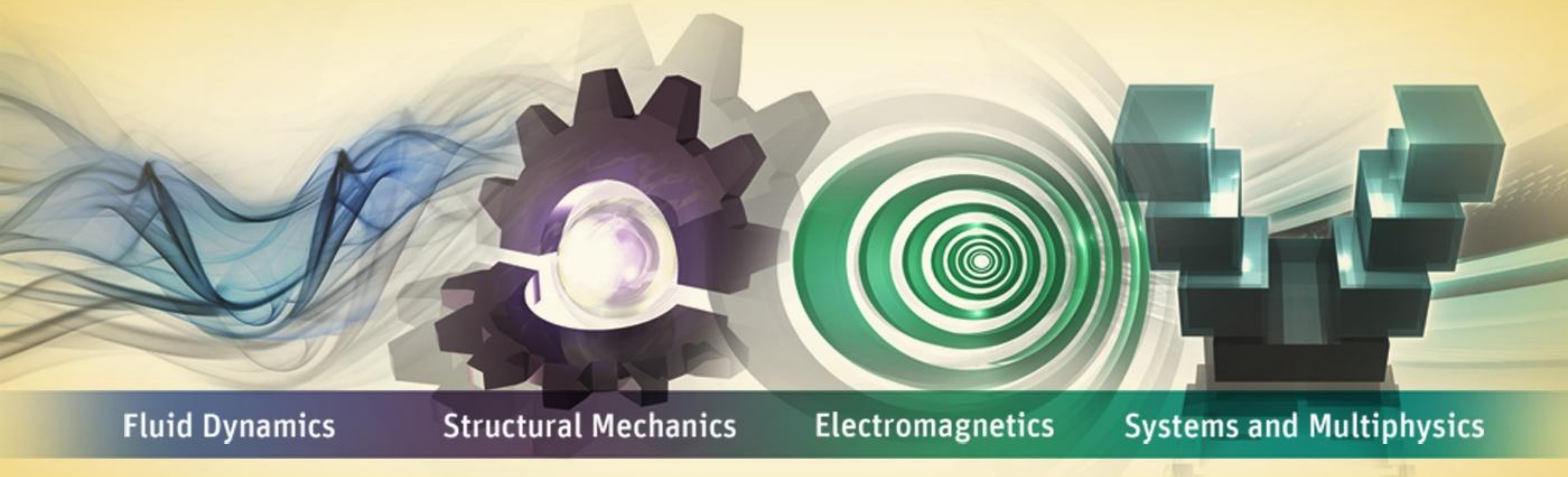
Workshop 2: Basic Eddy Current Analysis

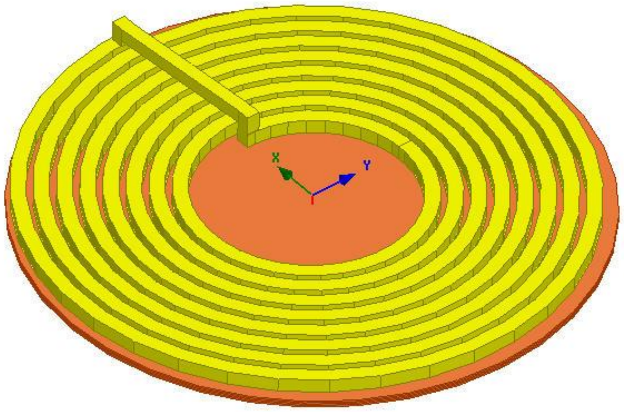


ANSYS Maxwell 3D V16

About Workshop

- **Eddy Current**
  - This exercise describes how to set up a Maxwell 3D Eddy Current project to evaluate eddy current in an object and resulting ohmic losses.
  - In Appendix, steps are show to transfer the resulting losses to ANSYS Mechanical for thermal analysis

- **Problem Definition**
  - A sinusoidal 500 Hz current will be passed through a spiral coil which will induce eddy currents in an iron disk causing it to heat up.



Problem Setup

- **Create Design**
  - Select the menu item ***Project →Insert Maxwell 3D Design***, or click on the  icon
- **Set Solution Type**
  - Select the menu item ***Maxwell 3D →Solution Type***
  - Solution Type Window:
    1. Choose **Magnetic > Eddy Current**
    2. Click the **OK** button
- **Set Model Units**
  - Select the menu item ***Modeler →Units***
  - In Set Modeler Units window,
    1. Select units: **cm (centimeters)**
    2. Press the **OK** button

Create Coil

- **Draw Spiral**
  - Select the menu item ***Draw → User Defined Primitive → SysLib → SegmentedHelix → PolygonHelix***
  - In User Defined Primitive Operation window,
    1. PolygonRadius: **1.5 cm**
    2. StartHelixRadius: **15 cm**
    3. RadiusChange: **3.1 cm**
    4. Pitch: **0 cm**
    5. Turns: **8**
    6. Press **OK**
  - Change the name of object to **Coil**
  - Change the color to **Yellow**
  - Change the material to **Copper**

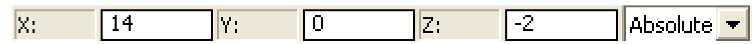
Parameters | Info

Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate System	Global			
DLL Name	SegmentedHelix/Polyg...			
DLL Location	syslib			
DLL Version	1.0			
PolygonSegments	4		4	Number of cross-sectio...
PolygonRadius	1.5	cm	1.5cm	Outer radius of cross-se...
StartHelixRadius	15	cm	15cm	Start radius from polygo...
RadiusChange	3.1	cm	3.1cm	Radius change per turn
Pitch	0	cm	0cm	Helix pitch
Turns	8		8	Number of turns
SegmentsPerTurn	36		36	Number of segments p...
RightHanded	1		1	Helix direction, non-zer...

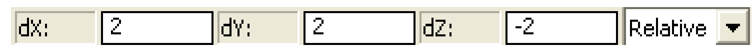
Note: Above operation creates a planar spiral when pitch is set to 0.

Create Coil (*Contd..*)

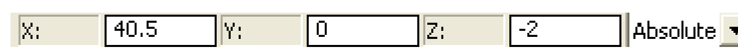
- **Draw Box**
  - Select the menu item ***Draw → Box***
    1. Using the coordinate entry fields, enter the box position
       - **X: 14, Y: 0, Z: -2**, Press the **Enter** key



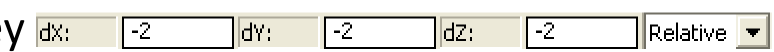
    2. Using the coordinate entry fields, enter the opposite corner
       - **dX: 2, dY: 2, dZ: -2**, Press the **Enter** key

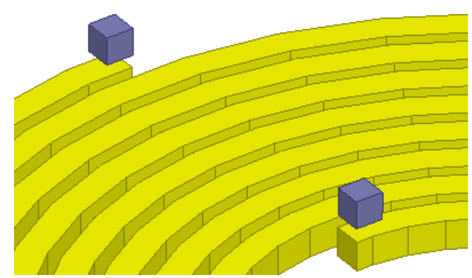


- **Draw another Box**
  - Select the menu item ***Draw → Box***
    1. Using the coordinate entry fields, enter the box position
       - **X: 40.5, Y: 0, Z: -2**, Press the **Enter** key



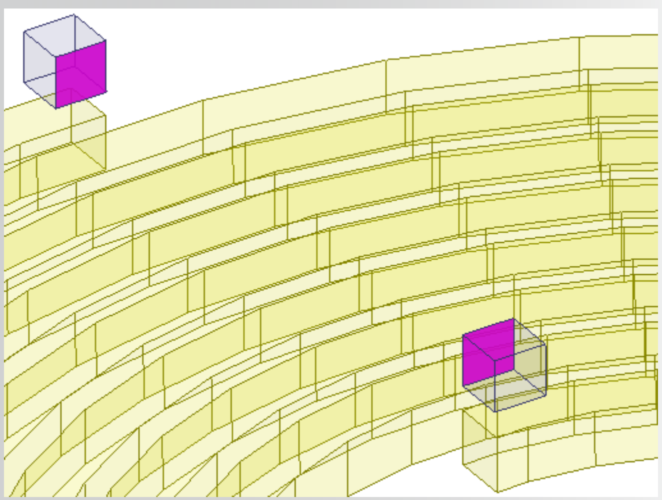
    2. Using the coordinate entry fields, enter the opposite corner
       - **dX: -2, dY: -2, dZ: -2**, Press the **Enter** key

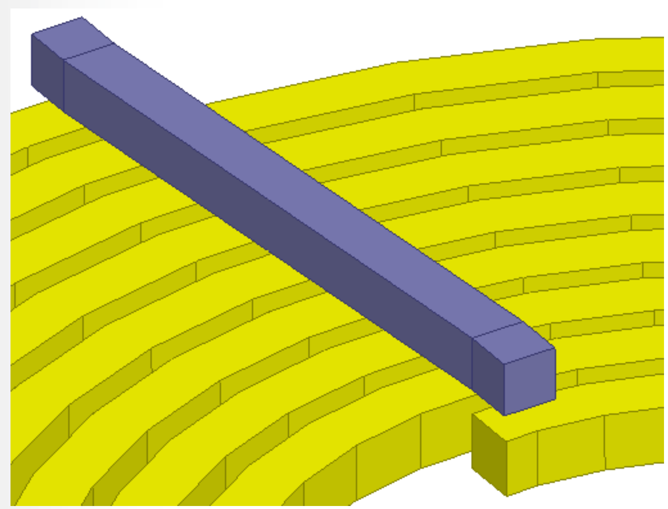




Create Coil (*Contd..*)

- **Connect Surfaces**
  - Select the menu item ***Edit→Select →Faces***
  - Select the faces of the box as shown in image below
  - Select the menu item ***Modeler →Surface →Create Object from Face***
  - Select the resulting sheet objects from the history tree
  - Select the menu item ***Modeler →Surface →Connect***





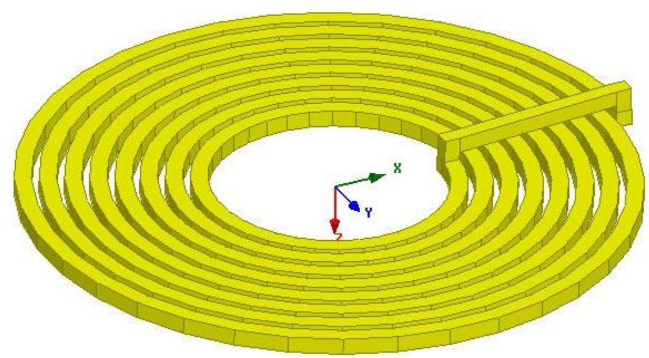
Note: The boxes created in last step and object created in this step will represent end connection of the coil

Create Coil (*Contd..*)

- **Duplicate Boxes**
  - Select **Box1** and **Box2** from the history tree
  - Select the menu item ***Edit → Duplicate → Along Line***
    1. Using the coordinate entry fields, enter the first point of duplicate vector
       - **X: 0, Y: 0, Z: 0**, Press the **Enter** key 

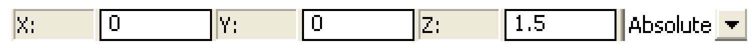
    2. Using the coordinate entry fields, enter the second point
       - **dX: 0, dY: 0, dZ: 1**, Press the **Enter** key 

    3. Total Number: **2**
    4. Press **OK**



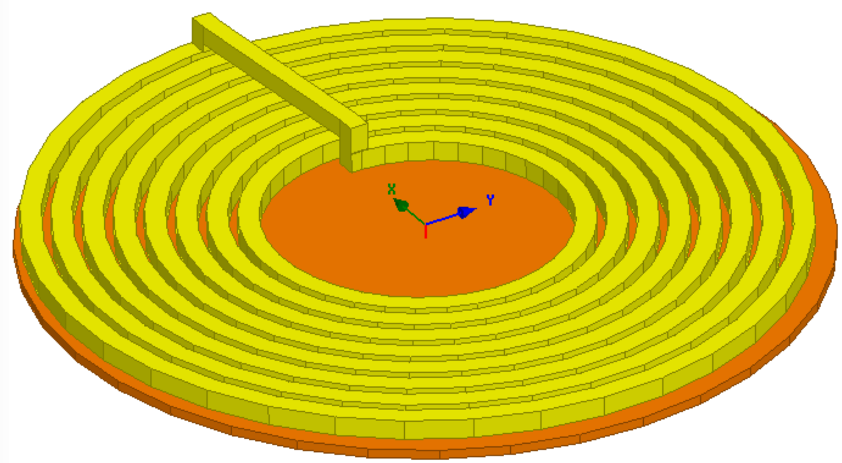
- **Unite Objects**
  - Select the menu item ***Edit → Select → Objects***
  - Select the menu item ***Edit → Select All***
  - Select the menu item ***Modeler → Boolean → Unite***

Create Disk

- **To Create Disk**
  - Select the menu item ***Draw → Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - **X: 0, Y: 0, Z:1.5**, Press the **Enter** key 

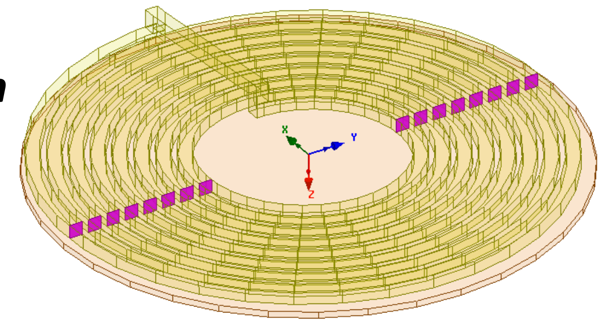
    2. Using the coordinate entry fields, enter the radius and height
       - **dX: 41, dY: 0, dZ:1**, Press the **Enter** key 

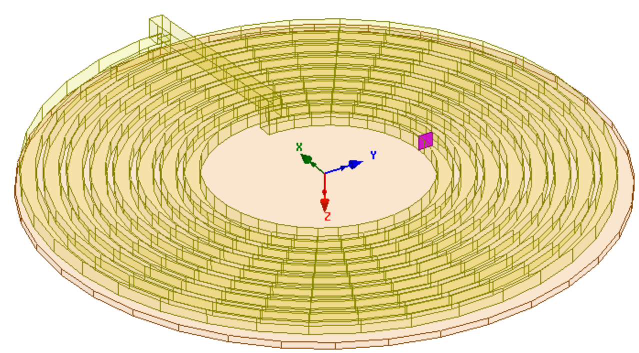
    3. Number of Segments: **36**
  - Change the name of the Object to **Disk** and color to **Orange**
  - Change the material of the object to **cast_iron**



Assign Excitations

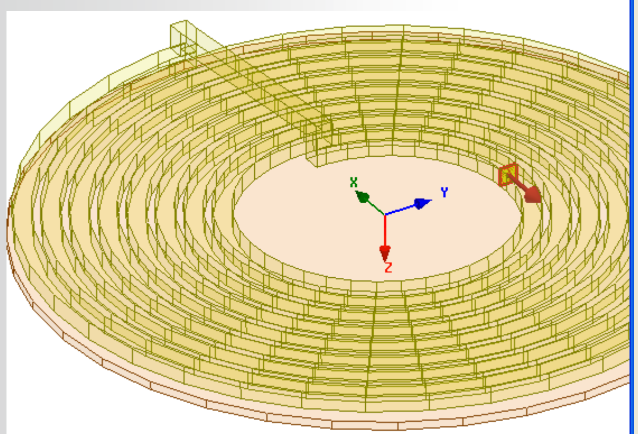
- **Create Coil Terminal**
  - Select the object **Coil** from the history tree
  - Select the menu item ***Modeler → Surface → Section***
  - In Section window,
    1. Section Plane: **YZ**
    2. Press the **OK** button
  - Change the name of the resulting object to **Coil_Terminal**
  - Select the sheet **Coil_Terminal** from the history tree
  - Select the menu item ***Modeler → Boolean → Separate Bodies***
  - Delete all the resulting sheets apart from **Coil_Terminal**

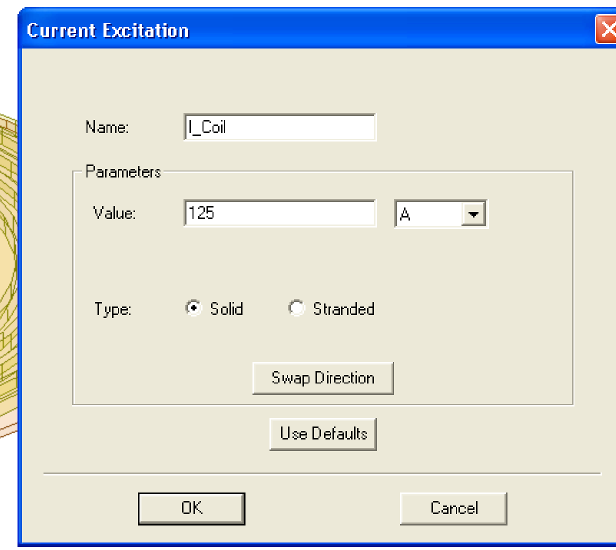




Assign Excitations (*Contd..*)

- **Assign Excitation**
  - Select the object **Coil_Terminal** from the history tree
  - Select the menu item ***Maxwell 3D → Excitations → Assign → Current***
  - In Current Excitation window,
    1. Name: **I_Coil**
    2. Value: **125 A**
    3. Type: **Solid**
    4. Press **OK**





Note: The current value assigned for static solvers is in Ampere-Turns. Users should multiply the current value by number of turns in winding and specify resulting value in Current Excitation window.

ANSYS

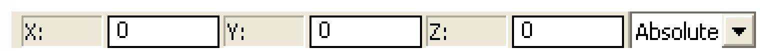
Resolve Skin Depth

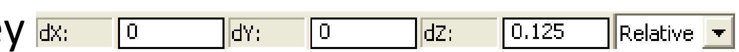
- **Compute Skin Depth**
  - Skin depth is a measure of how current density concentrates at the surface of a conductor carrying Alternating Current.
  - It is a function of the Permeability, Conductivity and frequency
  - Skin Depth in meters is defined as follows:

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

  - Where:
    - ω is the angular frequency, which is equal to $2\pi f$. (f is the source frequency which in this case is 500Hz).
    - σ is the conductor's conductivity; for Cast Iron its 1.5e6 S/m
    - μ_r is the conductor's relative permeability; for Cast Iron its 60
    - μ_o is the permeability of free space, which is equal to $4\pi\times10^{-7}$ A/m.
  - For our model the skin depth is approximately 0.24 cm.
  - After three skin depths, the induced current will become almost negligible.

Resolve Skin Depth (*Contd..*)

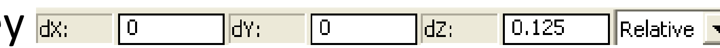
- **Create Surface layers to Assist with the Skin Depth Meshing**
  - Select the menu item ***Edit → Select → Faces***
  - Select the face on the disk that is closest to the coil
  - Select the menu item ***Modeler → Surface → Create Object from Face***
  - Select the resulting sheet objects from the history tree
  - Select the menu item ***Edit → Arrange → Move***
    1. Using the coordinate entry fields, enter the reference point of move vector
       - **X: 0, Y: 0, Z: 0**, Press the Enter key 

    2. Using the coordinate entry fields, enter the target point
       - **dX: 0, dY: 0, dZ: 0.125**, Press the Enter key 

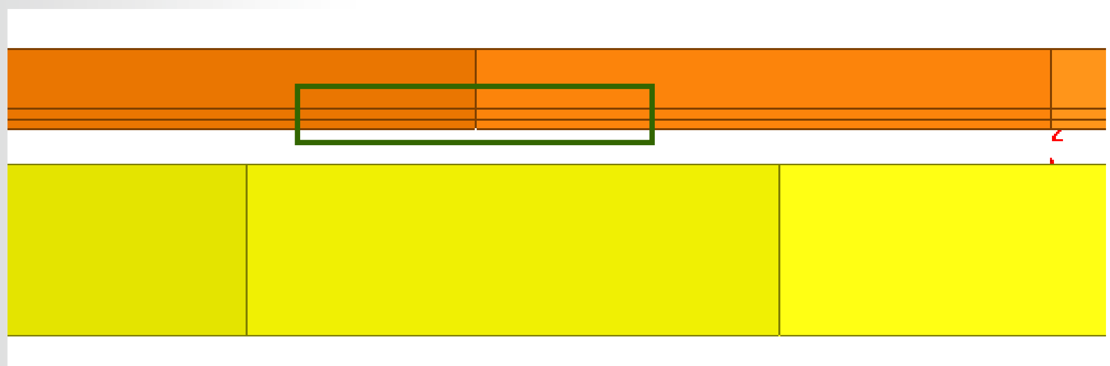
Note: As we saw on last slide that we need to resolve a skin depth of 0.24. Assigning mesh operations to resolve such a small skin depth region can lead to large mesh size. To resolve the skin depth effectively without need of too many elements, we are creating surfaces at a depth of 0.125. This will ensure that there are at least two elements in skin region

Resolve Skin Depth (*Contd..*)

- **Duplicate Sheet**
  - Select the moved sheet **Disk_ObjectFromFace1**
  - Select the menu item ***Edit → Duplicate → Along Line***
    1. Using the coordinate entry fields, enter the first point
       - **X: 0, Y: 0, Z: 0**, Press the **Enter** key 

    2. Using the coordinate entry fields, enter the second point
       - **dX: 0, dY: 0, dZ:0.125**, Press the **Enter** key 

    3. Total Number: **2**
    4. Press **OK**

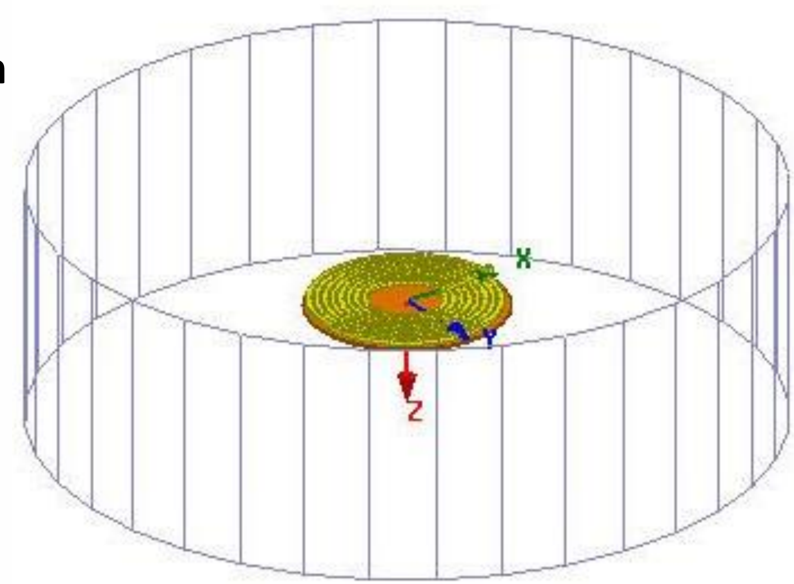


Draw Region

- **Create Simulation Region**
  - Select the menu item ***Draw*** **→** ***Regular Polyhedron***
    1. Using the coordinate entry fields, enter the center of the base
       - **X: 0, Y: 0, Z:-50**, Press the **Enter** key 

    2. Using the coordinate entry fields, enter the radius
       - **dX: 150, dY: 0, dZ: 100**, Press the **Enter** key 

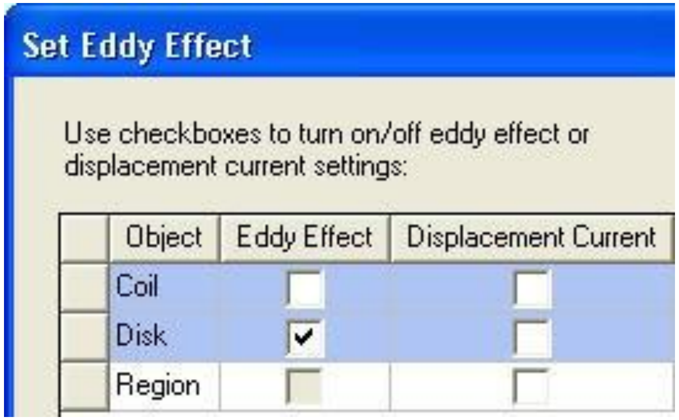
    3. Number of Segments: **36**
  - Change the name of the object to **Region**
  - Change Display Wireframe: ☑ **Checked**



Set Eddy Effect

- **Set Eddy Calculation for Disc**
  - Select the menu item ***Maxwell 3D → Excitations → Set Eddy Effects***
  - In Set Eddy Effects window,

    1. Coil
       - Eddy Effects: ☐ **Unchecked**
    2. Disk
       - Eddy Effects: ☑ **Checked**
       - Displacement Current: ☐ **Unchecked**
    3. Press **OK**



Note: Set Eddy Effects option enables users to turn off induced eddy current calculation in required objects. When a coil terminal is defined as Stranded in current definition, eddy effects are not calculated for the coil. But if coil terminal is defined as solid, users will have to turn off eddy effects manually if induced eddy current calculation in coil is not required

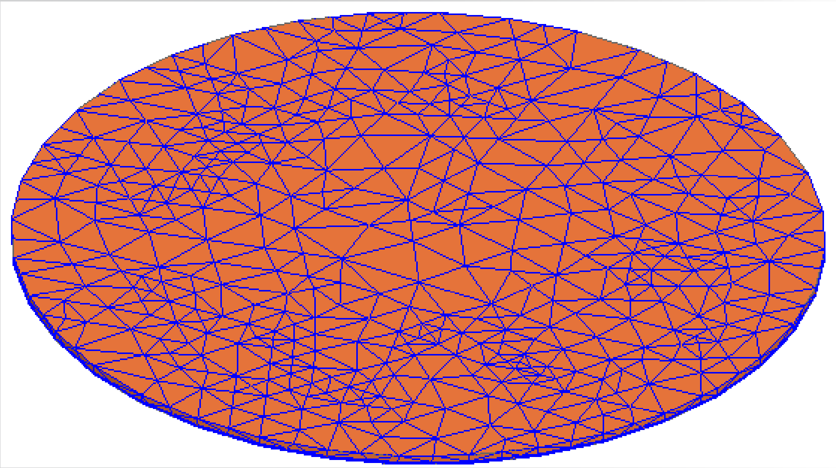
Analyze

- **Create an analysis setup:**
  - Select the menu item ***Maxwell 3D → Analysis Setup → Add Solution Setup***
  - Solution Setup Window:
    1. **General** Tab
       - Percentage Error: **2**
    2. **Convergence** Tab
       - Refinement Per Pass: **15 %**
    3. **Solver** Tab
       - Adaptive Frequency: **500 Hz**
    4. Click the **OK** button
- **To start the solution process:**
  - Select the menu item ***Maxwell 3D → Analyze All***

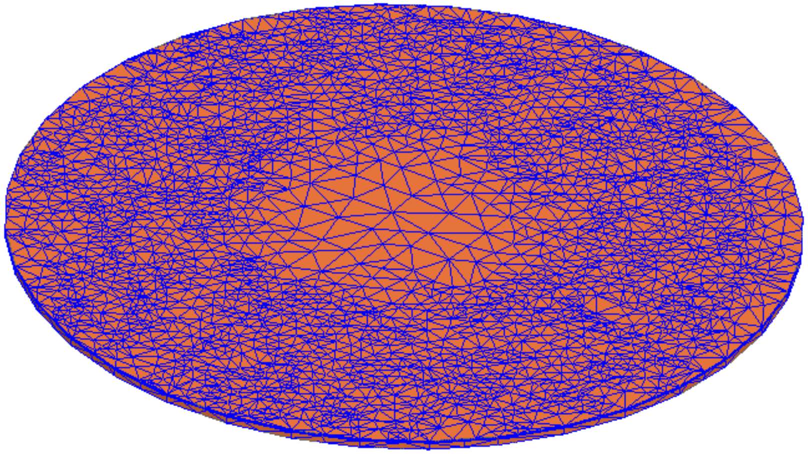
Note: Adaptive Frequency in Solution Setup window indicates the frequency of alternating magnetic field and hence frequency of applied excitation. Users can do frequency sweep for calculation at multiple frequency points

Plot Mesh

- **Plot Mesh on Disk**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D → Fields → Plot Mesh***
  - In Create Mesh Plot window,
    1. Press **Done**



Top View



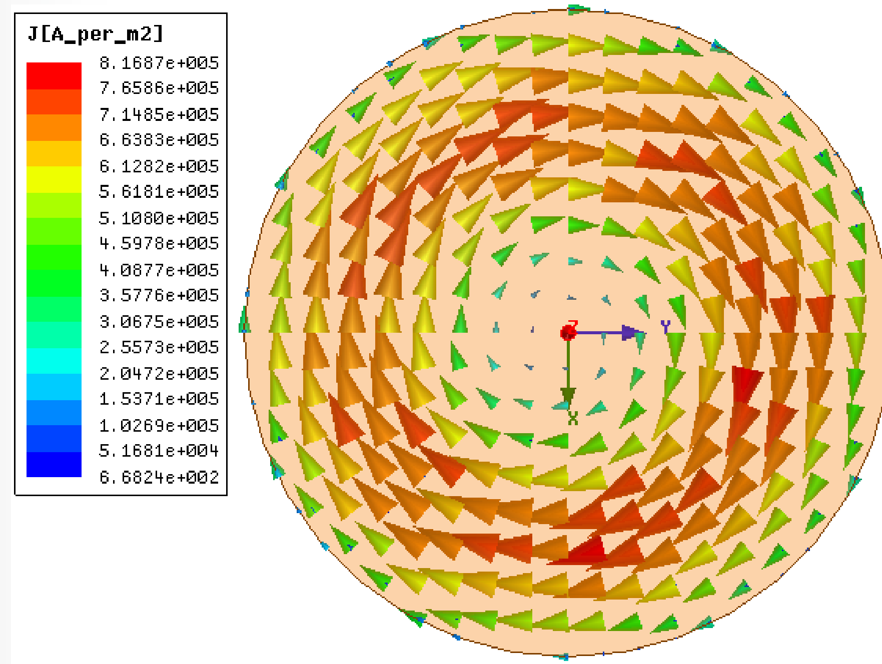
Bottom View: Notice the effect of the automatic adaptive meshing

Calculate Total Ohmic Loss

- **Calculate Ohmic Losses in Disk**
  – Select the menu item ***Maxwell 3D → Fields → Calculator***
  – In Fields Calculator window,
    1. Select **Input > Quantity > OhmicLoss**   Scl : Ohmic-Loss
    2. Select **Input > Geometry**
       – Select **Volume**
       – Select **Disk** from the list
       – Press **OK**   Vol : Volume(Disk)
    3. Select **Scalar >** **Integrate**
    4. Select **Output > Eval**   Scl : Integrate(Volume(Disk), Ohmic-Loss)
  – The evaluated value of losses in the Disk should be around **270.38 W**

Plot Current Density Vectors

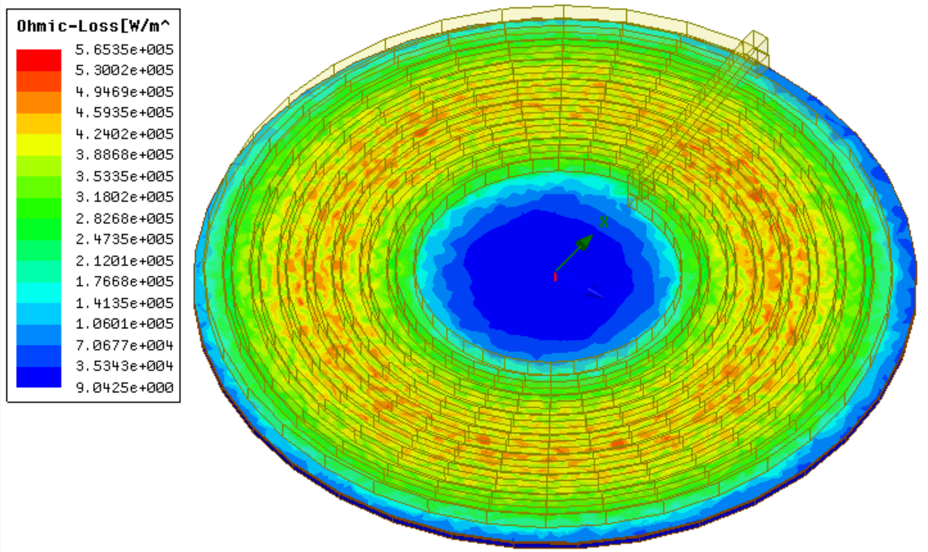
- **Plot Current Density Vectors**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D → Fields → Fields → J → Vector_J***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



Note: Default plot may not look as shown in image. Users can modify attributes of plot to make it look better. Double click on the legend to modify plot attributes.

Plot Ohmic Loss Distribution

- **Plot Ohmic Losses on Disk**
  - Select the object **Disk** from the history tree
  - Select the menu item ***Maxwell 3D → Fields → Fields → Other → Ohmic_Loss***
  - In Create Field Plot window,
    1. Plot on surface only: ☑ **Checked**
    2. Press **Done**



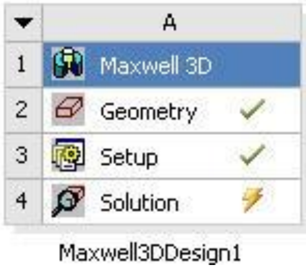
- **Save the file**
  - Select the menu item ***File → Save As***
  - Save the file with the name **"WS2_BasicEddyCurrentAnalysis.mxwl"**
  - Select the menu item ***File → Exit***

APPENDIX:

Maxwell to ANSYS Mechanical Coupling…

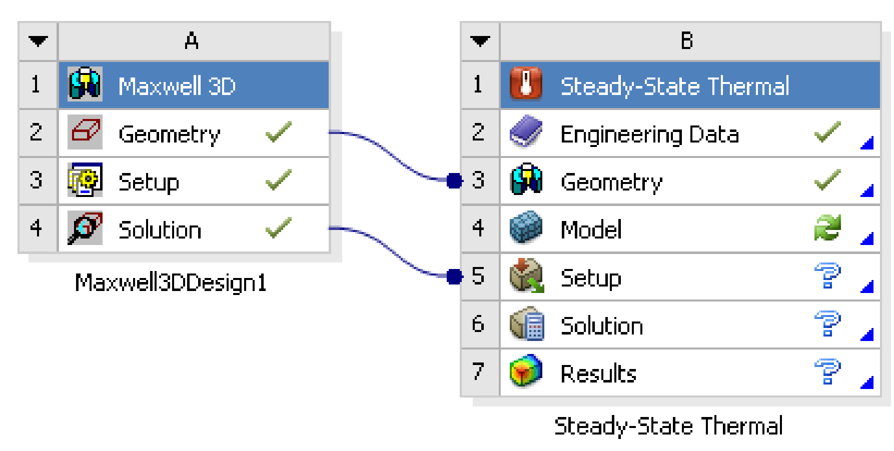
ANSYS Mechanical

- **ANSYS Mechanical**
  - In this section of workshop, we will map the losses calculated by Maxwell to ANSYS Mechanical Steady State Thermal Solver. Solution in ANSYS Mechanical will give final temperature distribution of the objects. Mapping data from Maxwell to ANSYS Mechanical will be done through Workbench interface
- **Launch ANSYS Workbench**
  - Select the Microsoft **Start** button and Select **Programs > ANSYS 14.5 > Workbench 14.5**
- **Import Maxwell Project**
  - In Workbench Project window, select the menu item ***File → Import***
  - Set the file type to **Maxwell Project File**
  - Browse to the file **WS2_EddyCurrent_LossCalculation.mxwl** and **Open** it
  - A Maxwell 3D Design is created as shown below

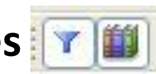


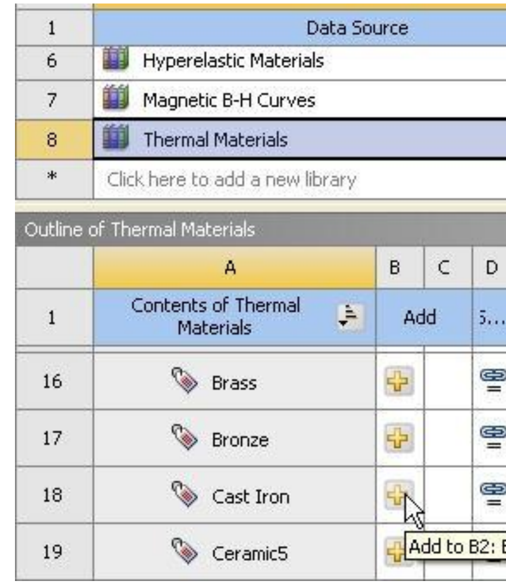
Create ANSYS Mechanical Design

- **Create Steady State Thermal Design**
  - Select a **Steady State Thermal** Analysis System from Analysis Systems list
  - Drag and drop it on the **Solution** tab of Maxwell 3D Design
  - Similarly drag and drop the **Geometry** tab of Maxwell on the **Geometry** tab of Steady State Thermal system. This will create a link between Maxwell Solution and Setup of ANSYS Mechanical
  - Right click on Solution tab of Maxwell 3D Design and Select **Update** to update the solution cell
  - Right click on the Geometry tab of Steady State Thermal system and select **Refresh**

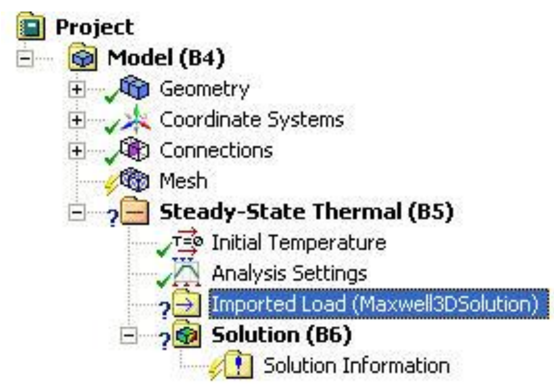


Define Material Database

- **Define Material Database**
  - Right click on the tab **Engineering Data** and select **Edit**
  - In Engineering Data window,
    1. Select the icon **Engineering Data Sources** 
    2. Select the tab **Thermal Materials** from Data Sources
    3. Locate **Cast Iron** material from the list and select Add
  - Select **Return to Project** button

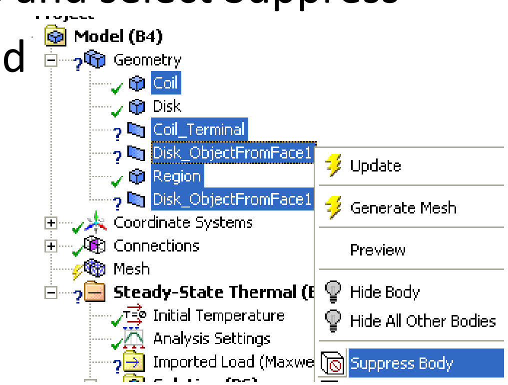


- **Launch ANSYS Mechanical**
  - Right click on **Model** tab of Steady State Thermal analysis system and select **Edit**
  - A tab corresponding to Maxwell data is automatically added in the tree to enable data mapping



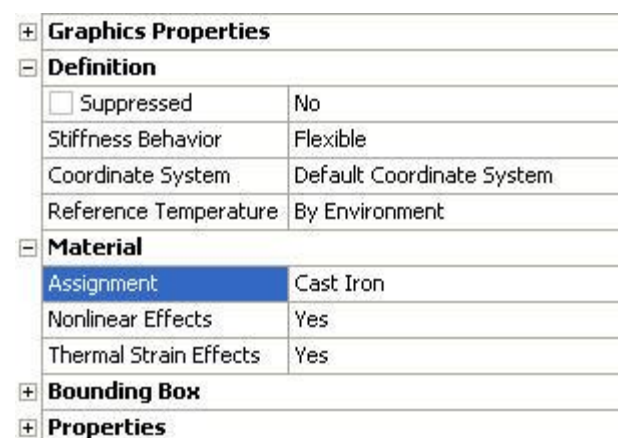
Setup Geometry

- **Suppress Unwanted Objects**
  - Expand the Project tree for **Geometry** under Model
  - Select all bodies apart from **Disk**, right click on the Bodies and select Suppress
  - This will keep only **Disk** for analysis and rest will be ignored



- **Specify Material**
  - Select the **Disk** from the tree and goto Details View window
  - In Details View window,

    1. Material
       - Assignment: set to **Cast Iron**

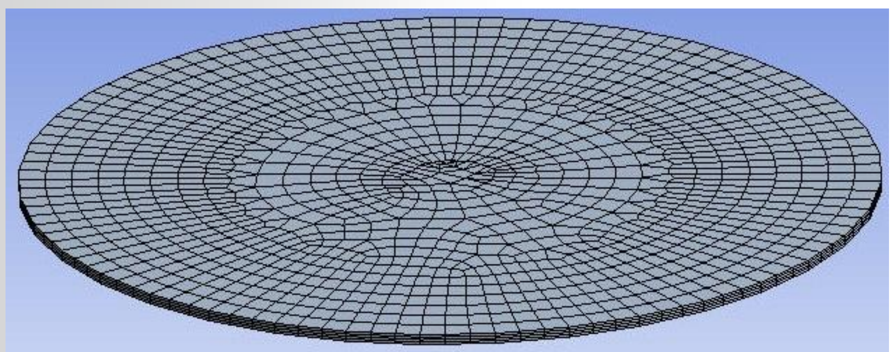


Generate Mesh

- **Set Mesh Parameters**
  - Right click on **Mesh** tab from specification tree and select ***Insert → Method***
  - In Details View window,
    1. Geometry: select the body **Disk**
    2. Method: **Sweep**
    3. Sweep Num Divs: **5**

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled
Target	Program Controlled
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	5
Sweep Bias Type	No Bias
Element Option	Solid

- **Generate Mesh**
  - Right click on **Mesh** tab and select **Generate Mesh**



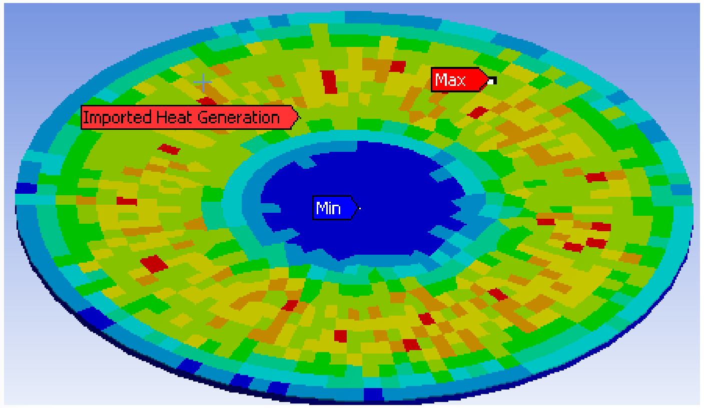
Note: Mesh specifications are important from the data mapping perspective to achieve good accuracy in mapped losses from Maxwell to ANSYS Mechanical. Above Settings will ensure mesh refinement close to mesh we have in Maxwell.

Map Data from Maxwell

- **Map Heat Loss Data from Maxwell**
  - Right click on the **Imported Load (Maxwell3DSolution)** and select ***Insert → Heat Generation***
  - In Details View window,
    1. Geometry: Select the body **Disk**
  - Right click on the tab **Imported Heat Generation** and select **Import Load**
  - Heat losses calculated in Maxwell be mapped to the mesh in ANSYS Mechanical
  - A Summery of mapped heat losses is shown below Imported Heat Generation tab. Scaling factor shown in this summery should be close to 1 to ensure correct data mapping. If not, mesh needs to be refined in important regions

Exporting Volume Loss Density With Scaling...

Object	Total Loss	Scaling Factor
Disk	270.381W	0.970063



Finalize Setup

- **Specify Convective Boundary**
  - Right click on **Steady State Thermal** tab from the specification tree and select ***Insert → Convection***
  - In Details View window,
    1. Geometry:
       - Change Selection Filter to **Faces**
       - Right click in Graphic window and select **Select All**
       - Press **Apply** in details view window
    2. Film Coefficient: **10 W/m²°C**

Scope	
Scoping Method	Geometry Selection
Geometry	38 Faces
Definition	
ID (Beta)	106
Type	Convection
Film Coefficient	10. W/m²·°C (ramped)
Ambient Temperature	22. °C (ramped)
Suppressed	No
Fluid Flow (Beta)	No

- **Create Temperature Plot for Disk**
  - Right click on **Solution** tab from specification tree and select ***Insert → Thermal → Temperature***

Solution

- **Run the Solution**
  - Right click on **Solution** tab from specification tree and select **Solve**
- **View Temperature Plot**
  - Select the **Temperature** plot from specification tree under **Solution**
  - Temperature distribution on Disk will be displayed in graphic window