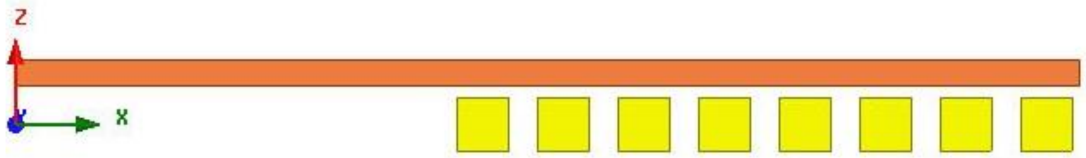
Workshop 2.1: Eddy Current Analysis

Release 2020R2

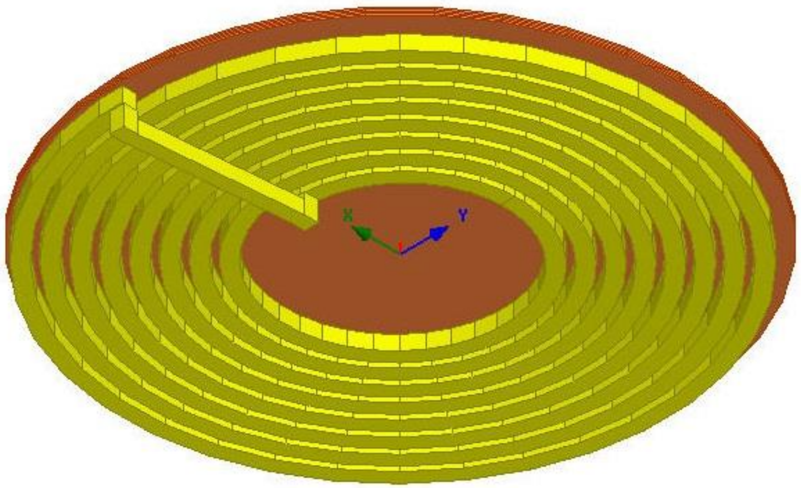
Ansys

Overview

- Introduction to the Eddy Current Solver
  - This workshop 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

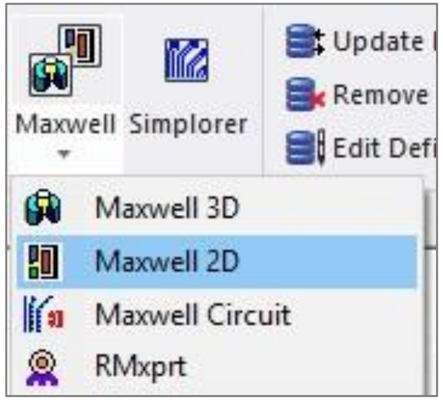


Actual 3D model

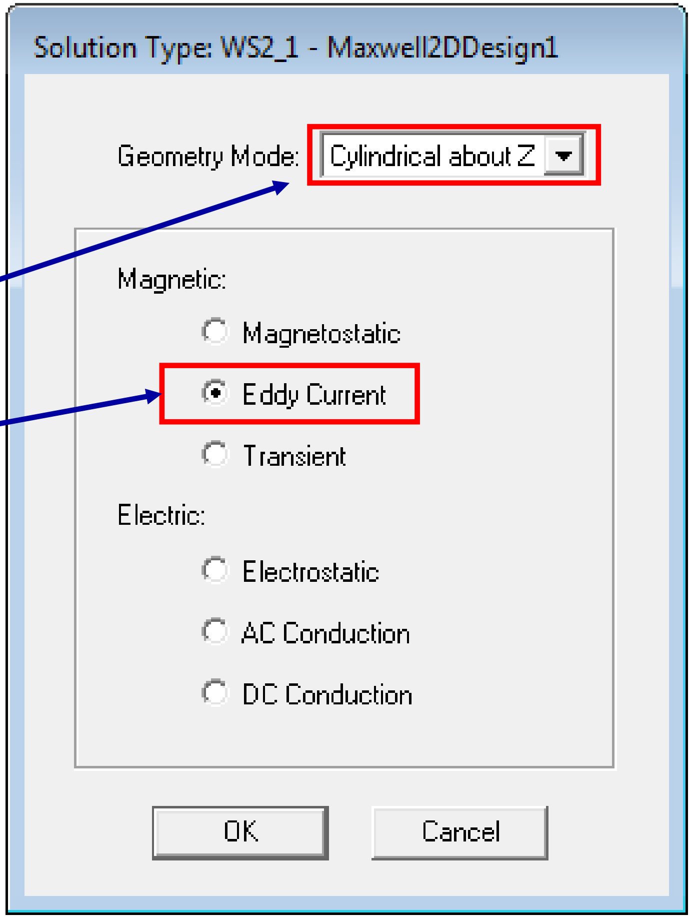
Ansys

Model Setup

- **Insert Design**
  - **Select the menu item *Project → Insert Maxwell 2D Des*ign, or click on the icon in drop down list Maxwell on panel Desktop**



- **Set Solution Type**
  - **Select the menu item *Maxwell 2D → Solution Type***
  - **Geometry Mode: *Cylindrical about Z***
  - **Choose *Magnetic → Eddy Current***
  - **Click the OK button**
- **Set Default Units**
  - **Select the menu item *Modeler → Units***
  - **Set units to cm (centimeters) and press OK**



Create Model

- Create Coil
  - Select the menu item *Draw → Rectangle*
    - Using the coordinate entry fields, enter the position of rectangle

      X: 17, Y: 0, Z: -1, Press the *Enter* key

    - 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*
      - Using the coordinate entry fields, enter the first point of duplicate vector

        X: 0, Y: 0, Z: 0, Press the *Enter* key

      - 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

Ansys

Create Model

- Create Plate
  - Select the menu item *Draw → Rectangle*
    - Using the coordinate entry fields, enter the position of rectangle

      X: 0, Y: 0, Z: 1.5, Press the *Enter* key

    - 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
- Create Solution Region
  - Select the menu item *Draw → Rectangle*
    - Using the coordinate entry fields, enter the position of rectangle

      X: 0, Y: 0, Z: -100, Press the *Enter* key

    - Using the coordinate entry fields, enter the opposite corner

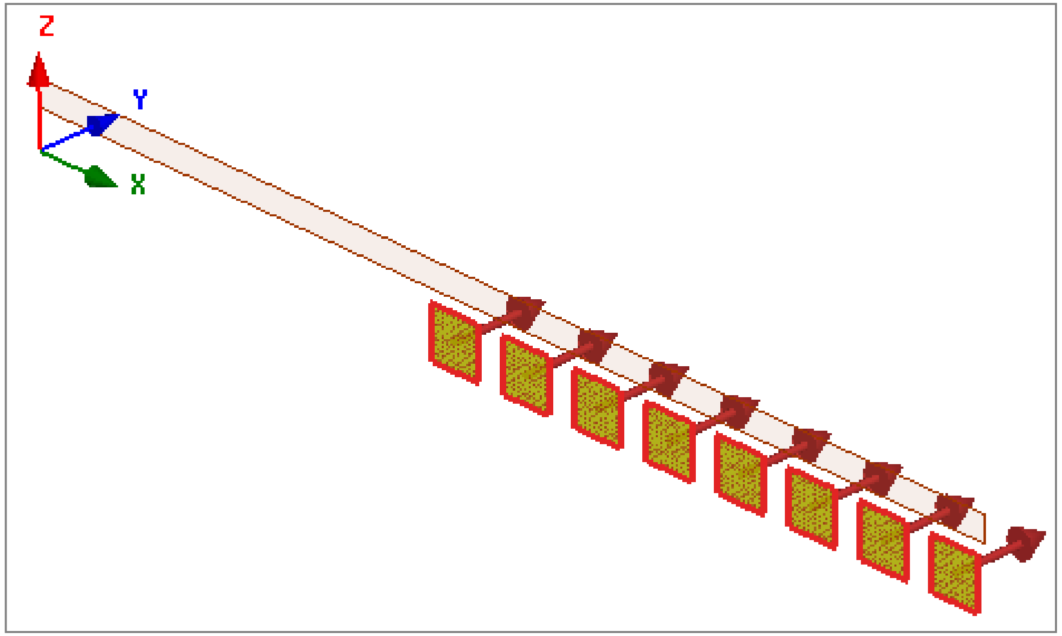
      dX: 120, dY: 0, dZ: 200, Press the *Enter* key

  - Change the name of resulting sheet to Region

Ansys

Excitations

- 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

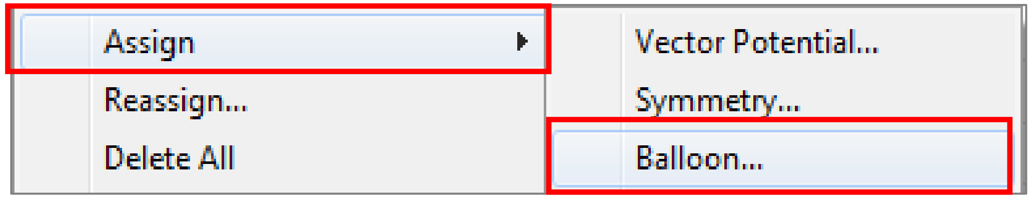


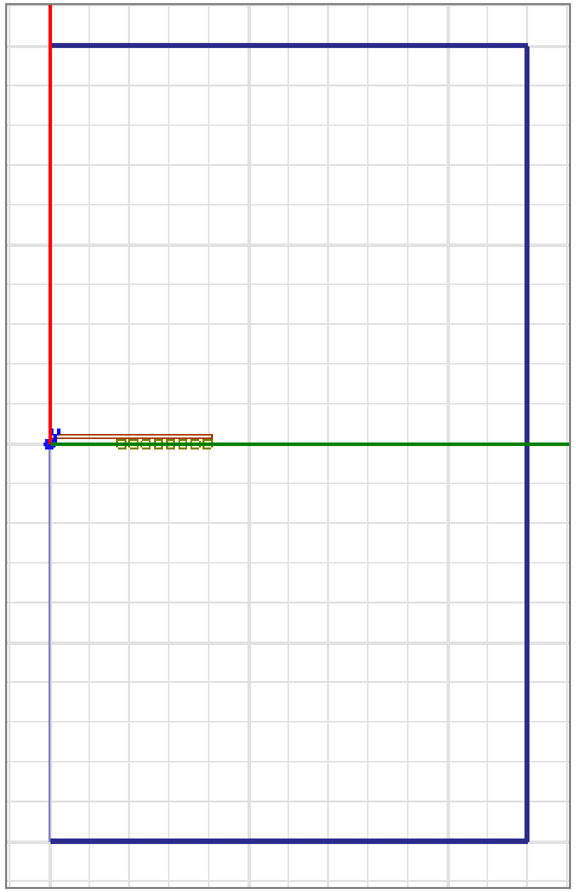
Note: Choosing Solid specifies that the eddy effects in the coil are considered. On the other hand, if Stranded is chosen, only the DC resistance is calculated and no AC effects in the coil are considered. Stranded is appropriate when skin depth is much larger than conductor thickness, for example when using Litz wires. Note that induced eddy effects in the plate will be calculated in either case

Ansys

Assign Boundaries and Parameters

- Assign Boundary
  - Select the object Region from history tree
  - Select the menu item *Edit → Select → Edges*
  - Select the three edges of the Region
  - Select the menu item *Maxwell 2D → Boundaries → Assign → Balloon*
  - In Balloon Boundary window, Press OK
- Assign Matrix Parameters
  - Select the menu item *Maxwell 2D → Parameters → Assign → Matrix*
  - In Matrix window, for all current Sources
  - Include: ☑ Checked
  - Press OK



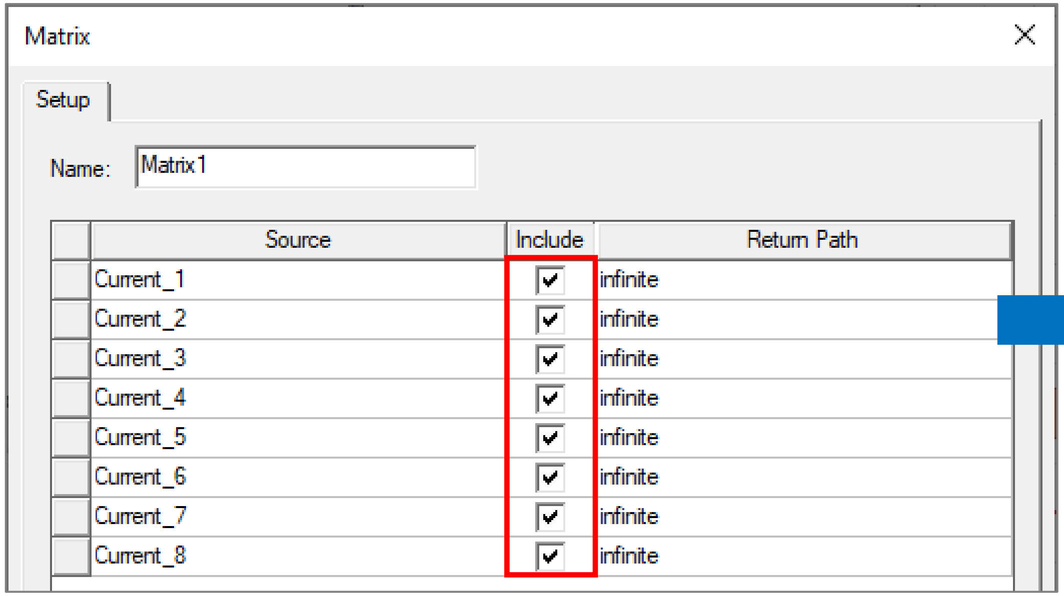


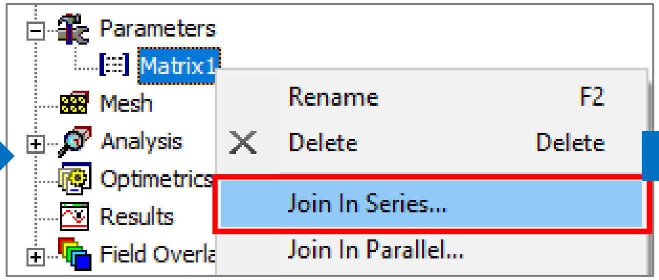
Note: Above settings will compute a [8x8] impedance matrix

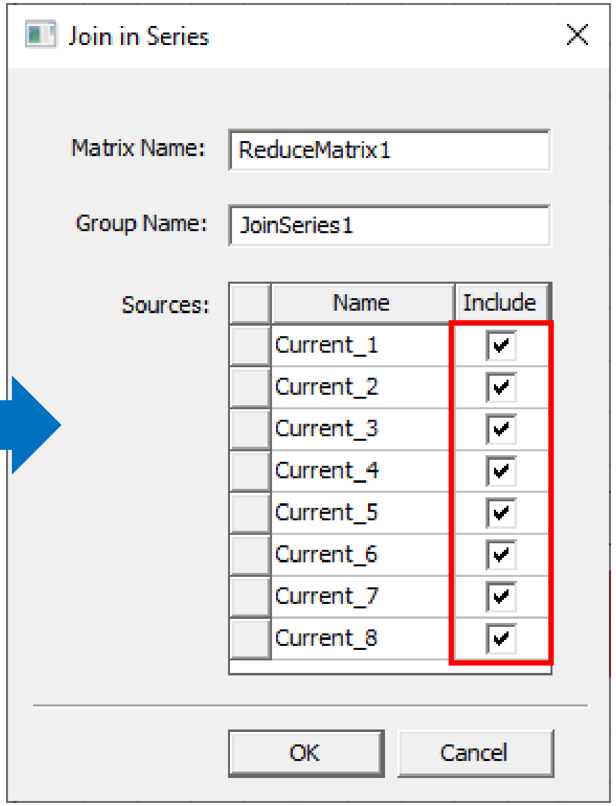
Ansys

Assign Matrix Parameter

- As the different turns are part of a single coil and are series-connected, the overall impedance can be found out by grouping the individual coils under a winding using the Matrix “Join in Series” reduction
- Each excitation (Current) represent one individual turn. Press Include to select all and then OK
- Use RMB on the just created Matrix and choose Join in Series option
- Check all currents (or press on Include) and press Ok button







Note: Above settings will compute a postprocessed [1x1] impedance matrix

Ansys

Skin Depth

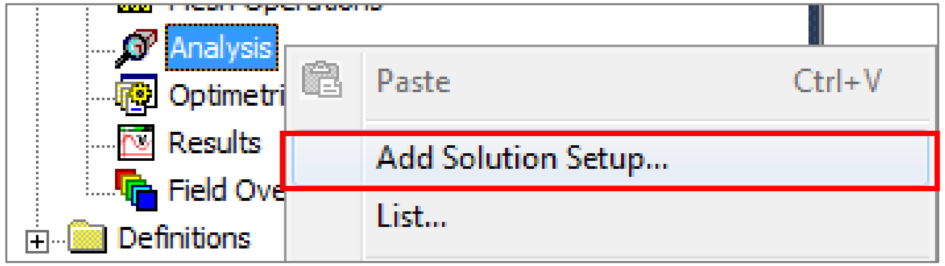
- **Compute the Skin Depth**
  - **Skin depth is a measure of how current density concentrates at the surface of a conductor carrying an alternating current. It is a function of permeability, conductivity and frequency**
  - **Skin depth d in meters is defined as follows:**

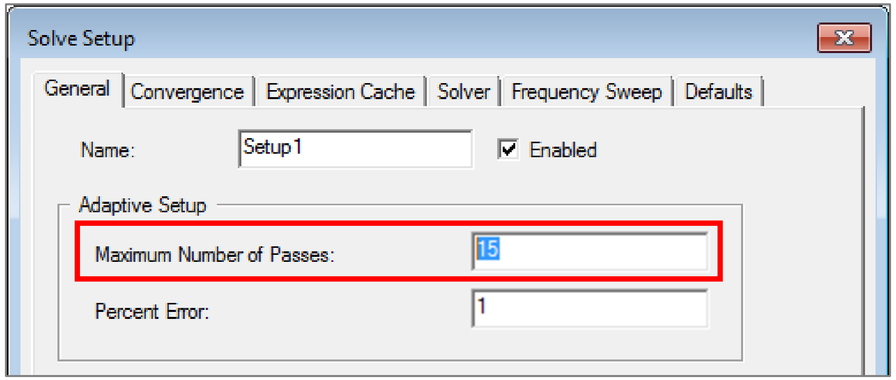
$$\delta = \sqrt{\frac{2}{\omega \mu_0 \mu_r \sigma}}$$

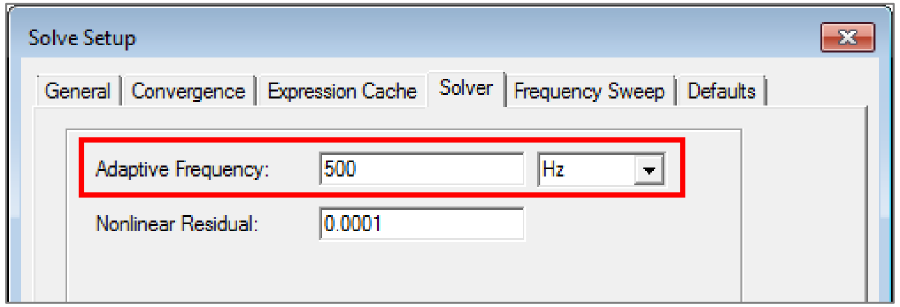
  - ω **is the angular frequency, which is equal to 2πf. (f is the frequency - in this case is 500Hz).**
  - σ **is conductor's conductivity; for cast iron it is 1.5e6 S/m**
  - μ **is conductor's relative permeability; for cast iron it is 60**
  - μ_0 **is permeability of free space, which is equal to 4*π*1e-7 H/m.**
  - **For the cast iron plate the skin depth result 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 inside the skin depth, so that mesh operations are not needed.**

Analyze

- **Create an analysis setup:**
  - Select the menu item ***Maxwell 2D→ Analysis Setup → Add Solution Setup***
  - ***General Tab***
    - **Maximum Number of Passes: 15**
  - **Solver Tab**
    - **Adaptive Frequency: 500 Hz**
  - **Click the OK button**
- **Start the solution process:**
  - **In the Project Manager window *RMB on Setup1 →Analyze***

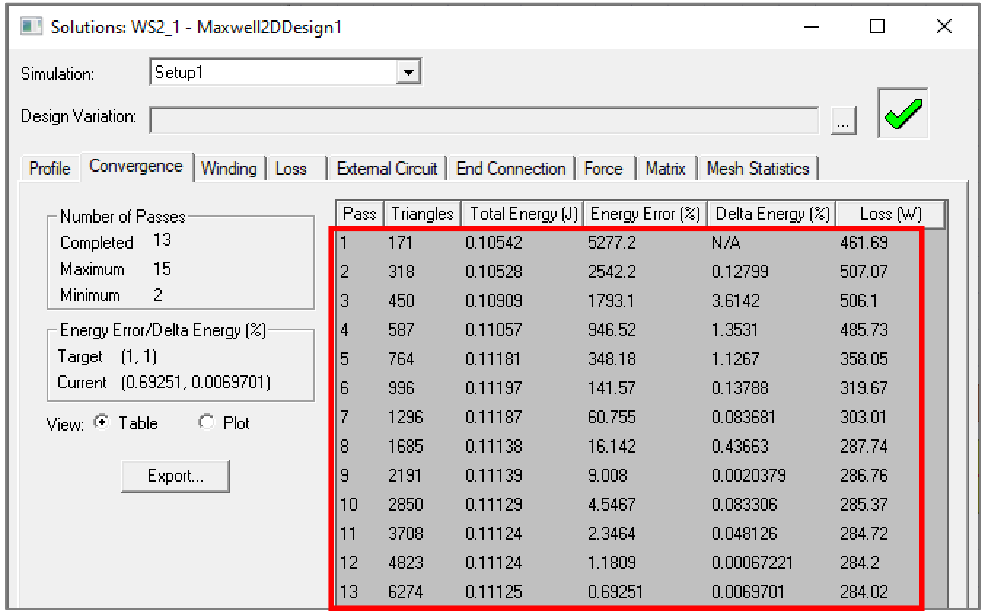


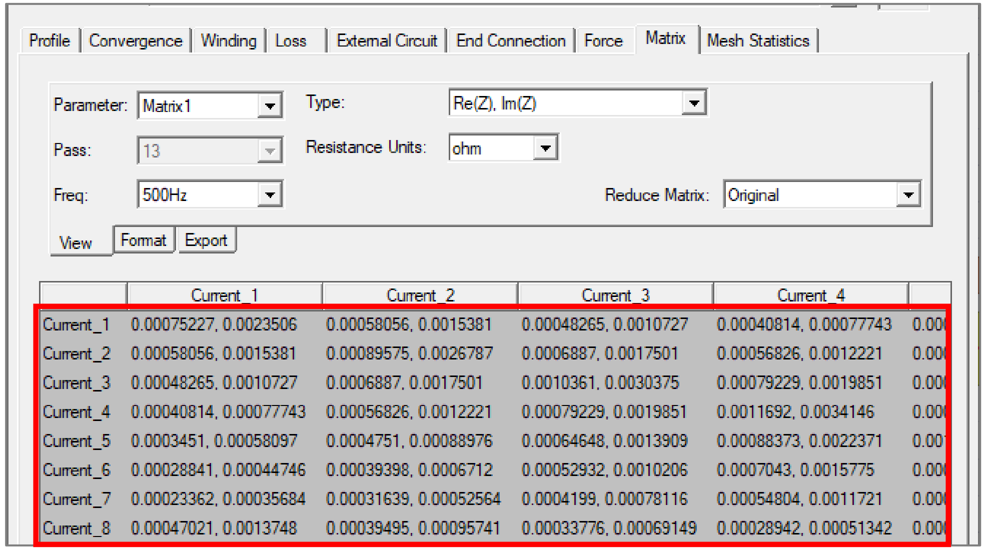


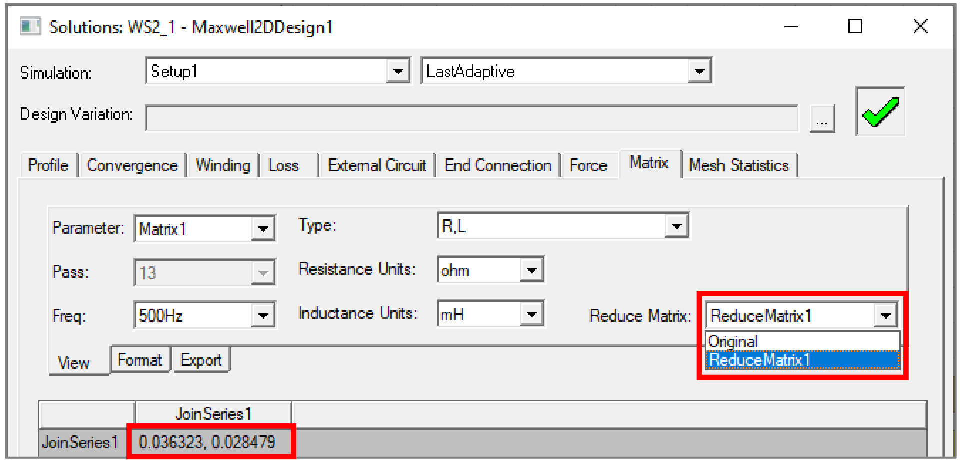


Solution data

- View Solution Information
- Select the menu ***Maxwell 2D → Results → Solution Data***
- Select the Convergence tab to view Convergence
- Select Matrix tab to view the Impedance Matrix
- By default, the results are displayed as [R,Z] but can be also shown as [R,L] or as coupling coefficients
- Enable the Reduce Matrix to check the overall Impedance (coupling coefficients are not available)



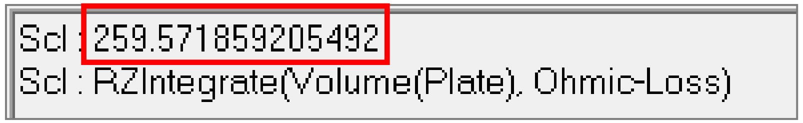




Ansys

Power Loss Computation

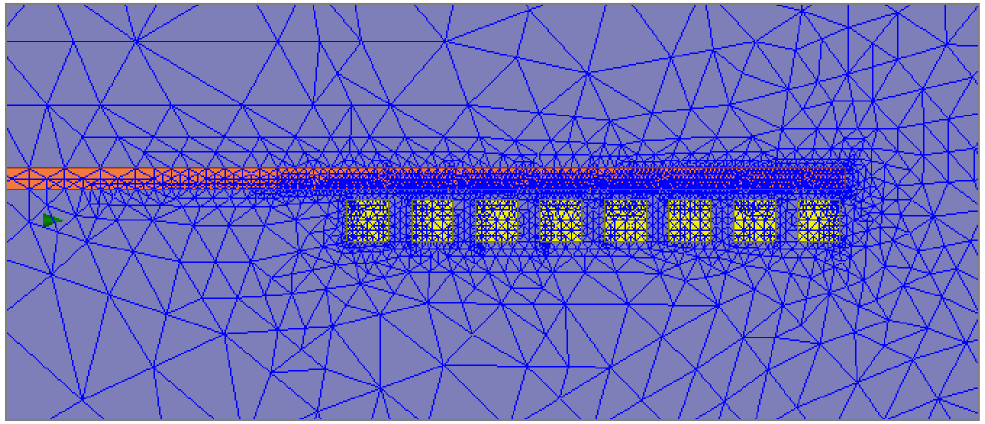
- Compute Total Power Loss in the Plate
  - Select the menu item *Maxwell 2D → Fields → Calculator*
  - In Fields Calculator window,
    - Select Input > Quantity > OhmicLoss
    - Select Input > Geometry
    - Select Volume
    - Select Plate
    - Press OK
    - Select Scalar > Integral > RZ
    - Select Output > Eval



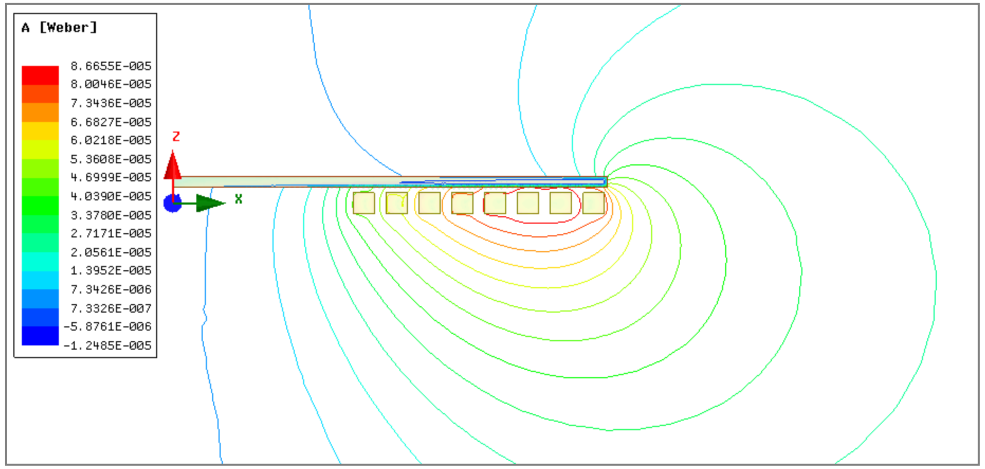
Note: The evaluated loss in the Plate should be about 260 W

Create Field Plots

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

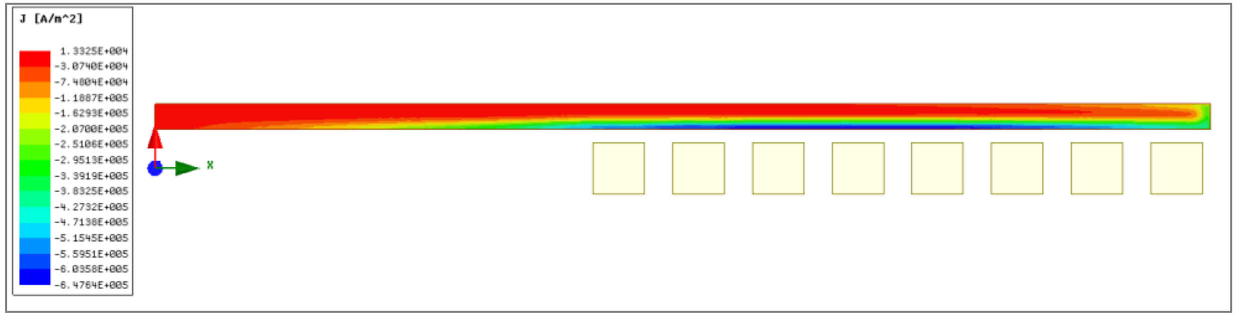


Note that the flux lines are attracted to the plate since it is magnetic. Also, skin effects are present in the plate since there are eddy currents flowing in it.

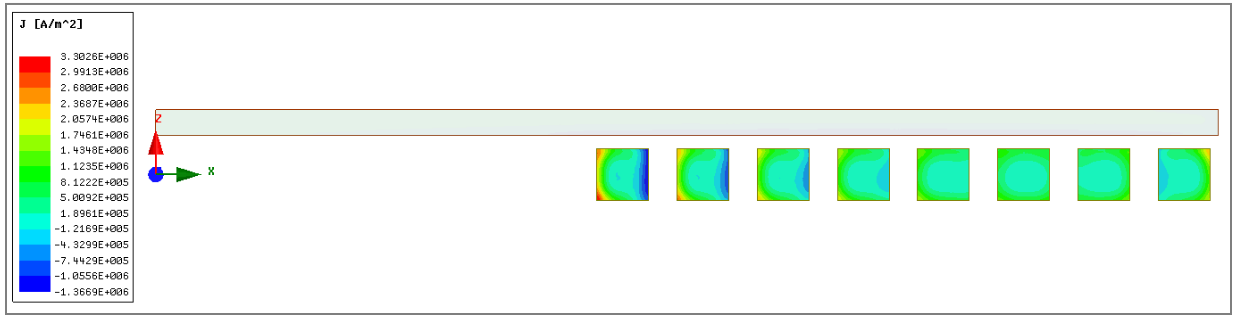
Ansys

Create Field Plots

- **Plot Current Density Scalar on Plate**
  - **Select the sheet Plate from history tree**
  - **Select the menu *Maxwell 2D → Fields → Fields → J → JAtPhase***
  - **In Create Field Plot window, Press Done**



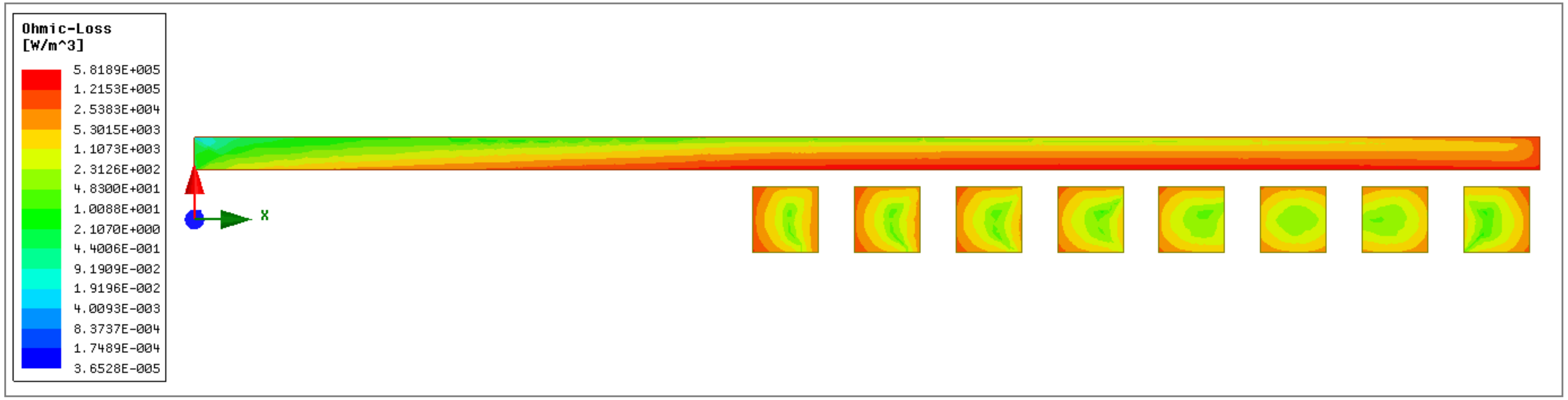
- **Plot Current Density Scalar on Coils**
  - **Press Ctrl and select all Coils from history tree**
  - **Select the menu *Maxwell 2D → Fields → Fields → J → JAtPhase***
  - **In Create Field Plot window, Press Done**



Note: Hide previous plots by selecting View → Active View Visibility → Fields Reporter and unchecking the previous plots

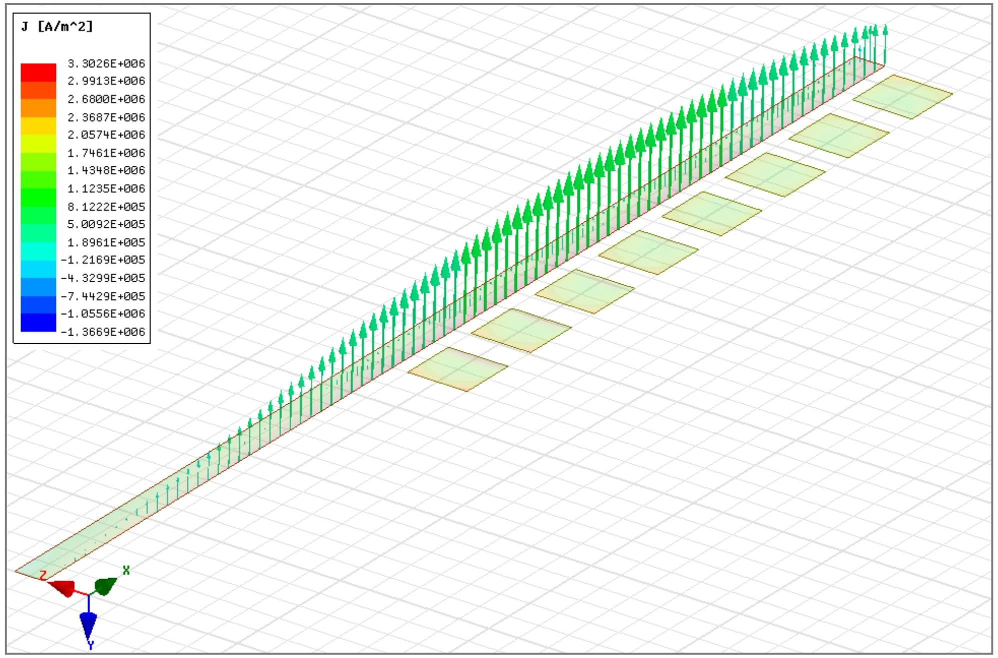
Plot Ohmic Loss Distribution

- Plot Ohmic Losses
  - Press Ctrl and select all Coils and Plate
  - Select the menu *Maxwell 2D → Fields → Fields → Other → Ohmic_Loss*
  - In Create Field Plot window, Press Done
- Modify Plot Attributes
  - Double click on the Legend to modify plot
  - Scale tab
    - Select Log
    - Press Apply and Close



Plot Current Density Vectors

- Plot Current Density vectors
  - Select the sheet Plate from history tree
  - Select the menu ***Maxwell 2D → Fields → Fields → J → J Vector***
  - In Create Field Plot window, Press Done
  - Double click on the Legend to modify plot
  - Plots tab
    - Plot: Change to J_Vector1
    - Change Vector plot spacing
    - Min: 0.5 - Max: 0.5
    - Press Apply and Close
- Animate Plot
  - Select the Vector plot from Project Manager tree, ***RMB → Animate***
  - In Setup Animation window, press OK with default settings
  - A window will appear to start, stop, pause or export the animation



Ansys

APPENDIX

Solve DC Problem

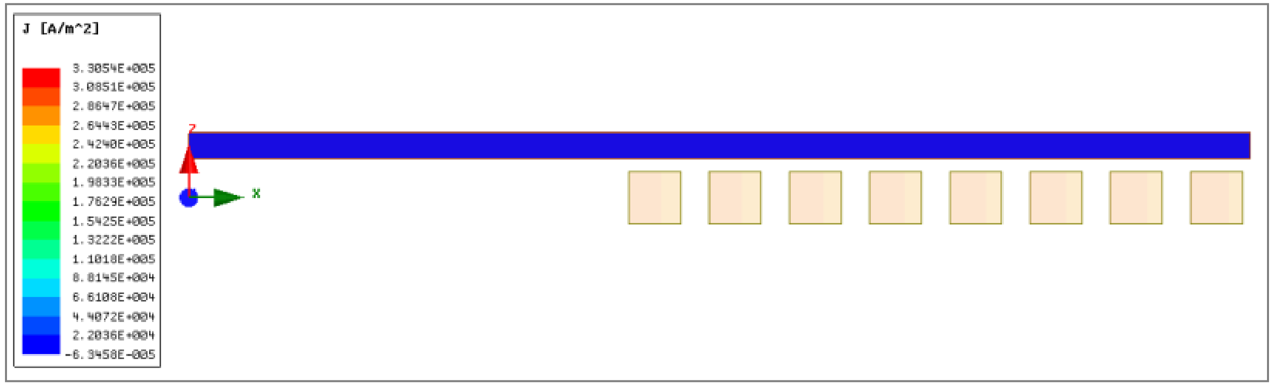
Ansys

Modify Setup and Solve

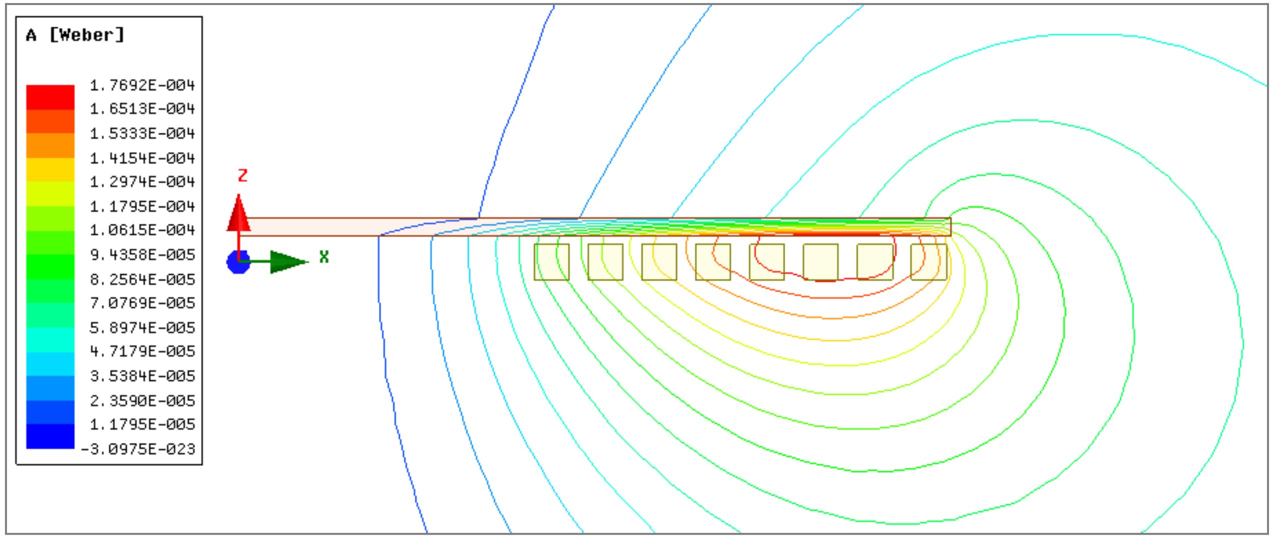
- Copy Design
  - Select the Maxwell2D Design from Project Manager tree, right click and select Copy
  - Right click on the Project name in Project Manager tree and select Paste
- Change Analysis Setup
  - Expand the project Manager tree for newly created design
  - Double click on Setup1 under Analysis in the tree and select the Solver tab
  - Adaptive Frequency: Change to 0.001 Hz
- Start the solution process:
  - In the Project Manager window *RMB on Setup1 → Analyze*

Results

- **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.**



Saving the Project

- This completes the workshop
- Save the file with the name Workshop_2_1 in the working folder

End of Presentation