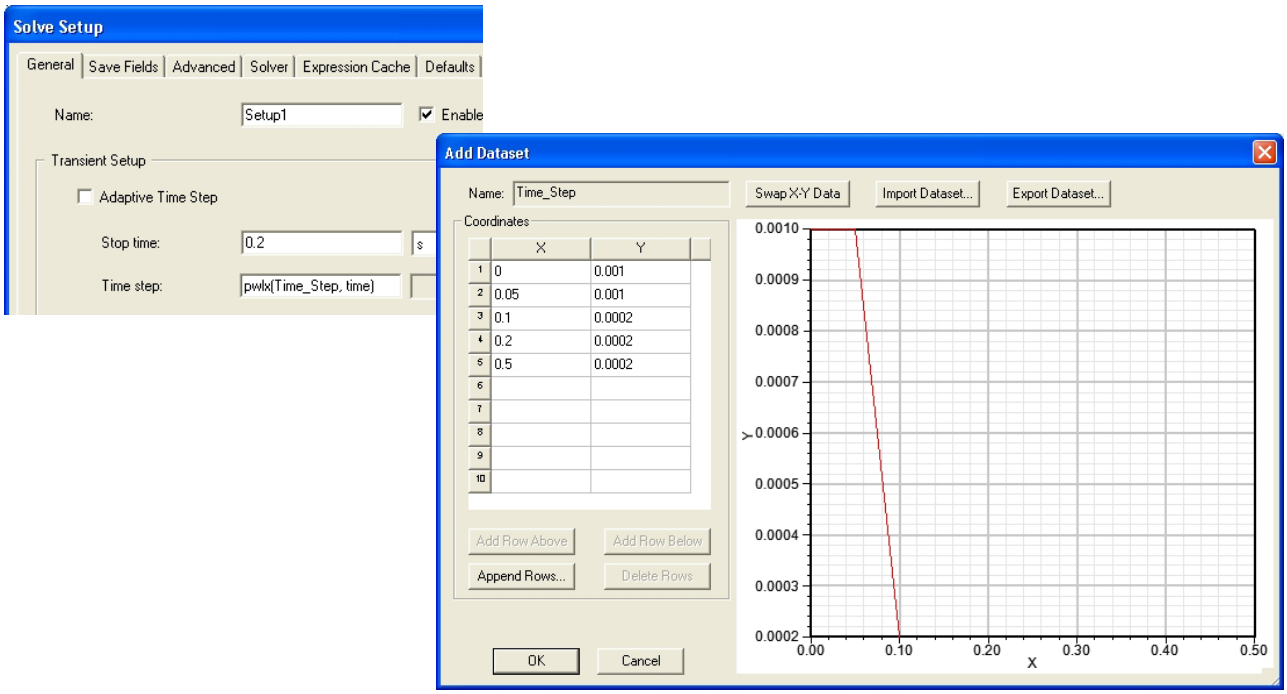




4. Modifying Maxwell 2D Design

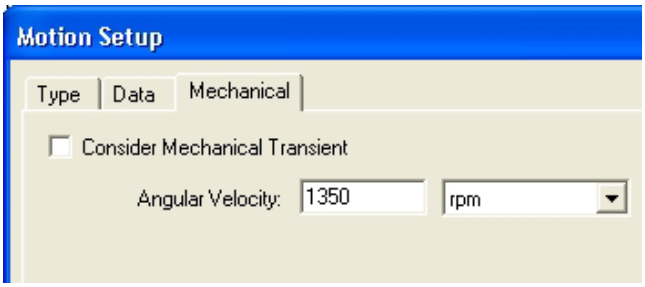
Setup Variable Simulation Time Step

- **Copy Design**
  - Select the design **"5_Nominal2"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Change the name of the design to **"6_Modified"**
- **Specify Variable Time Step**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Time Step: specify as **pwlx(Time_Step, time)**
    - Press **OK**
  - In Add Dataset window,
    - Specify X and Y Coordinates as shown in below image
    - Press **OK**



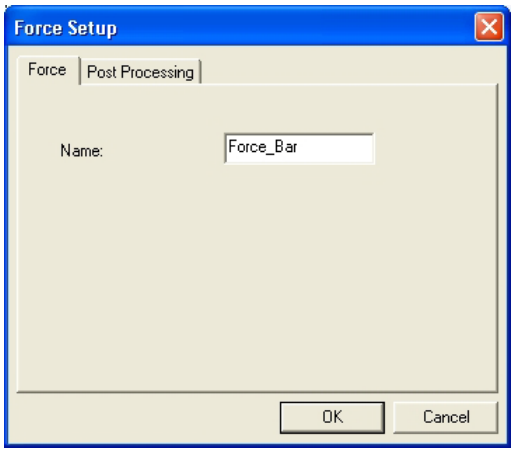
Change Angular Velocity

- **To Change Angular Velocity**
  - Expand the Project Manager tree to view **Model**
  - Double click on the tab **MotionSetup1** to modify motion
  - In Motion Setup window,
    - **Mechanical** tab
      - Angular Velocity: **1350 rpm**
    - Press **OK**



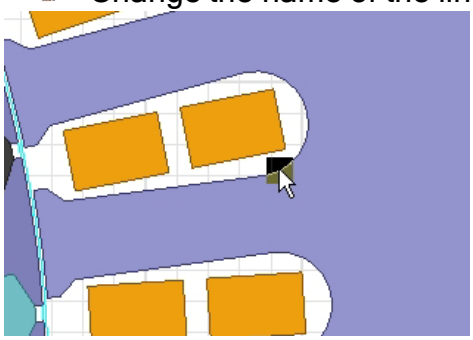
Set Force Calculation

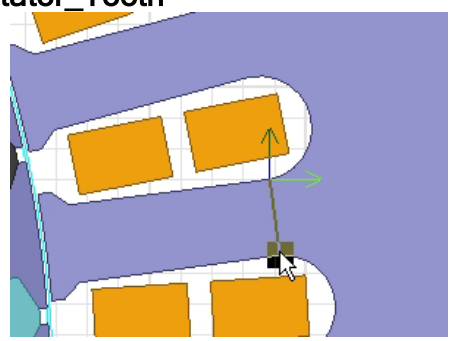
- **To Set Force Parameter**
  - Select the sheet **Bar** from history tree
  - Select the menu item *Maxwell 2D > Parameters > Assign > Force*
  - In Force Setup window,
    - Name: **Force_Bar**
    - Press **OK**
- There is no need to assign Torque calculation, moving torque is assigned automatically in Transient Solver with rotational motion
- You can assign as many Force calculations as you may need.



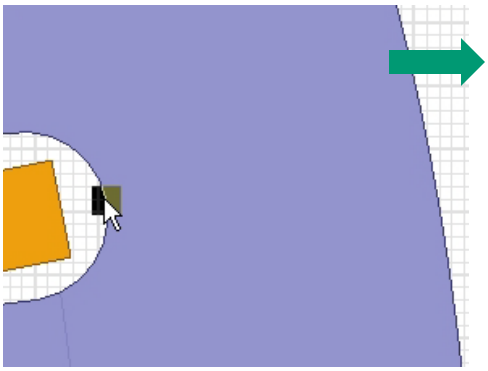
Create Output Variables for Flux Densities

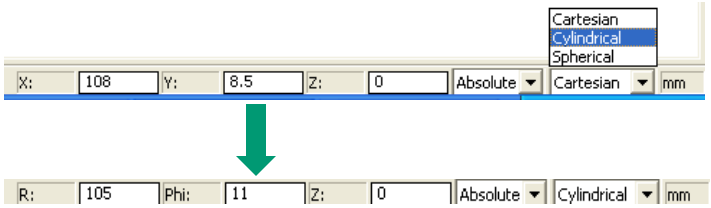
- **Create Line**
  - Select the menu item *Draw > Line*
  - Select the point from the graphic window to create a line as shown in below image
  - Right click and select **Done** to complete drawing
  - Change the name of the line to **Stator_Tooth**

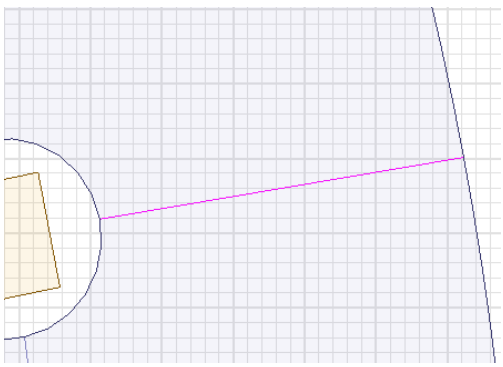




- **Create second Line**
  - Select the menu item *Draw > Line*
  - Select the first point from the graphic window as shown in below image
  - Change definition of the Coordinate Entry Field to **Cylindrical**

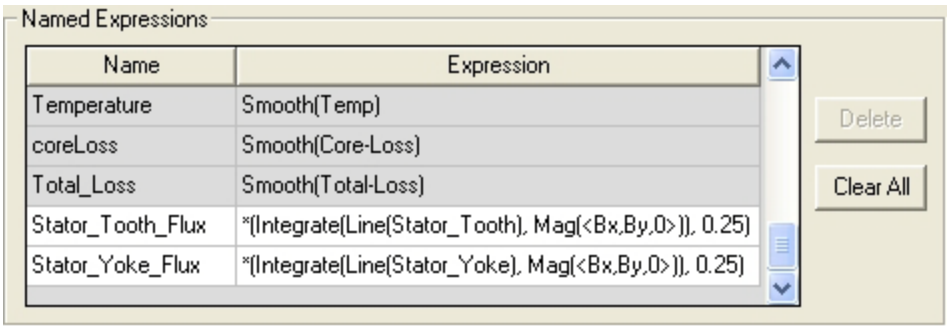






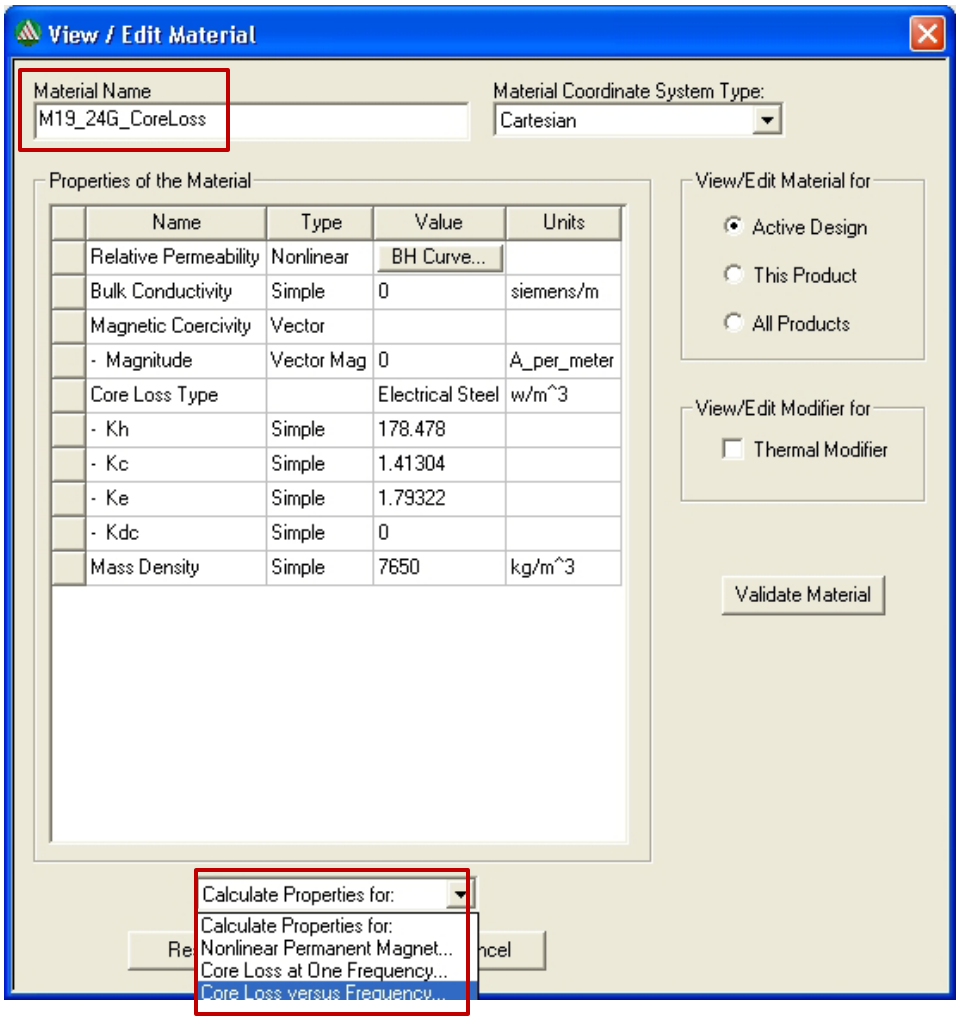
  - Enter the Coordinates as below
    - R: **105,** Phi: **11,** Z: 0, Press **Enter**
  - Press **Enter** again to exist the drawing mode
  - Change the name of the line to **Stator_Yoke**

- **Create Calculator Parameters**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Fields Calculator window,
    - Select **Input > Quantity > B**
    - Select **Vector > Mag**
    - Select **Input > Geometry**
      - Select the radio button to **Line**
      - Select the line **Stator_Tooth** from the list
      - Press **OK**
    - Select **Scalar > Integrate**
    - Select **Input > Number**
      - Type: **Scalar**
      - Value: **0.25** (stator stack length in meters)
      - Press **OK**
    - Select **General > ***
    - Select the button **Add...**
      - Specify the name as **Stator_Tooth_Flux**
      - Press **OK**
  - Similarly create flux parameter using the line **Stator_Yoke** and name the expression as **Stator_Yoke_Flux**



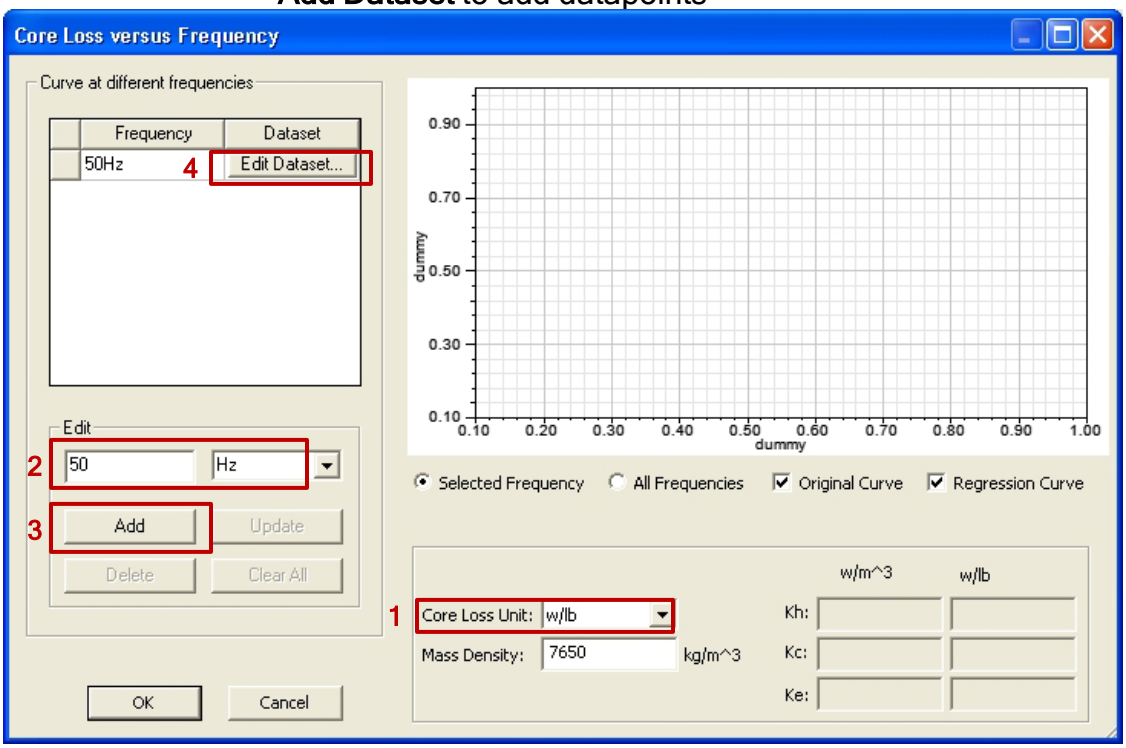
Set Core Loss Parameters

- **To Set Core Loss Parameters**
  - Press **Ctrl** and select the sheets **Stator** and **Rotor,** right click and select **Assign Material**
  - In Select Definition window,
    - Ensure that the material **M19_24G_2DSF0.920** is selected
    - Select the button **Clone Material**
    - In View/Edit Material window,
      - Name: **M19_24G_CoreLoss**
      - Select the option at the bottom Calculate Properties for **Core Loss versus Frequency**



Motors – Three Phase Induction Machine

- In Core Loss versus Frequency window,
  - Set Core Loss units to **w/lb**
  - In Edit field set frequency to **50 Hz** and select the button **Add**
  - Specified frequency gets added to the list. Select the button **Add Dataset** to add datapoints



- In Edit Dataset window,
  - Add datapoints as shown in below image
  - Press **OK**
- Similarly add three more datasets for frequencies as shown below

50 Hz

Coordinates

	B (Tesla)	P
1	0	0
2	0.25	0.08
3	0.5	0.2
4	0.75	0.4
5	1	0.6
6	1.25	0.85
7	1.5	1.2
8	1.7	1.5
9		
10		

100 Hz

Coordinates

	B (Tesla)	P
1	0	0
2	0.5	0.4
3	1	1.1
4	1.5	3
5	1.7	4
6		
7		
8		
9		
10		

200 Hz

Coordinates

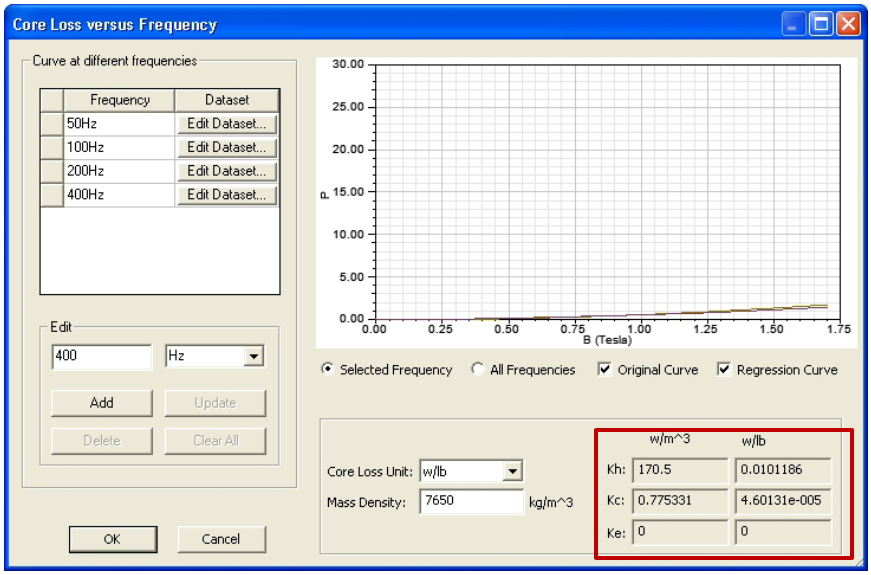
	B (Tesla)	P
1	0	0
2	0.5	1
3	0.75	2.2
4	1	3.75
5	1.25	5.75
6	1.5	8.75
7	1.7	12
8		
9		
10		

400 Hz

Coordinates

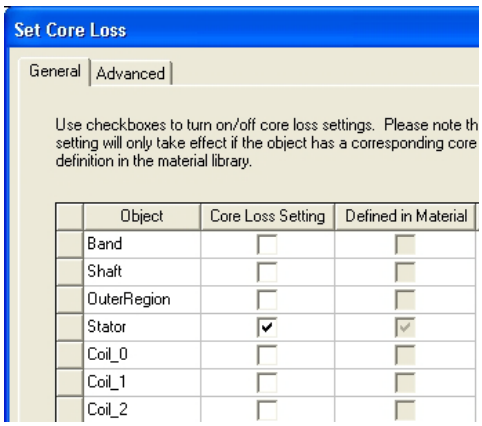
	B (Tesla)	P
1	0	0
2	0.25	1
3	0.5	3
4	0.75	6
5	1	11
6	1.25	17.5
7	1.5	26
8		
9		
10		

- In Core Loss versus Frequency window,
  - Core loss coeffecients are automatically calculated
  - Press **OK**



  - In View/Edit Material window, press **OK**
- Press **OK** to close Select Definition window.

- **Activate Core Loss**
  - Select the menu item *Maxwell 2D > Excitations > Set Core Loss*
  - In Set Core Loss window,
    - Check the Core Loss Settings for **Stator** and **Rotor**
    - Press **OK**



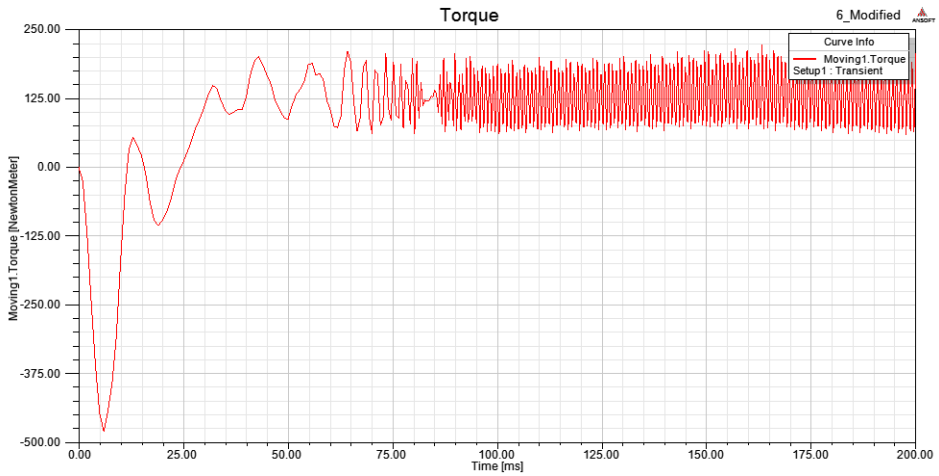
Analyze

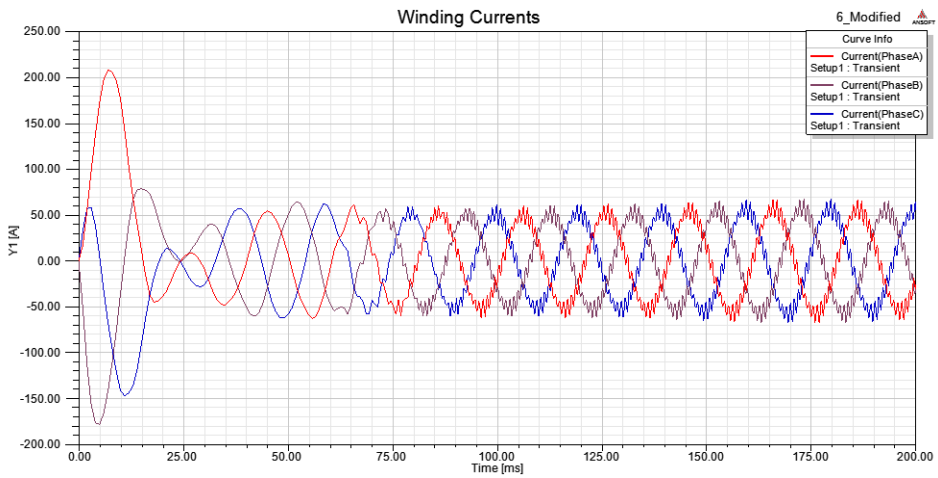
- **To Run the Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

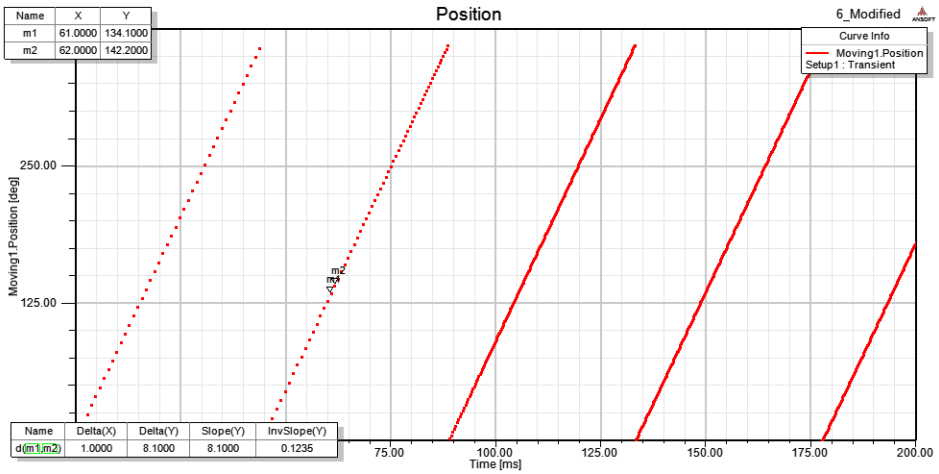
Results

View Existing Plots

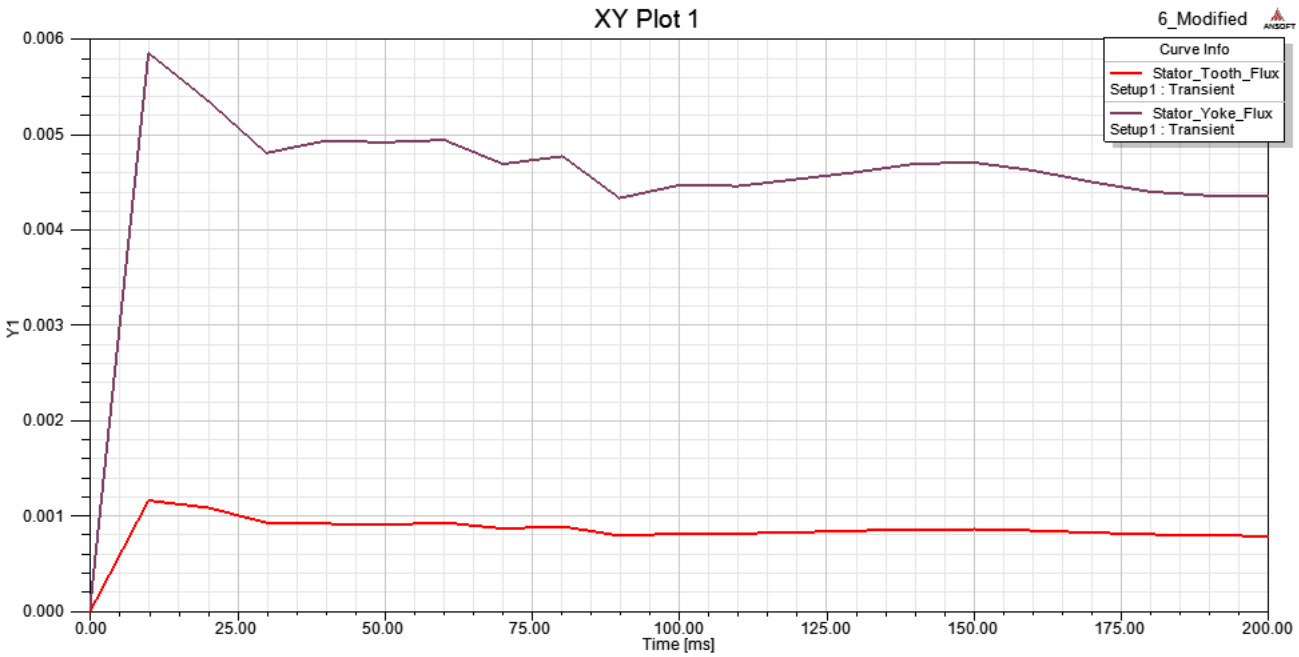
- torque, current, position vs. time, compare with results from Design 5







- **Plot Flux Density Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Field Report > Rectangular Plot*
  - In Report window,
    - Category: **Calculator Expressions**
    - Quantity: **Stator_Tooth_Flux**
    - Select **New Report**
    - Change Quantity to **Stator_Yoke_Flux**
    - Select **Add Trace**
    - Press **Close**



5. Locked Rotor Simulation

Modify Motion Setup

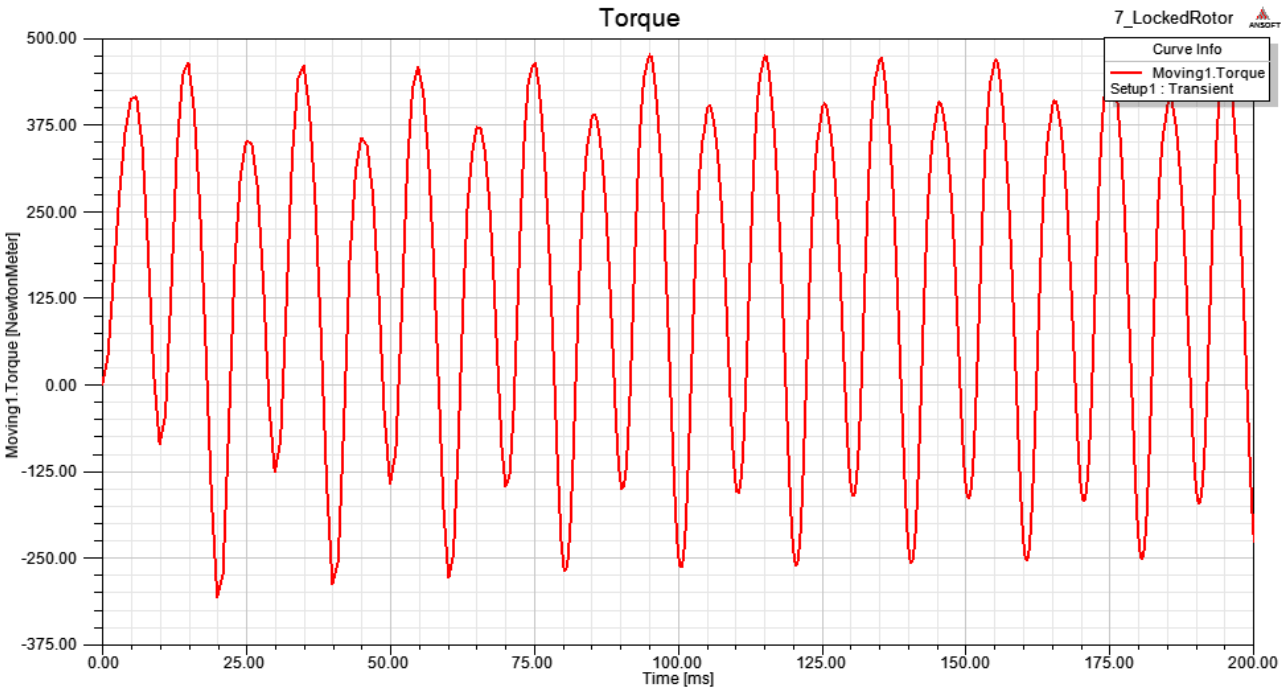
- **Copy Design**
  - Select the design **"6_Modified"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Change the name of the design to **"7_LockedRotor"**
- **To Modify Motion**
  - Expand the Project Manager tree to view **Model**
  - Double click on the tab **MotionSetup1** to modify motion
  - In Motion Setup window,
    - **Data** tab
      - Initial Position: **45 deg**
    - **Mechanical** tab
      - Angular Velocity: **0 rpm**
    - Press **OK**

Analyze

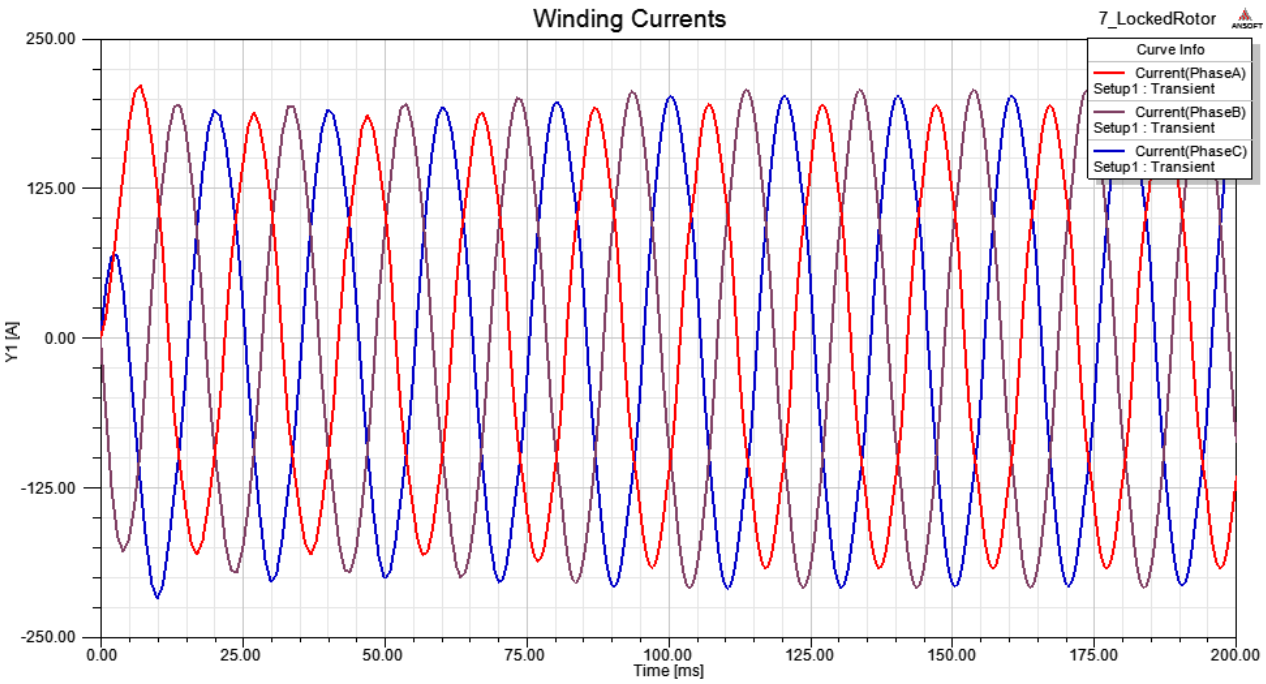
- **To Run the Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

Results

Torque Vs Time



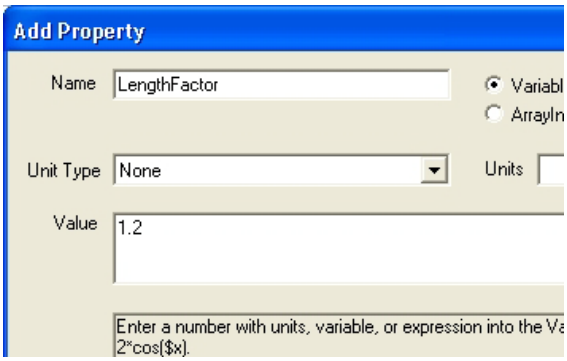
Current Vs Time



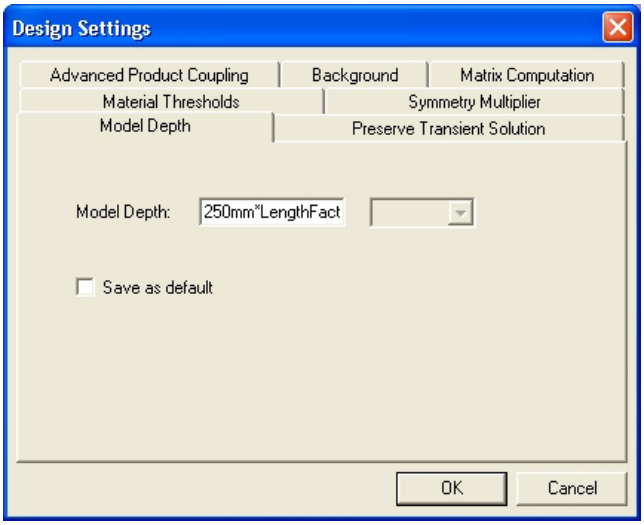
6. Design Sweep

Add Design Variables

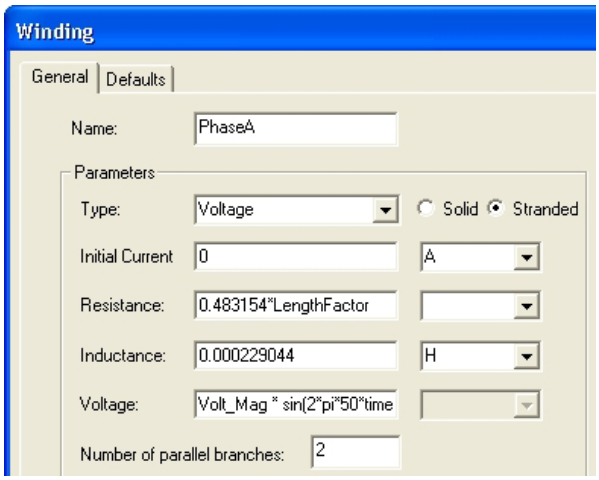
- **Copy Design**
  - Select the design **"6_Modified"** from Project Manager tree, right click and select **Copy**
  - Right click on the Project name in Project Manager tree and select **Paste**
  - Change the name of the design to **"8_DesignSweep"**
- **Add Design Variables**
  - Select the menu item *Maxwell 2D > Design Properties*
  - In Properties window, select **Add**
  - In Add Property window,
    - Name: **LengthFactor**
    - Value: **1.2**
    - Press **OK**
  - Press **OK** to close Properties window

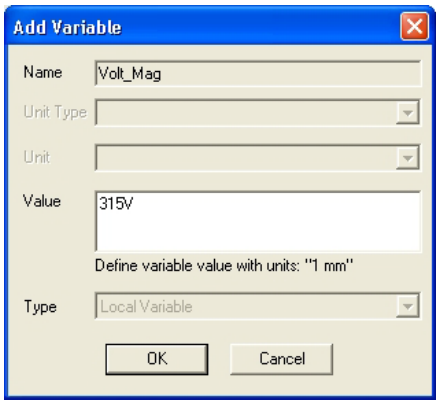


- **Assign variable**
  - Select the menu item *Maxwell 2D > Design Settings*
  - In Design Settings window,
    - **Model Depth** tab
      - Model Depth: **250mm*LengthFactor**
      - Press **OK**

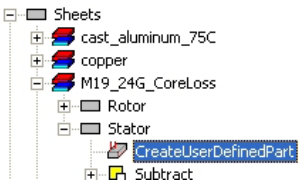


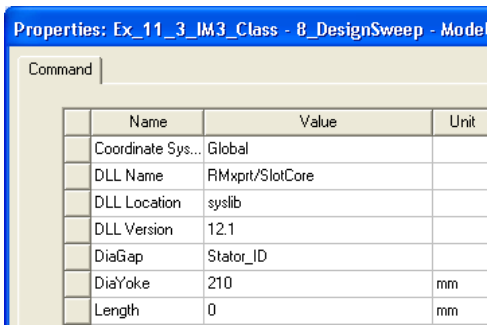
- **Add Variable to Excitation**
  - Expand the Project Manager tree to view **Excitations**
  - Double click on the tab **PhaseA** to modify its parameters
  - In Winding window,
    - Change resistance to **0.483154*LengthFactor**
    - Set peak voltage to **Volt_Mag** (Volt_Mag * sin(2*pi*50*time))
    - Press **OK**
  - In Add Variable window,
    - Set value to **315V**
    - Press **OK**





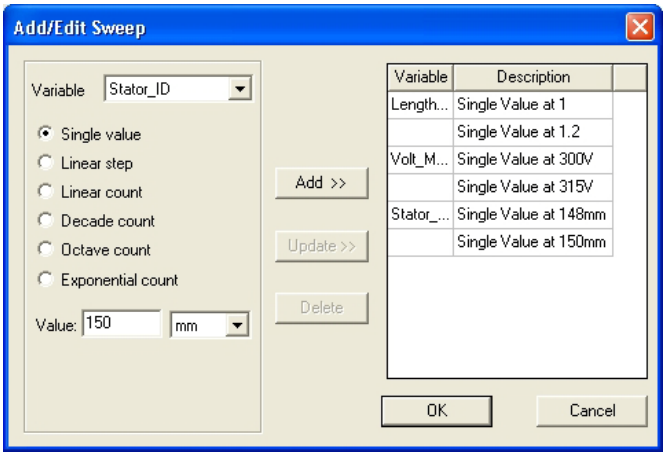
  - Similarly change Resistance and voltage for **PhaseB** and **PhaseC**
- **Assign Geometrical parameters**
  - Expand the History tree for **Stator**
  - Double click on the command **CraeteUserDefinedPart** to edit its properties
  - In Properties window,
    - Change DiaGap to **Stator_ID**
    - Press **OK**
  - In Add Variable window,
    - Set value to **150 mm**
    - Press **OK**

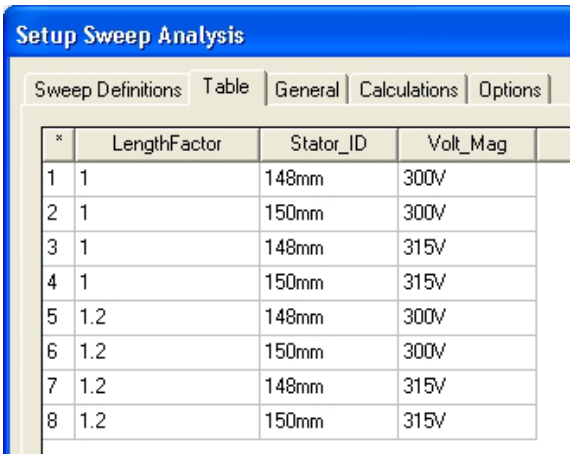




Set Parametric Sweep

- **To Set Parametric Sweep**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Add Parametric*
  - In Setup Sweep Analysis window, select **Add**
  - In Add/Edit Sweep window,
    - Variable: **LengthFactor**
    - Select **Single Value**
    - Set value to **1**
    - Select **Add>>**
    - Change Value to **1.2**
    - Select **Add>>**
    - Change Variable to **Volt_Mag**
    - Set value to **300V**
    - Select **Add>>**
    - Change Value to **315V**
    - Select **Add>>**
    - Change Variable to **Stator_ID**
    - Set value to **148mm**
    - Select **Add>>**
    - Change Value to **150mm**
    - Select **Add>>**
    - Select **OK**
  - In Setup Sweep Analysis window, select **Table** tab to view all variations
    - Press **OK**





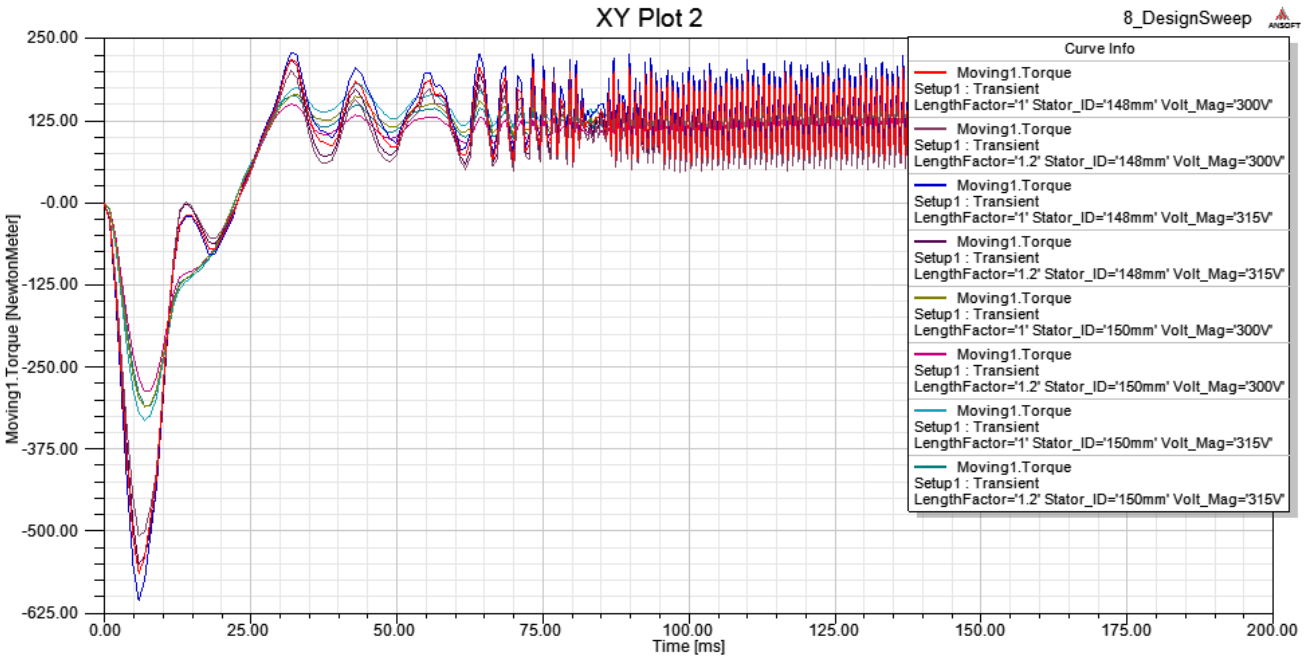
*	LengthFactor	Stator_ID	Volt_Mag
1	1	148mm	300V
2	1	150mm	300V
3	1	148mm	315V
4	1	150mm	315V
5	1.2	148mm	300V
6	1.2	150mm	300V
7	1.2	148mm	315V
8	1.2	150mm	315V

Run Parametric Analysis

- **To Run Parametric Setup**
  - Expand the Project Manager tree to view **Optimetrics**
  - Right click on **ParametricSetup1** and select **Analyze**

Results

- **Plot Torque Vs Time for all Variations**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In Report window,
    - Category: **Torque**
    - Quantity: **Moving1.Torque**
    - Select **New Report**
  - By default, all variations will be plotted on the same Plot



Save File

- Save the file at appropriate location.

7. Drive Design

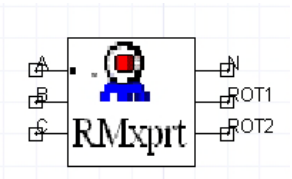
- **Note:** Simplorer V9 or higher is needed to perform this section

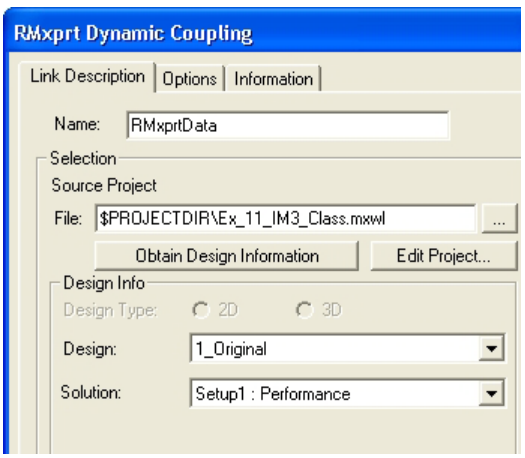
Launch Simplorer

- **To Launch Simplorer V10**
  - Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Simplorer 10.0** and select **Simplorer 10.0**
- **Change Design Name**
  - Change the name of the design to **"1_Link_with_RMxprt"**
- **Save File**
  - Select the menu item ***File > Save***
  - Save the file with the name **"IM3_Drive_withLoad.asmp"**

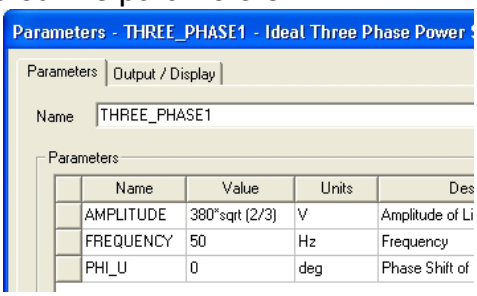
Add Components

- **Add RMxprt Component**
  - Select the menu item ***Simplorer Circuit > SubCircuit > Add RMxprt Dynamic Component***
  - In RMxprt Dynamic Coupling window,
    - Source File: Browse to the Maxwell Project File that was saved in last step
    - Design: Select **1_Original**
    - Press **OK**
  - Place the RMxprt component on Project Page
  - Press **Esc** to exit component addition

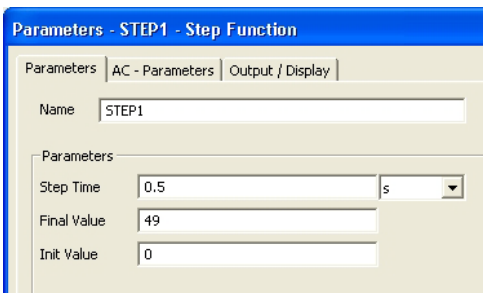




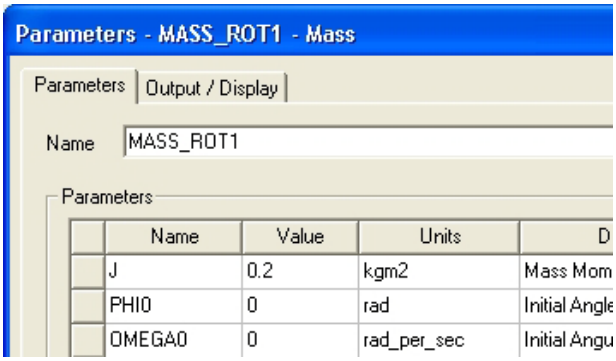
- **Add Three Phase Source**
  - Select the tab **Components** at the bottom of Project Manager window
  - Expand the tree to view **Simplorer Elements > Multiphysics > Power System > Power Source**
  - Drag and drop **THREE_PHASE: Ideal Three Phase Power Supply** on the project page
  - Press **Esc** to exit component addition
  - Double click on added component to edit its parameters
  - In Parameters window,
    - Amplitude: **380*sqrt (2/3)**
    - Press **OK**



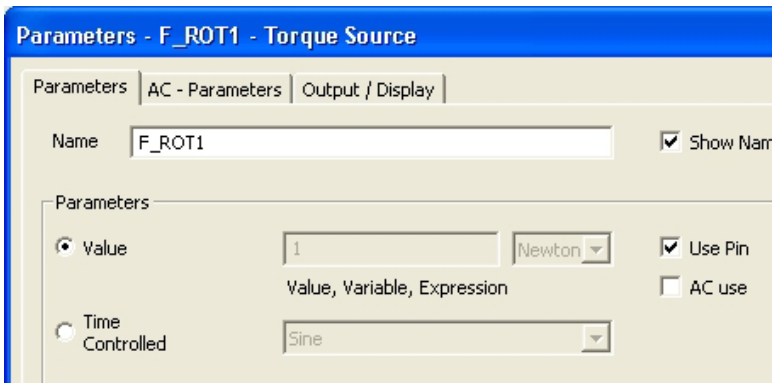
- **Add Other Components**
  - Similarly add more components as give below
  - Add **Ammeter** from
    - **Simplorer Elements > Basic Elements > Measurement > Electrical > Ammeter**
  - Add **Voltmeter** from
    - **Simplorer Elements > Basic Elements > Measurement > Electrical > Voltmeter**
  - Add **Torque Meter** from
    - **Simplorer Elements > Basic Elements > Measurement > Mechanical > Velocity-Force -Representation >Rotational_V > FM_ROT: Torque Meter**
  - Add **Step Function** from
    - **Simplorer Elements > Basic Elements >Blocks > Sources Blocks > STEP: Step Function**
    - Double click on the component to edit its Properties
      - Step Time: **0.5 s**
      - Final value: **49**
      - Press **OK**



- Add **Rotational Mass** from
  - **Simplorer Elements > Basic Elements > Physical Domains > Mechanical > Velocity-Force -Representation >Rotational_V > MASS_ROT: Mass**
  - Double click on the component to edit its Properties
    - J : **0.2 kgm^2**
    - Press **OK**



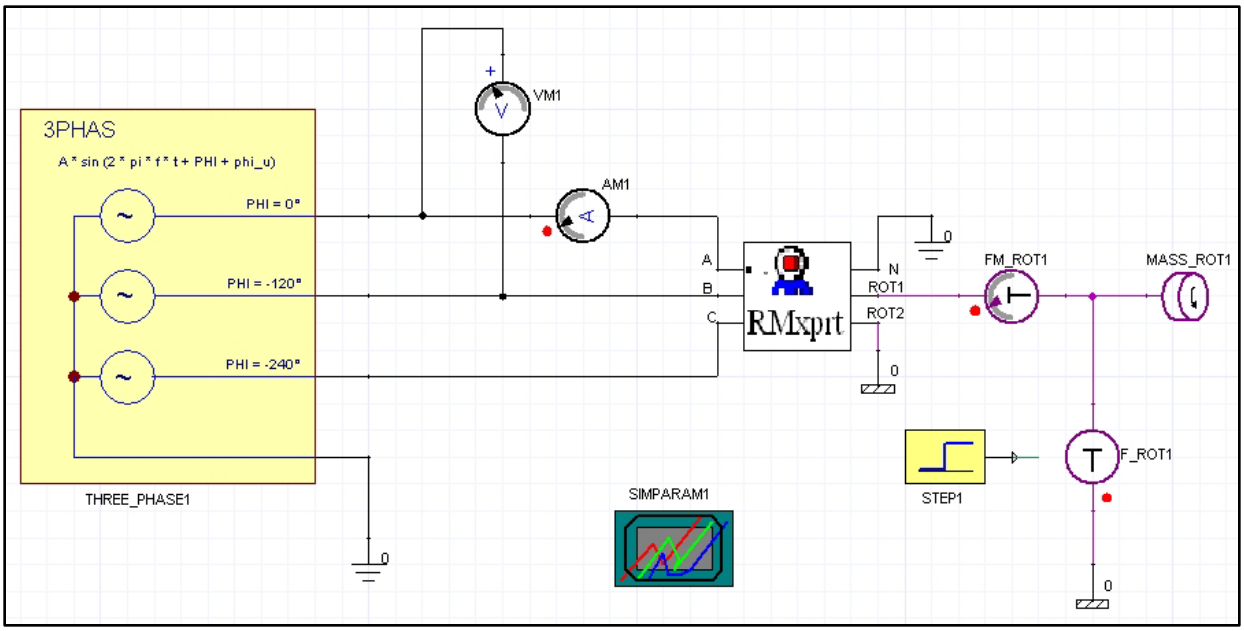
- Add **Torque Source** from
  - **Simplorer Elements > Basic Elements > Physical Domains > Mechanical > Velocity-Force -Representation >Rotational_V > F_ROT: Torque Source**
  - Double click on the component to edit its Properties
    - Use Pin: ☑ **Checked**
    - Press **OK**



- Add **Simulator Parameter Display** from
  - **Simplorer Elements > Basic Elements >Tools > Simulator Parameters > SIMPARAM: Display of Simulator Parameters**

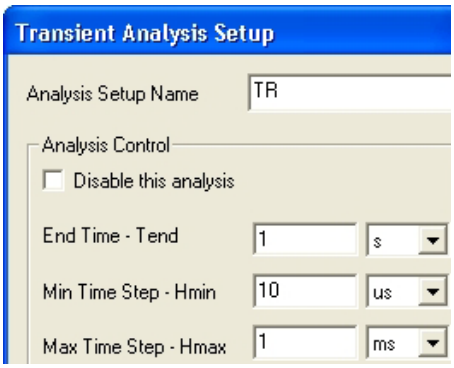
Build Circuit

- To Build Circuit
  - Position the components and join them using wires
  - Add **Ground** from *Draw > Ground*



Analysis Setup

- To Define Analysis Setup
  - Select the tab **Project** at the bottom of Project Manager window
  - Double click on the tab **TR** to modify analysis settings
  - In Transient Analysis Setup window,
    - Tend : **1 s**
    - Hmin : **10 us**
    - Hmax: **1 ms**
    - Press **OK**

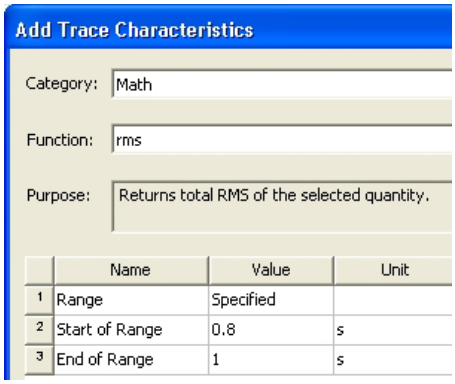


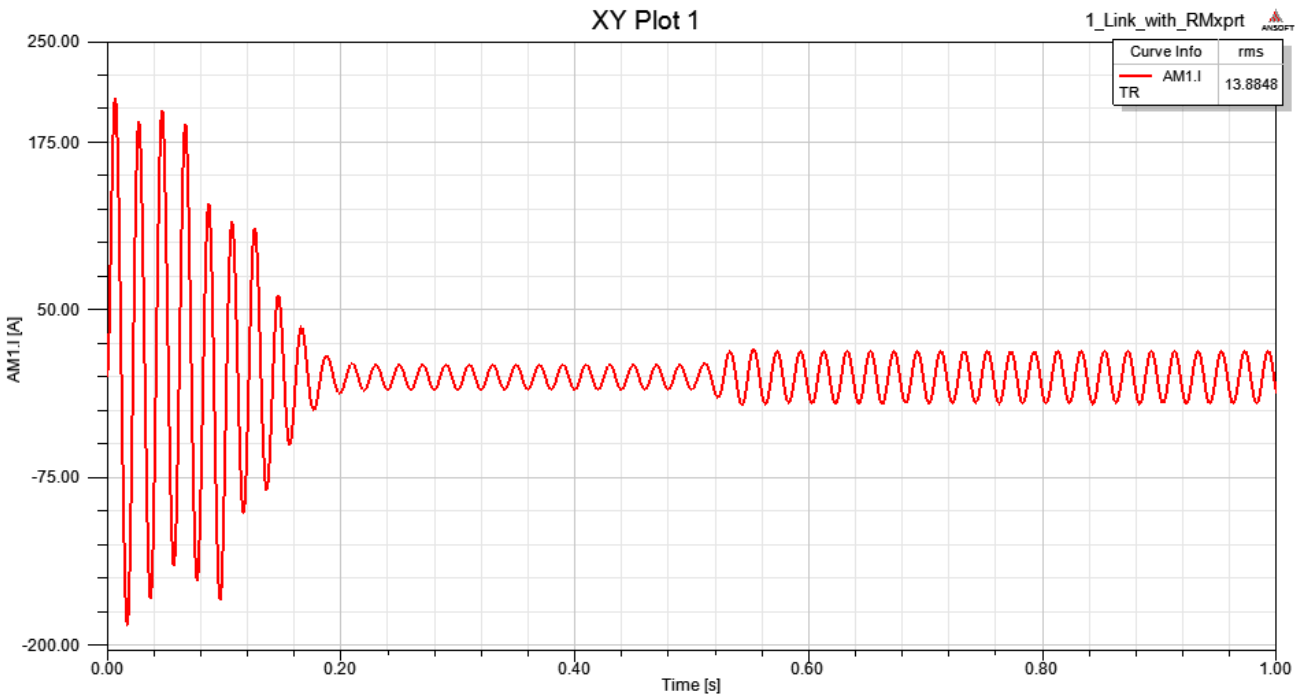
Analyze

- To Start Analysis
  - Select the menu item *Simplorer Circuit > Analyze*

Results

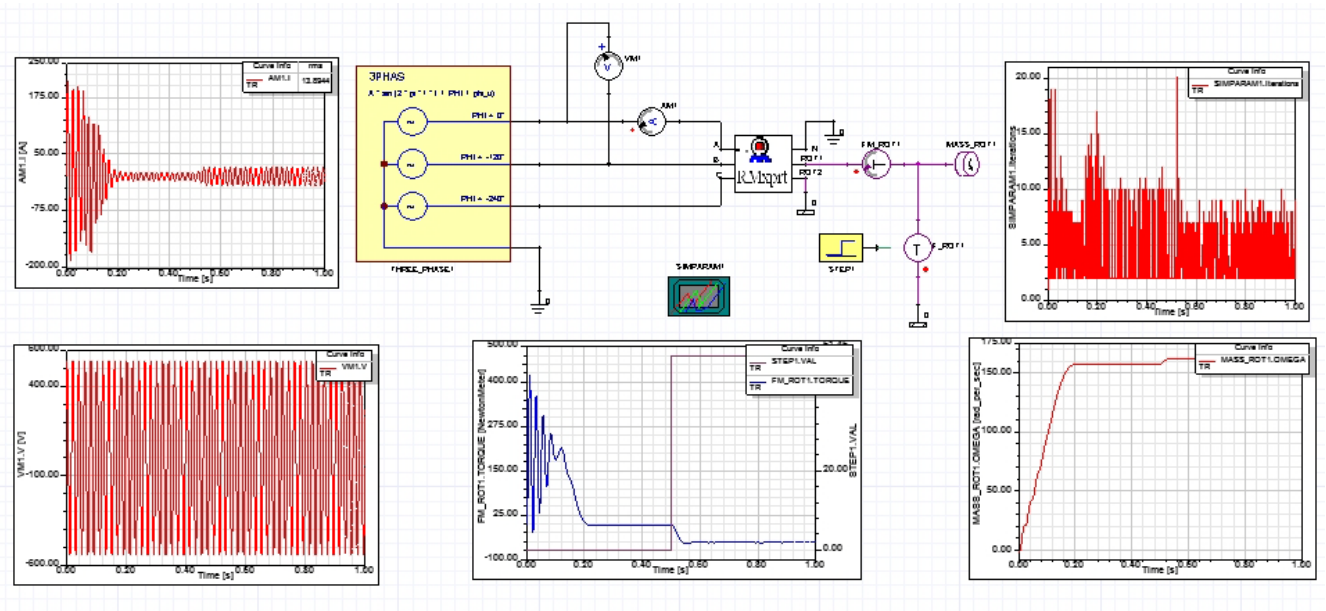
- **Plot AM1.I Vs Time**
  - Select the menu item *Simplorer Circuit > Create Standard Report > Rectangular Plot*
  - In Report window,
    - Category: **All**
    - Quantity: **AM1.I**
    - Select **New Report**
  - Right click on the Plot and select **Trace Characteristics > Add**
  - In Add Trace Characteristics window,
    - Category: **Math**
    - Function: **rms**
    - Range: **Specified**
      - Start of Range: **0.8 s**
      - End of Range: **1 s**
      - Select **Add** and **Done**





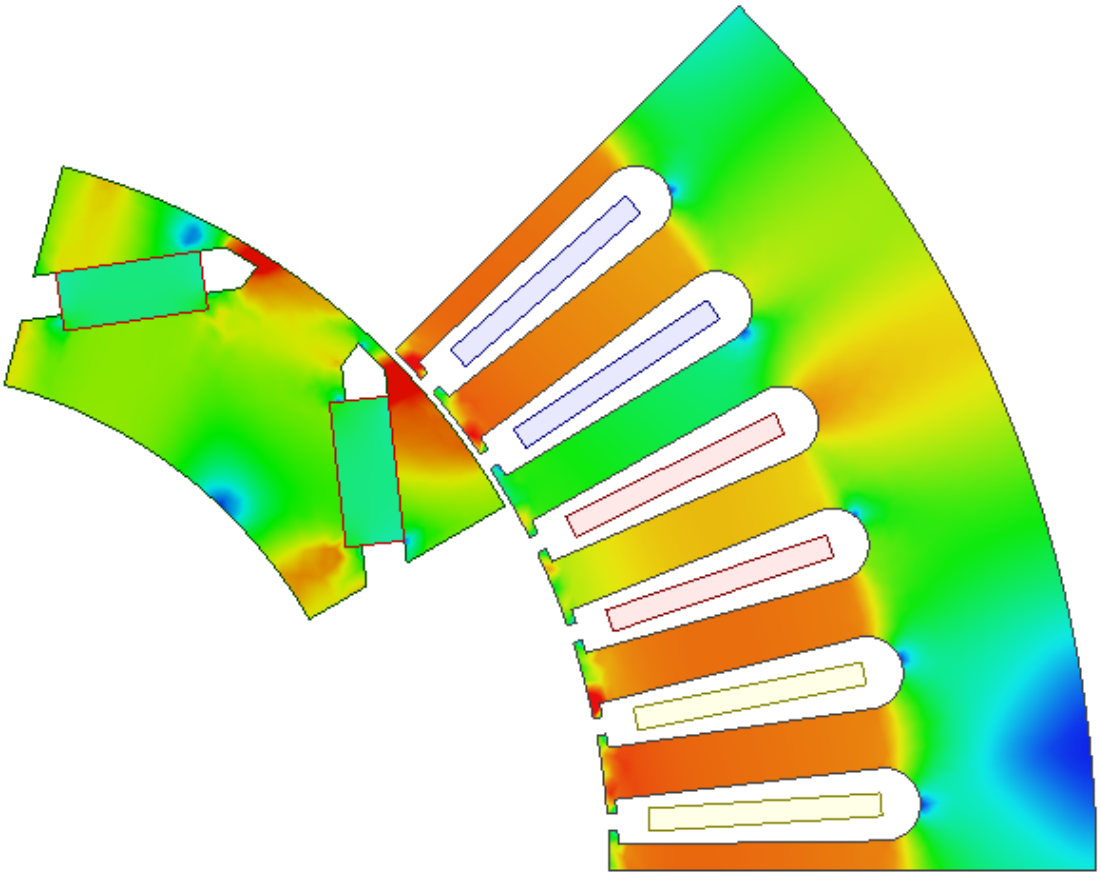
Motors - Three Phase Induction Machine

- Similarly create other plots like VM1.V, FM_ROT1.TORQUE, STEP1.VAL, MASS_ROT1.OMEGA (in rad/sec) and SIMPARAM1.Iterations.



- **This completes the one day training course on Three-phase Induction Machine using ANSYS RMxprt, Maxwell 2D and Simplorer**

Study of a Permanent Magnet Motor with MAXWELL 2D: Example of the 2004 Prius IPM Motor

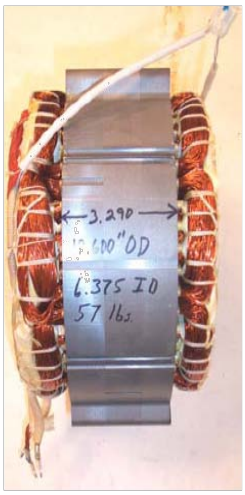


Motors - Permanent Magnet Motor (Prius IPM)

Study of an electrical machine

- The Electro Mechanical software package provided by Ansoft enables extensive electrical malchinesimulation. This application note details the simulaton of an electrical machine with Maxwel2D. 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
  - Creating the 3D Model
  - Reducing the size of the 3D Model
  - Material properties of the machine
  - Applying Master/Slave Boundary Condition
- STATIC ANALYSIS
- DYNAMIC ANALYSIS
- COGGING TORQUE

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 :
  - Select the menu item *Tools > Options > Maxwell 2D 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
  - 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

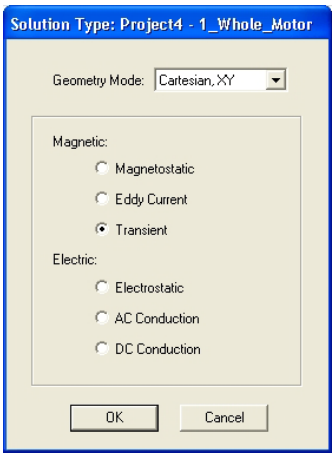
Motors - Permanent Magnet Motor (Prius IPM)

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 2D Design***, or click on the icon
- **Rename Design**
  - Right click on the design Maxwell2D Design1 and select **Rename**
  - Change the name of the design to **"1_Whole_Motor"**

Set Solution Type

- **To set the Solution Type:**
  - Select the menu item ***Maxwell 2D > Solution Type***
  - Solution Type Window:
    1. Geometry Mode: **Cartesian, XY**
    2. Choose **Magnetic > Transient**
    3. Click the **OK** button



Set Default Units

- **To Set Default Units**
  - Select the menu item ***Modeler > Units***
  - In Units window,
    - Select units: **mm**
    - Press **OK**



Creating the 2D 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

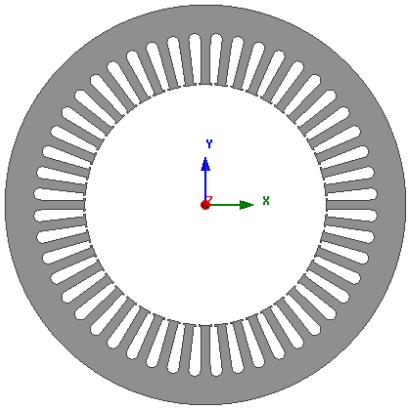
Parameters | Info

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	0	mm	0mm	Core length
Skew	0	deg	0deg	Skew angle in core len...
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 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...
HalfSlot	0		0	0 for symmetric slot, 1 f...
SegAngle	0	deg	0deg	Deviation angle for slot ...
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 100: region.

Show Hidden

Motors - Permanent Magnet Motor (Prius IPM)

- **Change Attributes**
  - Change the name of the sheet to **Stator**
  - Change the color of the sheet 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

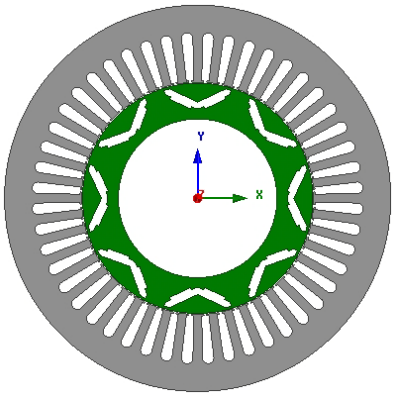
Parameters | Info

	Name	Value	Unit	Evaluated Value	Description
	DiaGap	160.4	mm	160.4mm	Core diameter on gap sid...
	DiaYoke	110.64	mm	110.64mm	Core diameter on yoke sid...
	Length	0	mm	0mm	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 du...
	O1	0	mm	0mm	Bottom width for separate...
	O2	7.28	mm	7.28mm	Distance from duct botto...
	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 betwee...
	ThickMag	6.48	mm	6.48mm	Magnet thickness
	WidthMag	32	mm	32mm	Total width of all magnet ...
	LenRegion	200	mm	200mm	Region length
	InfoCore	0		0	0: core; 1: magnets; 2: du...

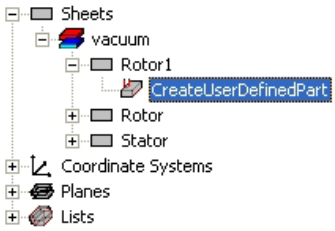
Show Hidden

Motors - Permanent Magnet Motor (Prius IPM)

- **Change Attributes**
  - Change the name of the sheet to **Rotor**
  - Change the color of the sheet to **Green**



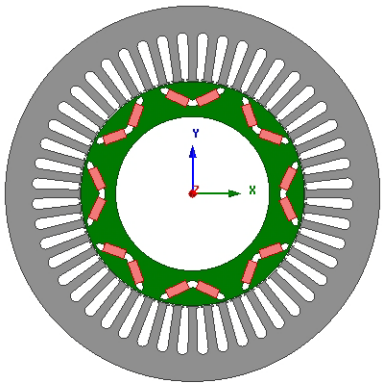
- **Create the Magnets**
  - Select the sheet **Rotor** from history tree
  - Select the menu item ***Edit > Copy***
  - Select the menu item ***Edit > Paste***
  - New sheet named **Rotor1** is created
  - Expand the history tree for **Roto1**



  - Double click on the command **CreateUserDefinedPart** to edit properties
  - In Properties window,
    - Change the value for InfoCore to **1** (magnets)
    - Press **OK**

	HRib	3	mm	3mm	Rib height
	DminMag	4.5	mm	4.5mm	Minimum distance bet
	ThickMag	6.48	mm	6.48mm	Magnet thickness
	WidthMag	32	mm	32mm	Total width of all magı
	LenRegion	200	mm	200mm	Region length
	InfoCore	1		1	0: core; 1: magnets; 2

- **Change Attributes**
  - Change the name of the sheet to **Magnet**
  - Change the color of the sheet to **Light Red**



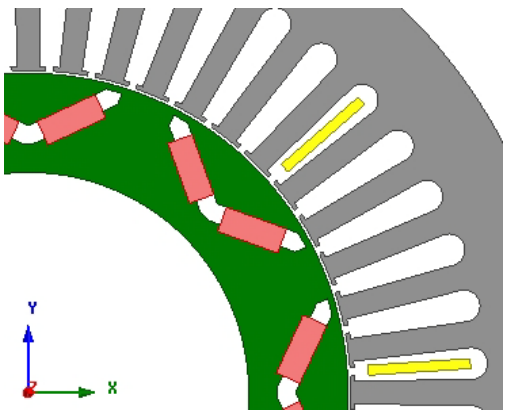
- **Create Windings**
  - A User Defined Primitive will be used to create the windings.
  - Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > LapCoil*
  - Use the values given in the panel below to create the coil

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	0	mm	0mm	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	1.28	mm	1.28mm	Slot opening height
Hs1	0.1	mm	0.1mm	Slot wedge height
Hs2	29.7	mm	29.7mm	Slot body height
Bs0	2.11	mm	2.11mm	Slot opening width
Bs1	3.14	mm	3.14mm	Slot wedge maximum w...
Bs2	5.65	mm	5.65mm	Slot body bottom width,...
Rs	2.5	mm	2.5mm	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	5	mm	5mm	One-side end extended...
SpanExt	18	mm	18mm	Axial length of end spa...
SegAngle	10	deg	10deg	Deviation angle for end...
LenRegion	200	mm	200mm	Region length
InfoCoil	1		1	0: winding; 1: coil; 2: te...

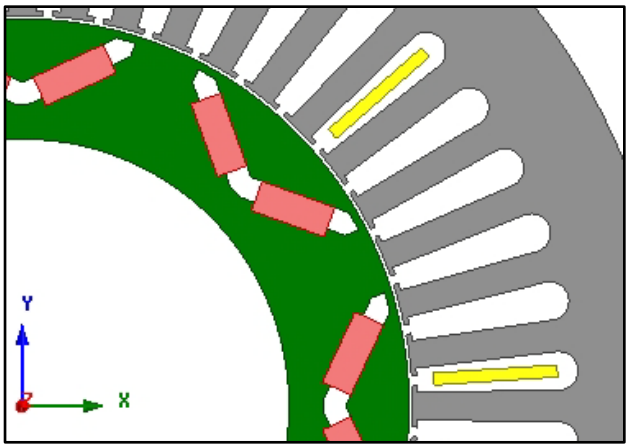
Show Hidden

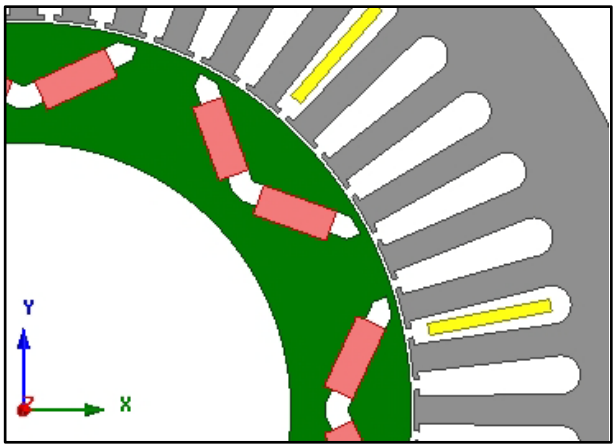
- **Change Attributes**
  - Change the material of the sheet to **Copper**
  - Change the color of the sheet to **Yellow**



Motors - Permanent Magnet Motor (Prius IPM)

- **Rotate Coil**
  - Select the sheet **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 sheet **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**



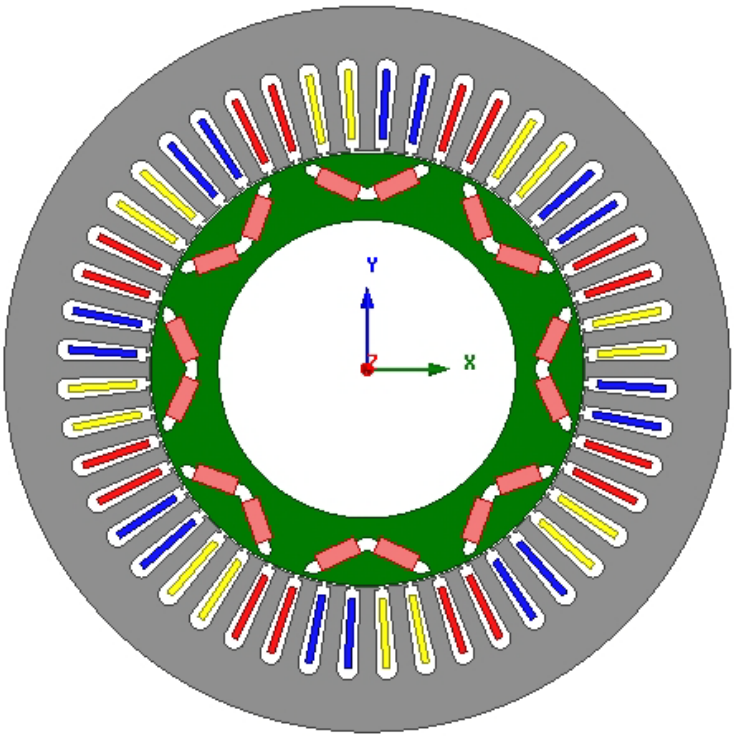
- **Change Attributes**
  - Change the name of the sheets LapCoil1, LapCoil1_1 and LapCoil1_2 to **PhaseA, PhaseC,** and **PhaseB** respectively
  - Change the color of the sheets PhaseB and Phase C to **Blue** and **Red**

Motors - Permanent Magnet Motor (Prius IPM)

- **Duplicate Coils**
  - Press **Ctrl** and select the sheets **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**



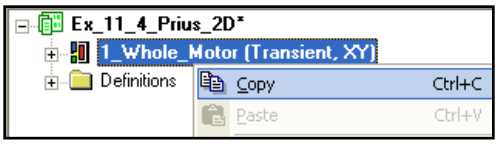
- Motor geometry is complete

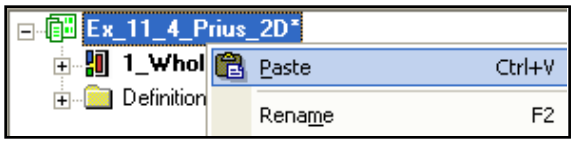


Motors - Permanent Magnet Motor (Prius IPM)

Copy Design

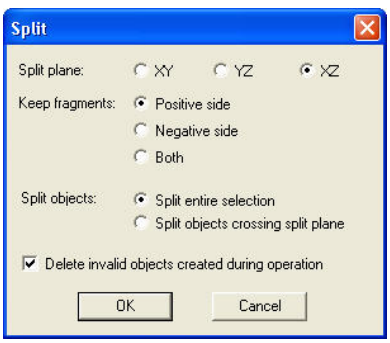
- **To Copy Design**
  - Select the Maxwell design **"1_Whole_Motor"** from the Project Manager tree, right click and select **"Copy"**
  - Select the project name from the Project manager tree, right click and select **"Paste"**
  - Change the name of the design to **"2_Partial_Motor"**



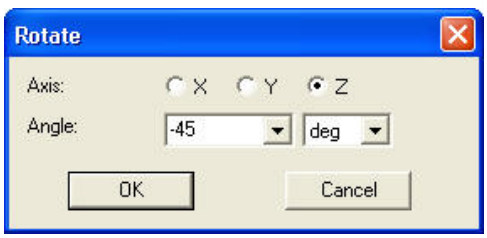


Reduce Model Size

- 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.
- **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**

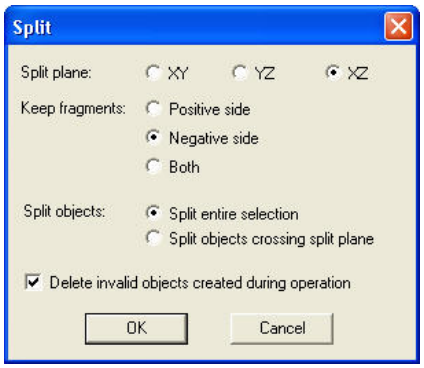


- **Rotate Model**
  - We have achieved half model now
  - Select the menu item *Edit > Select All*
  - Select the menu item *Edit > Arrange > Rotate*
  - In Rotate window,
    1. Axis: **Z**
    2. Angle: **-45 deg**
    3. Press **OK**

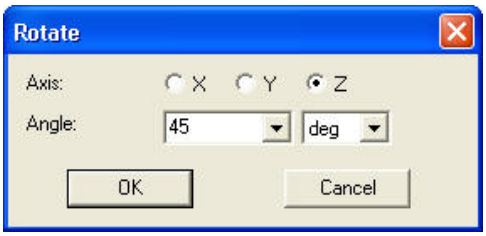


Motors - Permanent Magnet Motor (Prius IPM)

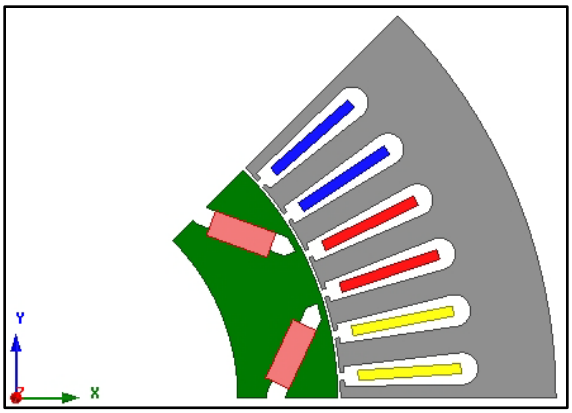
- **Split Model by XZ**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Modeler > Boolean > Split*
  - In Split window,
    1. Split Plane: **XZ**
    2. Keep fragments: **Negative Side**
    3. Split Objects: **Split entire Selection**
    4. Press **OK**



- **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**



- **Reduced Model**
  - The 3D model now looks like below



- **Change Attributes**
  - Change the name of the sheets PhaseA and PhaseA_7 to **PhaseA1** and **PhaseA2**
  - Change the name of the sheets PhaseB and PhaseB_7 to **PhaseB1** and **PhaseB2**
  - Change the name of the object PhaseC and PhaseC_7 to **PhaseC1** and **PhaseC2**

Create Region

Create Rectangle

- Select the menu item *Draw > Line*
  1. Using the coordinate entry field, enter the first vertex
     - **X: 0.0, Y: 0.0, Z: 0.0**, Press the **Enter** key
  2. Using the coordinate entry field, enter the second vertex
     - **X: 200.0, Y: 0.0, Z: 0.0**, Press the **Enter** key
  3. Press **Enter** a second time to finish the drawing

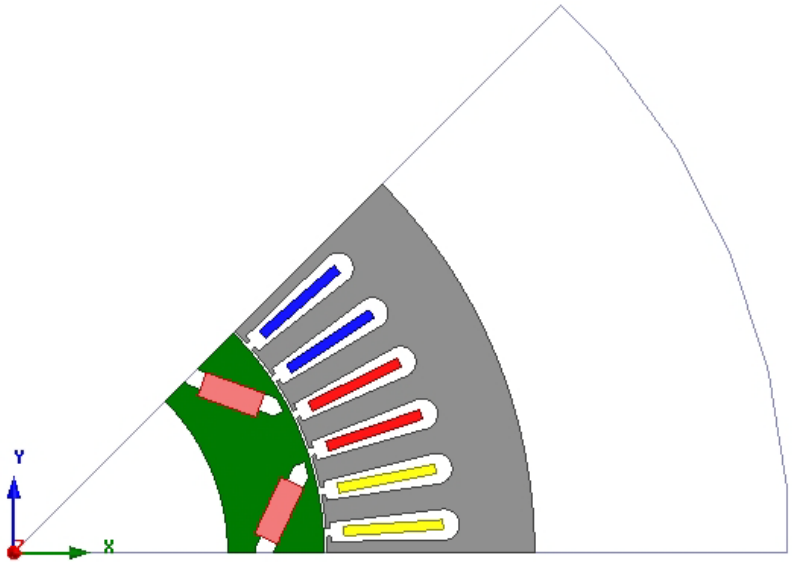
Sweep Line

- Select the line **Polyline1** from 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

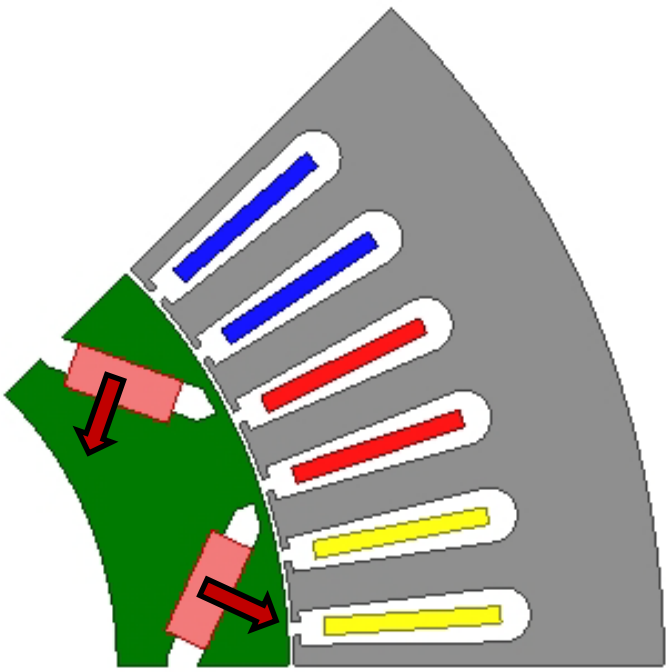
- Change the name of the resulting sheet to **Region**
- Select Display Wireframe: ☑ **Checked**



Material Properties for the Motor

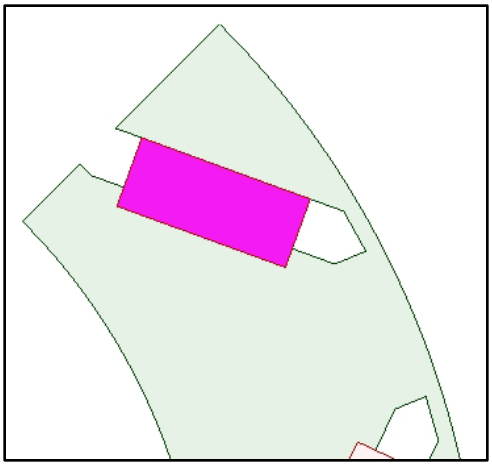
Permanent Magnet Characterization

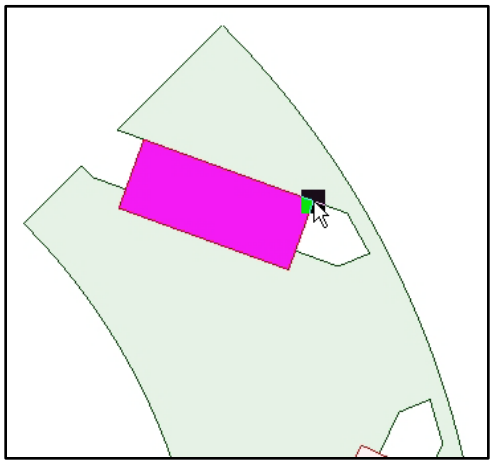
- The Prius Permanent Magnets (PMs) are high-strength magnets.
- In order to define PMs magnetization orientation, we need to create separate objects for each magnet.
- **Separate Objects**
  - Select the object **Magnets** from the history tree
  - Select the menu item *Modeler > Boolean > Separate 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.



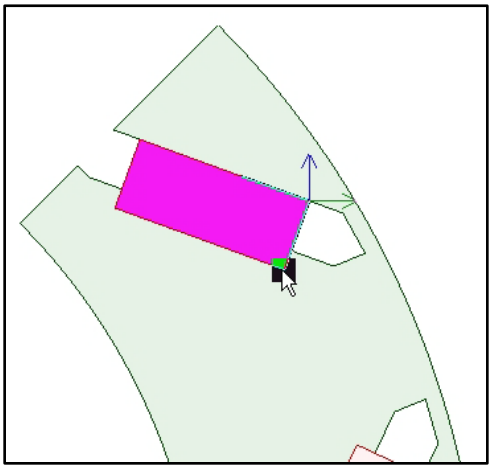
Motors - Permanent Magnet Motor (Prius IPM)

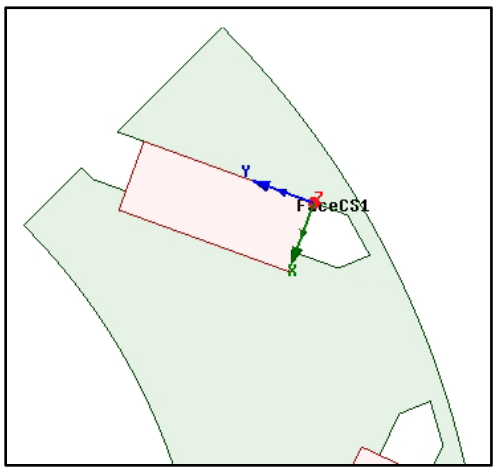
- **Hide All other Objects**
  - Select the menu item *View > Active View Visibility*
  - In the window,
    1. Disable the visibility for all objects except **Rotor, PM1** and **PM2**
    2. Press **Done**
- **To Create Face Coordinate System**
  - Select the menu item *Edit > Select > Faces* or press **"f"** from keyboard
  - Select the face of the sheet **PM1** as shown in below image
  - Select the menu item *Modeler > Coordinate System > Create > Face CS*
  - Select the vertex on the face of PM1 to define origin



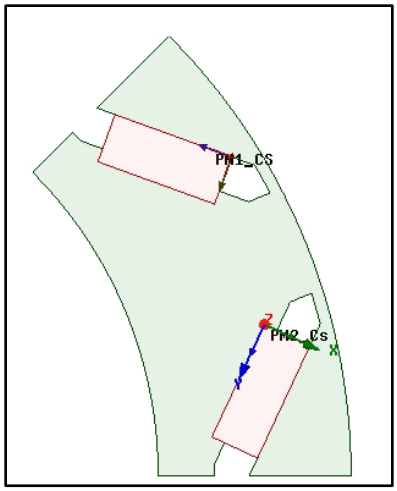


  - Select the vertex of the magnet as shown in below image to define X axis
  - A Face coordinate system will be created as shown in below image
  - Rename the FaceCS1 to **PM1_CS**



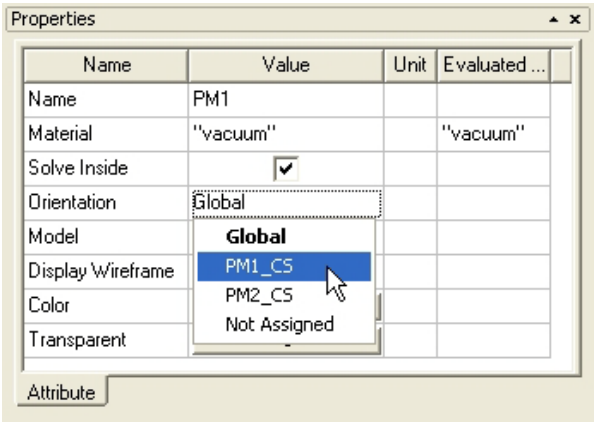


- Similarly Create a Face coordinate system for PM2 and name it as **PM2_CS.**
- Make sure to have the X axis looking toward the air gap



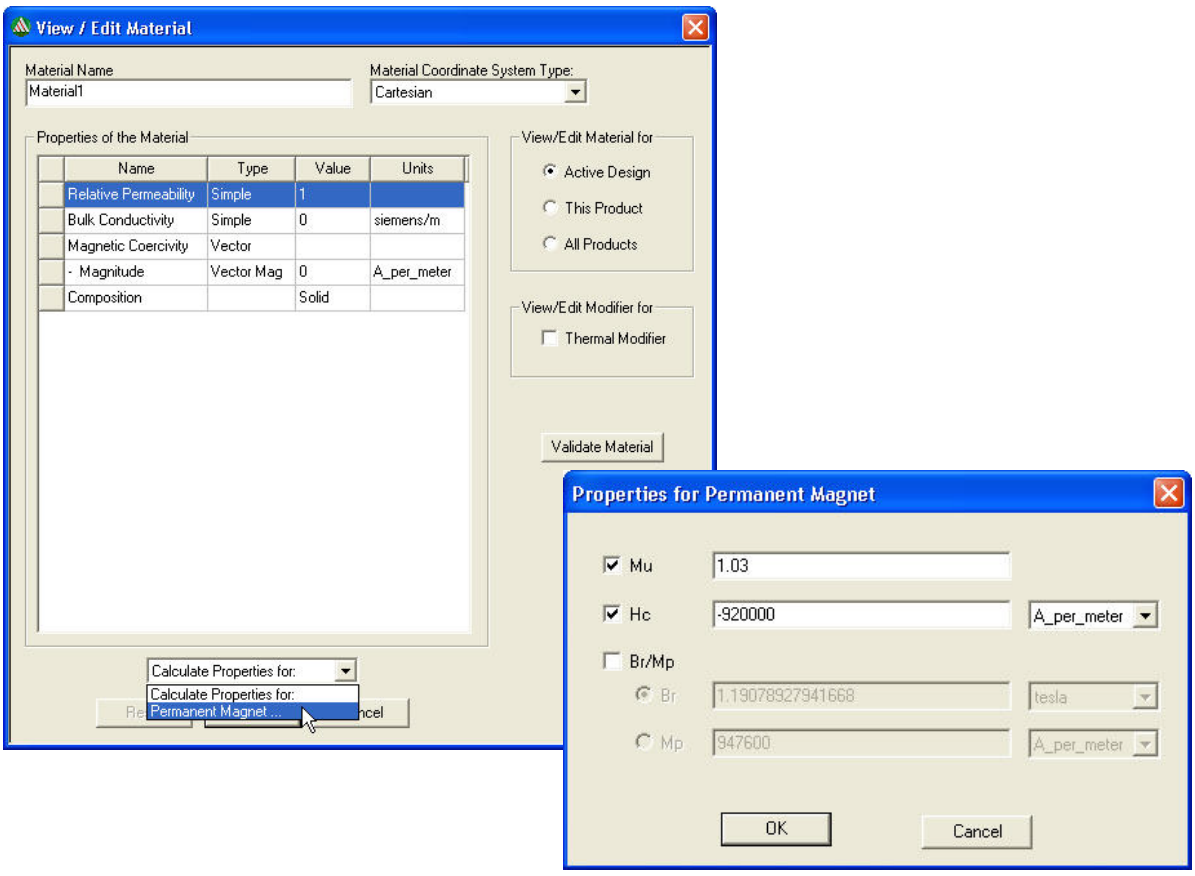
- Select the menu item *Modeler > Coordinate System > Set Work CS*
  1. Select **Global**

- **Set Magnet Orientations**
  - Select the sheet **PM1** and goto the Properties window,
  - Change the orientation coordinate system from Global to **PM1_CS**

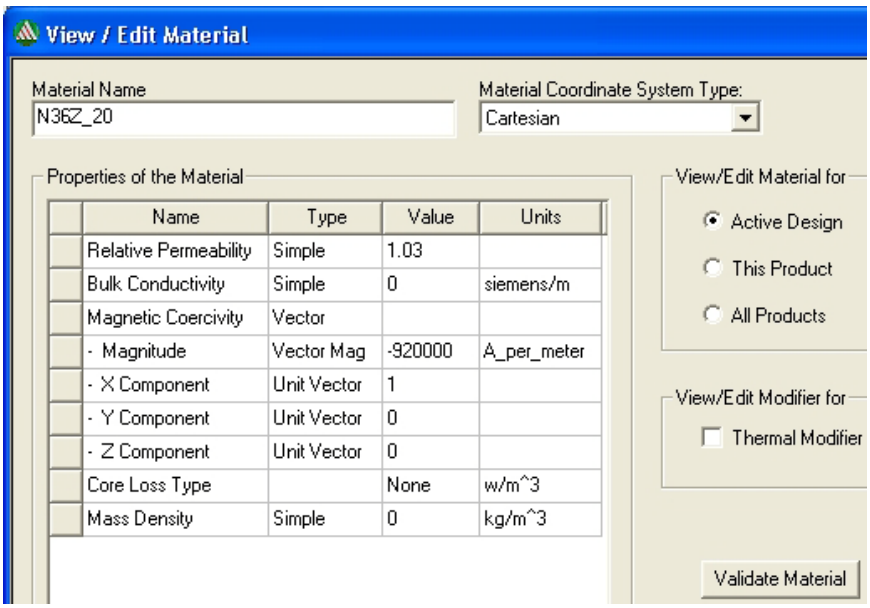


- Similarly change the orientation coordinate system for **PM2** to **PM2_CS**
- Select the menu item *View > Show All > Active View*
- Select the menu item *Edit > Select > Objects* or press **"o"** from keyboard

- **Create Material for PM1**
  - Select the sheet **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, leave 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**

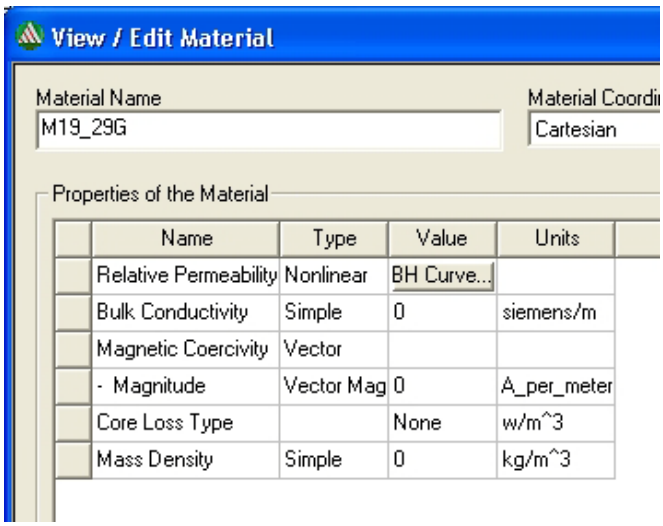


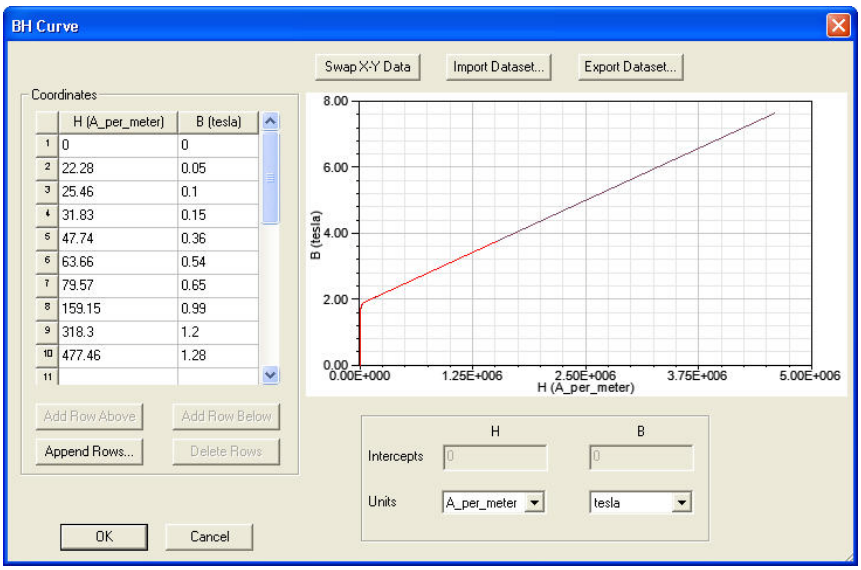
- **Material for PM2**
  - If the definition of *PM2_CS* is opposite with *PM1_CS* (X axis in the direction of the air gap), you can use the same material for *N36Z_20* for *PM2*. If it is not the case, you can clone the material *N36Z_20* and change the orientation to be consistent with the *PM2_CS* 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**

Motors - Permanent Magnet Motor (Prius IPM)

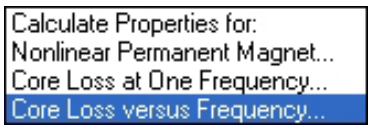
- **Material for Stator and Rotor**
  - Press **Ctrl** and select the sheets **Stator** and **Rotor,** right click and select **Assign Material**
  - In Select Definition window, select **Add Material**
  - In View/Edit Material window,
    1. Material Name: **M19_29G**
    2. Relative Permeability
       - 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

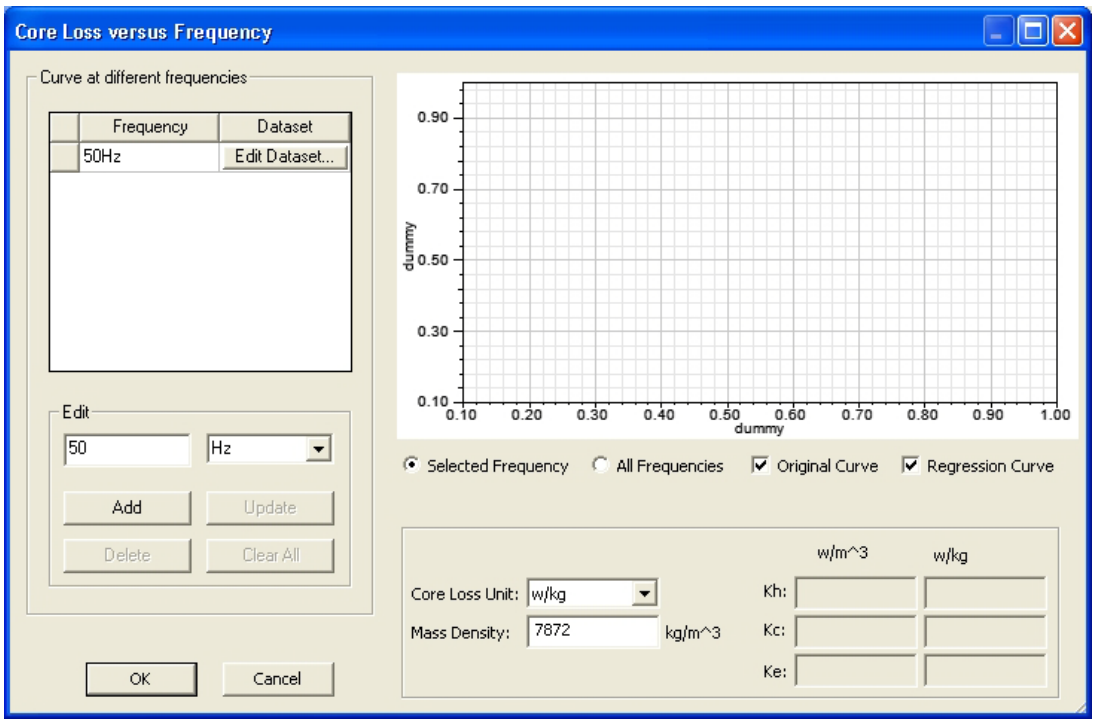




- **Material Properties for Core Loss calculations**
  - This section is only necessary if you wish to compute the coreloss of the motor.
  - In the Transient solver, we are able to compute coreloss (or hystereris loss), stranded loss and eddy current loss (or proximity loss). We will only consider coreloss in this document.
  - In View/Edit Material window,
    - Set Core Loss Type to **Electrical Steel**
    - Set the option at the bottom Calculate Properties for **Core Loss versus Frequency**



    - In Core Loss versus Frequency window,
      - Set Core Loss units to **W/kg**
      - Set Mass Density to **7872 kg/m^3**
      - In Edit window, set **50 Hz** and select the button **Add**
      - Select the button **Edit Dataset** to open dataset window

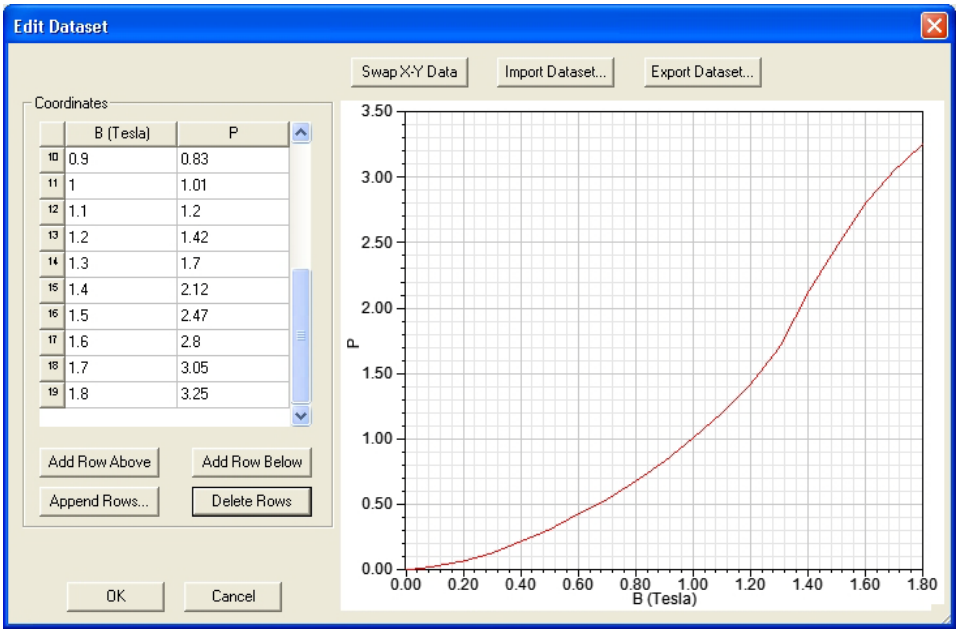


Motors - Permanent Magnet Motor (Prius IPM)

- In Edit Dataset window,
  - Enter the values as shown in below table
  - Press **OK**

50 Hz

B	P
0	0
0.1	0.03
0.2	0.07
0.3	0.13
0.4	0.22
0.5	0.31
0.6	0.43
0.7	0.54
0.8	0.68
0.9	0.83
1	1.01
1.1	1.2
1.2	1.42
1.3	1.7
1.4	2.12
1.5	2.47
1.6	2.8
1.7	3.05
1.8	3.25



- Similarly add more datasets for frequencies of **100, 200, 400** and **1000 Hz**

100Hz

B	P
0	0
0.1	0.04
0.2	0.16
0.3	0.34
0.4	0.55
0.5	0.8
0.6	1.08
0.7	1.38
0.8	1.73
0.9	2.1
1	2.51
1.1	2.98
1.2	3.51
1.3	4.15
1.4	4.97
1.5	5.92

200Hz

B	P
0	0
0.1	0.09
0.2	0.37
0.3	0.79
0.4	1.31
0.5	1.91
0.6	2.61
0.7	3.39
0.8	4.26
0.9	5.23
1	6.3
1.1	7.51
1.2	8.88
1.3	10.5
1.4	12.5
1.5	14.9

400Hz

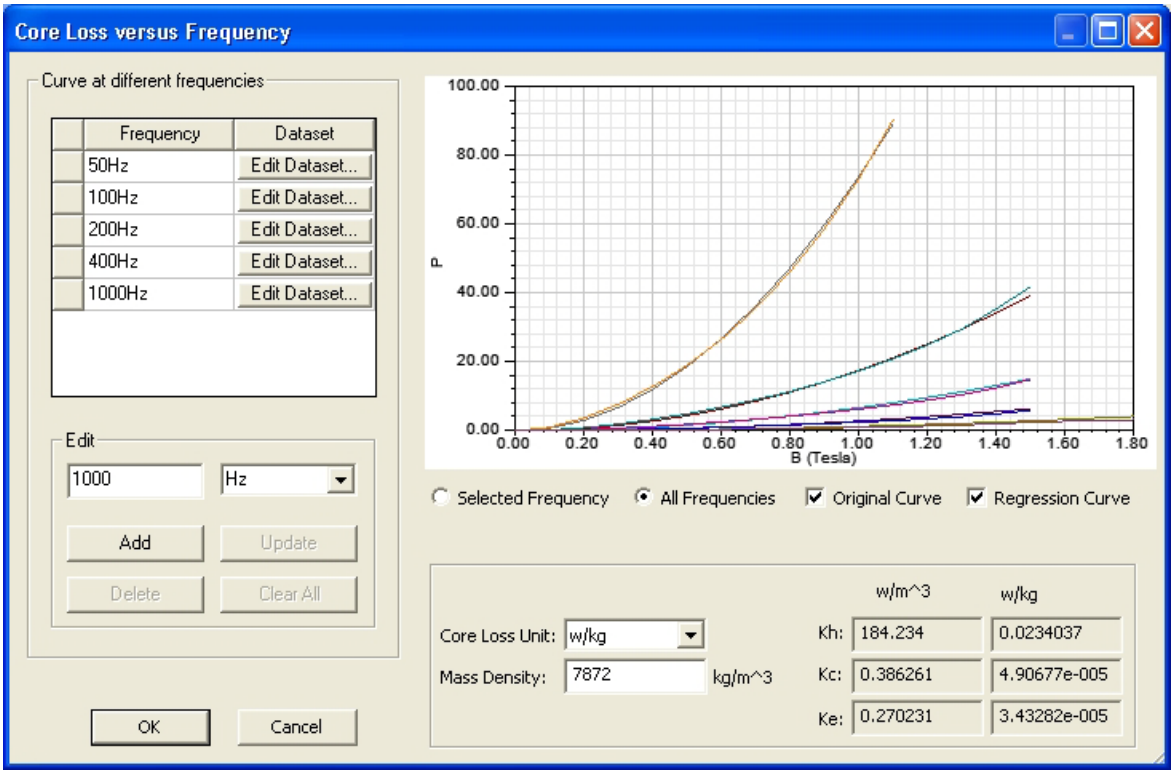
B	P
0	0
0.1	0.21
0.2	0.92
0.3	1.99
0.4	3.33
0.5	4.94
0.6	6.84
0.7	9
0.8	11.4
0.9	14.2
1	17.3
1.1	20.9
1.2	24.9
1.3	29.5
1.4	35.4
1.5	41.8

1000Hz

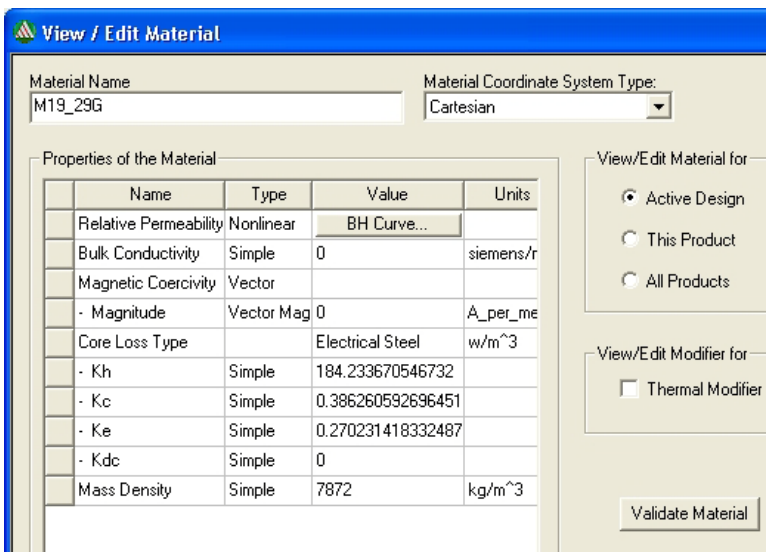
B	P
0	0
0.1	0.99
0.2	3.67
0.3	7.63
0.4	12.7
0.5	18.9
0.6	26.4
0.7	35.4
0.8	46
0.9	58.4
1	73
1.1	90.1

Motors - Permanent Magnet Motor (Prius IPM)

- The coreloss coefficient are automatically calculated

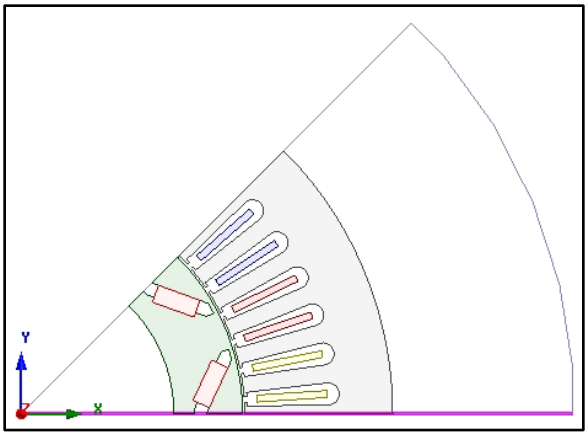


- Press **OK** to accept the setings
- In View/Edit Materials window,
  - Press **OK** to assign material

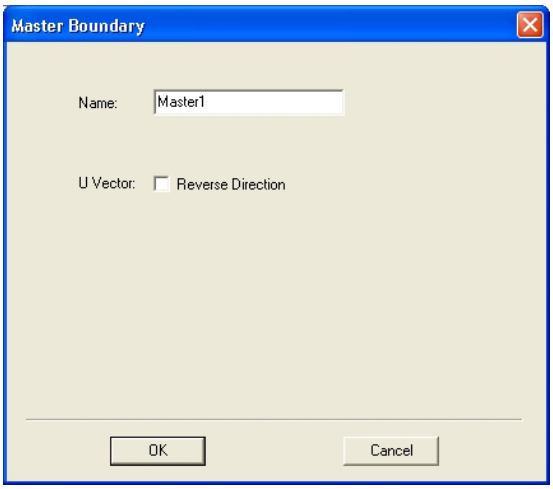


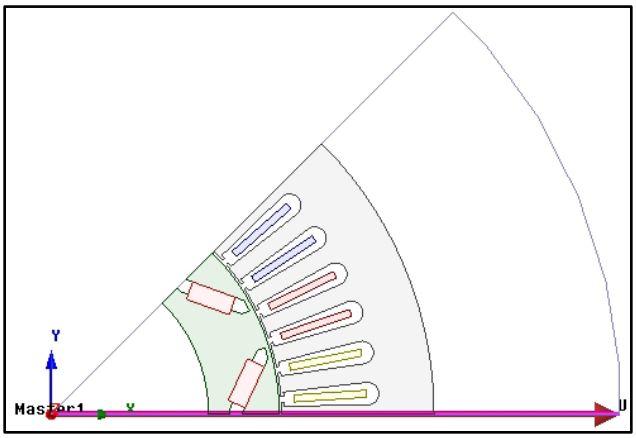
Applying Master / Slave Boundary

- 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 > Edges*
  - Select the edge of the region that coincides with Global X axis

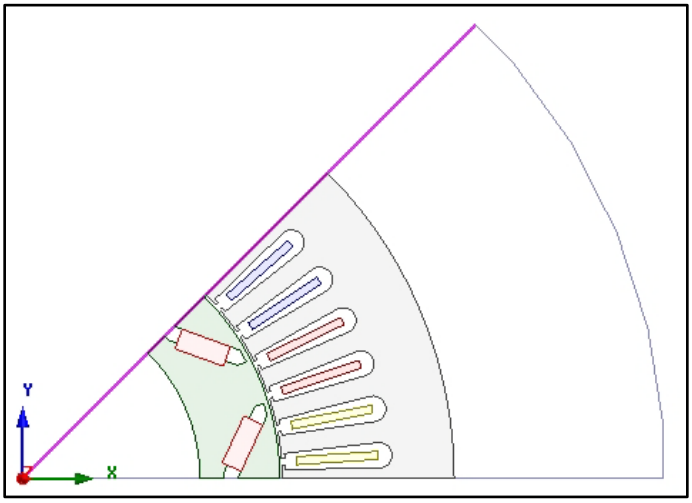


  - Select the menu item *Maxwell 2D > Boundaries > Assign > Master*
  - In Master Boundary window,
    1. Check the direction of U Vector shown in Graphic window
    2. If the direction is pointing away from the origin, use Reverse Direction
    3. Press **OK** to accept the settings

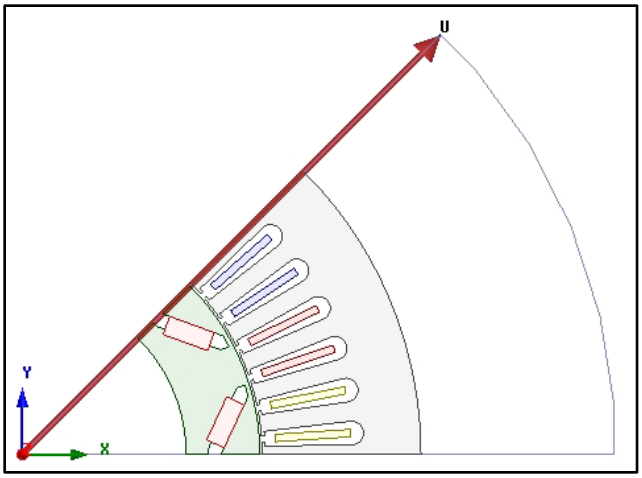


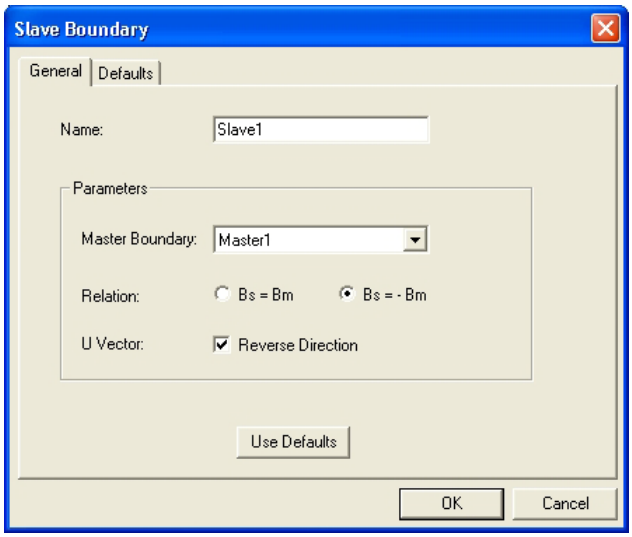


- **To Define Slave Boundary**
  - Select opposite edge of the region as shown in below image



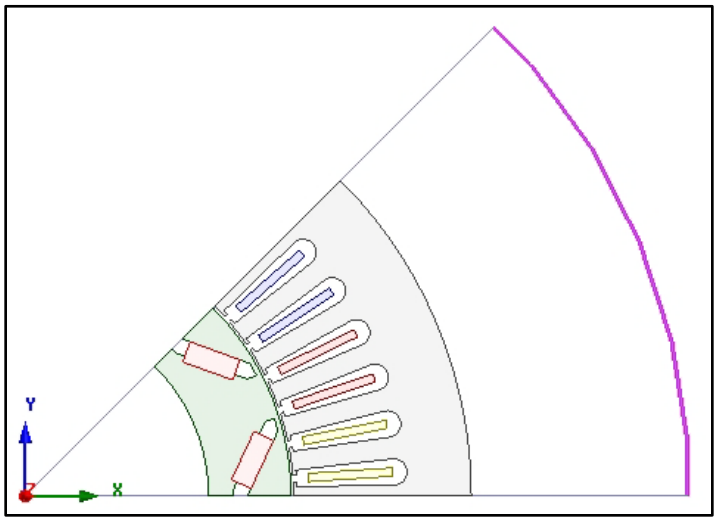
  - Select the menu item *Maxwell 2D > Boundaries > Assign > Slave*
  - In Slave Boundary window,
    1. Master Boundary: **Master1**
    2. Relation: select **Hs = -Hm**
    3. Use **Reverse Direction** if U vector direction is not consistent with Master
    4. Press **OK**



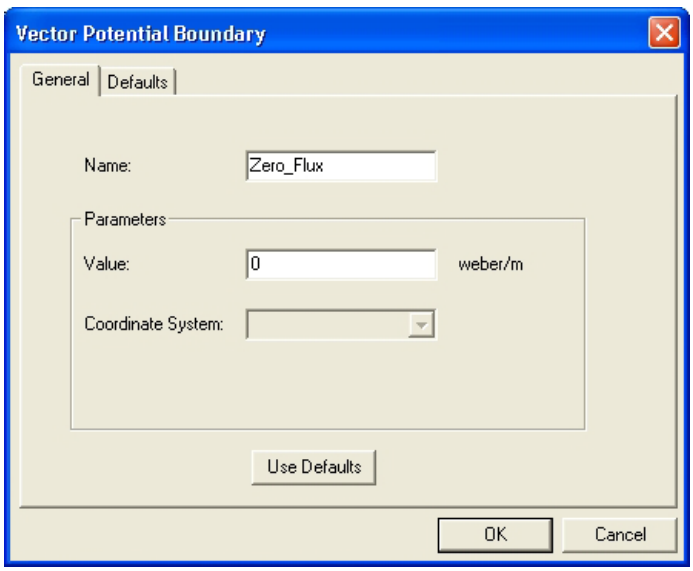


Assign Vector Potential Boundary

- **To Assign Vector Potential Boundary to region**
  - Select the menu item ***Edit > Select > Edges***
  - Select the edges of the Region as shown in below image



  - Select the menu item ***Maxwell 2D > Boundaries > Assign > Vector Potential***
  - In Vector Potential Boundary window,
    - Name: **Zero_Flux**
    - Set the value to **0 weber/m**
    - Select **OK**



Static Analysis

- We will study the different static parameters of the motor.

Copy Design

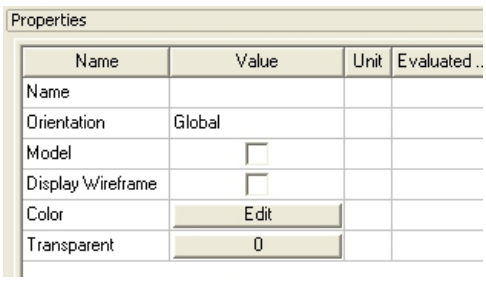
- **To Copy Design**
  - Select the Maxwell Design **" 2_Partial_Motor"** from the Project Manager tree, right click and select **Copy**
  - Right click on Project Name and select **Paste**
  - Rename the newly created design to **"3_Partial_Motor_MS"**

Set Static Analysis

- **To Set Static Analysis**
  - Select the menu item *Maxwell 2D > Solution Type*
    - Change solution type to **Magnetic > Magnetostatic**
    - Press **OK**

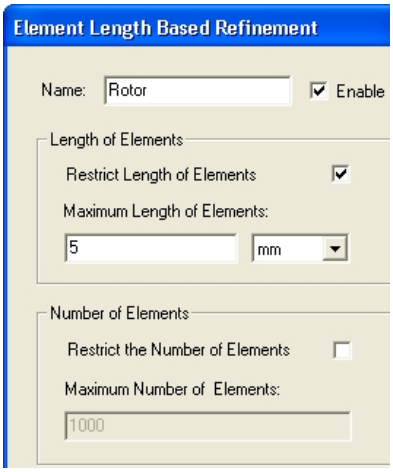
No Load Study

- The first analysis that will be performed consists in computing the fields due to the permanent magnets.
- The Coils are not needed in the model since no current is defined.
- **To Deactivate Coils**
  - Press **Ctrl** and select all coils from history tree
  - Goto the Properties window for selected sheets
  - Uncheck the box **Model** from Properties window to neglect the coils for simulation
  - Keep all the coils selected and select the menu item *View > Hide Selection > Active View*

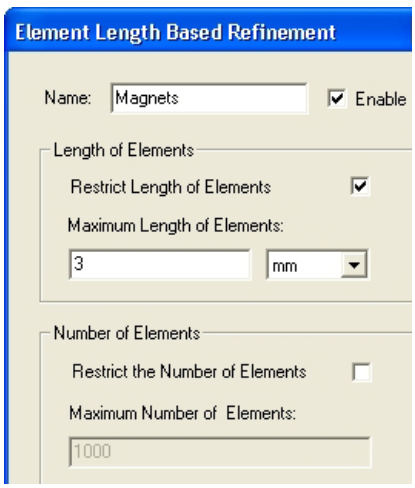


Apply Mesh Operations

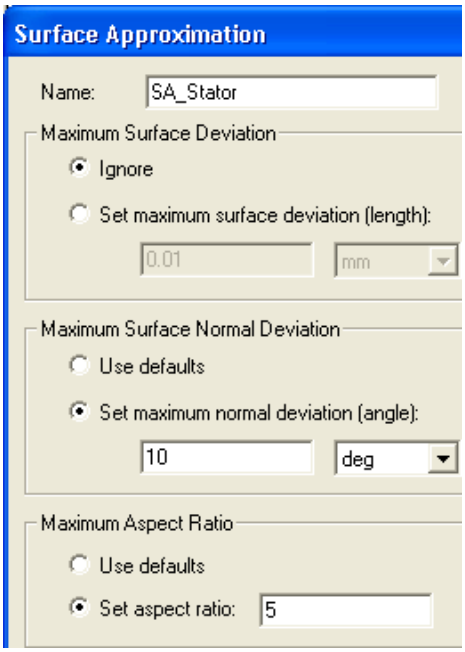
- The adaptive meshing is very effective, so it is not necessary to enter dedicated mesh operations. However, it is always a good idea to start with a decent initial mesh in order to reduce time computation since we know where the mesh needs to be refined for a motor. The non linear resolution will be faster with a small aspect ratios for the elements in the steel.
- **To Apply Mesh Operations for Rotor**
  - Select the sheet **Rotor** from history tree
  - Select the menu item ***Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Rotor**
    2. Restrict length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **5 mm**
    4. Press **OK**



- **To Apply Mesh Operations for Magnets**
  - Press **Ctrl** and select the sheets **PM1** and **PM2**
  - Select the menu item ***Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based***
  - In Element Length Based Refinement window,
    1. Name: **Magnets**
    2. Restrict length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **3 mm**
    4. Press **OK**



- **To Apply Mesh Operation for Stator**
  - Select the sheet **Stator** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Surface Approximation*
  - In Surface Approximation window,
    - Name: **SA_Stator**
    - Set maximum normal deviation (angle) : **30 deg**
    - Set aspect ratio: **5**
    - Press **OK**



Set Torque Parameter

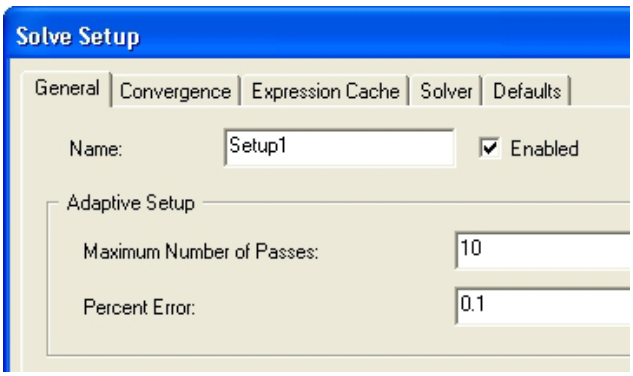
- **To Set Torque Calculation**
  - Press **Ctrl** and select the sheets **PM1, PM2** and **Rotor**
  - Select the menu item *Maxwell 2D > Parameters > Assign > Torque*
  - In Torque window,
    - Press **OK** to accept default settings

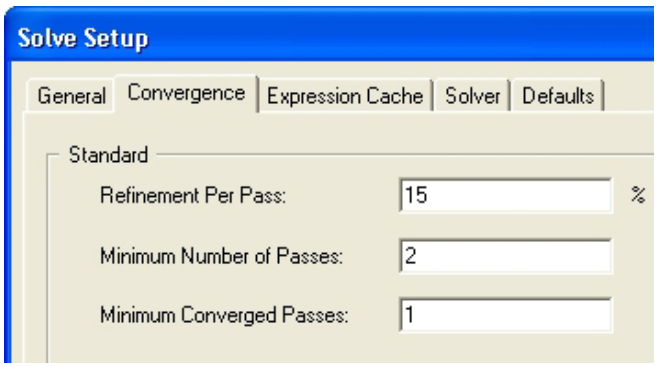


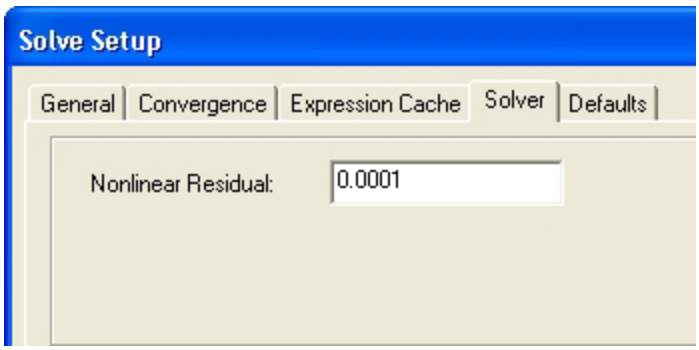
Analysis Setup

To Create Analysis Setup

- Select the menu item *Maxwell 2D > Analysis Setup > Add Solution Setup*
- In Solve Setup Window,
  1. **General** Tab
     - Percentage Error: **0.1**
  2. **Convergence** tab
     - Refinement Per pass: **15 %**
  3. **Solver** Tab
     - Non-linear Residuals: **0.0001**
  4. Press **OK**







Analyze

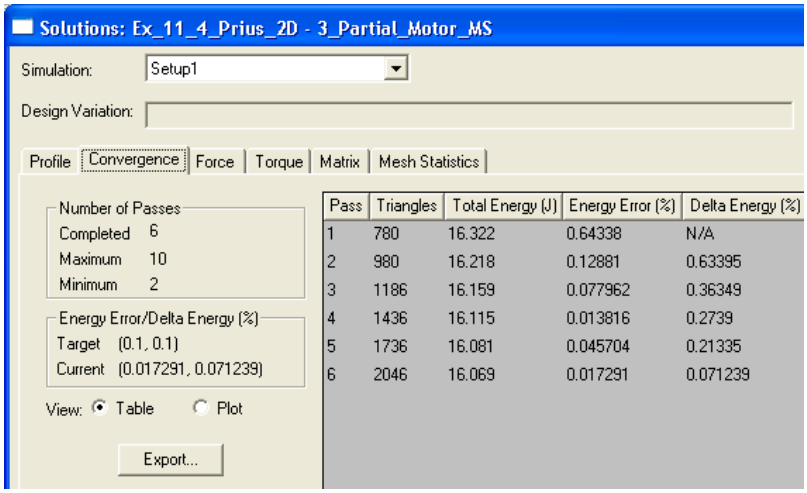
To Run the Solution

- Select the menu item *Maxwell 3D > Analyze All*

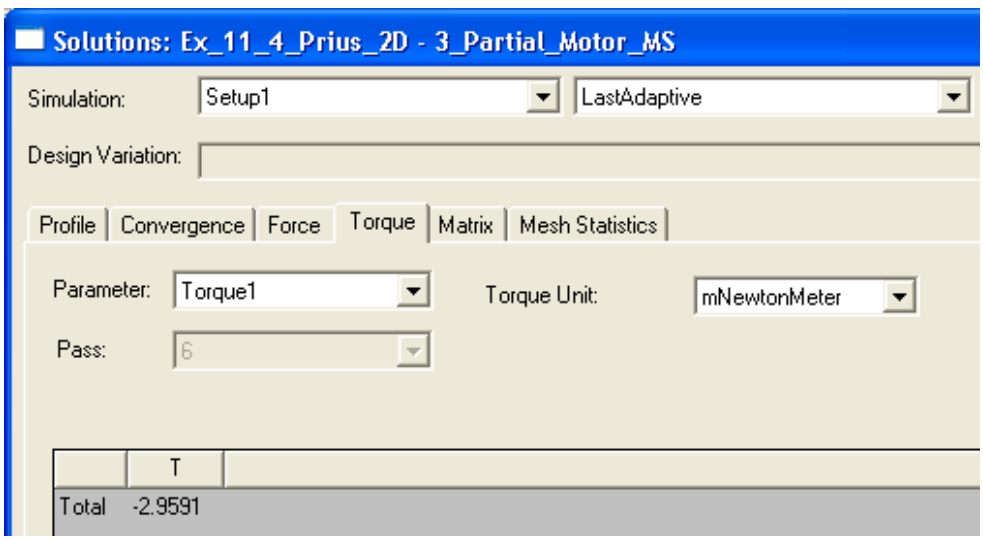
Post Processing

Solution Data

- Select the menu item *Maxwell 3D > Results > Solution Data*
- To View Convergence
  1. Select **Convergence** tab

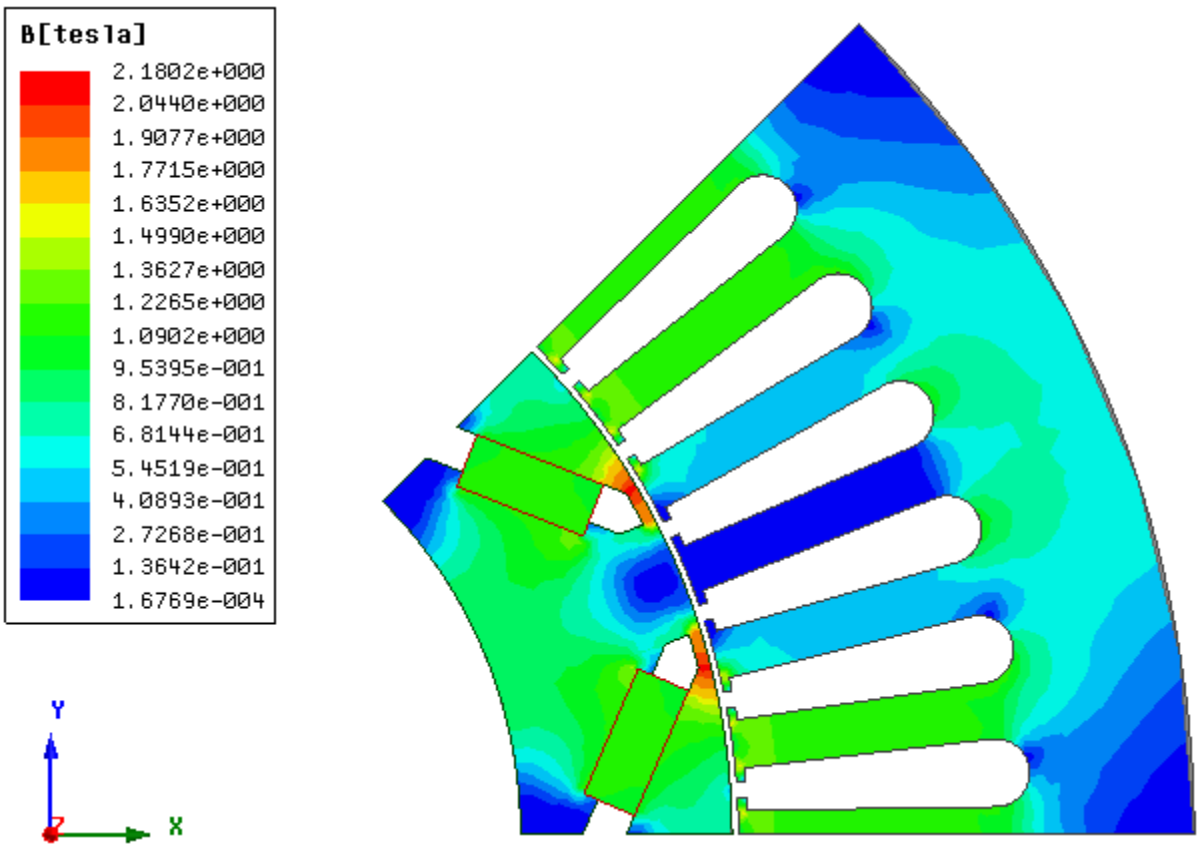


- To View Torque Values
  1. Select the **Torque** tab
     - The torque for the full motor needs to be multiplied by 8 (symmetry factor), then by 0.082 (to account for the motor length). The value is very small in regards to the full load operation. Different angles between the rotor and the stator would give different values.



Field Plots

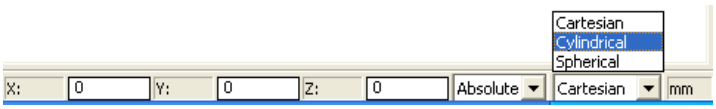
- **Plot Magnetic Flux Density**
  - Press **Ctrl** and select the sheets **PM1, PM2, Stator** and **Rotor**
  - Select the menu item *Maxwell 2D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    1. Press **Done**
  - The steel is highly saturated close to the magnets as expected. This saturations appears just because of the magnets strengths.



Vector Plot in Airgap

Create an Arc for Vector Plot

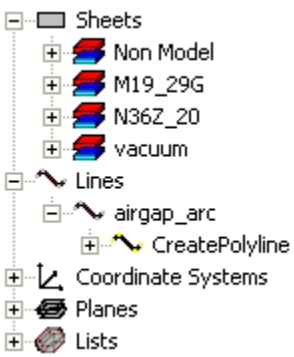
- Select the menu item *Draw > Arc > Center Point*
- A massage box appears asking if Geometry needs to be create d as a Non-model object. Press **Yes** to it
  1. Using Coordinate entry field, enter the center of arc
     - **X: 0, Y :0, Z: 0,** Press the **Enter** Key
  2. Change the definition coordinate system of the coordinate entry field to Cylindrical

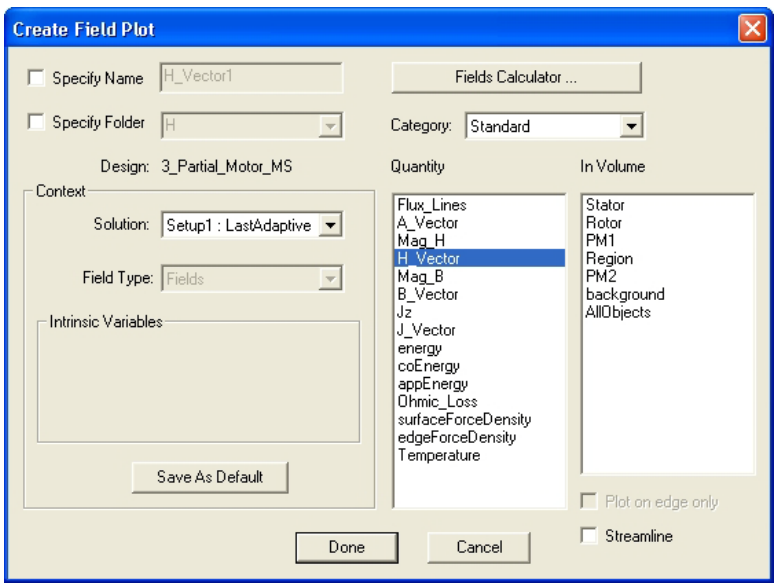


  3. Using Coordinate entry field, enter the first point of arc
     - **R: 80.575, Phi :0, Z: 0,** Press the **Enter** Key
  4. Using Coordinate entry field, enter the first point of arc
     - **R: 80.575, Phi : 45, Z: 0,** Press the **Enter** Key
  5. To finish the arc, move the mouse on the drawing area, right mouse click, and select the menu entry **Done**
- Rename the created Polyline to **airgap_arc**

Plot H_Vector on Arc

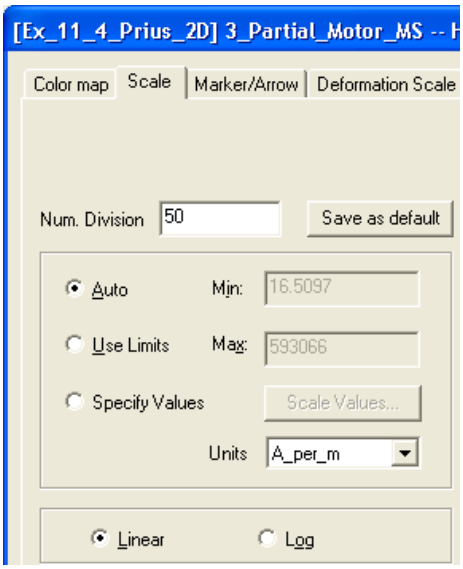
- Select the line **airgap_arc** from the history tree
- Select the menu item *Maxwell 2D > Fields > Fields > H > H_Vector*
- In Create Field Plot window,
  1. Press **Done**

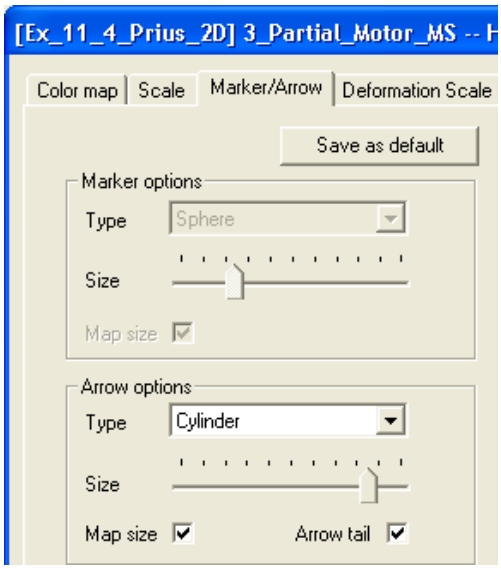


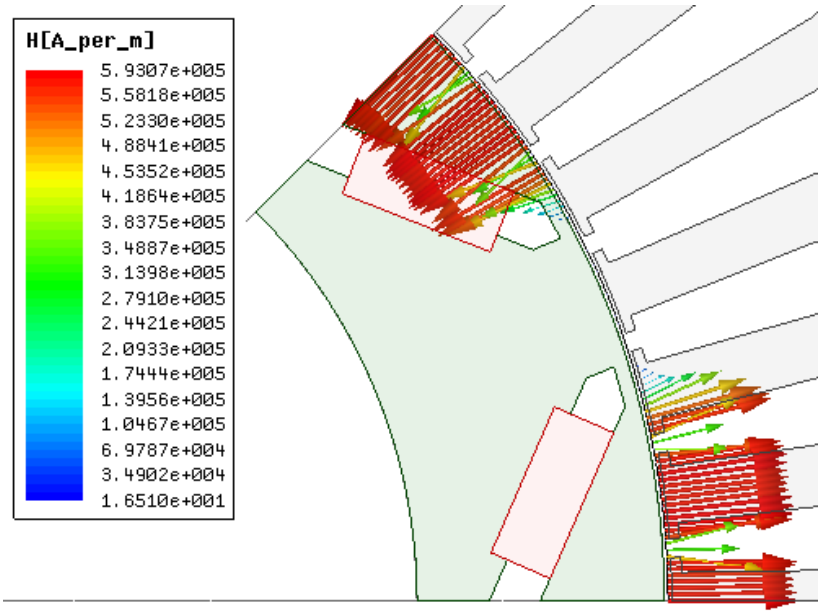


Motors - Permanent Magnet Motor (Prius IPM)

- **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 appropriate value**
    3. Press **Apply** and **Close**







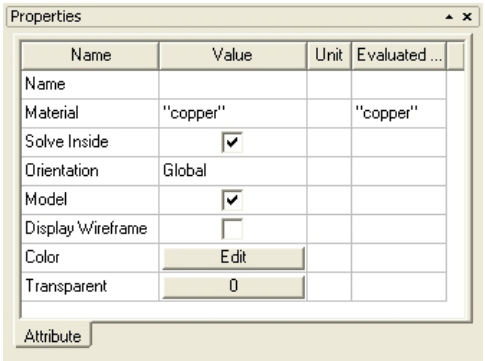
Full Load Study

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**
  - Rename the newly created design to **"4_Partial_Motor_MS2"**
  - In this design, we apply current in the coils

Activate Coils

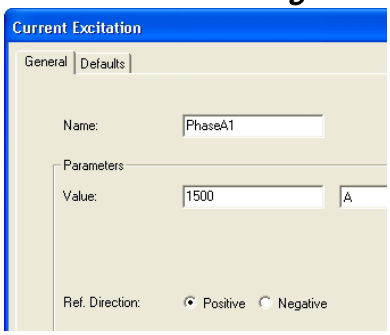
- **To Activate Coils**
  - Press **Ctrl** and select all coils from history tree
  - Goto the Properties window for selected object
  - Check the button **Model** to activate all coils



Apply Excitations

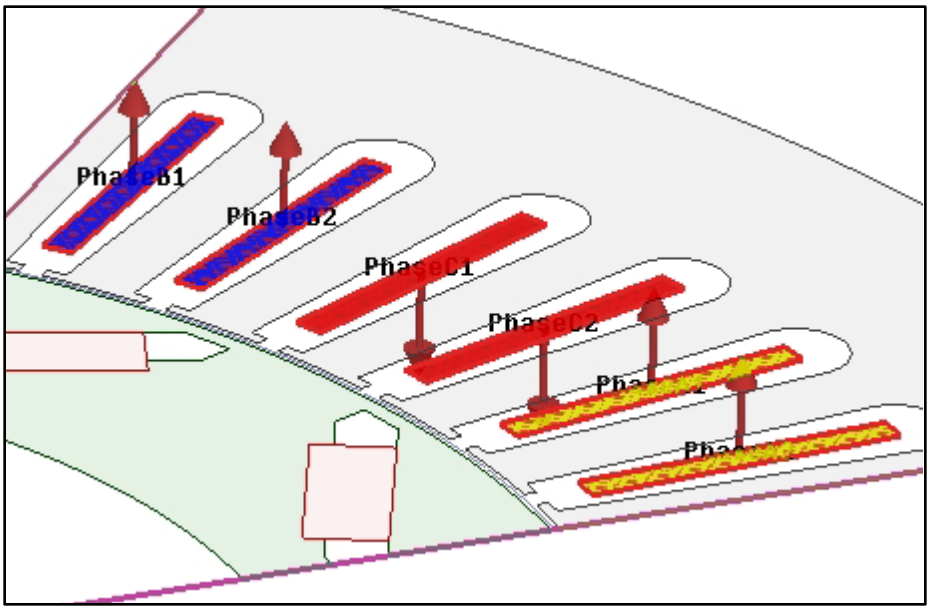
- The coils are partially represented in the model. We need to enter the current that flows in and out inside each coil. The excitation is realized through a balanced three phase system. For instance, in our example, we apply:
  - 1500 A to PhaseA
  - -750 A to PhaseB
  - -750 A to PhaseC.
- In the Magnetosatic 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.

- **Assign Excitations for Phase A**
  - Select the sheet **PhaseA1** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseA1**
    2. Value: **1500 A**
    3. Ref. Direction: **Positive**
    4. Press **OK**



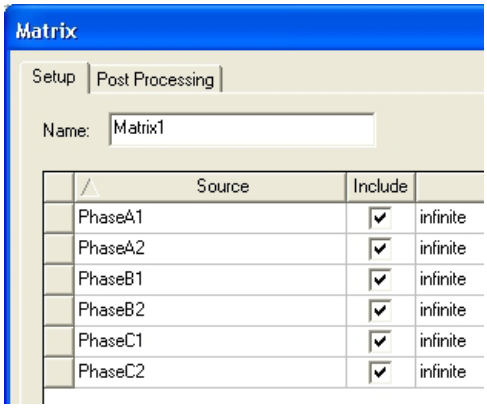
  - Similarly select the sheet **PhaseA 2** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseA2**
    2. Value: **1500 A**
    3. Ref. Direction: **Positive**
    4. Press **OK**
- **Assign Excitations for PhaseC**
  - Select the sheet **PhaseC1** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseC1**
    2. Value: **-750 A**
    3. Ref. Direction: **Negative**
    4. Press **OK**
  - Similarly select the sheet **PhaseC2** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseC2**
    2. Value: **-750 A**
    3. Ref. Direction: **Negative**
    4. Press **OK**

- **Assign Excitations for PhaseB**
  - Select the sheet **PhaseB1** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseB1**
    2. Value: **-750 A**
    3. Ref. Direction: **Positive**
    4. Press **OK**
  - Similarly select the sheet **PhaseC2** from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Current*
  - In Current Excitation window,
    1. Name: **PhaseB2**
    2. Value: **-750 A**
    3. Ref. Direction: **Positive**
    4. Press **OK**

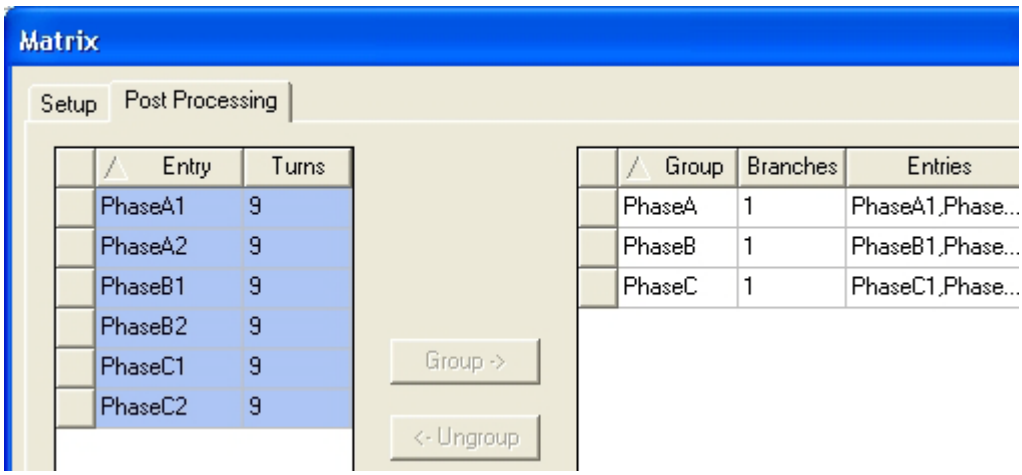


Set Parameters

- 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 2D > Parameters > Assign > Matrix*
  - In Matrix window,
    1. Setup Tab
       - For all sources:
         1. Include : ☑ **Checked**



    2. Post Processing Tab
       - Set Turns for all phases to **9**
       - Press **Ctrl** and select the sources **PhaseA1** and **PhaseA2**
       - Select the button **Group**
       - Rename the group as **PhaseA**
       - Similarly create the groups **PhaseB** (using PhaseB1 and PhaseB2) and **PhaseC** (using PhaseC1 and PhaseC2)



Analyze

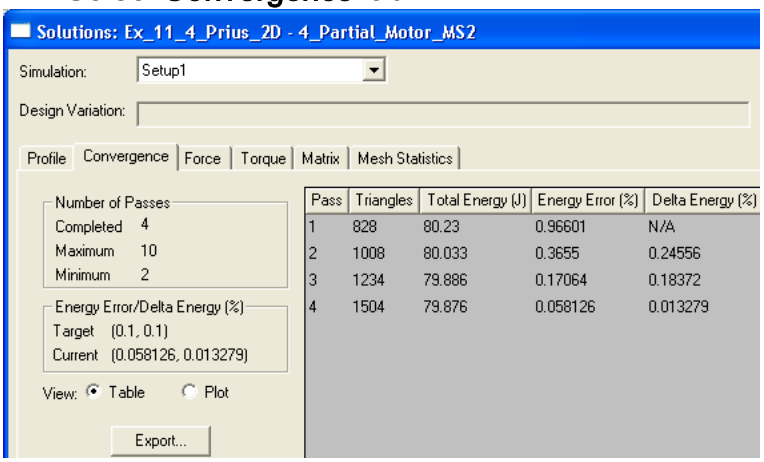
To Run the Solution

- Select the menu item *Maxwell 2D > Analyze All*

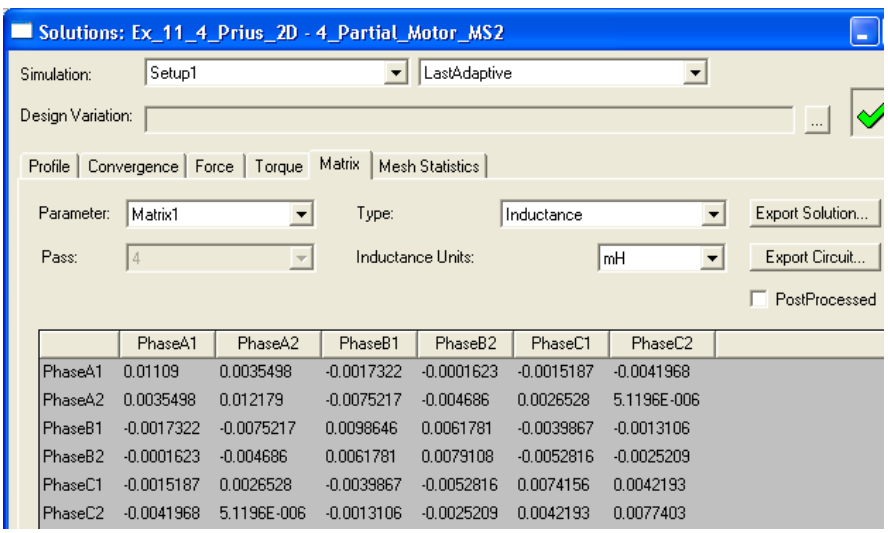
Post Processing

Solution Data

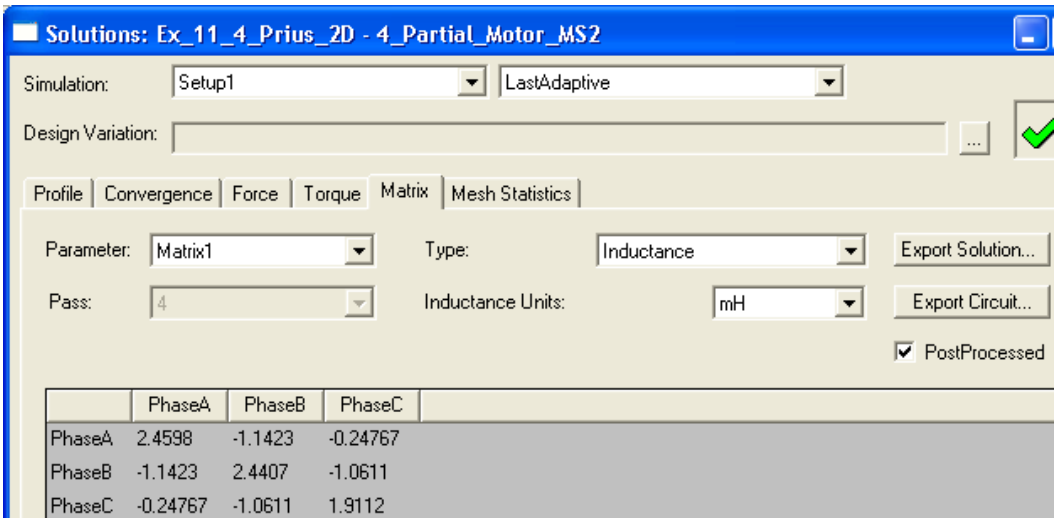
- Select the menu item *Maxwell 2D > Results > Solution Data*
- To View Convergence
  1. Select **Convergence** tab



- To View Inductance Values
  1. Select the **Matrix** tab

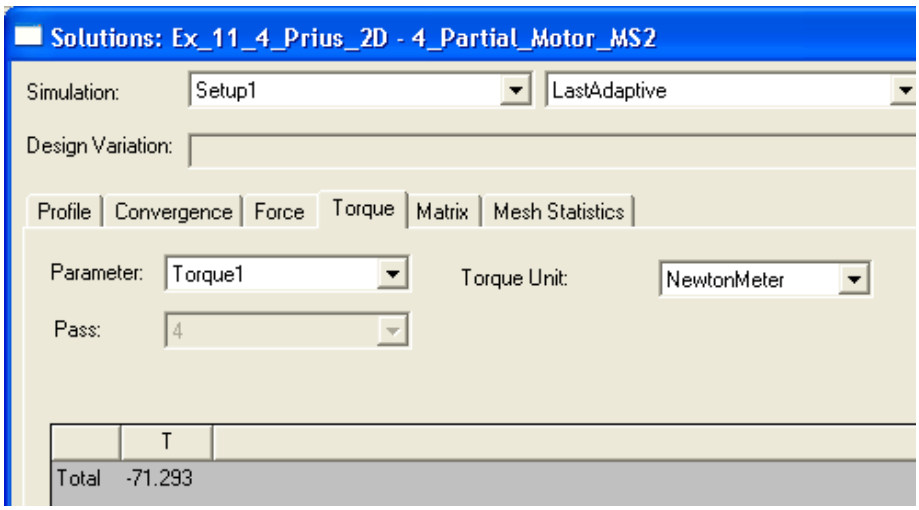


- The Values shown are Per Turn Values
- To View Total inductance, check the button **PostProcessed**



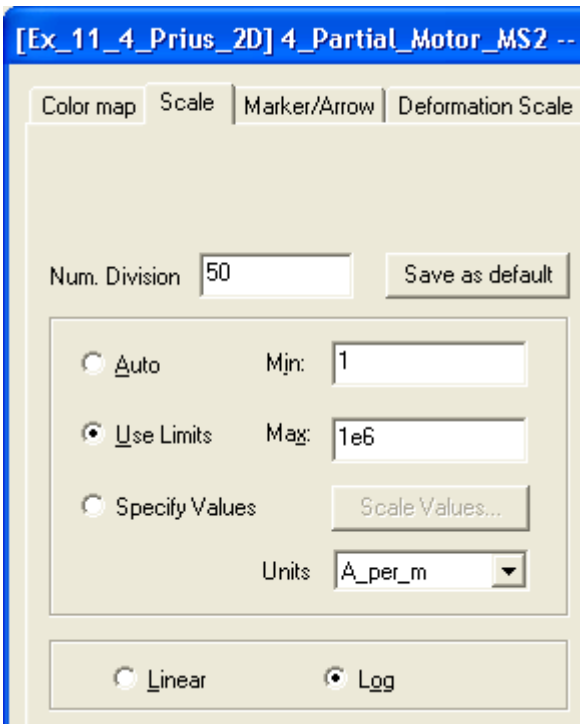
- **Note:** it is possible to export the inductance matrix to Simplorer using the *Export Circuit* button

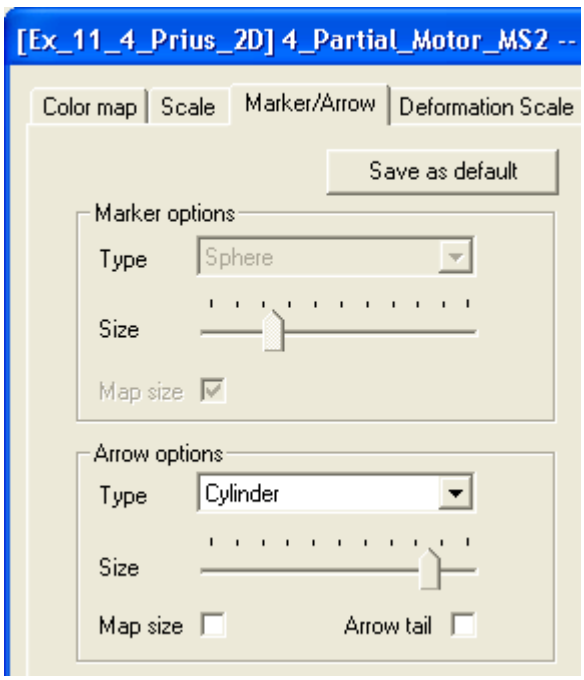
- To View Torque Values
  1. Select the **Torque** tab
  2. The torque for the full motor needs to be multiplied by 8 (symmetry factor), then multiplied by 0.083 (length of the motor). This gives around 47N.m. In this case, we have not synchronized the position of the rotor poles with the winding currents, 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.



H Vector Plot

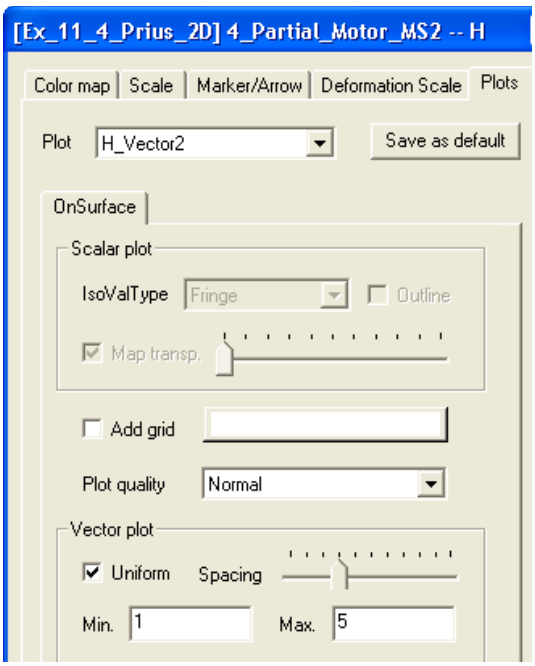
- **Plot H_Vector on XY Plane**
  - Select the Plane **Global:XY** from history tree
  - Select the menu item *Maxwell 2D > Fields > Fields > H > H_Vector*
  - Press **Done**
- **Modify Plot Attributes**
  - In the window,
    1. **Scale** Tab
       - Select **User Limits**
         - Min: **1**
         - Max: **1e6**
       - Select **Log**
    2. **Marker/Arrow** Tab
       - Map Size: ☐ **Unchecked**
       - Arrow Tail: ☐ **Unchecked**



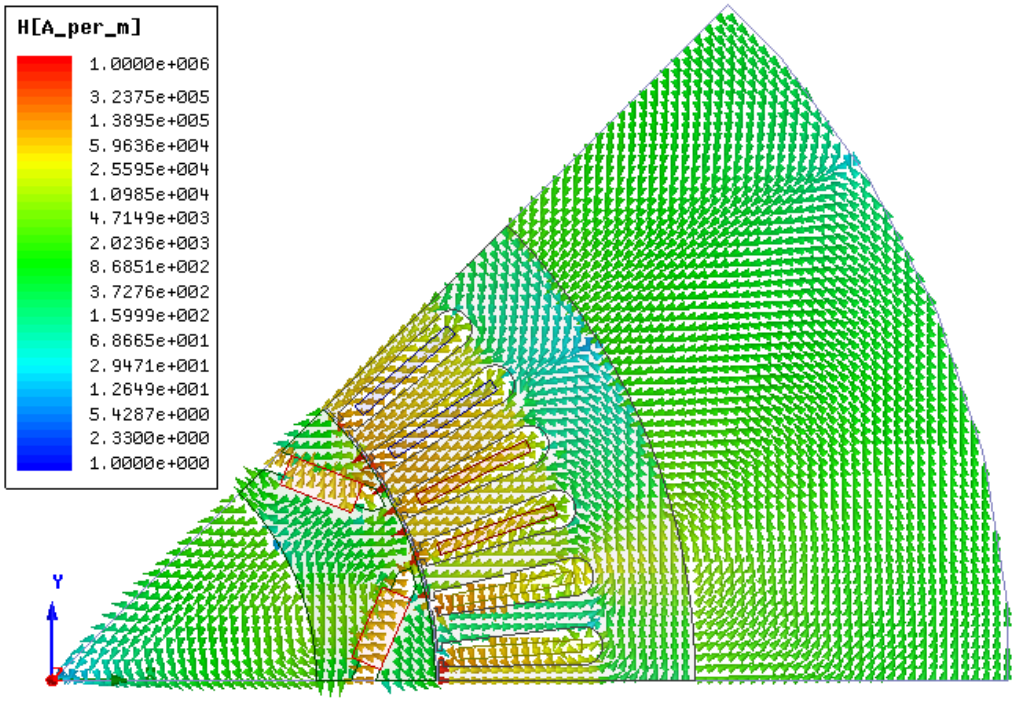


Motors - Permanent Magnet Motor (Prius IPM)

1. **Plots** Tab
   - Plot: **H_Vector2**
   - Spacing: **Four Spaces from Left**
     - Min: **1**
     - Max: **5**
2. Press **Apply** and **Close**



- Plot should look as below



Dynamic Analysis

- We will study the transient characteristic 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**
  - Change the name of the design to **"5_Partial_Motor_TR"**

About Transient Solver

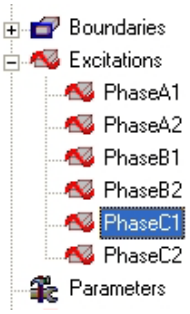
- The transient solver acts differently from the Magnetostatic solver mainly because:
  - There is not adaptive meshing. Since the relative position of objects changes at every time step, Maxwell does not re-mesh adaptively for obvious time saving. In transient analysis, we will build a good mesh valid 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.

Create Coils

- **To Create Coils**
  - Press **Ctrl** and select all coils from history tree
  - Select the menu item *Maxwell 2D > Excitations > Assign > Coil*
  - In Coil Excitation window,
    - Base Name: Leave default name as object names will be used automatically
    - Number of Conductors: **9**
    - Polarity: **Positive**
    - Press **OK**

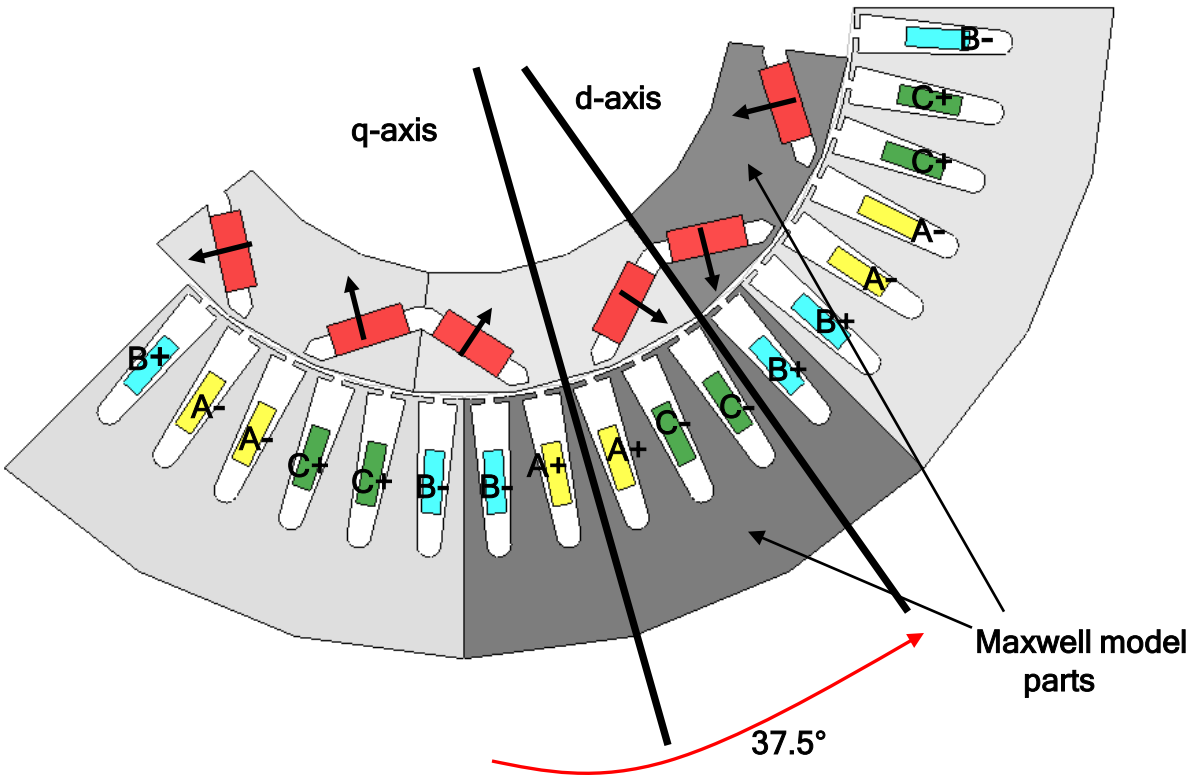
Motors - Permanent Magnet Motor (Prius IPM)

- **Modify Excitation Direction**
  - Direction of excitation for PhaseC needs to be reverses
  - Expand the Project Manager tree to view **Excitations**
  - Double click on the tab **PhaseC1** to modify it
  - In Coil Excitation window,
    - Change Polarity to **Negative**
  - Similarly change the polarity of **PhaseC2** to **Negative**



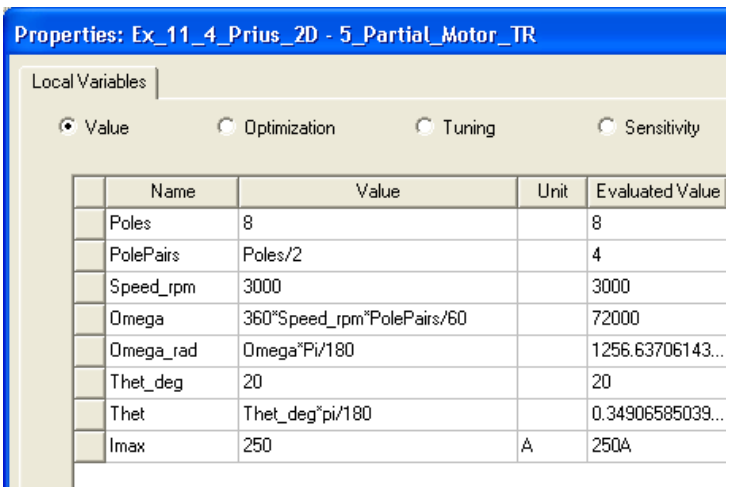
Motor excitation

- 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

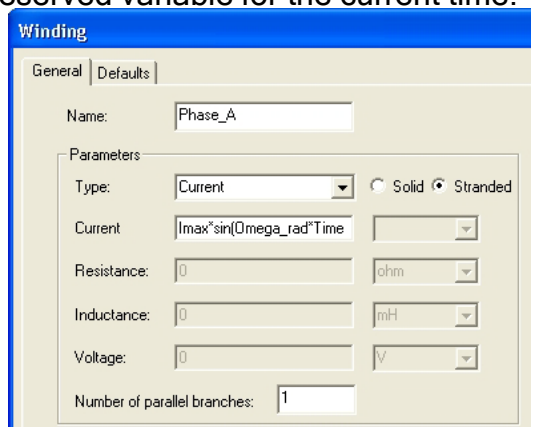
- To Create Parameters
  - Select the menu item *Maxwell 2D > Design Properties*
  - In Properties window, select **Add**
  - In Add Property window,
    - Name: **Poles** (number of poles of the motor)
    - Value: **8**
    - Press **OK**
  - Similarly add more properties as below
    1. Name: **PolePairs** (number of pair of poles)
       - Value: **Poles/2**
    2. Name: **Speed_rpm** (speed of the motor in rpm)
       - Value: **3000**
    3. Name: **Omega** (pulsation of the excitation in degrees/s)
       - Value: **360*Speed_rpm*PolePairs/60**
    4. Name: **Omega_rad** (pulsation in rad/s)
       - Value: **Omega * pi / 180**
    5. Name: **Thet_deg** (load angle of the motor)
       - Value: **20**
    6. Name: **Thet** (load angle in radian)
       - Value: **Thet_deg * pi /180**
    - **Note:** Do not specify unit to any of the above parameters
    7. Name: **Imax** (peak winding current of the motor)
       - Value: **250**
       - Unit: **A**



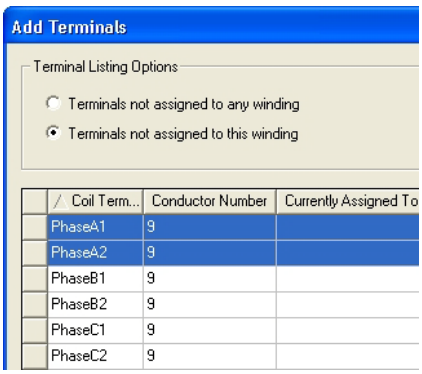
Name	Value	Unit	Evaluated Value
Poles	8		8
PolePairs	Poles/2		4
Speed_rpm	3000		3000
Omega	360*Speed_rpm*PolePairs/60		72000
Omega_rad	Omega*Pi/180		1256.63706143...
Thet_deg	20		20
Thet	Thet_deg*pi/180		0.34906585039...
Imax	250	A	250A

Create Windings

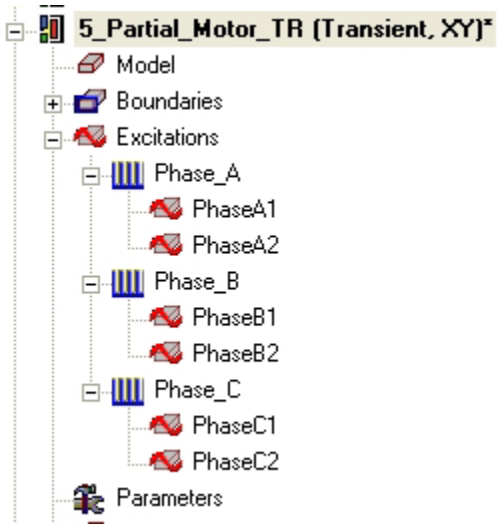
- The terminals are meant to define the excitation paths in and out of the model. The actual excitation is defined through the definition of windings. A winding needs to be defined for each parallel electrical excitation of the motor.
- The motor is excited with a balanced three phase connection. A sinusoidal 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 2D > 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 Coils**
  - Press **Ctrl** and select the terminals **PhaseA1** and **PhaseA2**
  - Press **OK**

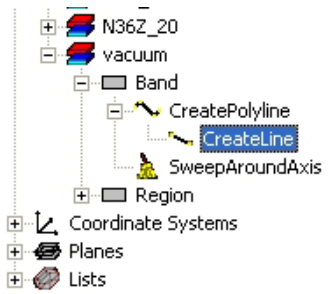


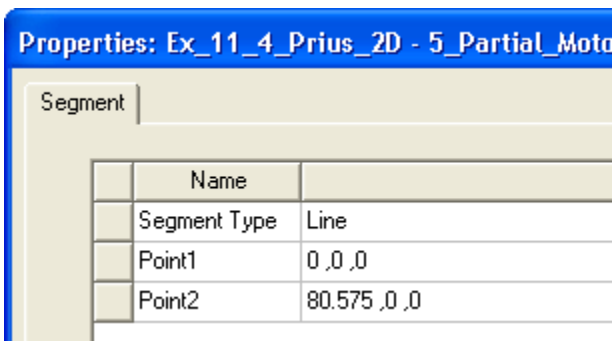
- **Add Winding for Phase B and Phase C**
  - Similarly add Windings for Phase B and Phase C
  - Phase B
    1. Name: **Phase_B**
    2. Type: **Current**
    3. Check **Stranded**
    4. Current: **Imax*sin(Omega_rad*Time-2*pi/3+Thet)**
       - It is shift by -120 degrees from PhaseA.
    5. Press **OK**
    6. Add terminals **PhaseB1** and **PhaseB2** 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 terminals **PhaseC1** and **PhaseC2** to this winding



Create Band Object

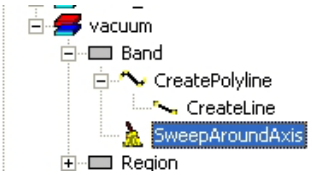
- The moving parts (rotor and permanent magnets) need to be enclosed in an air object, the band. This will separated the moving part from the fixed part of the project. Some rules apply for the definition of the band object for motor applications:
  - The band object must be somewhat larger than the rotating parts in all directions (except at the boundaries)
  - The band object should be a facetted type cylinder of wedge
  - It is very advisable to have an air object that encloses all the moving object inside the band object. This will facilitate the mesh handling around the air gap
- To create the Band object, we will clone the region and adapt the parameters
- **To Duplicate Region**
  - Select the sheet **Region** from history tree
  - Select the menu item ***Edit > Copy***
  - Select the menu item ***Edit > Paste***
  - Change the name of the sheet Region1 to **Band**
- The rotor radius is 80.2mm. The inner diameter of the stator 80.95mm. We pick the middle for Band object .
- **To Edit Band Object**
  - Expand the history tree for the sheet **Band**
  - Double click on the command **CreateLine** to open its properties
  - In Properties window,
    - Change Point2 position to **80.575,0,0**
    - Press **OK**

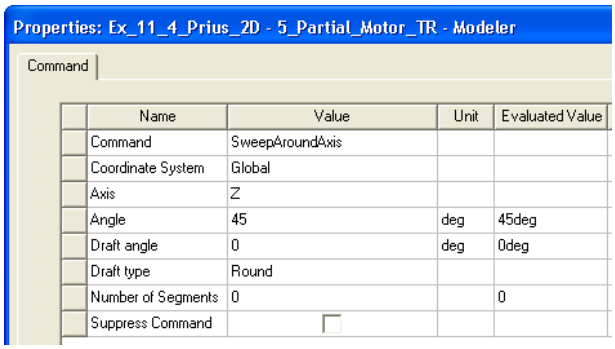


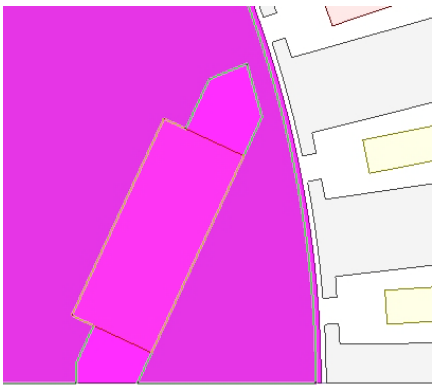


Motors - Permanent Magnet Motor (Prius IPM)

- Double click on the command **SweepAroundAxis** to open its properties
- In Properties window,
  - Change Number of Segments to **0**
  - Press **OK**

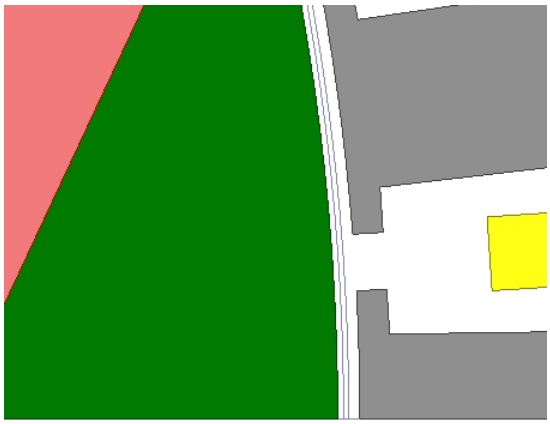






Create Inner Band

- We now create an object that enclosed the moving objects inside the *Band*.
- **Copy Band**
  - Select the sheet **Band** from history tree
  - Select the menu item *Edit > Copy*
  - Select the menu item *Edit > Paste*
  - Change the name of the sheet to **Band_in**
- **To Edit Band_in Object**
  - Expand the history tree for the sheet **Band_in**
  - Double click on the command **CreateLine** to open its properties
  - In Properties window,
    - Change Point2 position to **80.4,0,0**
    - Press **OK**

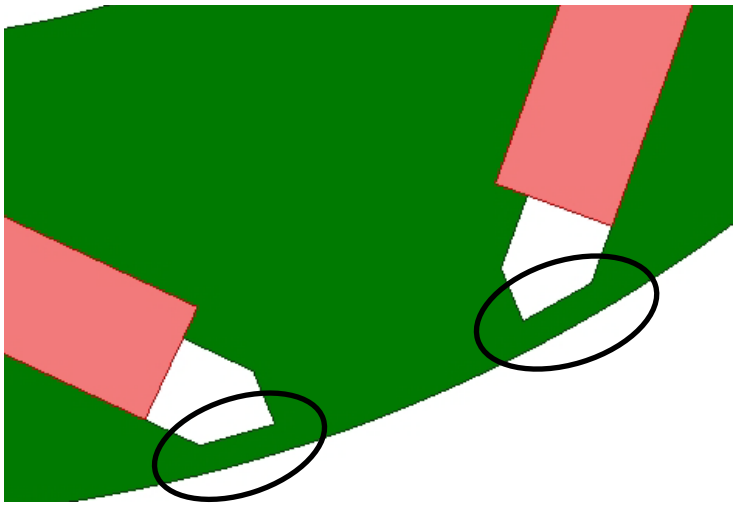


Mesh Operations

- 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.

Mesh Operation for Rotor

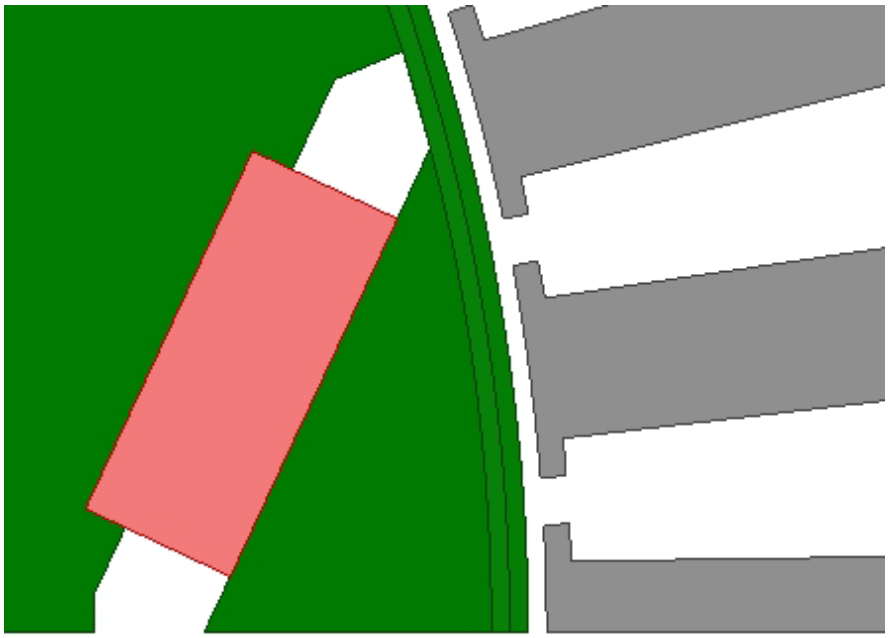
- The Rotor is designed to be highly saturated around the permanent magnets, close to the air gap. It is required to have a good mesh density around this area.
- To achieve this requirement, we create a couple of objects inside the rotor; then mesh operations will be applied to these objects in order to have a nice mesh around the ducts.



Highly saturated zones

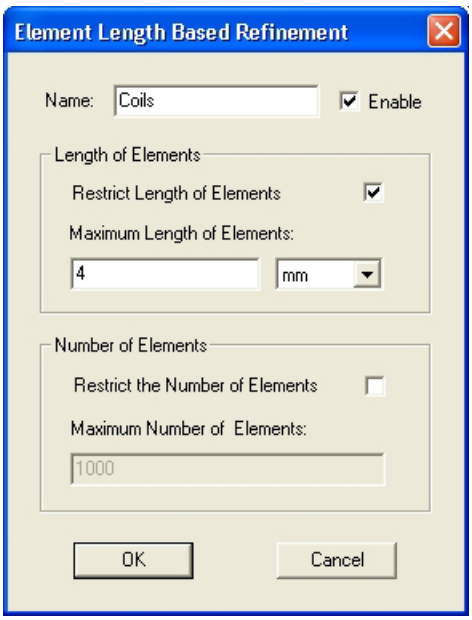
- **Create Line**
  - Select the menu item *Draw > Line*
    1. Using the coordinate entry field, enter the first vertex
       - **X: 78.72, Y: 0.0, Z: 0.0**, Press the **Enter** key
    2. Using the coordinate entry field, enter the second vertex
       - **X: 80.2, Y: 0.0, Z: 0.0**, Press the **Enter** key
  - Press **Enter** again to end drawing
  - Change the name of the resulting line to **Rotor2**

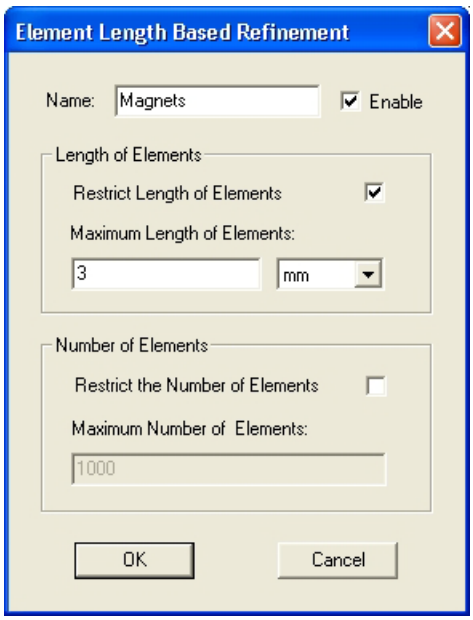
- **To Sweep Line**
  - Select the line **Rotor2** from the history tree
  - Select the menu item *Draw > Sweep > Around Axis*
  - In Sweep Around Axis window,
    1. Axis: **Z**
    2. Angle of sweep: **45**
    3. Number of segments: **0**
    4. Press **OK**
  - Change the material property of **Rotor2** to **M19_29G**. Also, assign the same color and transparency as the object Rotor.
- **Create Rotor3**
  - Using same process as for Rotor2, create Rotor3
  - Use line coordinates as mentioned below to create line
    - **X: 78.72, Y: 0.0, Z: 0.0**
    - **X: 79.46, Y: 0.0, Z: 0.0**
  - Change the material property of **Rotor3** to **M19_29G**. Also, assign the same color and transparency as the object Rotor.



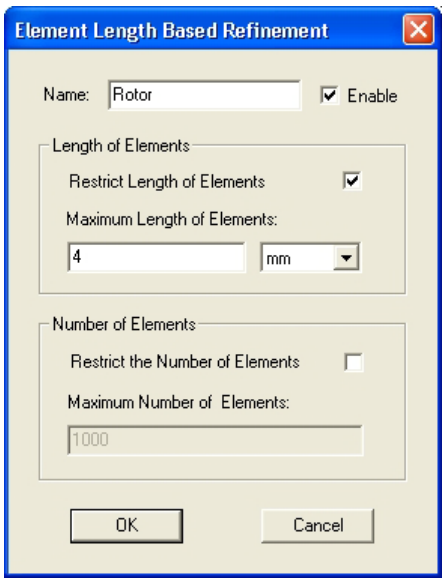
Mesh Operations for Rest

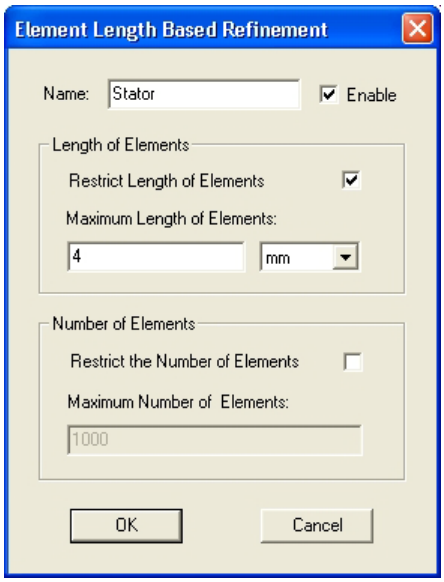
- **Assign Mesh Operation for Coils**
  - Press **Ctrl** and select all **Coils** from history tree
  - Select the mesh item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length based Refinement window,
    1. Name: **Coils**
    2. Restrict the Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **4 mm**
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**
- **Mesh Operation for Magnets**
  - Press **Ctrl** and select the sheets **PM1** and **PM2** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length based Refinement window,
    1. Name: **Magnets**
    2. Restrict the Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **3 mm**
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**





- **Mesh Operation for Rotor**
  - Press **Ctrl** and select the sheets **Rotor, Rotor2** and **Rotor3** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length based Refinement window,
    1. Name: **Rotor**
    2. Restrict the Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **4 mm**
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**

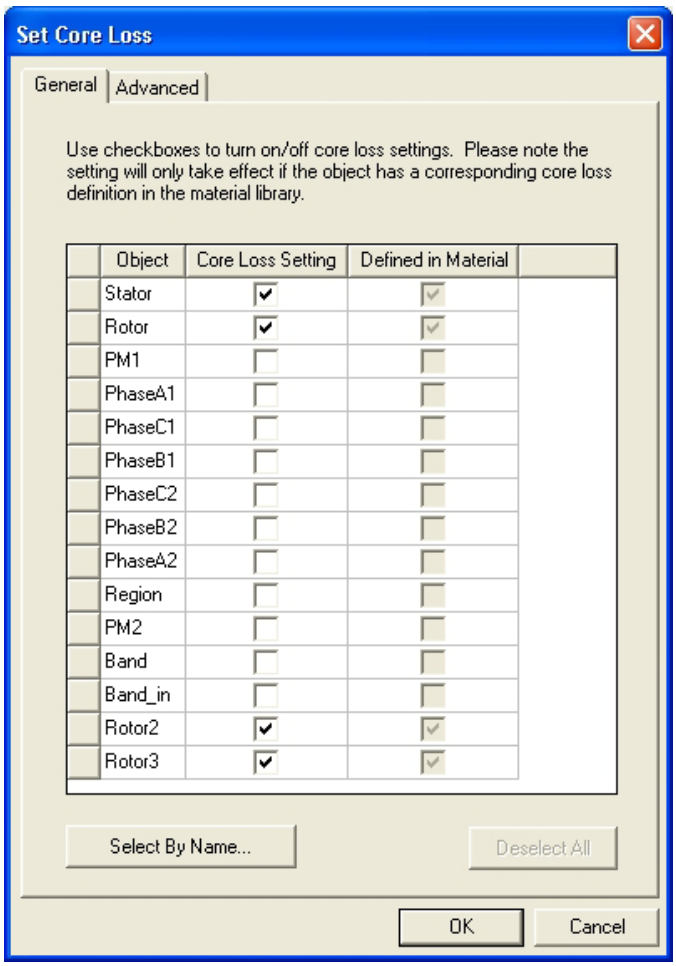




- **Mesh Operation for Stator**
  - Select the sheet **Stator** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based*
  - In Element Length based Refinement window,
    1. Name: **Stator**
    2. Restrict the Length of Elements: ☑ **Checked**
    3. Maximum Length of Elements: **4 mm**
    4. Restrict the Number of Elements: ☐ **Unchecked**
    5. Press **OK**

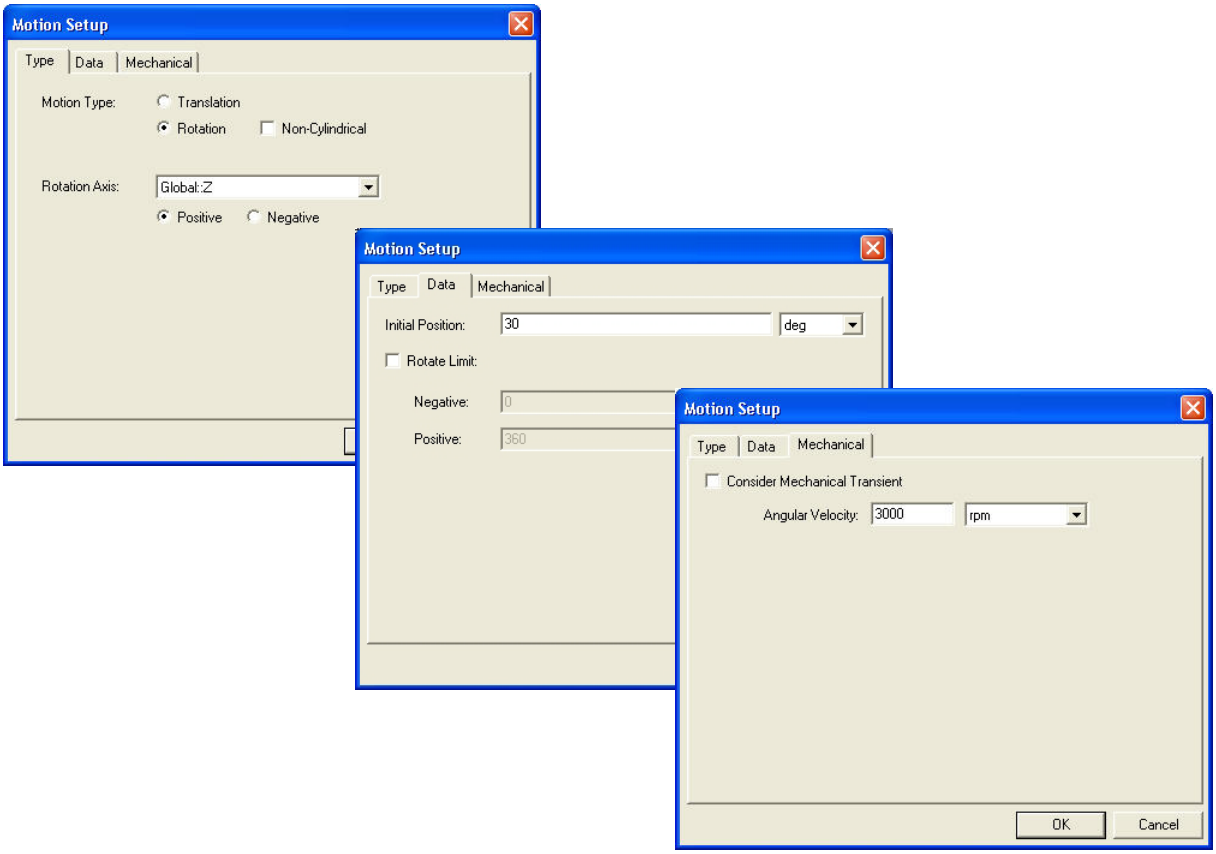
Set Core Loss

- The coreloss are not activated by default.
- Core Loss parameters must be defined to all objects in which core loss is to be calculated
- Since we have already defined core loss parameters in material definition, we can set core loss calculation for all those objects
- **To Set Core Loss Calculations**
  - Select the menu item *Maxwell 2D > Excitations > Set Core Loss*
  - In Set Core Loss window,
    - Enable Core Loss Settings for **Stator, Rotor, Rotor2** and **Rotor3**
    - Press **OK**



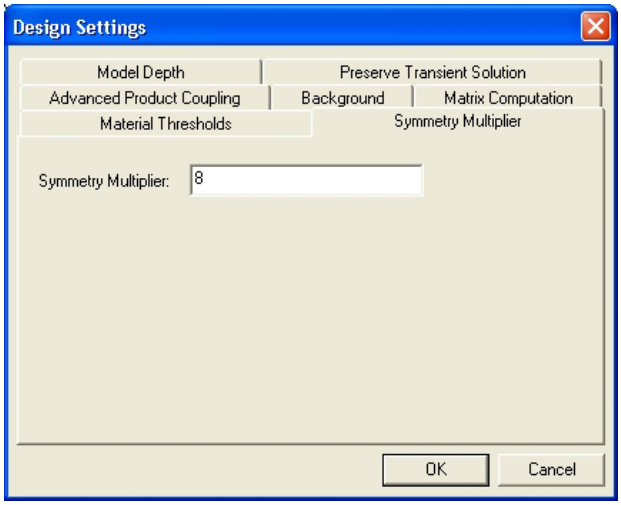
Assign Motion

- **To Assign Motion**
  - Select the sheet **Band** from the history tree
  - Select the menu item *Maxwell 2D > 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: **30 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**

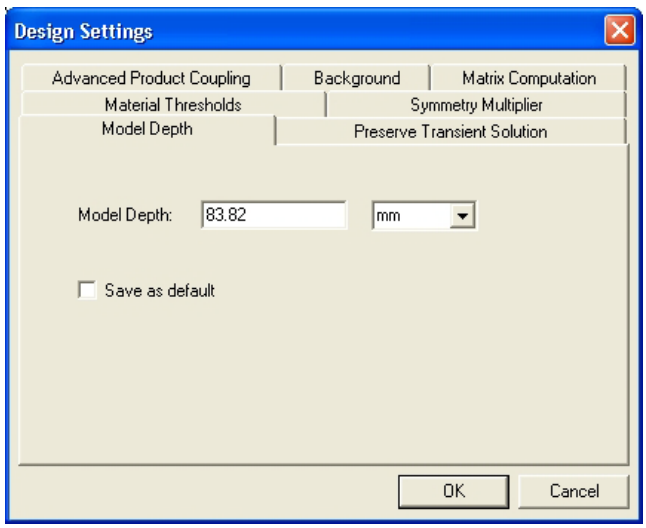


Set Symmetry Multiplier and Model Depth

- **To Set Symmetry Multiplier**
  - Select the menu item *Maxwell 2D > Design Settings*
  - In Design Settings window,
    1. **Symmetry Multiplier** tab
       - Symmetry Multiplier: **8**
       - Since we model 1/8th of the motor (our model spans on 45°), The force, torque will be rescaled to take into account the full model.



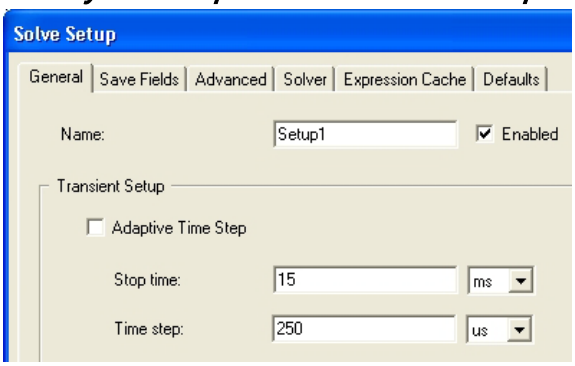
- **To Set Model Depth**
  - In Design Settings window,
    2. Select **Model Depth** tab
       - Model Depth: **83.82 mm**
    - Press **OK**

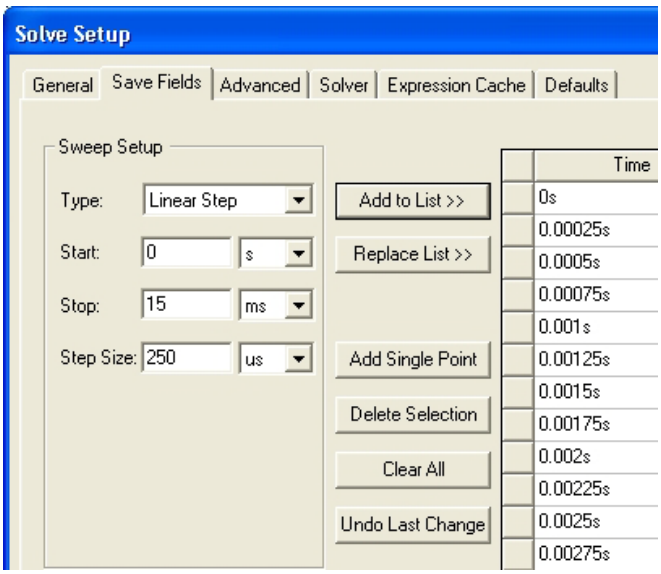


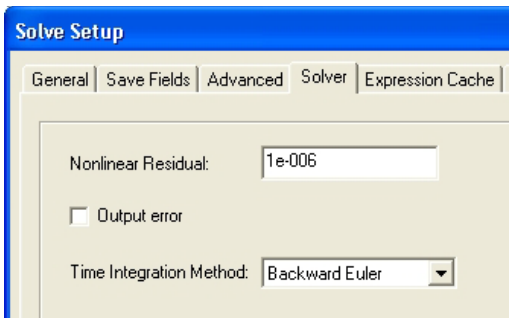
Solution Setup

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 1 or 2 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 2D > Analysis Setup > Add Solution Setup***
- In Solve Setup window,
  1. **General** tab
     - Stop Time: **15 ms**
     - Time Step: **250 us**
  2. **Save Fields** tab
     - Type : **Linear Step**
     - Start: **0 s**
     - Stop: **15 ms**
     - Step Size: **250 us**
     - Select **Add to List**
  3. **Solver** tab
     - Non-linear Residuals: **1e-6**
  4. Press **OK**



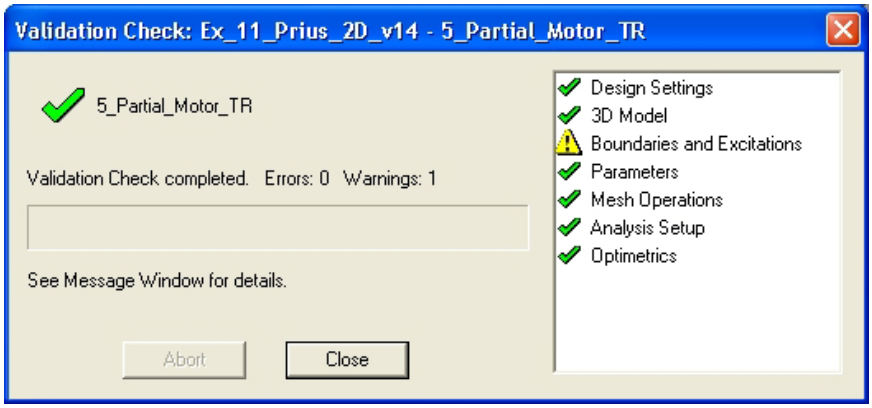




Validation Check

To Check the Validity of the Setup

- Select the menu item *Maxwell 2D > 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. Eddy effect are not taken into account in our design which is what we decided

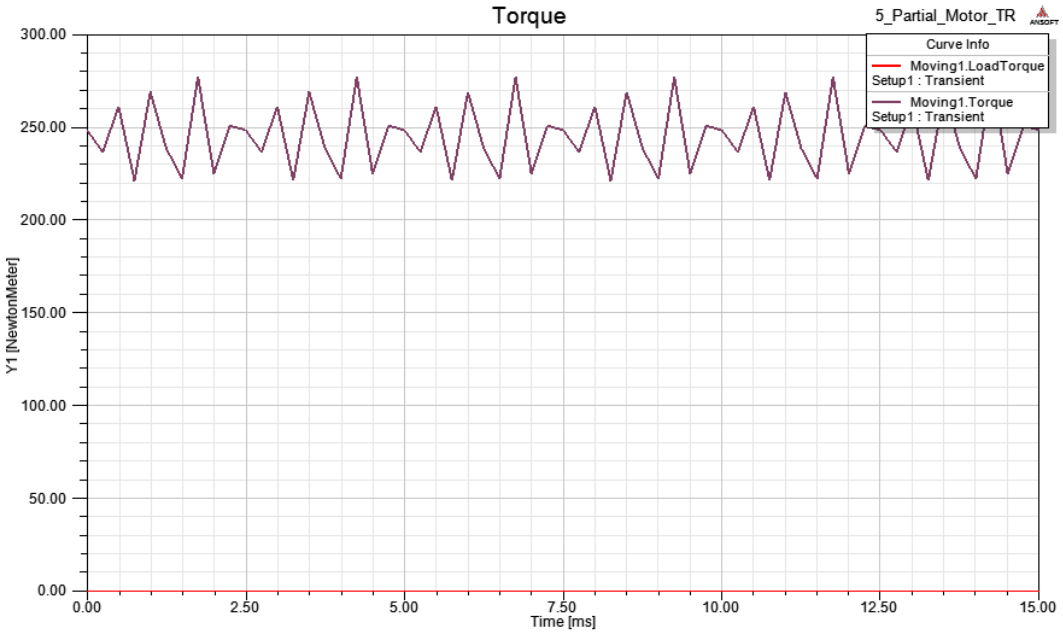
Solve

To Run Solution

- Select the menu item *Maxwell 2D > Analyze All*

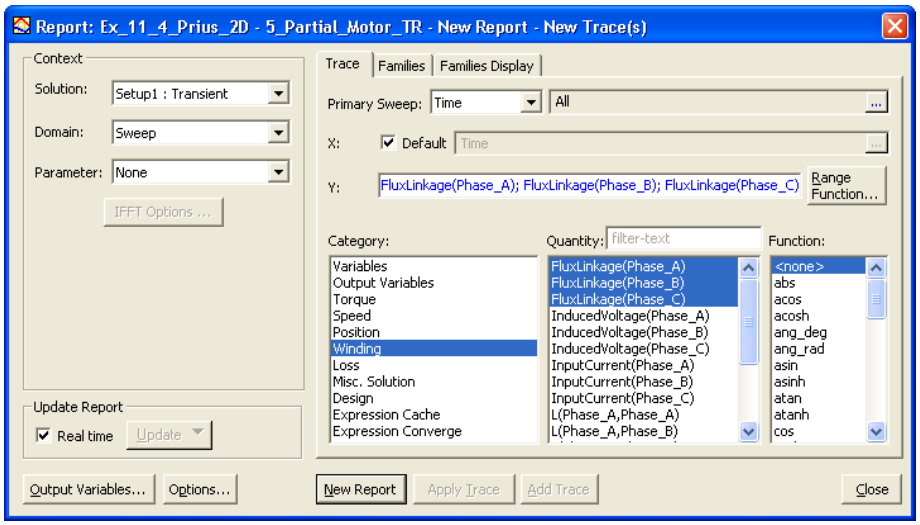
Postprocessing

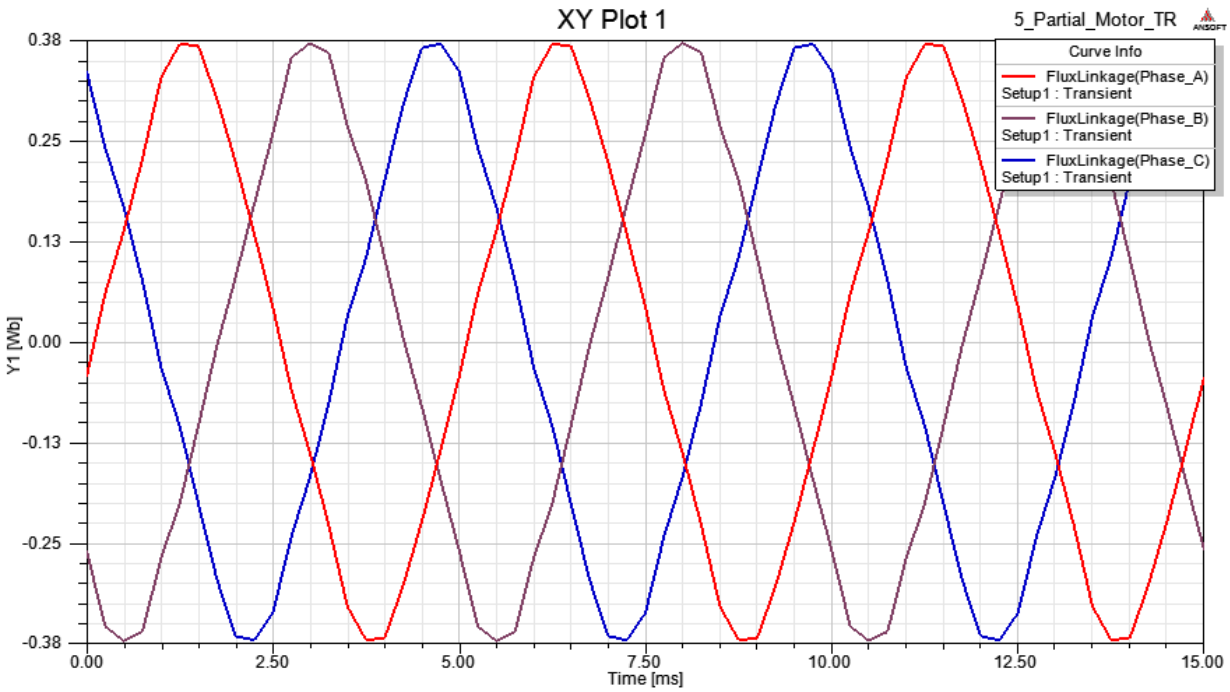
- Solve information appear in the profile of simulation
- **To View Solution Data**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
- Performance curves can be displayed during the simulation. They are updated at the end of each time steps.
- **To Create Torque Vs Time Plot**
  - Select the menu item *Maxwell 2D > Results > Create Quick Reports*
  - In Quick report window,
    1. Select **Torque**
    2. Press **OK**



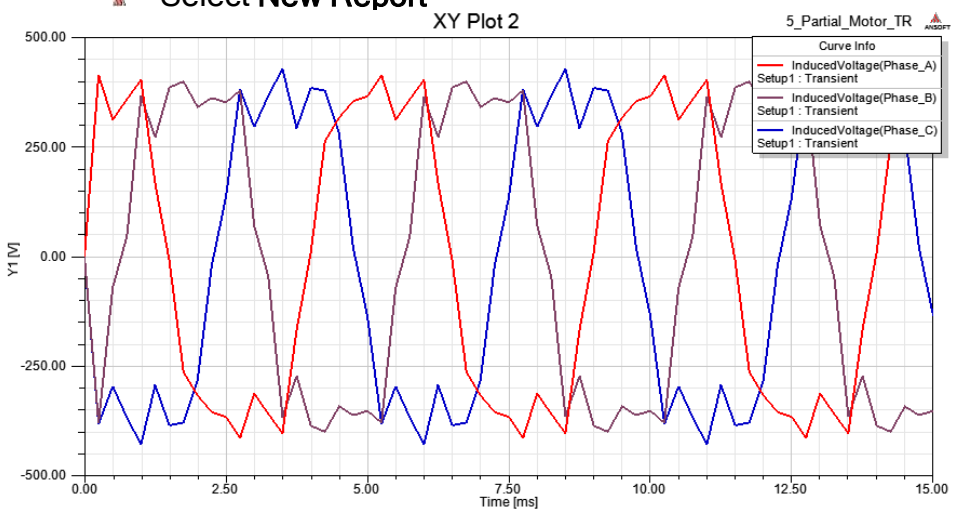
- The *LoadTorque* (in red) is zero as we are in motor mode.
- We can see that there are a lot of ripples in the Torque. The ratio between the torque and the torque ripples is almost 10 percent. This is due to the unique structure of the IPM motor (Internal Permanent Magnets). To limit the ripple, some manufacturers modify slightly the rotor shape around the magnets or add a second layer of internal magnets. Also the control strategy plays a big role into preventing the ripples.
- The torque value is around 240 N.m. This value is compatible with measurement.

- **Plot Flux Linkages Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **FluxLinkage(Phase_A)**, **FluxLinkage(Phase_B)** and **FluxLinkage(Phase_C)**
    - Select **New Report**

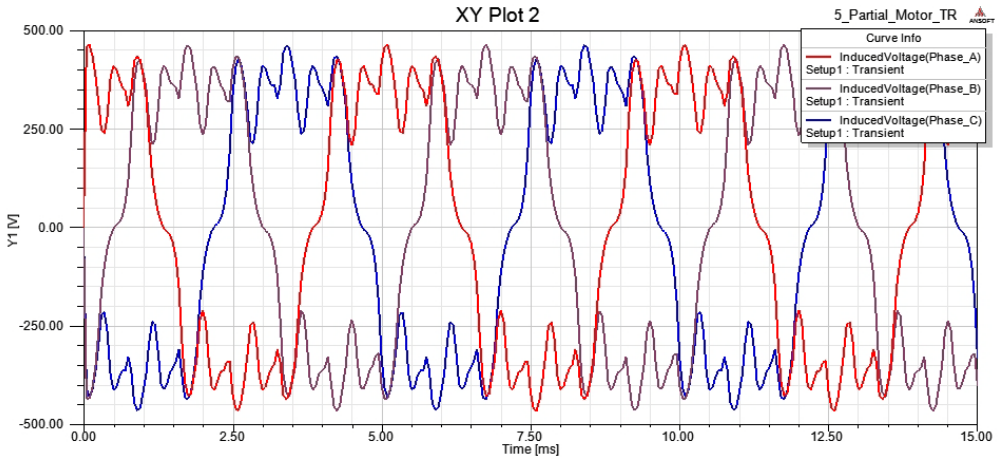




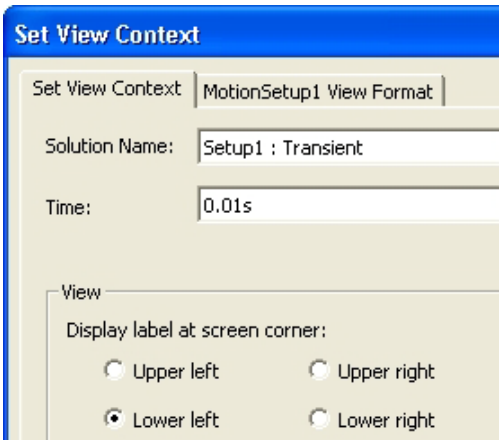
- **Plot Induced Voltages Vs Time**
  - Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
  - In report window,
    - Category: **Winding**
    - Quantity: Press **Ctrl** and select **InducedVoltage(Phase_A)**, **InducedVoltage(Phase_B)** and **InducedVoltage (Phase_C)**
    - Select **New Report**



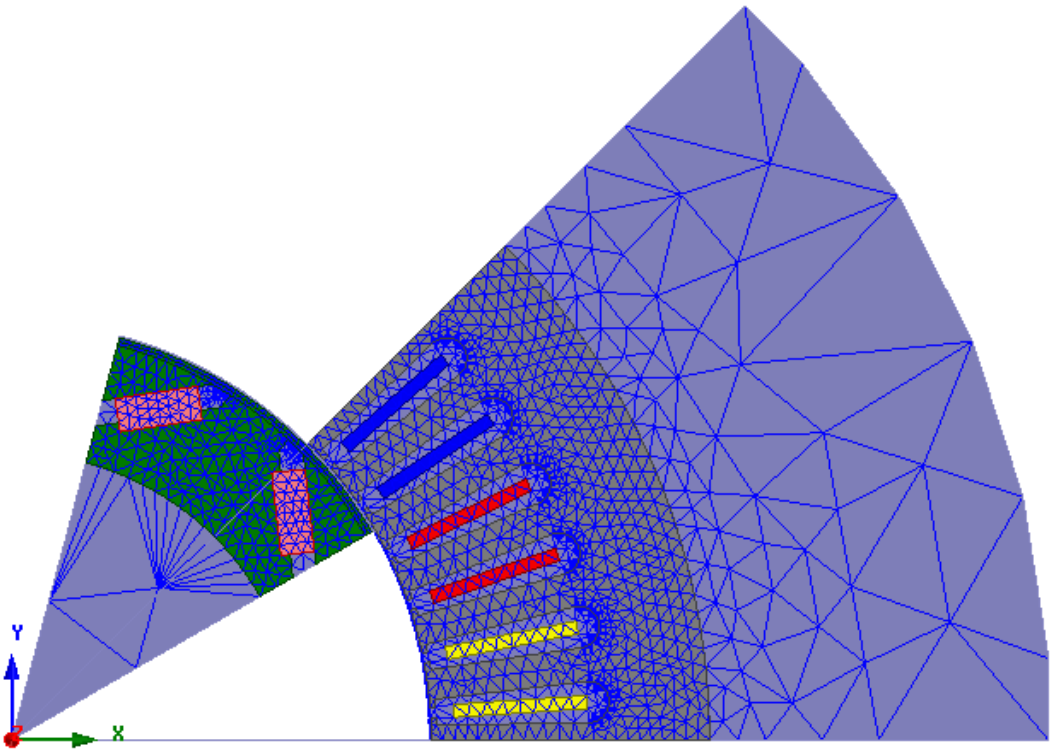
  - The curves are not really smooth. The reason is that the time step is too high. As the induced voltage is a derived quantity, Maxwell needs to derive the total flux ; the time steps is way to high to have accurate Induced Voltage. If you re run the simulation with a time steps of 50us (instead of 250 us), the Induced Voltage will have a more realistic shape:



- **Plot Mesh**
  - Select the menu item *View > Set Solution Context*
  - In Set View Context window,
    1. Set Time to **0.01 s**
    2. Select **OK**

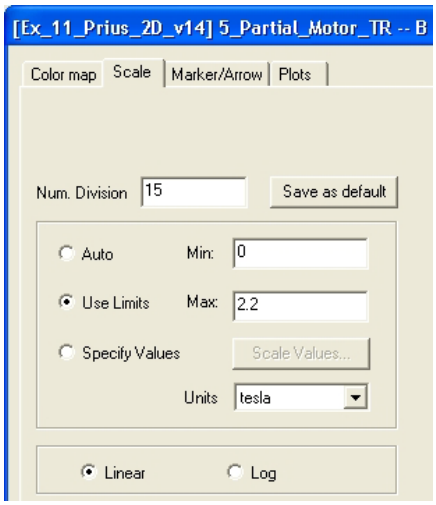


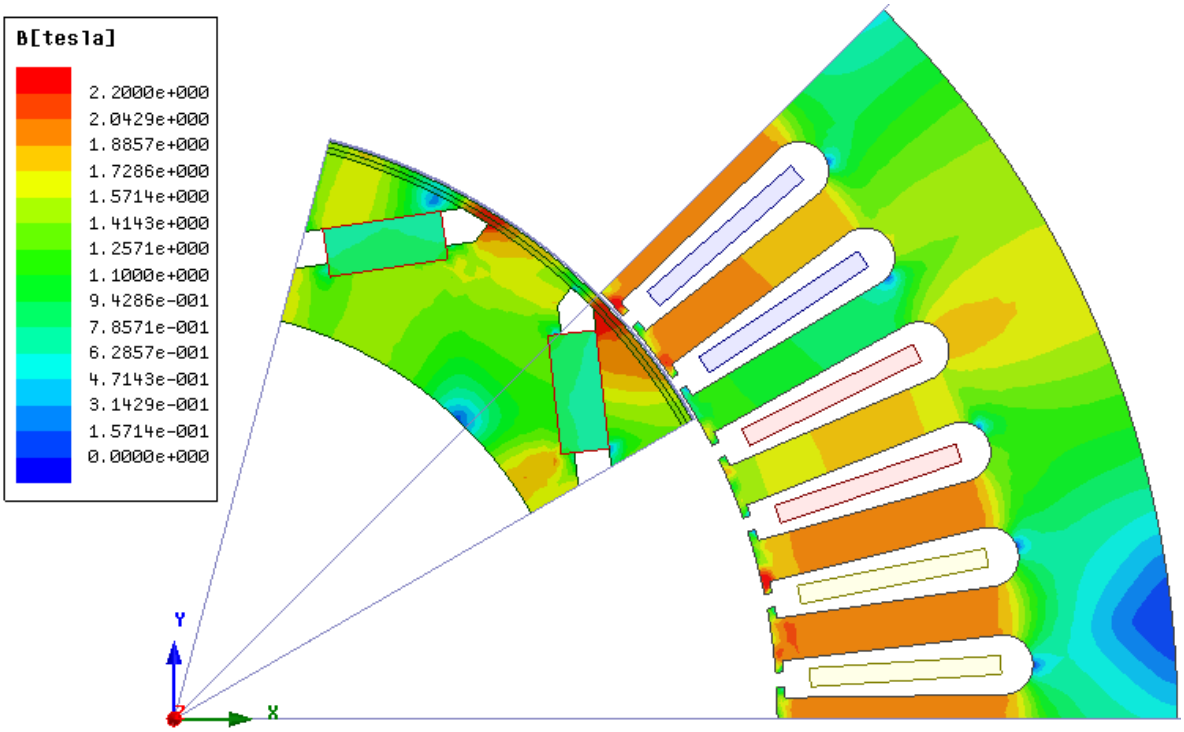
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Plot Mesh*
  - In Create Mesh Plot window,
    - Press **Done**



Motors - Permanent Magnet Motor (Prius IPM)

- **Plot Flux Density**
  - Press **Ctrl** and select the objects **Rotor, Rotor1, Rotor2, PM1, PM2** and **Stator**
  - Select the menu item *Maxwell 2D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    - Press **Done**
  - Double click on the legend to modify its attributes
  - In the window,
    - **Scale** tab
      - Select **User Limits**
        - Min: **0**
        - Max: **2.2**
    - Press **Apply** and **Close**



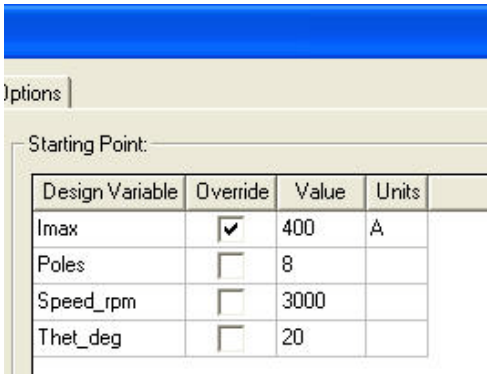


Parametric Study

- 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 2D > 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**
    4. Stop: **60**
    5. Step: **15**
    6. Select **Add**

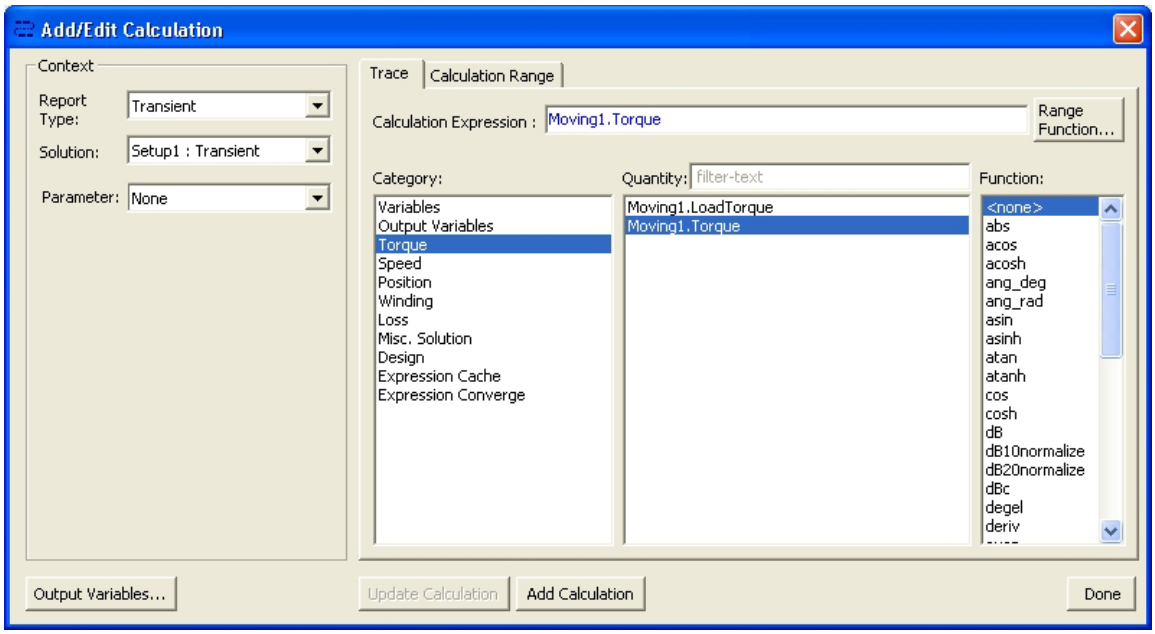


  - In Setup Sweep Analysis window, **General** Tab
    - This panel enables the user to change a design variable. For instance, if you wish to run the parametric sweep with a peak winding current of **400 A**, select the *Override* button, and change the current value.

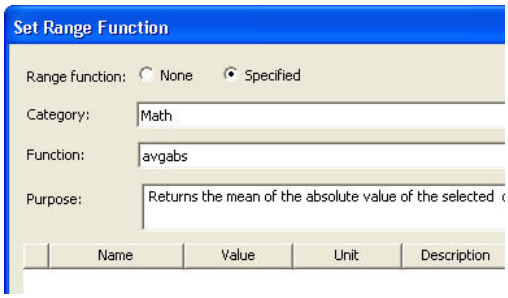


Motors - Permanent Magnet Motor (Prius IPM)

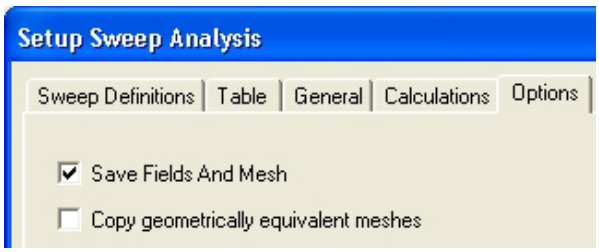
- **Calculation** tab, Select **Setup calculation**
- In Add/Edit Calculation window,
  - Category: **Torque**
  - Quantity: **Moving1.Torque**



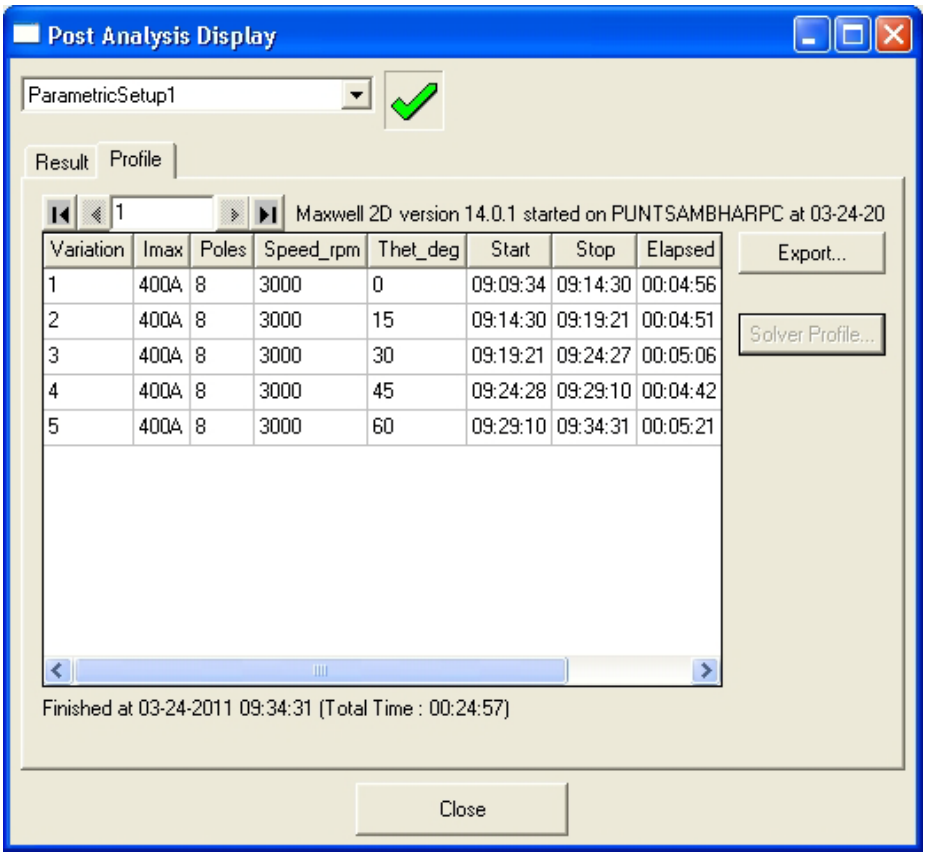
  - Select the button **Range Function**
  - In Set Range Function window,
    - Category: **Math**
    - Function: **avgabs**
    - Press **OK**
  - Select **Add Calculation**
  - Press **Done**



- In Setup Sweep Analysis window, select **Options** tab
  - Save Fields And Mesh: ☑ **Checked**
  - Press **OK**



- **Run Parametric Solution**
  - Expand the Project Manager tree to view **Optimetrics**
  - Right click on the tab **ParametricSetup1** and select **Analyze**
  - The Calculation Process will be done for all variations
- **View Results**
  - Select the menu item *Maxwell 2D > Optimetrics Analysis > Optimetrics Results*
  - The results can also be monitored while solution is running



  - All the plots are now available for any variation

Cogging Torque

- The Cogging Torque corresponds to the torque due to the shape of the teeth and the permanent magnets, when all the coils excitations are 0. The torque is a very small value in regard to the full load torque. Its computation is very sensitive to the mesh, as its value is in the same order of magnitude of the mesh noise.
- To compute accurately the cogging Torque, one could solve a parametric sweep in Magnetostatic (the input parameter being the angle between rotor and stator). This method will not lead to excellent results as the error due to the mesh will be different for each position (the mesh will change for every row).
- The preferred method is the use of the transient solver with motion:
  - We will move the rotor at the speed of 1 deg/s
  - The mesh will remain unchanged for all the positions thanks to the Band object : the mesh inside the Band object will rotate with the rotor
  - Each time step will be independent of the other
  - The adaptive mesh will not be used therefore the simulation time will be shorter

Copy Design

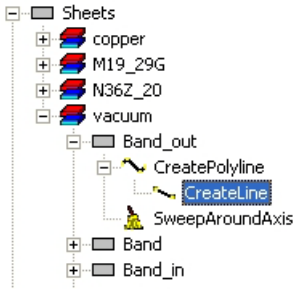
- **To Copy Design**
  - Select the Maxwell Design **"5_Partial_Motor_TR"** from the Project Manager tree, right click and select **Copy**
  - Right click on Project Name and select **Paste**
  - Change the name of the design to **"6_Partial_Motor_CT"**

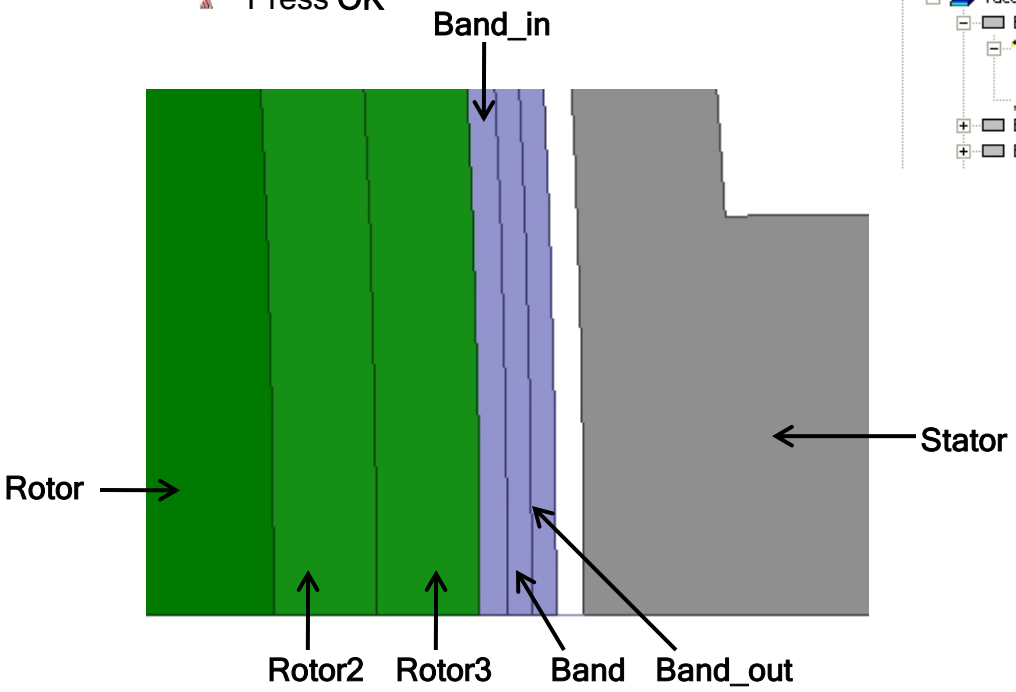
Create Outer Band

- We derive the set up for the cogging torque calculation from the Full load setup. We will change the speed, the excitations and some meshing operations.
- Since the mesh has to be well defined in the air gap, we will add an object so that we have enough layers of element:
- **Copy Band**
  - Select the sheet **Band** from history tree
  - Select the menu item *Edit > Copy*
  - Select the menu item *Edit > Paste*
  - Change the name of the sheet to **Band_out**

Motors - Permanent Magnet Motor (Prius IPM)

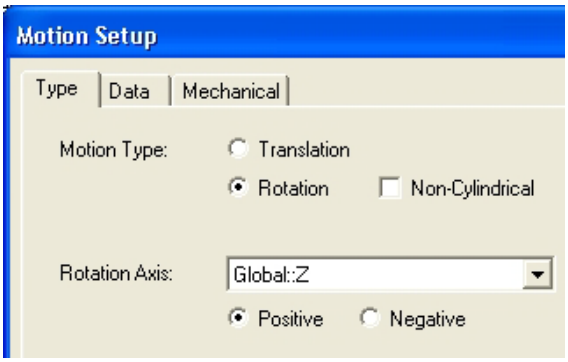
- **To Edit Band_out Object**
  - Expand the history tree for the sheet **Band_out**
  - Double click on the command **CreateLine** to open its properties
  - In Properties window,
    - Change Point2 position to **80.75,0,0**
    - Press **OK**

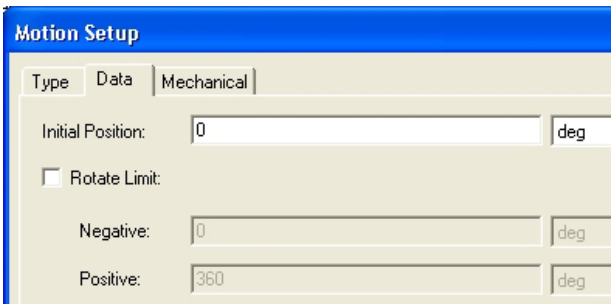


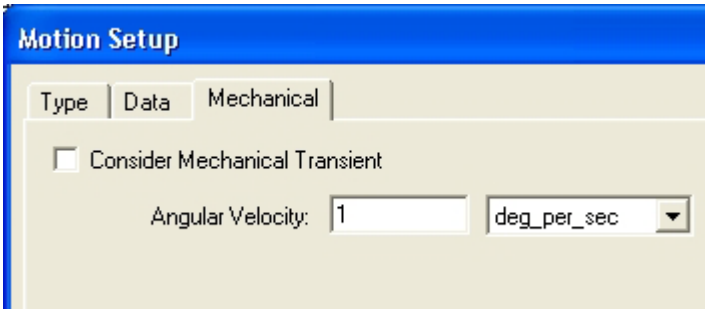


Modify Motion Setup

- **Delete Existing Motion Setup**
  - Select the menu item *Maxwell 2D > Model > Motion Setup > Delete All*
- **Assign Motion**
  - Select the sheet **Band** from the history tree
  - Select the menu item *Maxwell 2D > Model > Motion Setup > Assign Band*
  - In Motion Setup window,
    1. **Type** Tab
       - Motion Type: **Rotation**
       - Rotation Axis: **Global:Z**
       - Select **Positive**
    2. **Data** Tab
       - Leave it unchanged
    3. **Mechanical** Tab
       - Angular Velocity: **1 deg_per_sec**
    4. Press **OK**

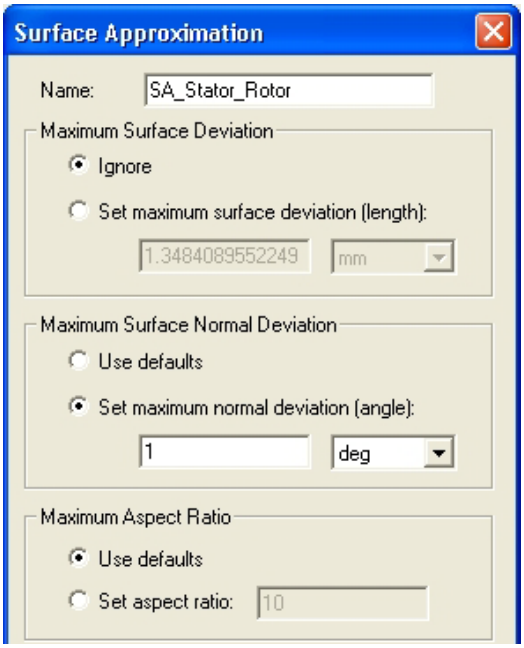






Edit Mesh Operations

- We also need to change the meshing operations. The mesh density that was good enough to compute the full load torque won't be accurate enough for the cogging torque
- **To Edit Mesh Operations**
  - Expand the Project Manager tree to view **Mesh Operations**
  - Double click on the mesh operation **Stator** to change its parameters
  - In Element Length Based Refinement window,
    - Change Maximum Length of Elements to **3mm**
  - Similarly change Maximum Length of Elements for operation **Rotor** to **3mm**
- **Add New Mesh Operation**
  - Press **Ctrl** and select the sheets **Stator, Rotor, Rotor2** and **Rotor3** from history tree
  - Select the menu item *Maxwell 2D > Mesh Operations > Assign > Surface Approximation*
  - In Surface Approximation window,
    - Name: **SA_Stator_Rotor**
    - Set Maximum Normal Deviation (angle): **1 deg**
    - Press **OK**



Remove Coils

- We can delete the coils as they are not needed in the cogging torque simulation
- **To Delete Coils**
  - Press **Ctrl** and select all coils from history tree
  - Select the menu item *Edit > Delete* or press **Delete** button from keyboard
  - Select the menu item *Maxwell 2D > Excitations > Delete All*

Modify Analysis Setup

- One pole pair takes 7.5 mech degrees. We will solve over *15 s* in order to have two periods (remember: the speed is 1 deg/s)
- We set the time step to *0.125 s* to have a very smooth curve. An higher time step is still valid if you want a faster result.
- **To Modify Analysis Setup**
  - Expand the Project Manager tree to view **Analysis**
  - Double click on the tab **Setup1** to modify its parameters
  - In Solve Setup window,
    - **General** tab
      - Stop Time: **15 s**
      - Time Step: **0.125 s**
    - **Save Fields** tab
      - Select the button **Clear All**
    - **Solver** tab
      - Nonlinear Residuals: **1e-4**
    - Press **OK**

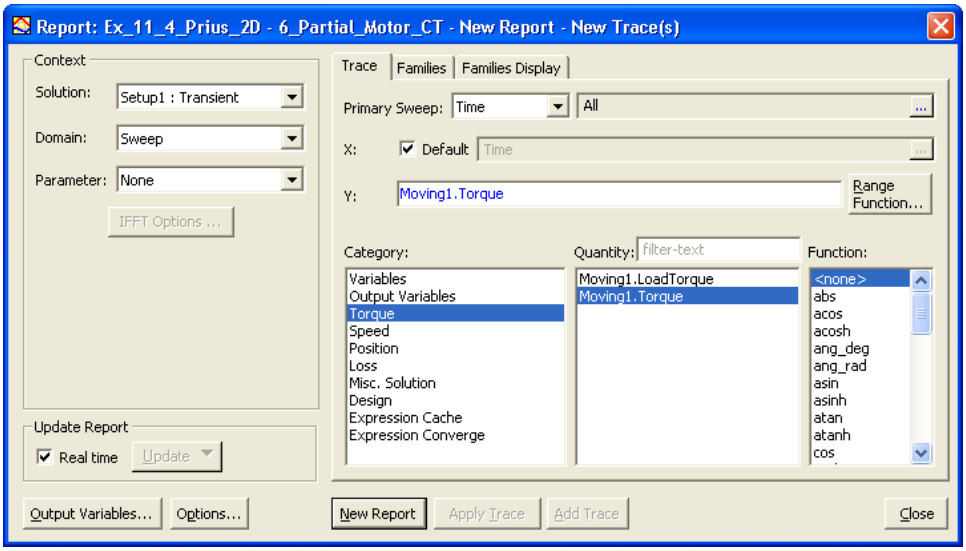
Solve

- **To Run Solution**
  - Expand the Project Manager tree to view **Analysis**
  - Right click on the tab **Setup1** and select **Analyze**

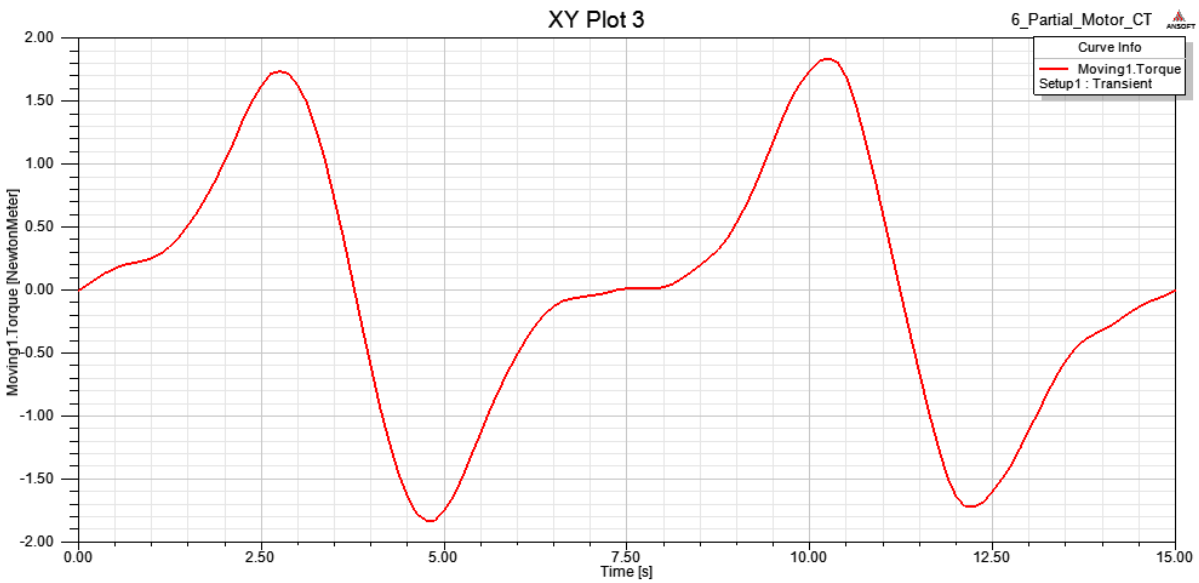
PostProcess

Plot Torque Vs Time

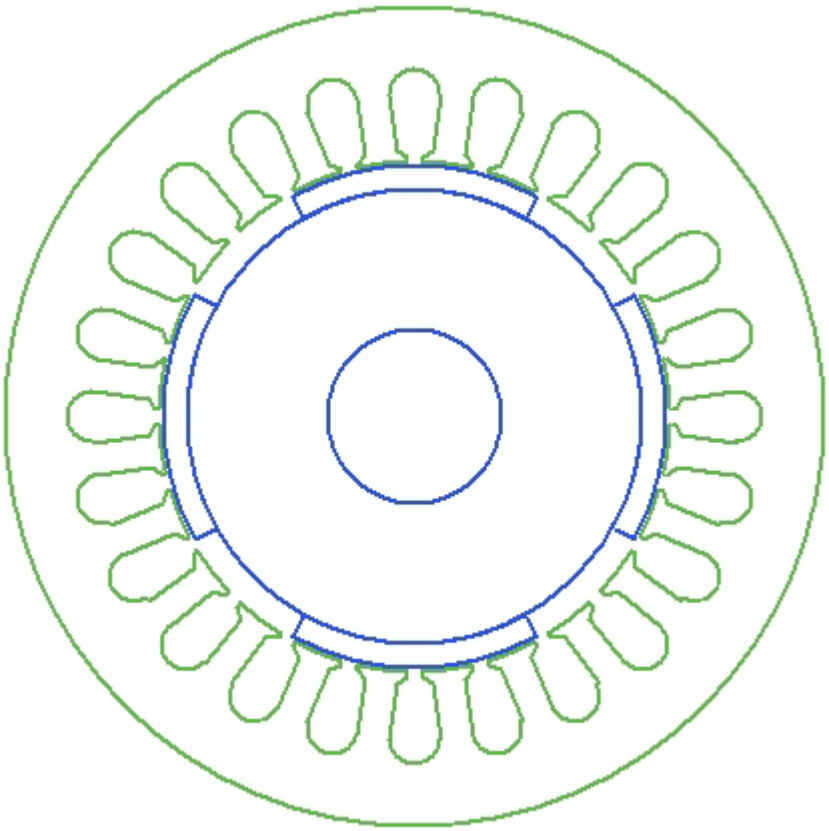
- Select the menu item *Maxwell 2D > Results > Create Transient Report > Rectangular Plot*
- In Report window,
  - Category: **Torque**
  - Quantity: **Moving1.Torque**
  - Select **New Report**



- The Torque trace appears. As expected, the cogging torque is periodical. The peak value is about *1.75 N.m*.



Optimetrics Analysis with RMxprt



Brushless Permanent Magnet DC Motor

Launching Maxwell

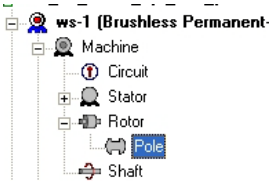
- To access Maxwell:
  1. Click the Microsoft **Start** button, select **Programs**, and select **Ansoft > Maxwell 15.0** and select **Maxwell 15.0**

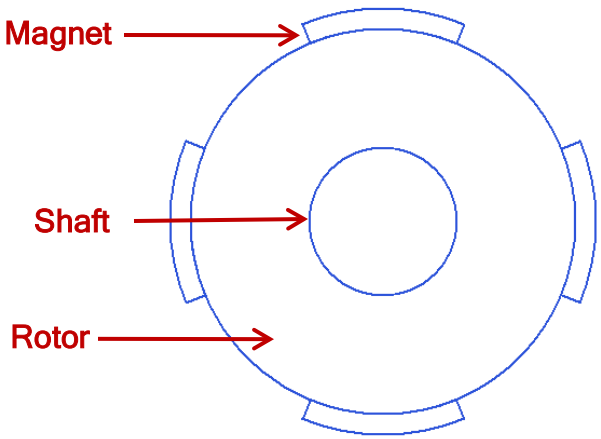
Open Input File

- To Open File
  - Select the menu item *File > Open Example*
  - Browse to the location **RMxprt / bldc** and open the file **"ws-1.mxwl"**
  - Save the file at any other location with the name **"Ex_11_5_BLDC_Opt.mxwl"**

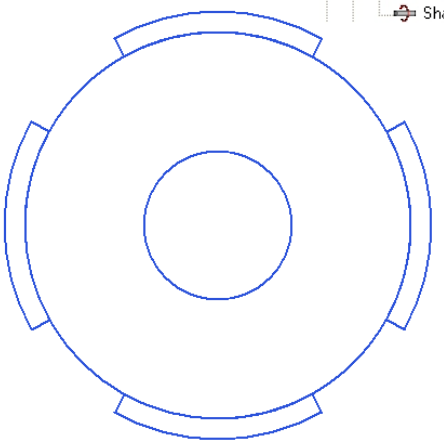
Variables for Optimetrics

- **Pole Embrace**
  - Pole Embrace indicates how much of a pole is covered by a magnet
  - We will study the effect of variation in Pole Embrace on **Cogging Torque**
  - The value of PoleEmbrace is set to **0.65** in input file
  - This can be checked by expanding Project Manager tree to view **Machine > Rotor > Pole**
  - Double click on the tab **Pole** to view Pole Embrace value
    - Change the value to **0.5** to see the effect.
    - Set the value to **0.65**
    - Press **OK**





Pole Embrace: 0.5



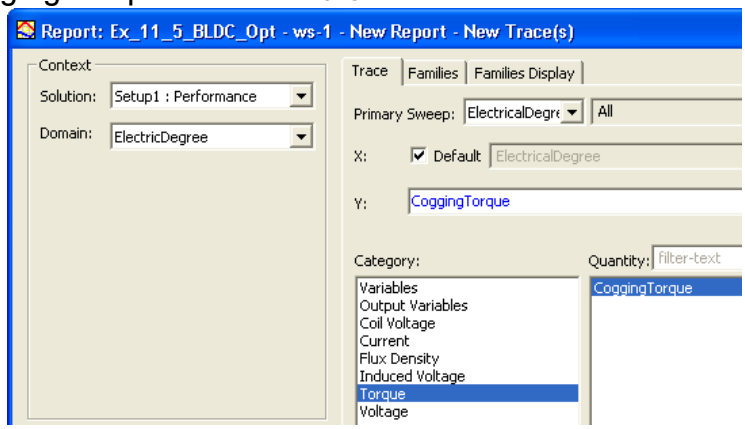
Pole Embrace: 0.65

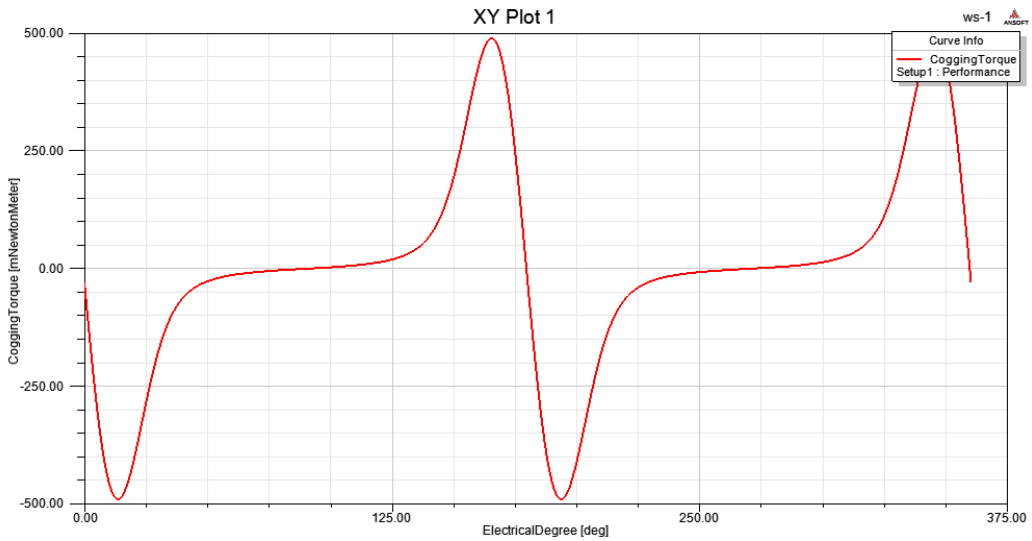
Analyze

- To Run the Solution
  - Select the menu item *RMxprt > Analyze All*

Results

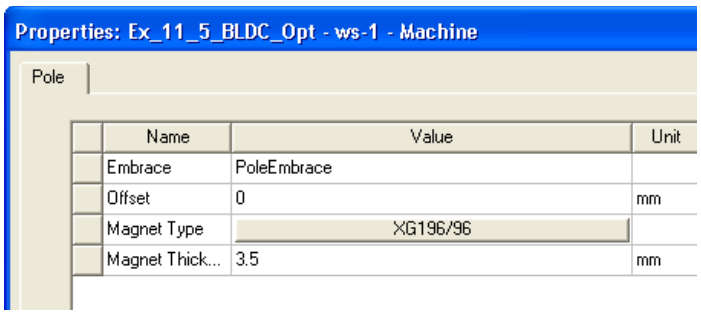
- Plot Torque Vs Time
  - Select the menu item *RMxprt > Results > Create RMxprt Report> Rectangular Plot*
  - In Report window,
    - Category: **Torque**
    - Quantity: **Cogging torque**
    - Select **New Report**
  - Maximum value of Cogging Torque is around **0.5Nm**



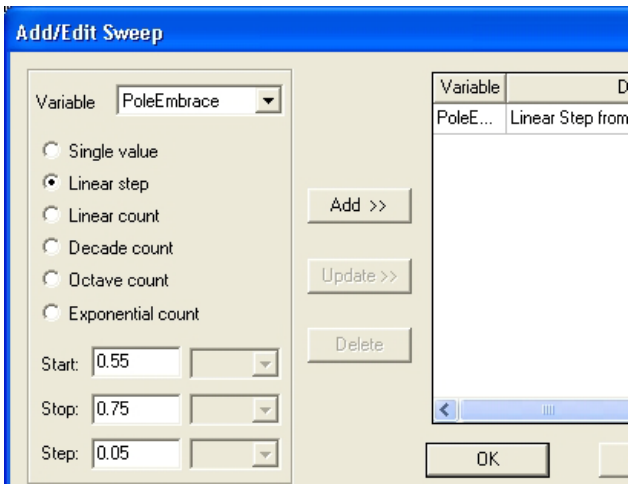


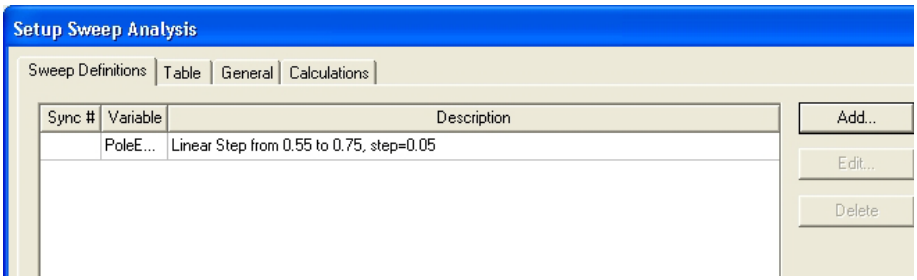
Parametric Analysis

- **Set Parameter for Pole Embrace**
  - Expand the Project Manager tree to view **Machine > Rotor > Pole**
  - Double click on the tab **Pole** to modify its parameter
  - In Properties window,
    - Set Embrace value to **PoleEmbrace**
    - In Add variable window,
      - Value: **0.55**
      - Press **OK**
    - Press **OK**

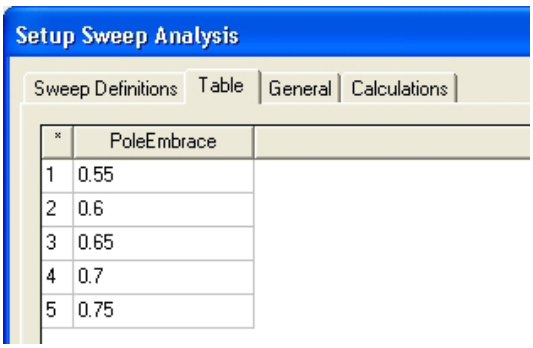


- **Set Parametric Sweep**
  - Select the menu item ***RMxprt > Optimetrics Analysis > Add Parametric***
  - In Setup Sweep Analysis window, select **Add**
  - In Add/Edit Sweep window,
    - Variable: **PoleEmbrace**
    - Select **Linear Step**
    - Start: **0.55**
    - Stop: **0.75**
    - Step: **0.05**
    - Select **Add>>**
    - Press **OK**

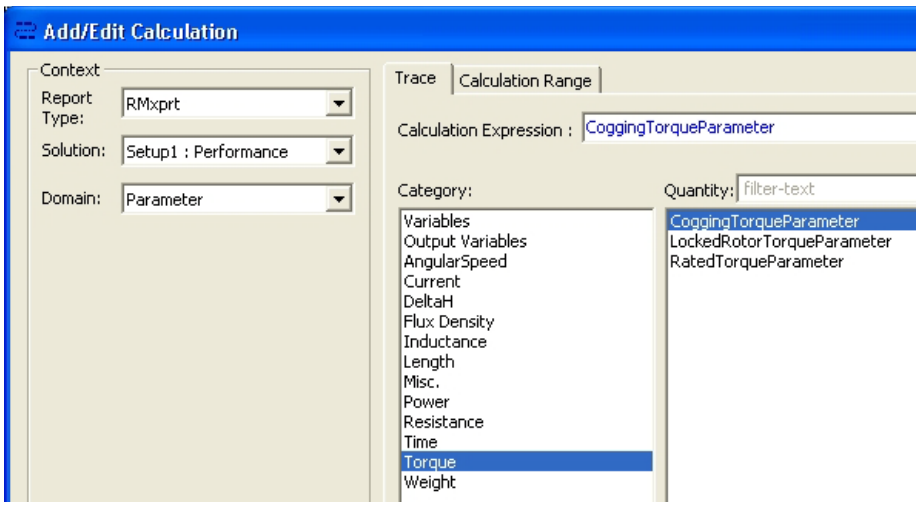


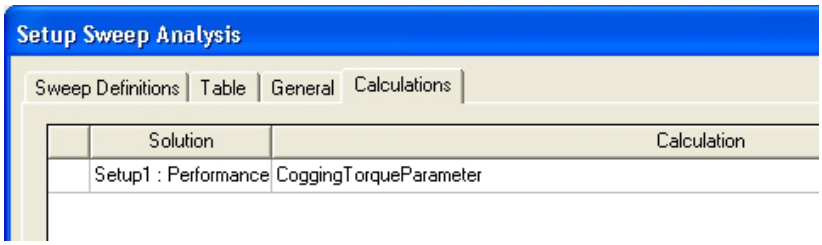


- In Setup Sweep Analysis window, select **Table** tab to view all variations for parametric analysis



- Select the **Calculations** tab and select the button **Setup calculations**
- In Add/Edit Calculations window,
  - Category: **Torque**
  - Quantity: **CoggingTorqueParameter**
  - Select **Add Calculation**
  - Press **Done**
- Press **OK** to close Setup Sweep Analysis window



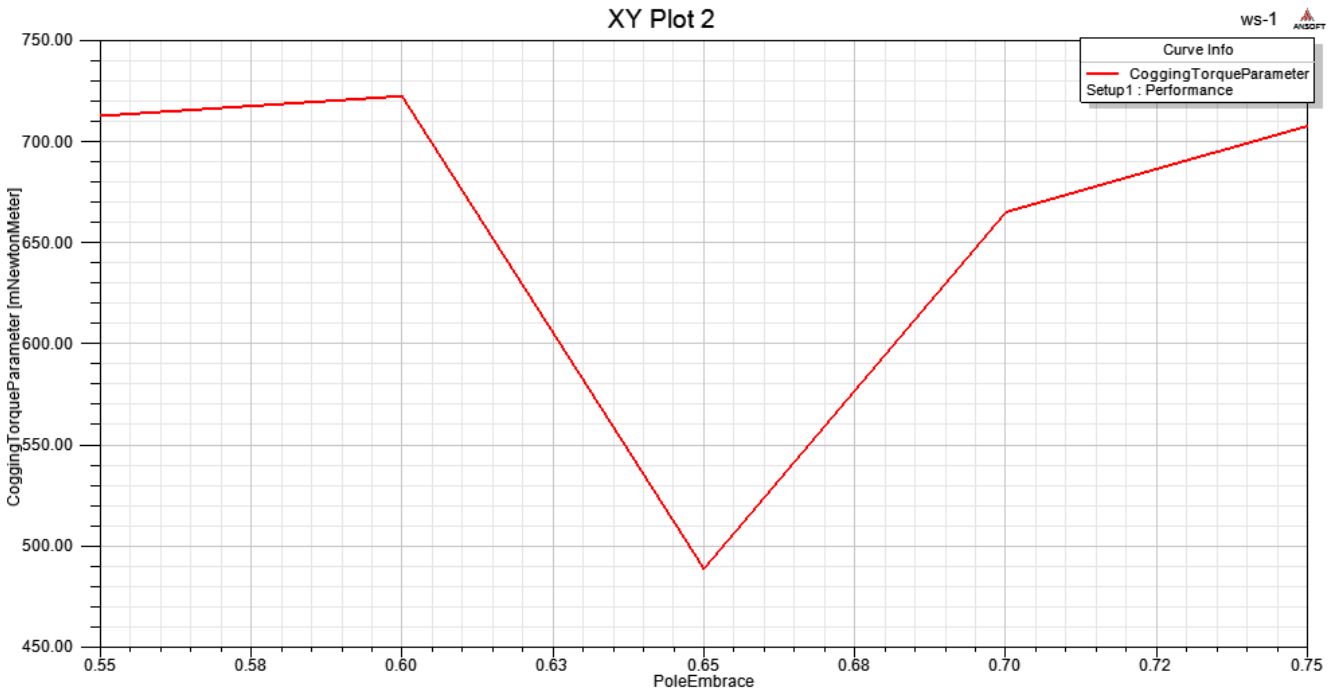


Run Parametric Analysis

- To Run Parametric Setup
  - Expand the Project Manager tree to view **Optimetrics**
  - Right click on the tab **ParametricSetup1** and select **Analyze**

View Parametric Results

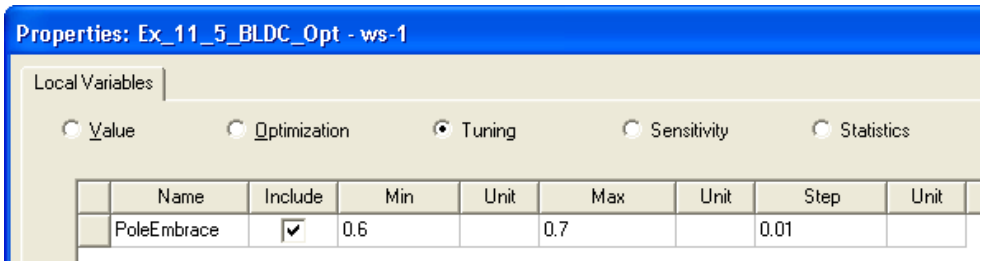
- To View Results
  - Select the menu item *RMxprt > Results > Create RMxprt Report > Rectangular Plot*
  - In Report window,
    - Domain: **Parameter**
    - Category: **Torque**
    - Quantity: **CoggingTorqueParameter**
    - Select **New Report**



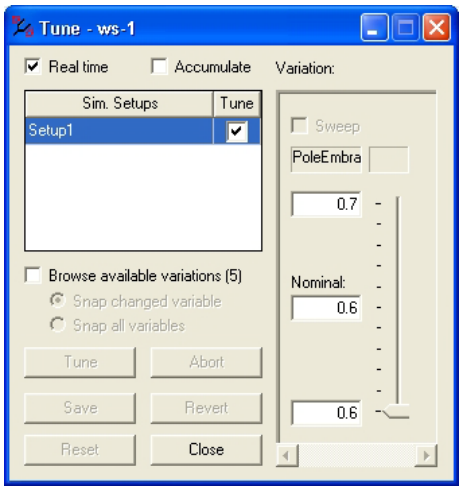
- **Note:** Plot is showing the value is minimum at **0.65.** But since no other variation has been solved between **0.6** and **0.7,** minimum location can not be predicted exactly

Setup Tuning Analysis

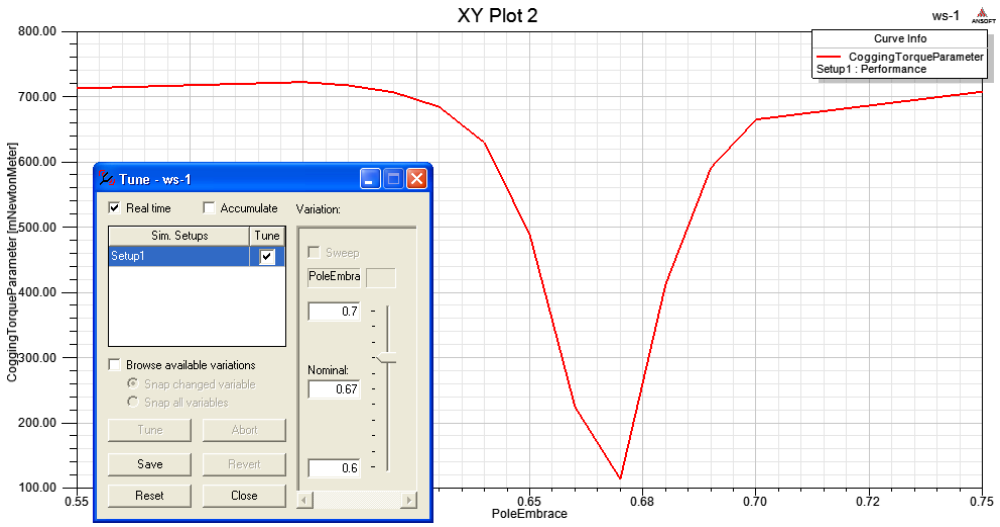
- Tuning Analysis allows to fine tune the pole embrace value between 0.6 and 0.7 to the real minimum value of cogging torque
- **To Set Parameters for Tuning Analysis**
  - Select the menu item *RMxprt > Design Properties*
  - In Properties window,
    - Select the radio button to **Tuning**
    - Check the **Include** field for the parameter PoleEmbrace
      - Min: **0.6**
      - Max: **0.7**
      - Step: **0.01**
    - Press **OK**



- **To Set Tuning Analysis**
  - Select the menu item *RMxprt > Optimetrics Analysis > Tune*
  - In Tune window,
    - Real Time: ☑ **Checked**
    - Browse Available Variations: ☐ **Unchecked**
    - The value of the variable is adjusted by sliding the slider bar

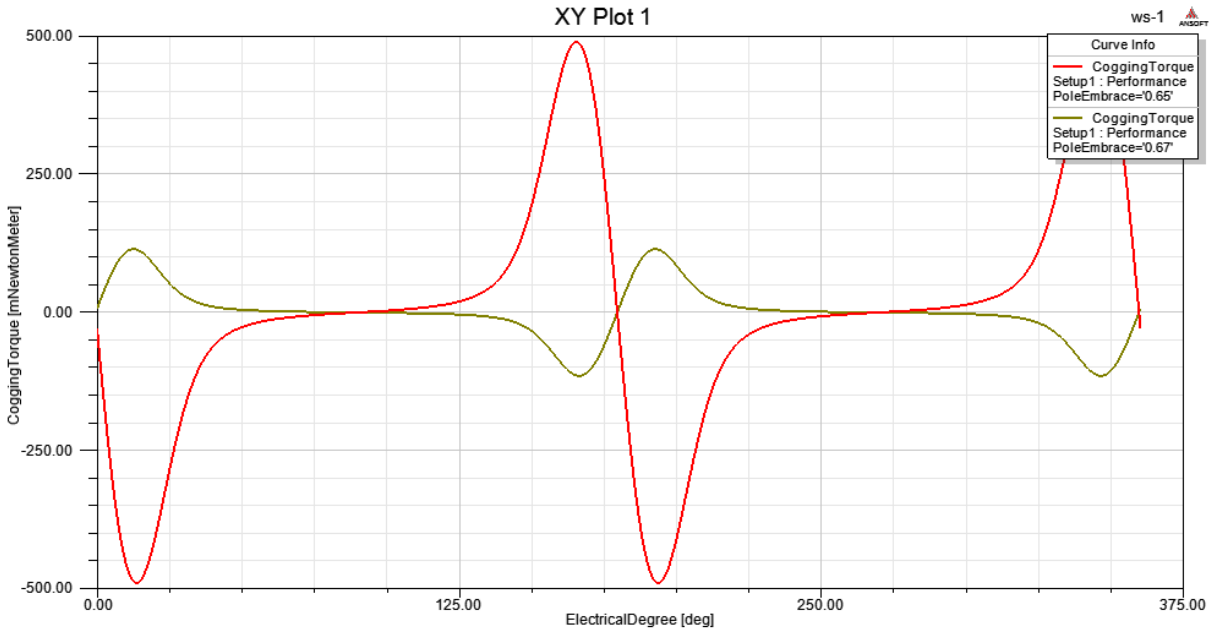


- The changes of the curve can be observed in real time immediately as the variable is being changed
- The analysis reveals that the real minimum occurs for the value of 0.67



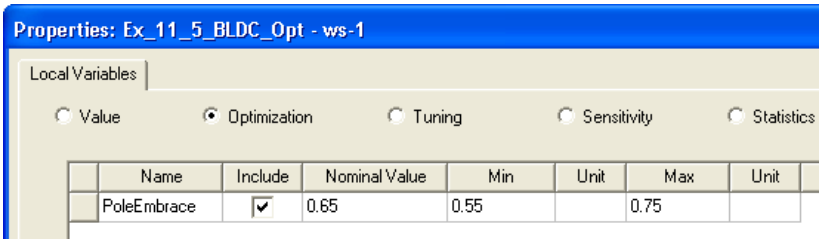
Compare Torque Plots

- **To Compare Previous and Tuned Torque Plot**
  - Right click on the Torque plot created previously and select **Modify Report**
  - In report window, select **Families** tab
    - Press **Ctrl** and select the value for PoleEmbrace to **0.65** and **0.67**
    - Select **Add Trace**

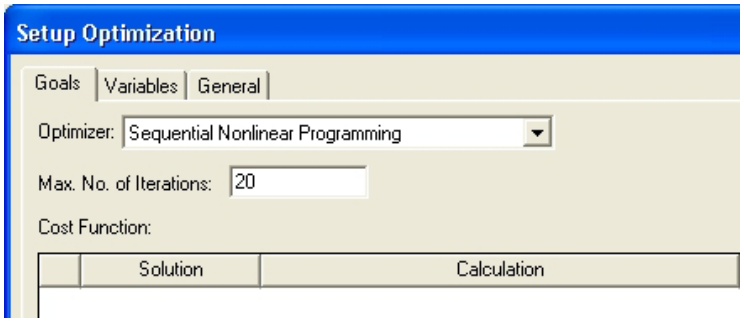


Optimization Analysis in RMxprt

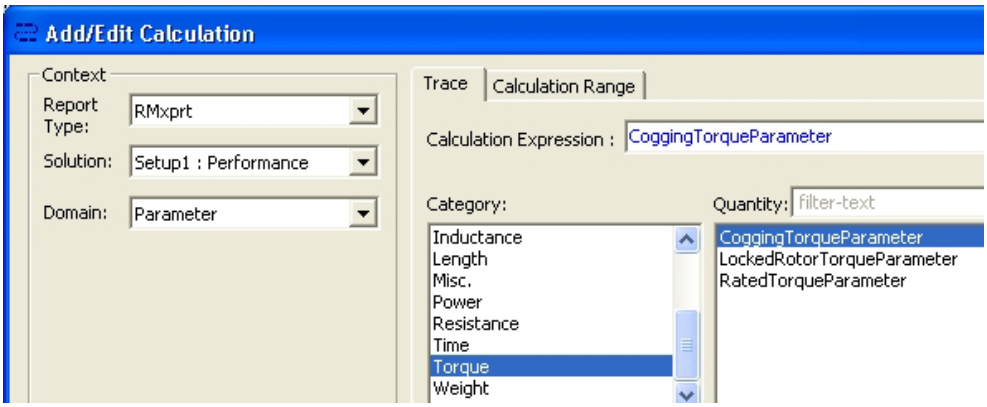
- Parametric analysis involves all possible variations of the input variable(s)
- For a refined analysis and for the analysis involving many variables, the number of needed variations can grow rapidly and may become computationally demanding
- An alternative approach is to use optimization analysis, where a defined algorithm searches for the optimum value automatically without a need of going through all possible variations
- The following example shows the application of Sequential Nonlinear Programming optimization algorithm for the minimization of the cogging torque
- **To Set Variables for Optimization**
  - Select the menu item *RMxprt > Design Properties*
  - In Properties window,
    - Change the value of PoleEmbrace to **0.65**
    - Set the radio button to **Optimization**
    - Include: ☑ **Checked**
    - Min: **0.55**
    - Max: **0.75**



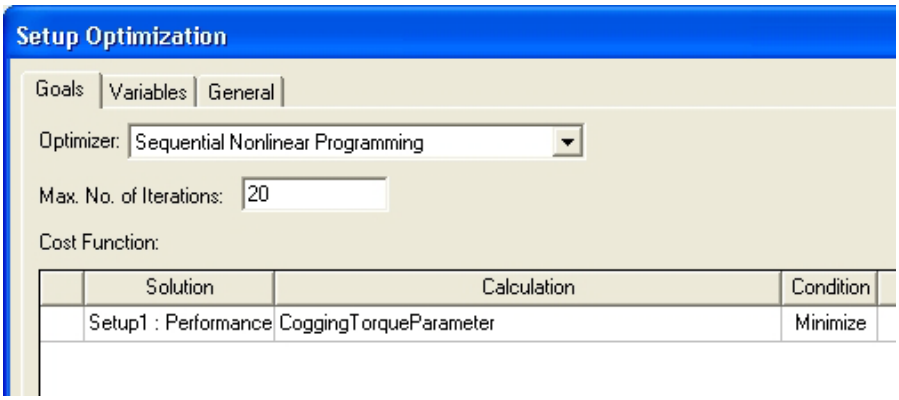
- **Setup Optimization**
  - Select the menu item *RMxprt > Optimetrics Analysis > Add Optimization*
  - In Setup Optimization window,
    - Optimizer: **Sequential Nonlinear Programming**
    - Max. No. of Iterations: **20**
    - Select **Setup Calculations**



- In Add/Edit Calculations window,
  - Category: **Torque**
  - Quantity: **CoggingTorqueParameter**
  - Select **Add Calculation** and **Done**



- Set the Condition to **Minimize**
- Press **OK**



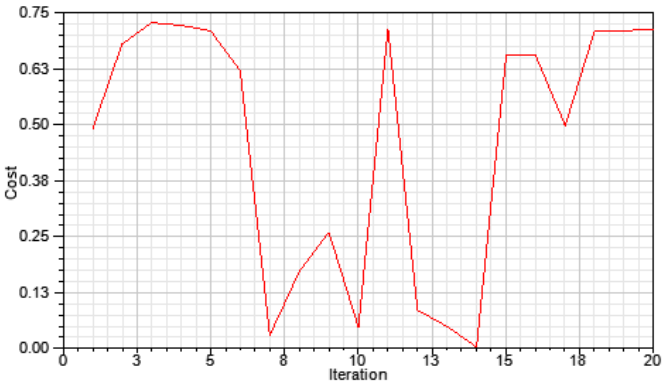
Run Optimization

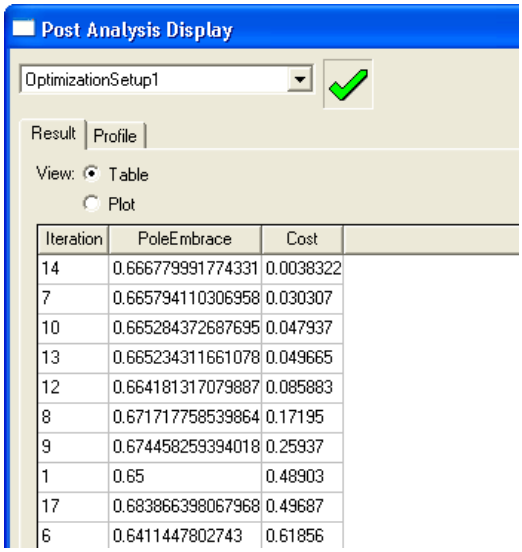
To Run Optimization Analysis

- Expand the Project Manager tree to view **Optimetrics**
- Right click on the tab **OptimizationSetup1** and select **Analyze**

View Optimization Results

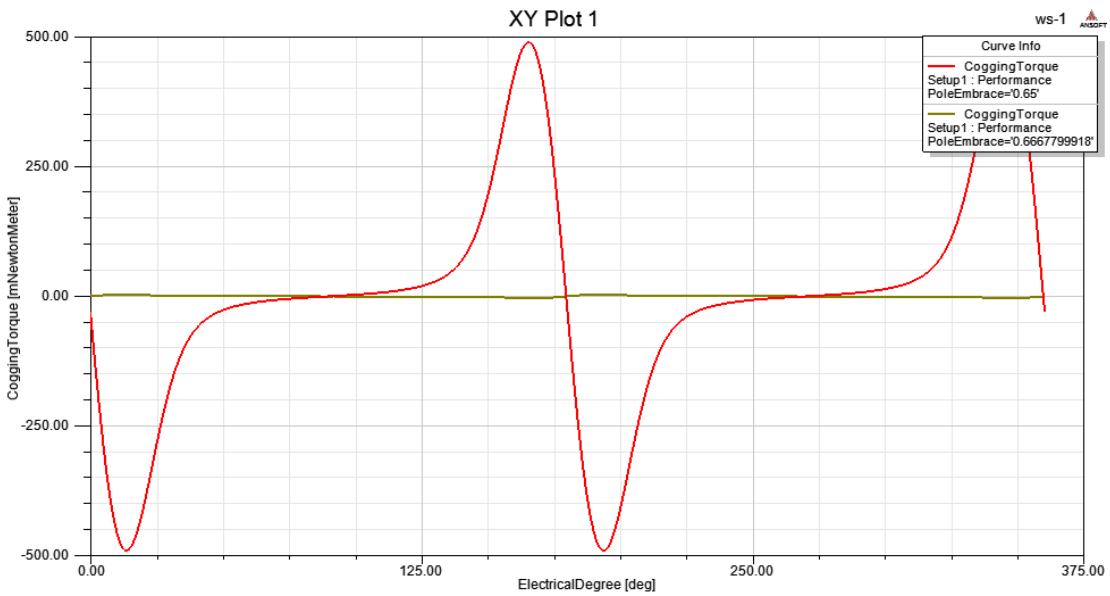
- To View Results
  - Right Click on the tab **OptimizationSetup1** and select **View Analysis Results**
  - In Post Analysis Display window,
    - Default view is set to **Plot**
    - Change the view to **Table**
    - It can be seen that Cost is minimum for **0.666**





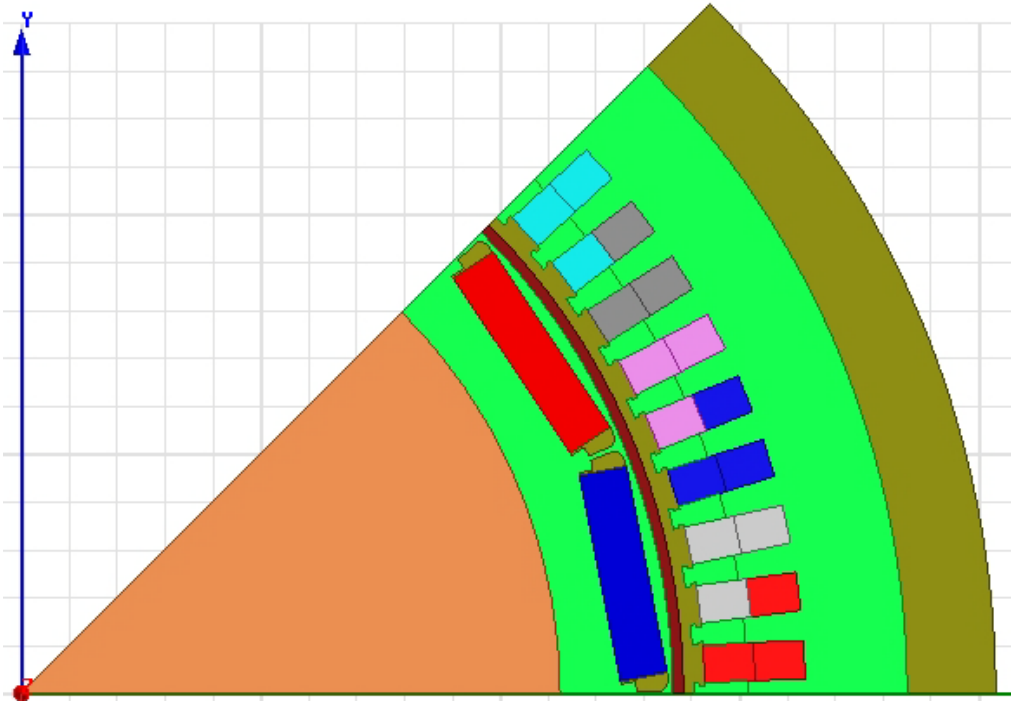
Iteration	PoleEmbrace	Cost
14	0.666779991774331	0.0038322
7	0.665794110306958	0.030307
10	0.665284372687695	0.047937
13	0.665234311661078	0.049665
12	0.664181317079887	0.085883
8	0.671717758539864	0.17195
9	0.674458259394018	0.25937
1	0.65	0.48903
17	0.683866398067968	0.49687
6	0.6411447802743	0.61856

- Compare Torque Plots
  - Compare Original values of Cogging Torque with Optimized value



About DesignXplorer

- DesignXplorer (DX) is a tool for designing and understanding the analysis response of parts and assemblies enabling the user to account for uncertainties in product design and use, and to determine how best to improve product reliability. These tasks are accomplished trough the use of response surfaces.
- DesingXplorer uses parameters as its basic language. Parametric data come from Maxwell, DesignModeler (DM) and numerous CAD systems in order to explore a wide range of responses based on a limited number of actual solutions.



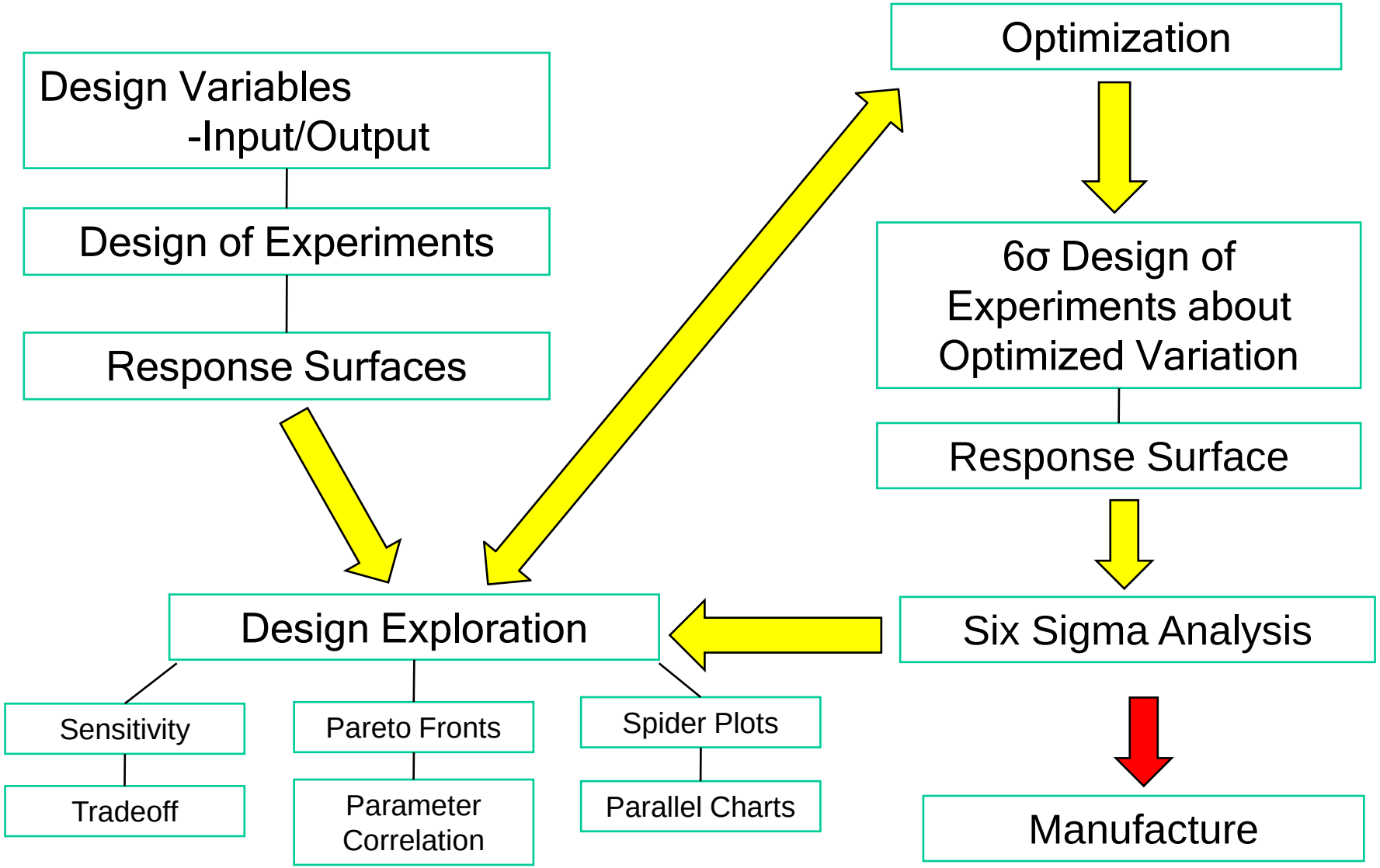
Solution Approach: ANSYS DesignXplorer

- Describes the relationship between:
  - Design variables
  - Performance
- Uses Design of Experiments (DOE), combined with Response Surfaces
- Information can be shared in easily understandable terms:
  - curves- surfaces – sensitivities

Steps

- **Step 1:** Create DOE
- **Step 2:** Create Response Surfaces
- **Step 3:**
  - view the parameters associated with the minimum and maximum values of your outputs
  - create a correlation matrix that shows the strength of the coupling between input/output parameters
  - set output objectives and see what input parameters will meet those objectives, i.e. optimize your design
- **Step 4:**
  - Perform Six Sigma analysis

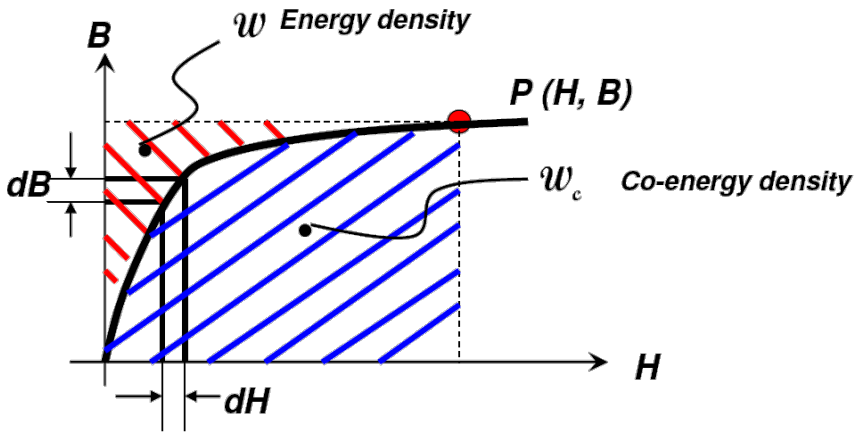
Solution Approach: ANSYS DesignXplorer



Torque Optimization Using DesignXplorer

- The Example used in this tutorial is of a Internal Permanent Magnet Motor
- The Objective of the Exercise is
  - Maximize Torque with a fixed line current
  - Minimize Magnet Volume
  - Maximize ratio: Torque / Volume
  - Obtain a maximum of the flux density in the stator tooth of 1.9 Tesla

$$T = \frac{dW(\Theta, i)}{d\Theta}\Big|_{i=const} = \frac{d}{d\Theta}\left(\int_v \left(\int_0^H B \bullet dH\right) dV\right)$$

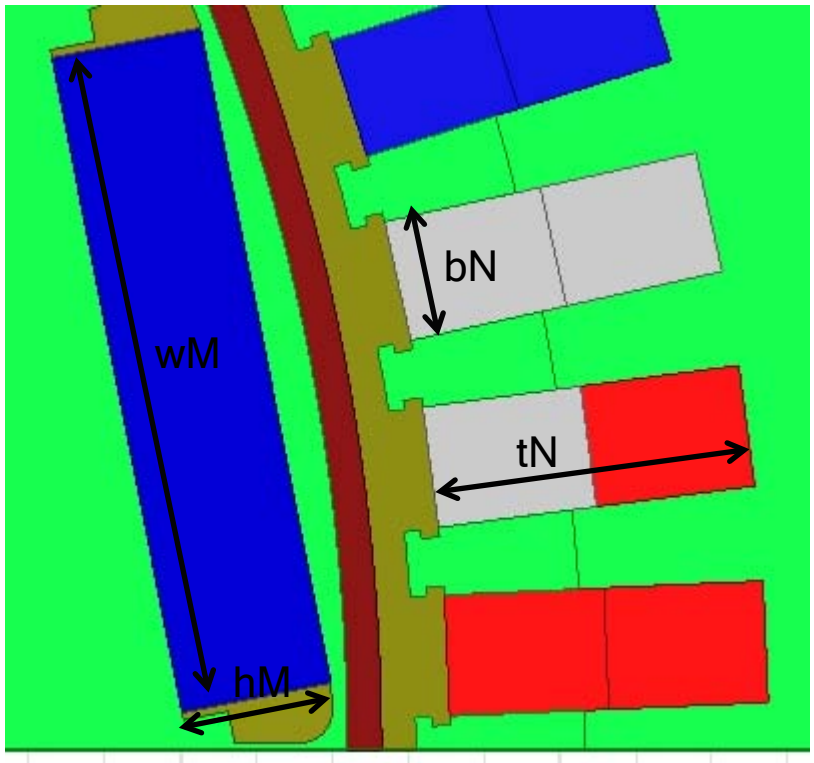


  - $W(\Theta, i)$ is the magnetic co-energy of the system and the current, i, is held constant

- The idea behind the tutorial is to show capabilities of DesignXplorer with minimum time. Users can refine the results by modifying Maxwell settings for more accuracy.

Design Parameters

- Four Parameters used for Design Optimization
  - Magnet Width (wM)
  - Magnet Thickness (hM)
  - Stator Slot Width (bN)
  - Stator Slot Depth (tN)
- Range of Variations for Parameters
  - tN: 20mm to 36mm
  - bN: 3mm to 11mm
  - wM: 25mm to 44mm
  - hM: 5mm to 16mm



Ouput Parameters

- Four Output Parameters
  - Torque
  - Magnet Area
  - Torque Per unit Magnet Area
  - Maximum Flux Density in Stator Tooth

Launch ANSYS Workbench

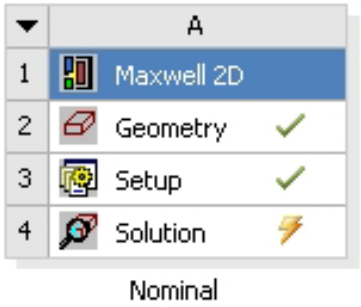
- **To Launch Workbench**
  - Click the Microsoft **Start** button, select **Programs**, and select **ANSYS 14.0** and select **Workbench 14.0**

Import Maxwell Project

- **To Import Maxwell Project**
  - In Workbench window, select the menu item *File > Import*
  - In Import window,
    - Set the file type to Maxwell Project File
    - Locate the file **"Ex_11_6_IPM_Opt_DX.mxwl"** and **Open** it
    - A Maxwell 3D Design is automatically created on Workbench project page.

Launch Maxwell

- **To Launch Maxwell**
  - Right click on **Solution** tab of Maxwell Design and Select **Edit**



About Maxwell Project

- **In Imported Maxwell Project File**
  - Mesh Operations, Excitations and Boundaries are already defined
  - Design Parameters wM, hM, bN and tN are also defined

Properties

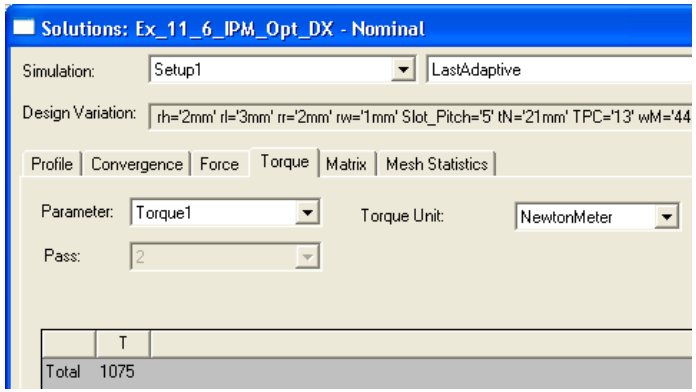
Name	Value	Unit	Evaluated
bN	8	mm	8mm
tN	21	mm	21mm
wM	44	mm	44mm
hM	10	mm	10mm

Run Maxwell Solution

- **To Run Maxwell Solution**
  - Select the menu item *Maxwell 2D > Analyze All*

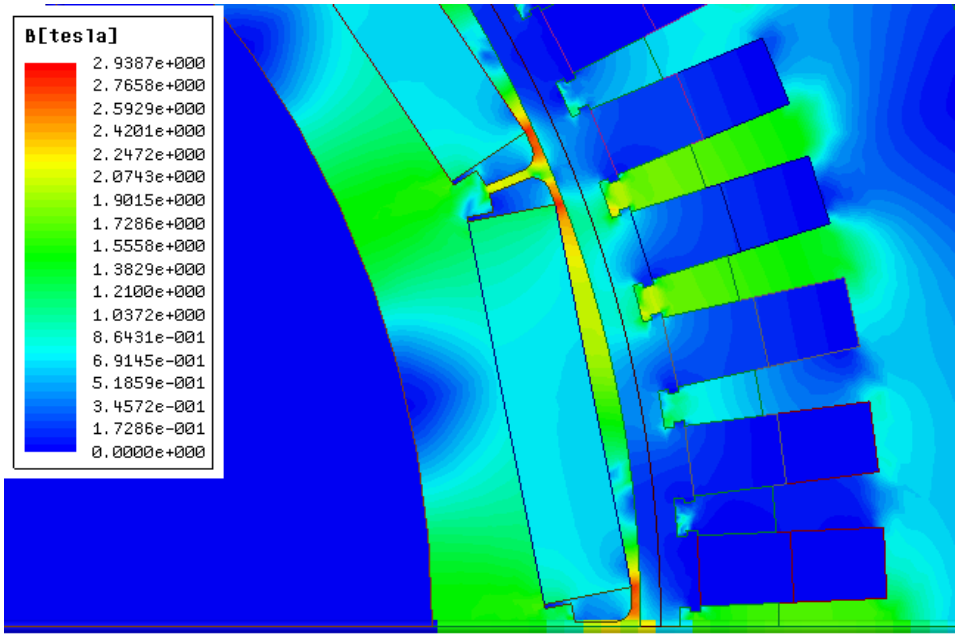
View Results

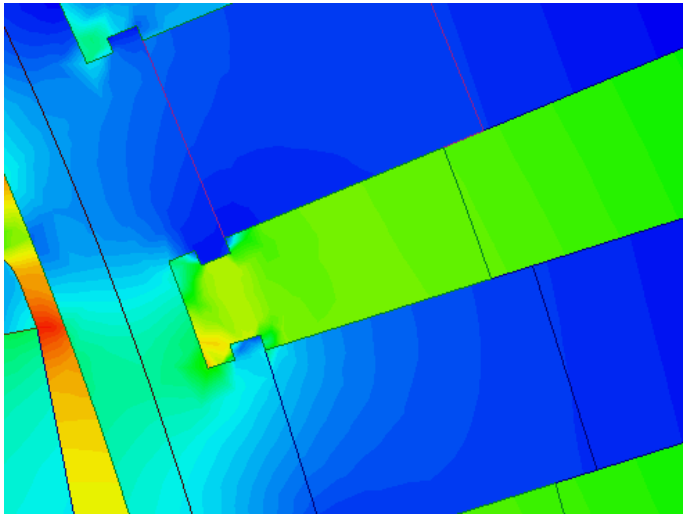
- **To View Solution results**
  - Select the menu item *Maxwell 2D > Results > Solution Data*
  - In Solution window,
    - Select **Torque** tab to view torque values
    - Reported value is around 1KNm



Plot Flux Density

- **To Plot Flux Density**
  - Select the menu item *Edit > Select All*
  - Select the menu item *Maxwell 2D > Fields > Fields > B > Mag_B*
  - In Create Field Plot window,
    - Press **Done**
  - Modify plot attributes for better display
  - From the plot, it can be seen that Maximum Flux Density in Stator occurs at tooth5





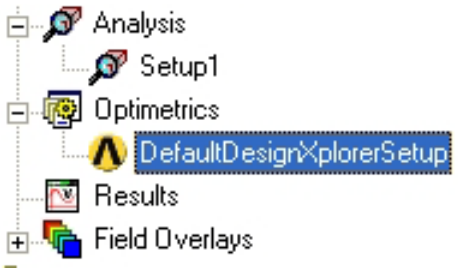
Create Output Parameter for Flux Density

- **Create Parameter for Flux Density at Tooth5**
  - Select the menu item *Maxwell 2D > Fields > Calculator*
  - In Calculator window,
    - Select **Input > Quantity > B**
    - Select **Vector > Mag**
    - Select **Input > Geometry >Volume**
      - Select **Tooth5** from list and press **OK**
    - Select **Scalar > ∫**
    - Select **Input > Number > 1**
    - Select **Input > Geometry >Volume**
      - Select **Tooth5** from list and press **OK**
    - Select **Scalar > Integrate**
    - Select **General > /**
    - Select the button **Add ...**
      - Specify Name of the expression as **Bavg_T5**
      - Press **OK**

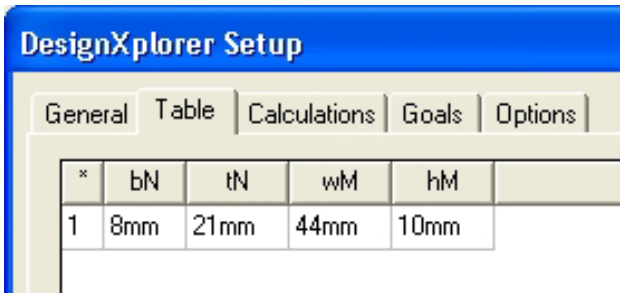
      ```
      Scl : /(Integrate(Volume(Tooth5), Mag(<Bx,By,0>)), Integrate(Volume(Tooth5), 1))
      ```
    - Press **Done** to close calculator

Add Parameters to DX

- **To Add Parameters to DX**
  - Expand the Project Manager tree to view Optimetrics
  - Double click on the tab **DefaultDesignXplorerSetup**



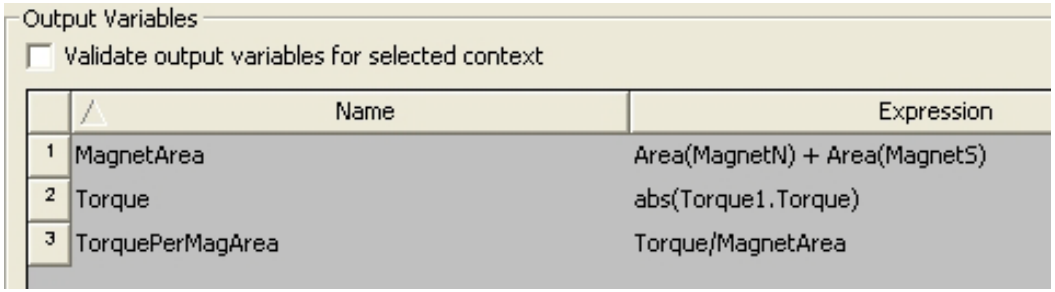
- **To Define Input Variables**
  - In DesignXplorer Setup window,
    - Select **General** tab
      - Include the Parameters bM, hM, tN and wN to specify them as Input Variables
    - Select **Table** tab
      - To view all included parameters



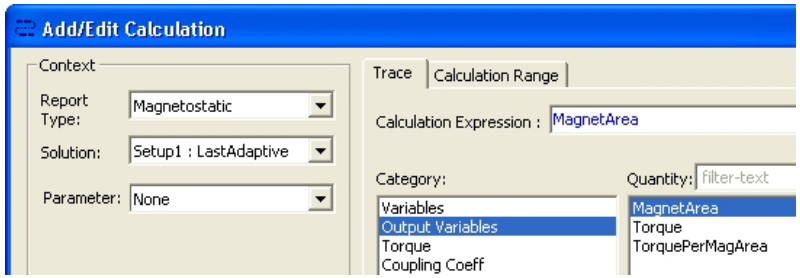
Input Variables:

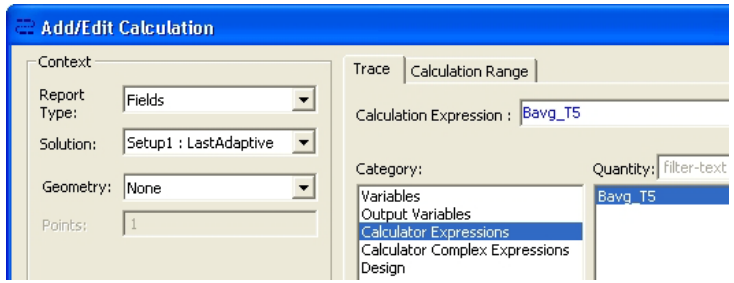
Design Variable	Include	Override	Value	Units
Slot_Pitch	☐	☐	5	
TPC	☐	☐	13	
bN	☑	☐	8	mm
delta	☐	☐	9	mm
gy	☐	☐	22	mm
hM	☑	☐	10	mm
hN	☐	☐	32.5	mm
lFe	☐	☐	400	mm
rh	☐	☐	2	mm
rl	☐	☐	3	mm
rr	☐	☐	2	mm
rw	☐	☐	1	mm
tN	☑	☐	21	mm
wM	☑	☐	44	mm
ww	☐	☐	0.5	mm

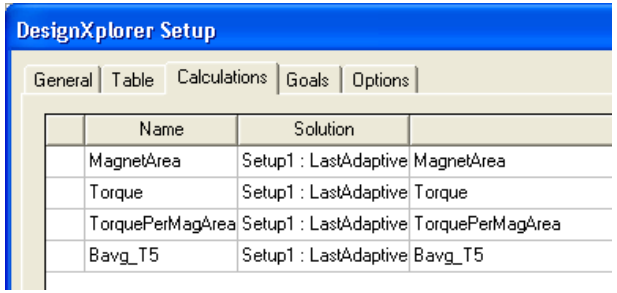
- **To Define Output Parameters**
  - In DesignXplorer Setup window,
    - Select **Calculations** tab and select **Setup Calculations ...**
    - In Add/Edit Calculations window, select **Output Variables ...** in the lower left corner of window
    - In the new Output Variables window, add variables as give below
      - Name: **MagnetArea**
      - Expression: **Area(MagnetN) + Area(MagnetS)**
      - Select **Add**
      - Name: **Torque**
      - Expression: **abs(Torque1.Torque)**
      - Select **Add**
      - Name: **TorquePerMagArea**
      - Expression: **Torque/MagnetArea**
      - Select **Add**
      - Press **Done**



- **To Define Output Parameters *(Contd...)***
  - In Add/Edit Calculation window,
    - Category: **Output variables**
    - Quantity: **MagnetArea**
    - Select **Add Calculation**
    - Change Quantity to **Torque**
    - Select **Add Calculation**
    - Change Quantity to **TorquePerMagArea**
    - Select **Add Calculation**
    - Change Report Type to **Fields**
    - Category: **Calculator Expressions**
    - Quantity: **Bavg_T5**
    - Select **Add Calculation**
    - Press **Done** to Add/Edit Calculations window
  - Press **OK** to Close DesignXplorer Setup window





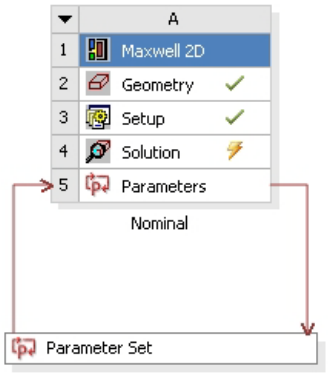


Close Maxwell Window

- **To Close Maxwell window**
  - Select the menu item *File > Close Desktop*

View Parameters

- **To View Parameters**
  - In Project page a new tab appears below Maxwell Design which Corresponds to parameter set
  - Double click on the tab **Parameter Set** to view added parameters
  - Maxwell Parameters can be seen in Parameter Set
  - Select the button Return to Project to exit Parameter Set



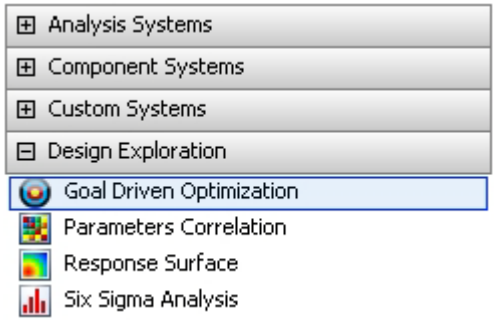
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Nominal (A1)			
4	P1	bN [mm]	8	
5	P2	tN [mm]	21	
6	P3	wM [mm]	44	
7	P4	hM [mm]	10	
*	New input parameter	New name	New expression	
9	Output Parameters			
10	Nominal (A1)			
11	P5	MagnetArea		
12	P6	Torque		
13	P7	TorquePerMagArea		
14	P8	Bavg_T5		
*	New output parameter		New expression	
16	Charts			

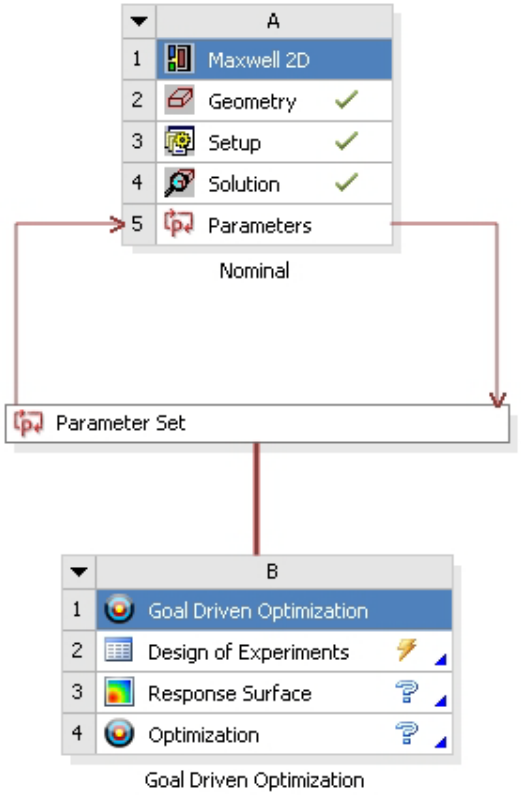
Update Project

- **To Update Project**
  - Select the button Update Project

Add Optimization

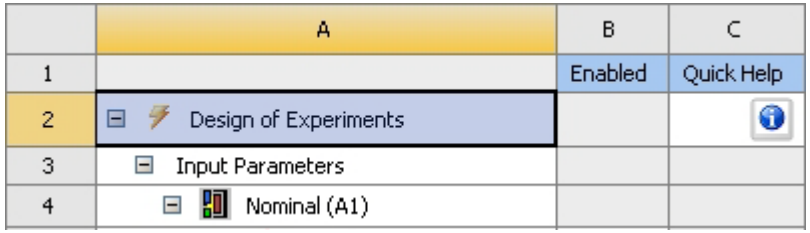
- **To Add Optimization Analysis**
  - Expand the Toolbox for **Design Exploration**
  - Select the tab **Goal Driven Optimization,** drag and drop it on Project page
  - Added system will automatically connect with Parameter Set to draw input and output variables for Parametric Analysis

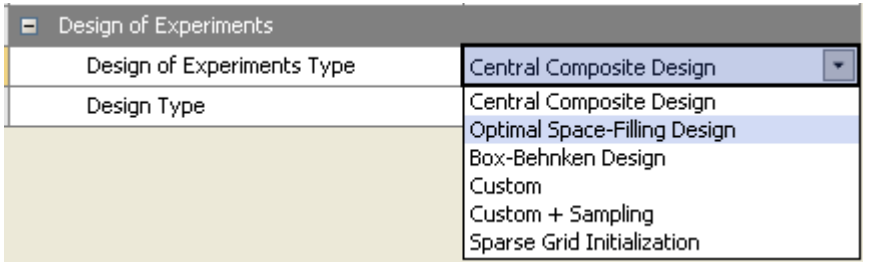




Set Design of Experiments (DOE)

- Based on the number of input parameters, a given number of solutions (design points) are required to build a response surface (DX uses a factorial method to calculate the necessary solutions). This is achieved through Design of Experiments or DOE
- **To Set Design of Experiments**
  - Double click on the tab Design of Experiments from Goal Driven Optimization Analysis system to enter the Design of Experiments window
  - Select the tab Design of Experiments in the window to view options available for setting Design of Experiments





  - A Design of Experiments method determines how many and which design points should be solved for the most efficient approach to optimization (many different DOE methods have been developed).
  - Default Method is Central Composite Design
  - Keep the method to Default

- **To Set Variations of Input Parameters**
  - Select the Parameter **bN** from DOE window
    - In Properties window,
      - Set Lower Bound to **3**
      - Set Upper Bound to **11**
  - Similarly Set Bands for other parameters
    - **tN:** Lower Bound : **20,** Upper Bound : **36**
    - **wM:** Lower Bound : **25,** Upper Bound : **44**
    - **hM:** Lower Bound : **5,** Upper Bound: **16**

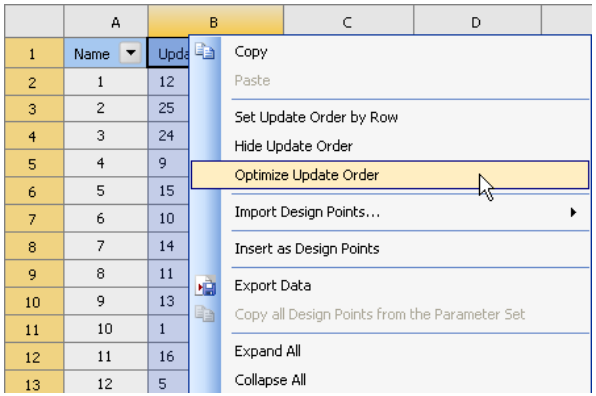
	A	B
1		Enabled
2	Design of Experiments	
3	Input Parameters	
4	Nominal (A1)	
5	P1 - bN [mm]	☑
6	P2 - tN [mm]	☑
7	P3 - wM [mm]	☑
8	P4 - hM [mm]	☑

	A	B
1	Property	Value
2	General	
3	Units	
4	Type	Design Variable
5	Classification	Continuous
6	Values	
7	Lower Bound	3
8	Upper Bound	11
9	Initial Value	8

  - Select the button 

 to view all variations
  - In total 26 variations are created. Right click on the table and select **Optimize Update Order** to set the Update priorities of the design of experiments

	A	B	C	D	E	F	G	H	I
1	Name	P1 - bN [mm]	P2 - tN [mm]	P3 - wM [mm]	P4 - hM [mm]	P5 - MagnetArea	P6 - Torque	P7 - TorquePerMagArea	P8 - Bavg_T
2	1	7	34.4	26.14	9.62	⚡	⚡	⚡	⚡
3	2	10.52	33.76	32.98	10.94	⚡	⚡	⚡	⚡
4	3	6.68	22.24	41.34	7.86	⚡	⚡	⚡	⚡
5	4	7.32	35.68	36.78	8.3	⚡	⚡	⚡	⚡
6	5	4.12	26.72	31.46	14.9	⚡	⚡	⚡	⚡
7	6	7.64	28	25.38	14.02	⚡	⚡	⚡	⚡
8	7	8.92	33.12	42.86	12.7	⚡	⚡	⚡	⚡
9	8	10.2	24.16	36.02	6.98	⚡	⚡	⚡	⚡
10	9	10.84	25.44	30.7	13.14	⚡	⚡	⚡	⚡



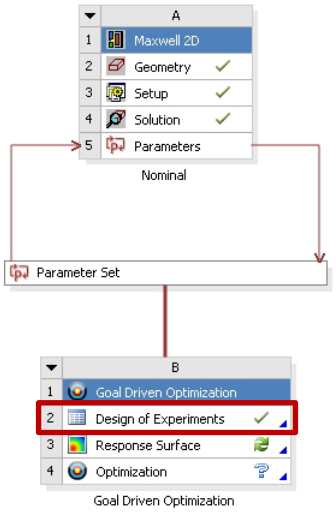
Save Project

- **To Save Project**
  - Select the menu item *File > Save*
  - Save the file with appropriate name

Solve Design of Experiments

- **To Solve Design of Experiments**
  - Select the button Update .
  - It should take about 20min to solve all variations
  - Results are displayed in the Table after each run

	A	B	C	D	E	F	G	H	I	J
1	Name	Update Order	P1 - bN [mm]	P2 - tN [mm]	P3 - wM [mm]	P4 - hM [mm]	P5 - MagnetArea	P6 - Torque	P7 - TorquePerMagArea	P8 - Bavg_T5
2	10	1	4.1832	22.366	27.81	6.6269	0.00036859	798.8	2.1672E+06	1.0545
3	18	2	4.1832	22.366	27.81	14.373	0.00079943	922.49	1.1539E+06	0.93464
4	14	3	4.1832	22.366	41.19	6.6269	0.00054592	1029.1	1.8851E+06	1.3157
5	22	4	4.1832	22.366	41.19	14.373	0.0011841	1194.2	1.0085E+06	1.2582
6	12	5	4.1832	33.634	27.81	6.6269	0.00036859	712.49	1.933E+06	1.0071
7	20	6	4.1832	33.634	27.81	14.373	0.00079943	863.13	1.0797E+06	0.94251
8	16	7	4.1832	33.634	41.19	6.6269	0.00054592	929.9	1.7034E+06	1.285
9	24	8	4.1832	33.634	41.19	14.373	0.0011841	1112.1	9.3919E+05	1.2594
10	4	9	7	20	34.5	10.5	0.0007245	995.31	1.3738E+06	1.2962
11	6	10	7	28	25	10.5	0.000525	770.43	1.4675E+06	1.045
12	8	11	7	28	34.5	5	0.000345	803.05	2.3277E+06	1.4505
13	1	12	7	28	34.5	10.5	0.0007245	994.98	1.3733E+06	1.3118
14	9	13	7	28	34.5	16	0.001104	1027.4	9.3063E+05	1.2352
15	7	14	7	28	44	10.5	0.000924	1133.3	1.2265E+06	1.633
16	5	15	7	36	34.5	10.5	0.0007245	899.39	1.2414E+06	1.2667
17	11	16	9.8168	22.366	27.81	6.6269	0.00036859	645.96	1.7525E+06	1.6925



Exit Design of Experiments

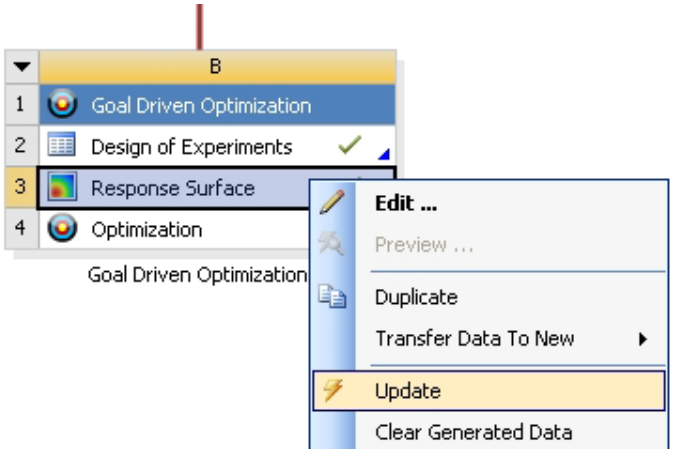
- **To Return to Project Page**
  - Select the button Return to Project

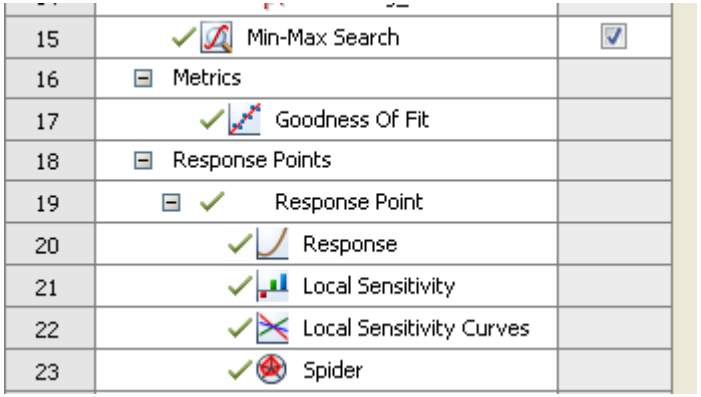
Save Project

- **To Save Project**
  - Select the menu item *File > Save*

Create Response Surface

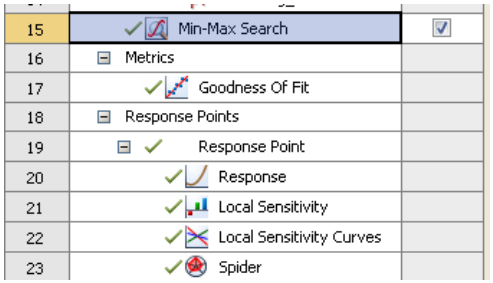
- **To Create Response Surface**
  - Right click on the cell Response Surface and select **Update**
  - Double click on the tab Response Surface to enter the window
  - In the Outline of Schematic window, various methods are listed to examine solved variations
  - Lets have a look at each of them one by one





Min-Max Search

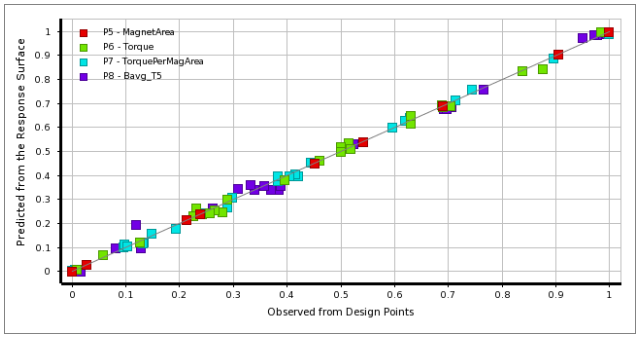
- Min-Max Search enables to search minima and maxima of each of the output parameters from DOEs. Corresponding Input parameters are also listed
- Select the tab **Min-Max Search** to get the table of minimum and maximum values



	A	B	C	D	E	F	G	H	I
1	Name	P1 - bN [mm]	P2 - tN [mm]	P3 - wM [mm]	P4 - hM [mm]	P5 - MagnetArea	P6 - Torque	P7 - TorquePerMagArea	P8 - Bavg_T5
2	Output Parameter Minimums								
3	P5 - MagnetArea Minimum Design Point	3.04	20.08	25	5	**0.00025**	658.56	2.6577E+06	1.3163
4	P6 - Torque Minimum Design Point	11	20	25	5	0.00025	**456.22**	1.7597E+06	2.0175
5	P7 - TorquePerMagArea Minimum Design Point	11	20	44	16	0.001408	748.52	**5.3879E+05**	2.2221
6	P8 - Bavg_T5 Minimum Design Point	4.6027	29.08	25	14.507	0.00072536	832.95	1.1286E+06	**0.87822**
7	Output Parameter Maximums								
8	P5 - MagnetArea Maximum Design Point	6.56	23.33	44	16	**0.001408**	1189.5	8.686E+05	1.5693
9	P6 - Torque Maximum Design Point	3	20	44	14.733	0.0012965	**1260.6**	9.8412E+05	1.5548
10	P7 - TorquePerMagArea Maximum Design Point	3.201	20	25	5	0.00025	658.55	**2.6584E+06**	1.2885
11	P8 - Bavg_T5 Maximum Design Point	11	35.705	44	5	0.00044	592.37	1.2452E+06	**2.2549**

Goodness of Fit

- After the variations are run with DOE, a curve fitting approach is used to determine Response Surfaces. The accuracy of the Curve Fitting can be analyzed used the option Goodness of Fit
- Select the tab **Goodness of Fit** to view Curve Fitting and various measures of Goodness of Fit



	A	B	C	D	E
1	Name	P5 - MagnetArea	P6 - Torque	P7 - TorquePerMagArea	P8 - Bavg_T5
2	Goodness Of Fit				
3	Coefficient of Determination (Best Value = 1)	1	0.99672	0.99839	0.9937
4	Adjusted Coeff of Determination (Best Value = 1)	1	0.99474	0.99773	0.9916
5	Maximum Relative Residual (Best Value = 0%)	0	2.5076	2.7655	7.1502
6	Root Mean Square Error (Best Value = 0)	4.473E-14	8.6391	16826	0.025516
7	Relative Root Mean Square Error (Best Value = 0%)	0	0.98251	1.3568	2.1522
8	Relative Maximum Absolute Error (Best Value = 0%)	0	12.557	7.7112	22.775
9	Relative Average Absolute Error (Best Value = 0%)	0	4.288	3.3667	6.0158

- **Response Surface**
  - Response Surfaces are useful tools to analyze the impact that parameters have on one another
  - The Plots can be 2D or 3D
  - Displays are fully interactive.
  - Sliders bars can be used to change the values of input parameters to explore other designs

15	Min-Max Search
16	Metrics
17	Goodness Of Fit
18	Response Points
19	Response Point
20	Response
21	Local Sensitivity
22	Local Sensitivity Curves
23	Spider

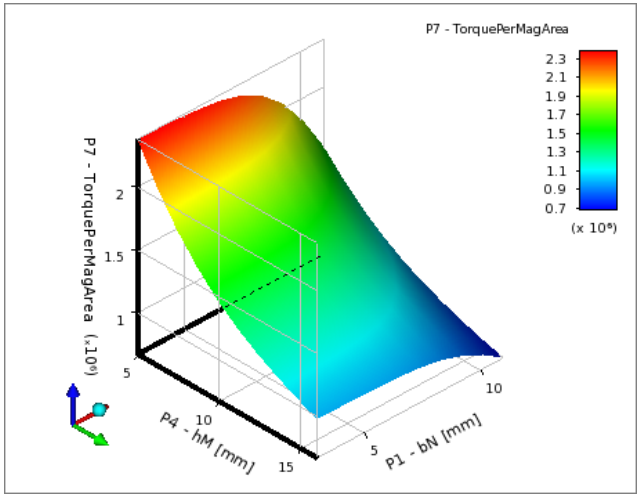
	A	B
1	Property	Value
2	Chart	
3	Display Parameter Full Name	☑
4	Mode	2D
5	Number of Points on X	10
6	Show Design Points	☐
7	Axes	
8	X axis	P1 - bN [mm]
9	Y axis	P5 - MagnetArea
10	Input Parameters	
11	P1 - bN [mm]	7
12	P2 - tN [mm]	28
13	P3 - wM [mm]	34.5
14	P4 - hM [mm]	10.5
15	Output Parameters	
16	P5 - MagnetArea	0.0007245
17	P6 - Torque	978.46
18	P7 - TorquePerMagArea	1.3354E+06
19	P8 - Bavg_T5	1.3533

Plots types 2D or 3D

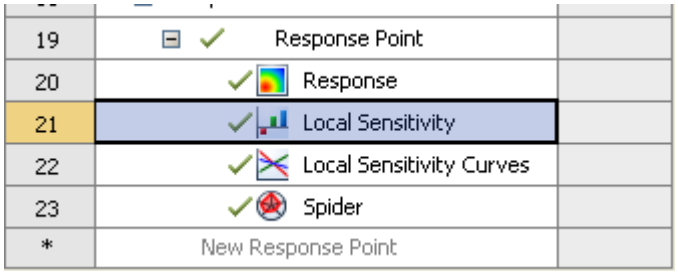
X and Y Axis of Plot (Z Axis in 3D)

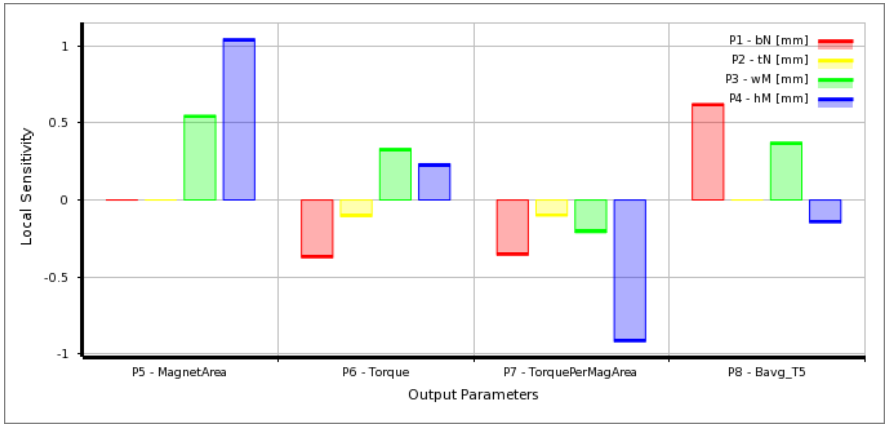
Input parameters values and slider bars to change the values

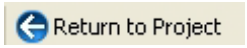
- **Plot Response Surface for bN Vs hM Vs TorquePerMagArea**
  - Change Plot mode to **3D**
  - X axis: Set to **bN**
  - Y axis: Set to **hM**
  - Z axis: Set to **TorquePerMagArea**
  - 3D plots can be rotated for better view



- **Local Sensitivity**
  - Local Sensitivity charts show sensitivity of each of the output parameters with respect to input parameters
  - Select the tab **Local Sensitivity** to view the chart in Bar or Pie mode

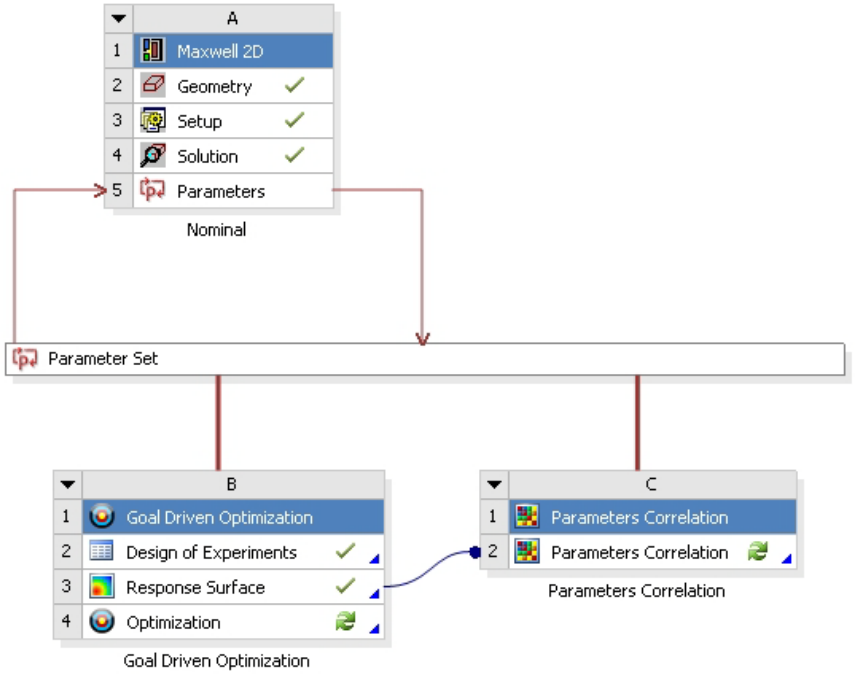


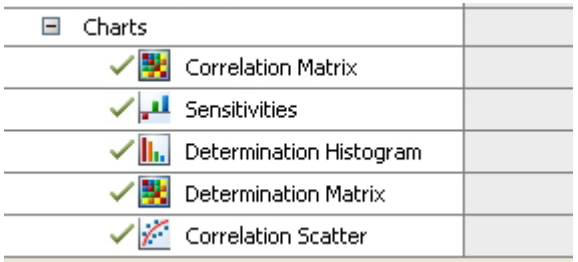


- **To Return to Project Page**
  - Select the button 

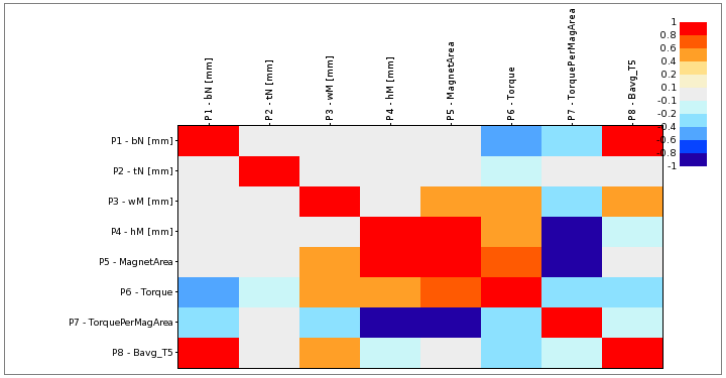
Add Parameters Correlation

- **To Add Parameters Correlation Analysis**
  - Select the system **Parameters Correlation** from toolbox, drag and it on the Response Surface tab of Goal Driven Optimization System
  - Right click on the tab **Parameters and Correlations** and select **Update**
  - Double click on the tab Parameters and Correlations to enter the window
  - Different types of correlation analysis can be done in this interface

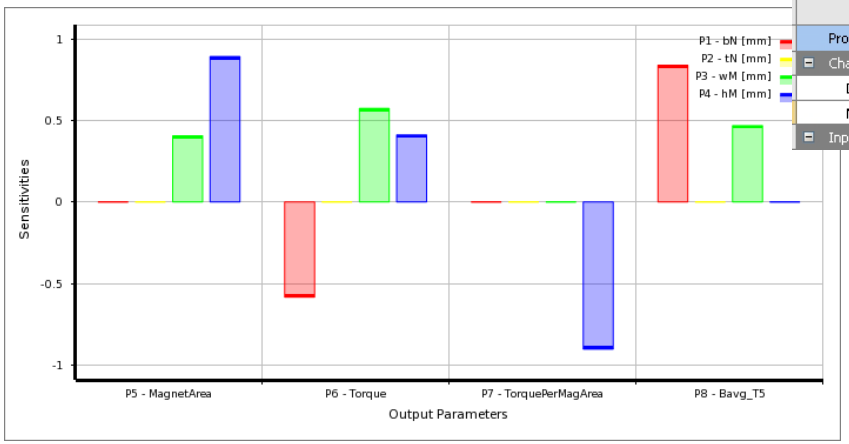


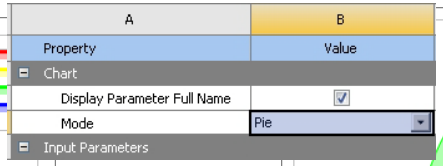


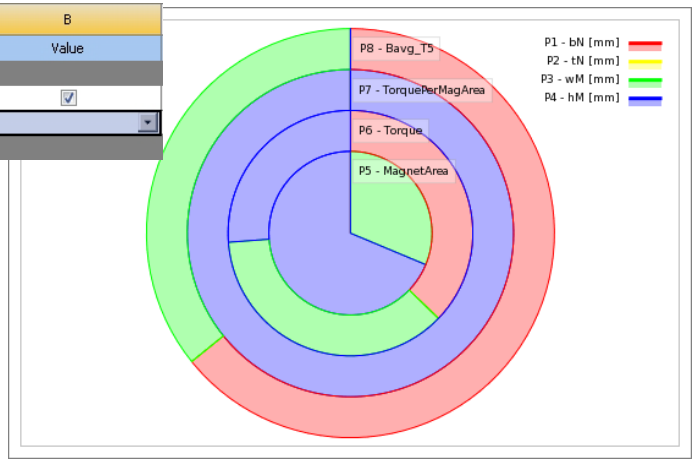
- **Correlation Matrix**
  - Select the tab Correlation Matrix
  - This Matrix shows how closely various input and output parameters are coupled. Strength of correlation is indicated by color in the Matrix



- **Sensitivities**
  - The Sensitivity chart shown in previous slides is local sensitivity (corresponding to particular values of input parameters) while this is Global Sensitivities
  - Sensitivity chart can be shown in Bar or Pie Chart

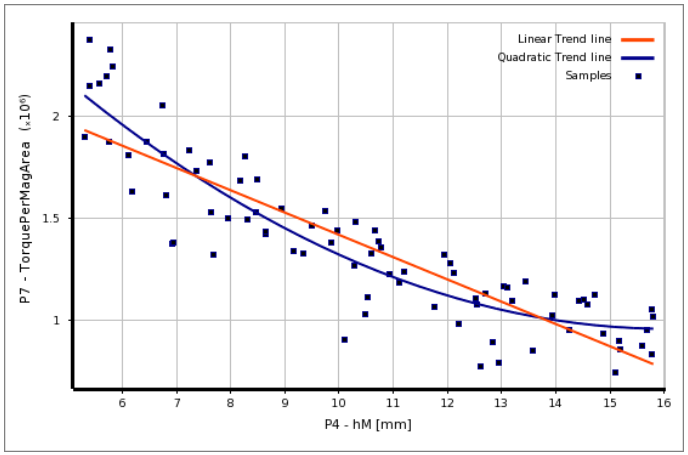






- **Correlation Scatter**
  - Correlation Scatter shows the sample values of one parameter vs. another. A trend line is drawn showing the correlation of the sample values.
  - Select the tab **Correlation Scatter**
  - Set X axis to **hM**
  - Set Y axis to **TorquePerMagArea**
  - The Plot is displayed with trend line in linear as well as quadratic interpolation

	A	B
1	Property	Value
2	Chart	
3	Display Parameter Full Name	☑
4	Axes	
5	X axis	P4 - hM [mm]
6	Y axis	P7 - TorquePe...
7	Trend Lines	
8	Linear	R2 = 0.74961
9	Quadratic	R2 = 0.89091



- **To Return to Project Page**
  - Select the button Return to Project

Save Project

- **To Save Project**
  - Select the menu item *File > Save*

Optimization Analysis

Optimization Analysis

- Double click on the tab **Optimization** from Goal Driven Optimization System
- Select the tab **Optimization** to view options available under it
- DX uses three different algorithms for Optimization
  - **MOGA:** Multi-Objective Genetic Algorithm – can optimize problems with continuous input parameters. It is an iterative approach
  - **NLPQL:** Gradient based single objective optimizer; based on quasi-Newton methods
  - **Screening:** A non-iterative direct sampling method by a quasi-random number generator based on the Hammersley algorithm.

	A	B
1		Enabled
2	Optimization	
3	Input Parameters	
4	Nominal (A1)	
5	P1 - bN [mm]	☑
6	P2 - tN [mm]	☑
7	P3 - wM [mm]	☑
8	P4 - hM [mm]	☑
9	Output Parameters	
10	Nominal (A1)	
11	P5 - MagnetArea	
12	P6 - Torque	
13	P7 - TorquePerMagArea	
14	P8 - Bavg_T5	

5	Design Points	
6	Preserve Design Points After DX Run	☐
7	Optimization	
8	Optimization Method	Screening
9	Number of Samples	MOGA NLPQL Screening
10	Size of Generated Sample Set	
11	Verify Candidate Points	☐

- **To Set Optimization Analysis**
  - In Properties window
    - Select the tab **Optimization**
    - Set Optimization Method to **Screening**
  - In Table of Optimization
    - Set Objectives for
      - MagnetArea: **Minimize**
      - Torque: **Maximize**
      - TorquePerMagArea: **Maximize**
      - Bavg_T5: **Seek Target**
        - Set Target to **1.5**
    - Set Importance to all of them to **Default**

	A	B	C	D	E	F	G	H	I
1		P1 - bN [mm]	P2 - tN [mm]	P3 - wM [mm]	P4 - hM [mm]	P5 - MagnetArea	P6 - Torque	P7 - TorquePerMagArea	P8 - Bavg_T5
2	Optimization Domain								
3	Lower Bound	3	20	25	5				
4	Upper Bound	11	36	44	16				
5	Optimization Objectives								
6	Objective	No O...	No O...	No ...	No...	Minimize	Maximize	Maximize	Seek Target
7	Target Value								1.5
8	Importance					Default	Default	Default	Default
9	Constraint Handling								

- **To Run Optimization**
  - Select the button 

  - The solution takes in the order of seconds

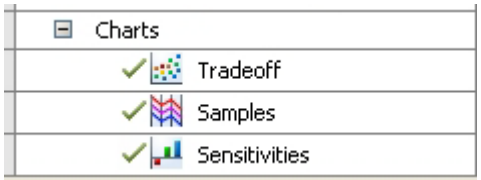
- **Optimization results**
  - Three candidates are listed based on the Goals set for Optimization

10	Candidate Points								
11	Candidate A	3.404	24.758	41.429	5.1815	★★ 0.00042933	★ 967.74	★★ 2.1869E+06	★★★ 1.4369
12	Candidate B	3.644	20.633	43.775	5.7095	★★ 0.00049987	★ 1029.1	★ 2.0226E+06	★★★ 1.4509
13	Candidate C	6.204	20.602	35.435	5.1463	★★ 0.00036472	– 860.17	★★ 2.3422E+06	★★ 1.3487

- **Set Importance for Bavg_T5**
  - Change the importance for **Bavg_T5** to **Higher**
  - Run Optimization again

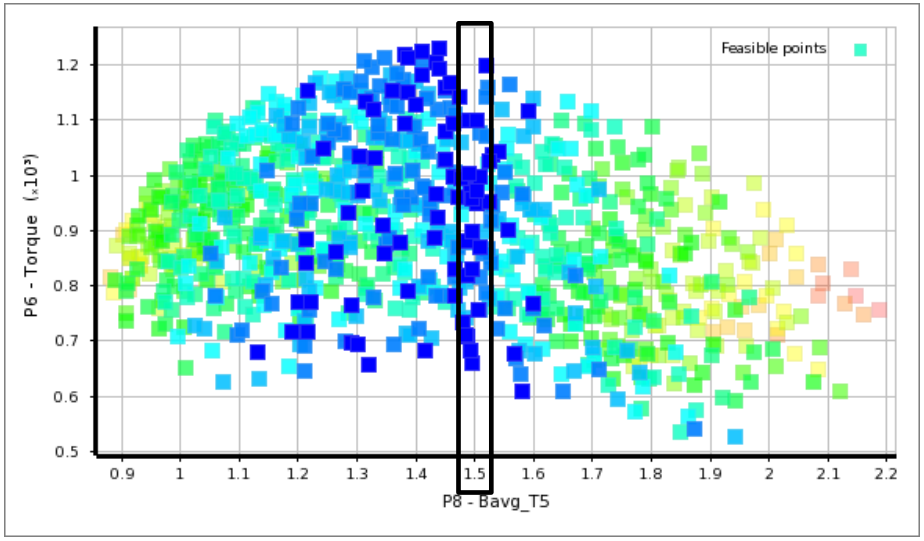
10	Candidate Points								
11	Candidate A	3.404	24.758	41.429	5.1815	★★ 0.00042933	★ 967.74	★★ 2.1869E+06	★★★ 1.4369
12	Candidate B	3.644	20.633	43.775	5.7095	★★ 0.00049987	★ 1029.1	★ 2.0226E+06	★★★ 1.4509
13	Candidate C	7.404	26.274	32.568	5.2519	★★★ 0.00034209	× 784.12	★★ 2.2941E+06	★★★ 1.4665

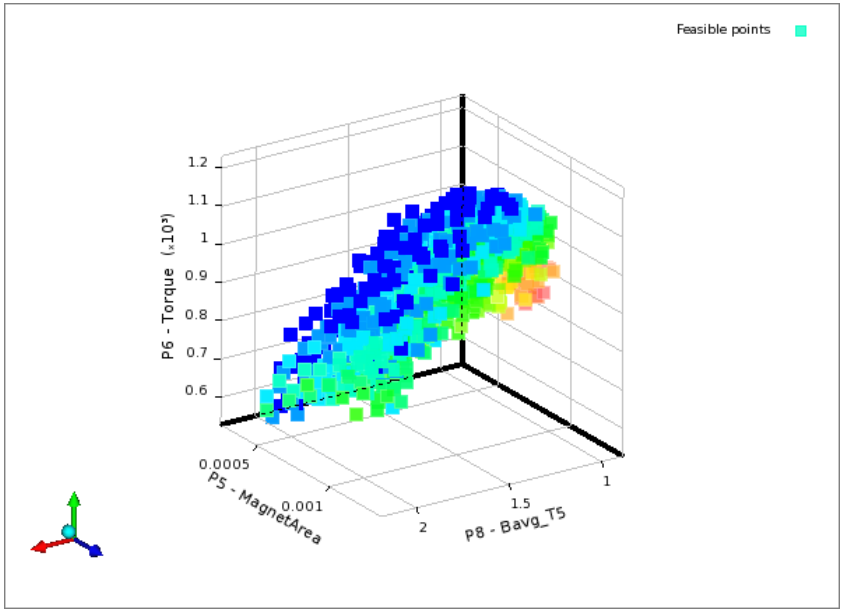
  - Note that candidates have changed as the importance of optimization is changed
  - Symbols in each field indicate agreement with set Goals for the parameters
- **Analyze Optimization results**
  - Optimization Results can be checked using below charts



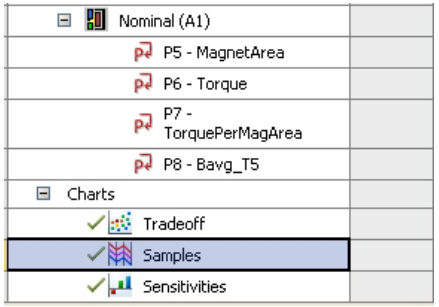
- **Tradeoff**
  - The trade off plot shows the line along which no improvement in one parameter's goal can be achieved without sacrificing the other
  - Tradeoff plots can be 2D or 3D
  - Select the tab **Tradeoff**
  - In Properties window,
    - Set Mode to **2D**
    - Set X axis to **Bavg_T5**
    - Set Y axis to **Torque**
    - 13 Pareto fronts shown, the best to worst go from blue to red respectively
    - Change Mode to 3D

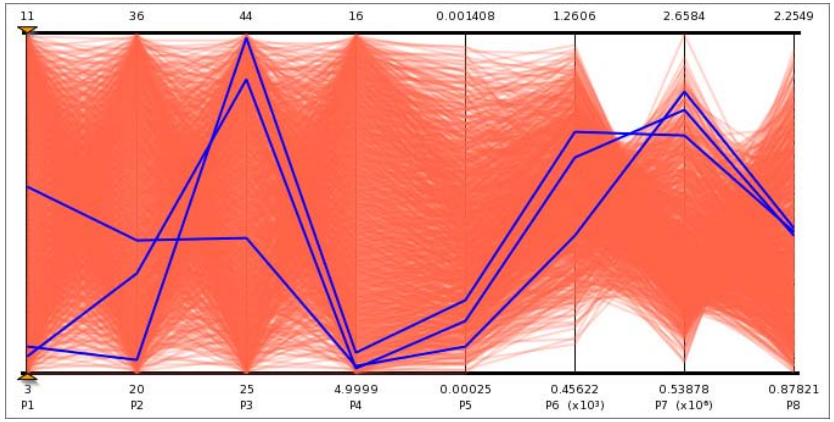
	A	B
1	Property	Value
2	Chart	
3	Display Parameter Full Name	☑
4	Mode	2D
5	Number of Pareto Fronts to Show	13
6	Axes	
7	X Axis	P8 - Bavg_T5
8	Y Axis	P6 - Torque



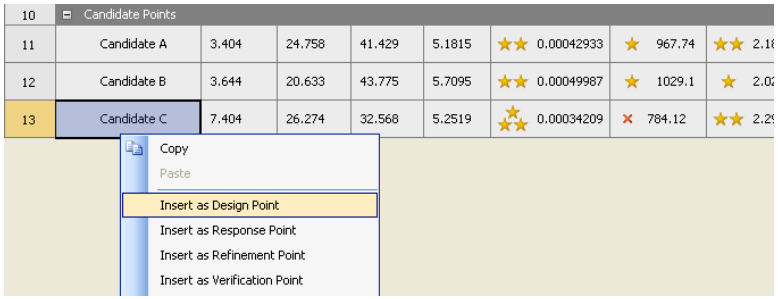


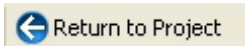
- **Samples**
  - Samples chart helps to indicate values of parameters graphically
  - X axis lists all defined parameters while Y axis indicates their position in range of variation of it





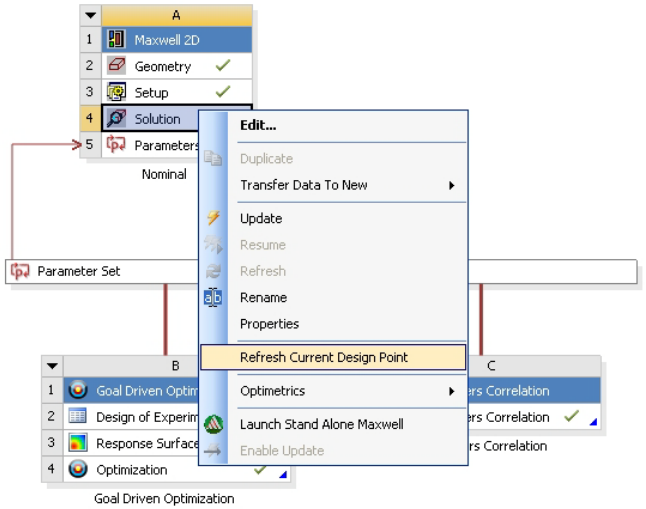
- **Choose Optimized Design**
  - Next step is to choose best of the Candidates listed after Optimization
  - This step involves manual judgment of users based on tradeoff that he is willing to make for a design
  - For the purpose of this tutorial we will choose **Candidate C**
  - Select Candidate C from table, right click and select **Insert as Design point**



- **To Return to Project Page**
  - Select the button 

Verify Optimized Design

- **To Add Optimized Design as Current**
  - Double click on the tab **Parameter Set**
  - A Design Point corresponding to optimized parameters is added as DP1
  - Right click on DP1 and select **Copy Inputs to Current**
  - Select the button Return to Project
- **Transfer Optimized Parameters to Maxwell**
  - Right click on **Solution** tab of Maxwell Design and select **Refresh Current Design Point**



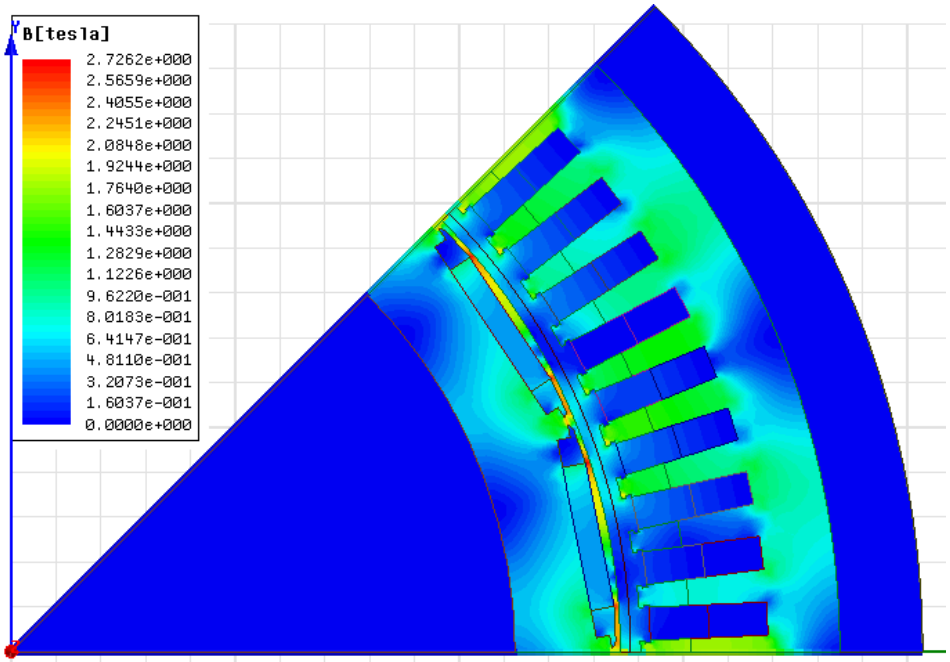
- **Update Maxwell Project**
  - Right click on **Solution** tab of Maxwell Design and select **Update**

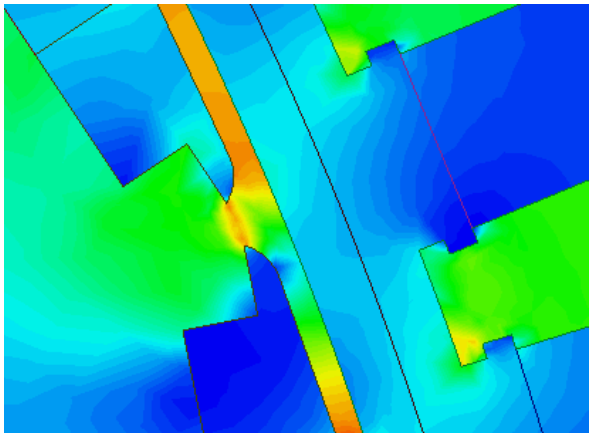
Check Optimized Results

- **To Launch Maxwell**
  - Right click on **Solution** tab of Maxwell Design and select **Edit**
  - Optimized parameters can be seen in Design Variables
  - Check the B Field Plot that we created earlier

Properties

Name	Value	Unit	Evaluated Val...	Type
bN	7.004	mm	7.004mm	Design
tN	22.97675	mm	22.97675mm	Design
wM	40.54310768...	mm	40.54310768...	Design
hM	5.0759	mm	5.0759mm	Design

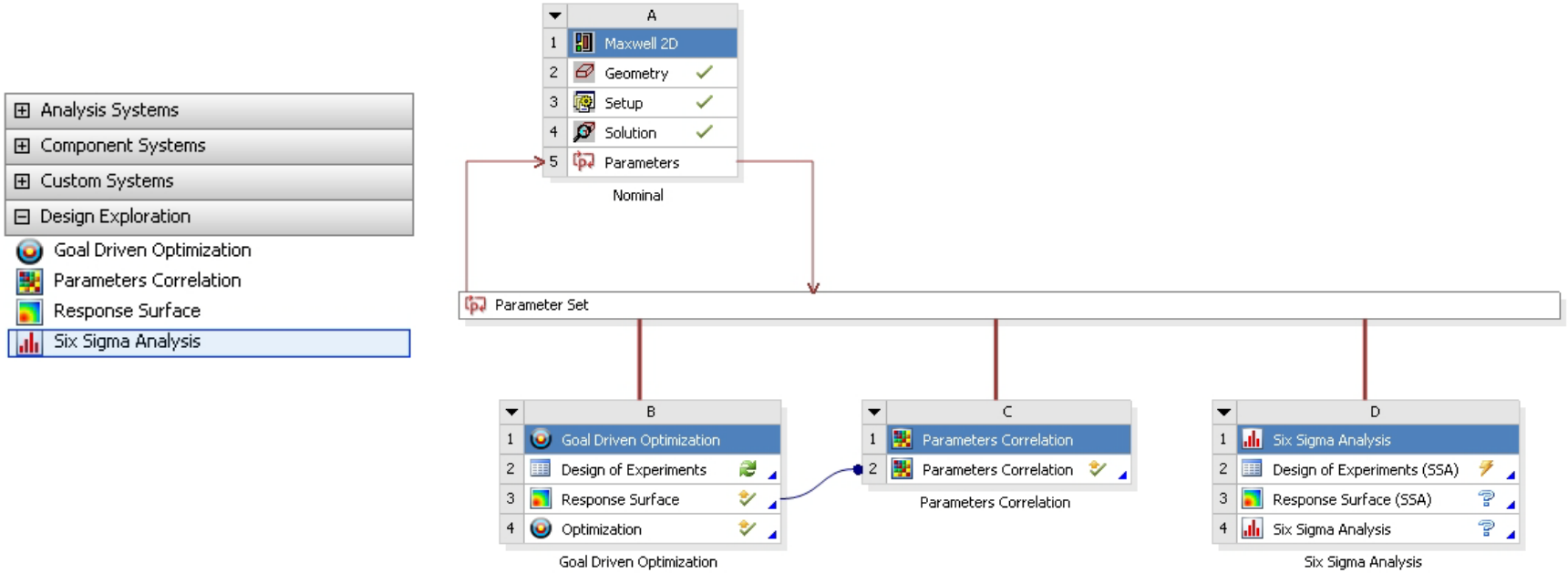




- **To Close Maxwell**
  - Select the menu item *File > Close Desktop*

Six Sigma Analysis

- In This Section, we will perform a six sigma analysis of the optimized design
- **To Add Six Sigma Analysis**
  - Select the tab **Six Sigma Analysis** from toolbox, drag and drop it on Project Page
  - Six Sigma Analysis system will automatically connect to Parameter set to draw input and output variables



- **Set Design Of Experiments for Six Sigma**
  - Double click on the tab **Design of Experiments (SSA)** to enter the window
  - Select the parameter **bN** from the list
  - In Properties window,
    - Set Standard Deviation to **0.14808**
      - The values yields to 6% tolerance, giving standard deviation value as 2% of mean value

	A	B	C
1		Enabled	Quick Help
2	Design of Experiments (SSA)		
3	Input Parameters		
4	Nominal (A1)		
5	P1 - bN [mm]	☑	
6	P2 - tN [mm]	☑	
7	P3 - wM [mm]	☑	
8	P4 - hM [mm]	☑	

	A	B
1	Property	Value
5	Classification	Continuous
6	Values	
7	Initial Value	7.404
8	Lower Bound	6.9464
9	Upper Bound	7.8616
10	Distribution	
11	Distribution Type	Normal
12	Distribution Lower Bound	-Infinity
13	Distribution Upper Bound	Infinity
14	Mean	7.404
15	Standard Deviation	0.14808
16	Skewness	0

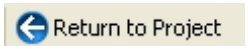
  - Similarly Set Standard Deviation for other parameters as below
    - For tN : **0.52548**
    - For wM: **0.65136**
    - For hM: **0.105038**
  - Select the button 

	A	B	C
1	Quantile	Probability Density Function	Cumulative Distribution Function
2	4.9496	0.060361	0.002
3	4.9556	0.071005	0.0023925
4	4.9616	0.083256	0.0028533
5	4.9675	0.097303	0.0033929
6	4.9735	0.11335	0.0040223
7	4.9795	0.13162	0.0047544
8	4.9855	0.15234	0.0056031
9	4.9915	0.17574	0.0065837
10	4.9975	0.20208	0.0077131
11	5.0035	0.23162	0.0090096

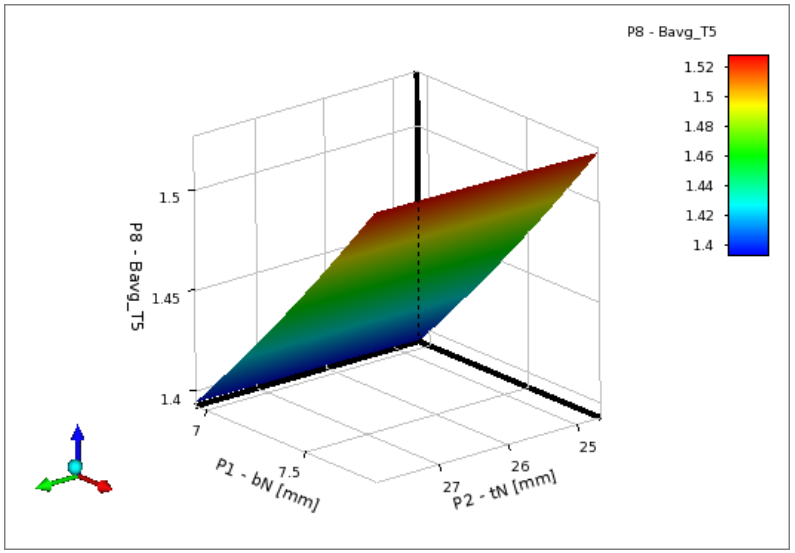
- **Run Design Of Experiments**
  - Select the button 

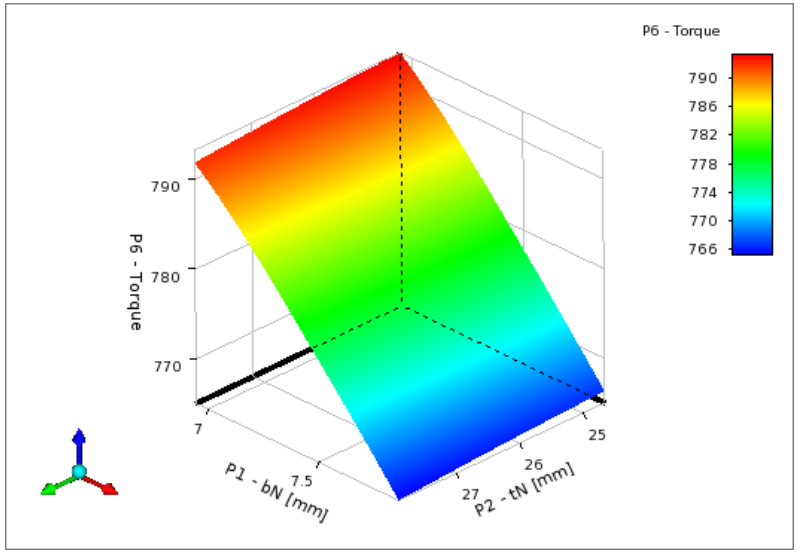
  - Solved Design of Experiments can be seen by selecting Design of Experiments tab

	A	B	C	D	E	F	G	H	I
1	Name	P1 - bN [mm]	P2 - tN [mm]	P3 - wM [mm]	P4 - hM [mm]	P5 - MagnetArea	P6 - Torque	P7 - TorquePerM...	P8 - Bavg_T5
2	1	7.404	26.274	32.568	5.2519	0.00034209	780.28	2.281E+06	1.4568
3	2	6.9464	26.274	32.568	5.2519	0.00034209	792.71	2.3173E+06	1.3929
4	3	7.8616	26.274	32.568	5.2519	0.00034209	765.77	2.2385E+06	1.5271
5	4	7.404	24.65	32.568	5.2519	0.00034209	780.88	2.2827E+06	1.4555
6	5	7.404	27.897	32.568	5.2519	0.00034209	779.31	2.2781E+06	1.4569
7	6	7.404	26.274	30.555	5.2519	0.00032094	751.43	2.3413E+06	1.4217
8	7	7.404	26.274	34.581	5.2519	0.00036323	805.98	2.2189E+06	1.4937
9	8	7.404	26.274	32.568	4.9273	0.00032094	763.77	2.3798E+06	1.4736
10	9	7.404	26.274	32.568	5.5765	0.00036323	794.48	2.1873E+06	1.4367
11	10	7.0818	25.13	31.15	5.0233	0.00031296	759.22	2.426E+06	1.4026
12	11	7.7262	25.13	31.15	5.0233	0.00031296	740.28	2.3654E+06	1.494
13	12	7.0818	27.417	31.15	5.0233	0.00031296	758.34	2.4232E+06	1.4042
14	13	7.7262	27.417	31.15	5.0233	0.00031296	739.44	2.3628E+06	1.4956
15	14	7.0818	25.13	33.985	5.0233	0.00034144	797.06	2.3344E+06	1.4484
16	15	7.7262	25.13	33.985	5.0233	0.00034144	776.58	2.2744E+06	1.5384
17	16	7.0818	27.417	33.985	5.0233	0.00034144	796.09	2.3316E+06	1.45
18	17	7.7262	27.417	33.985	5.0233	0.00034144	775.54	2.2714E+06	1.5402
19	18	7.0818	25.13	31.15	5.4805	0.00034144	778.62	2.2804E+06	1.3715

- **To Return to Project Page**
  - Select the button 

- **Plot Response Surfaces**
  - Right click on the tab **Response Surface (SSA)** and select **Update**
  - Double click on Response Surface tab to enter the window
  - Using Method described previously, create 3D plots for
    - **tN Vs bN Vs Bavg_T5**
    - **tN Vs bN Vs Torque**





- **To Return to Project Page**
  - Select the button Return to Project

- **Six Sigma Analysis**
  - Right click on the tab Six Sigma Analysis and select **Update**
  - Double click on the tab Six Sigma Analysis to enter the window
  - Select the parameter **Torque** to view its Probability Density plot
  - Change parameter to **Bavg_T5** to view its Probability Density plot

