# ANSYS MAXWELL SIMULATIONS FOR WIRELESS POWER APPLICATIONS

Christian Merz, Technical Lead WPT

# AGENDA

- Introduction to Ansys Maxwell
- Introduction to ICEPAK
- Simulation steps of WE Qi A10 WPCC coils (inductance,$R_{AC}$,field distribution, coupling) with Ansys Maxwell
- Thermal simulation of a WE Qi A10 WPCC coil
- Comparison between measurements and simulations

# INTRODUCTION

## ANSYS Maxwell

- high-performance software package that uses finite element analysis to solve electromagnetic problems
- is integrated into the Ansys Electronics Desktop (AEDT), together with all the other ANSYS Electromagnetic Tools
- solves the electromagnetic field problems by solving Maxwell's equations in a finite region of space
- geometries can be created directly inside Maxwell or imported from external CAD tools (e.g. SolidEdge)
- this presentations shows the workflow of a 3D simulation of a Qi-coil with Ansys Maxwell

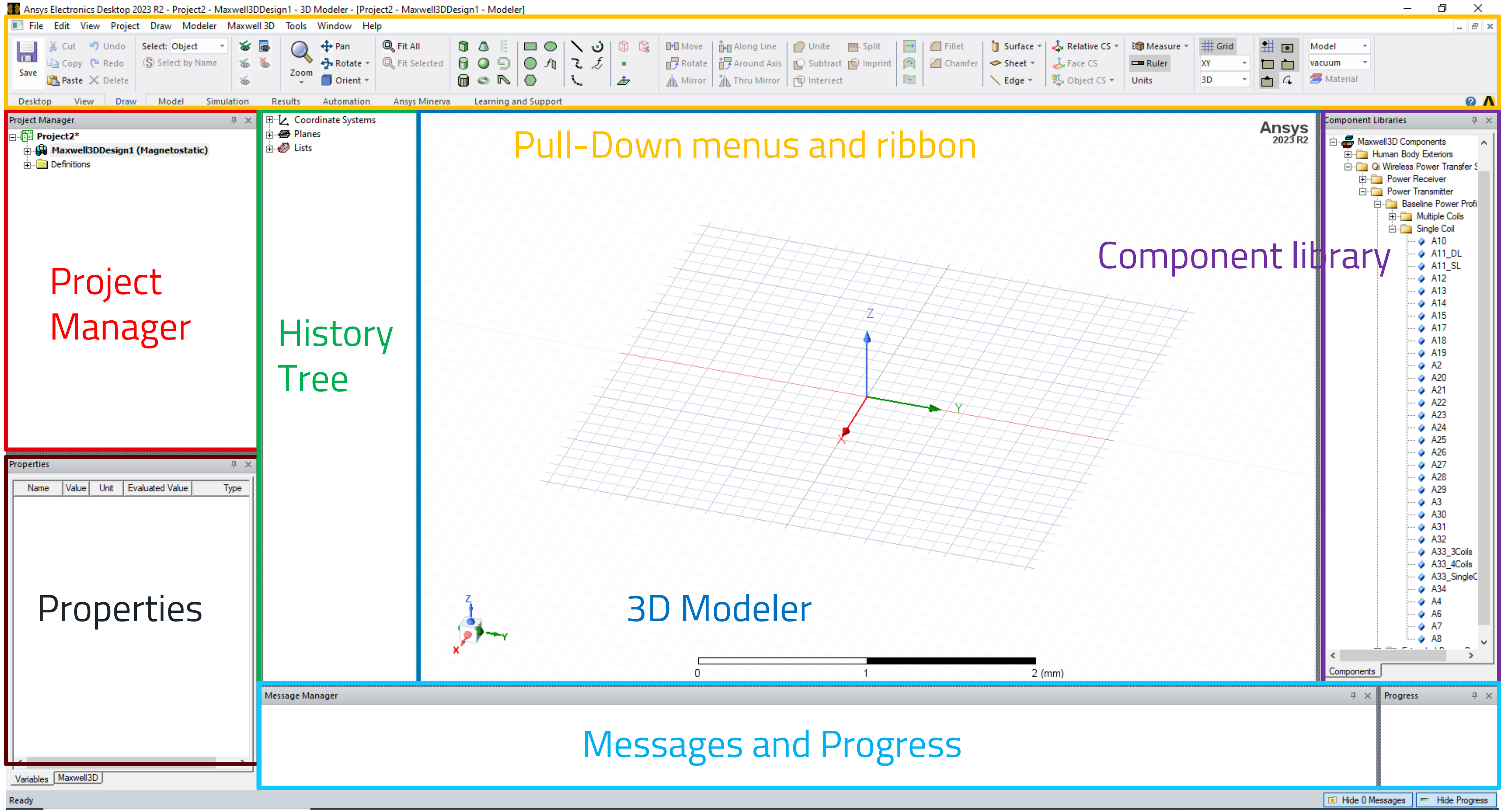
# INTRODUCTION

Ansys ICEPAK

- is a computational fluid dynamics (CFD) software for predicting and managing thermal performance in electronic devices
- simulates airflow, temperature and heat transfer for electronic devices
- supports conjugate heat transfer analysis, modeling conduction, convection and radiation in steady-state and transient modes
- is closely integrated with the AEDT
- is used by WE to simulate the wireless power coils at a certain AC current flowing through the respective coil

# ANSYS MAXWELL 3D SIMULATION

Ansys Electronic Desktop: Graphical User Interface



# WPC QI A10 DESIGN

- A10 design due to the Qi 1.3 Standard :

Inductance:
L = 24 µH ± 10 %

Ferrite Material: Mn-Zn or Ni-Zn with at least 0.5 mm thickness

Shield should extend at least 2 mm beyond the outer diameter of the primary coil

## Primary Coil

The Primary Coil is of the wire-wound type, and consists of no. 17 AWG (1.15 mm diameter) type 2 litz wire having 105 strands of No. 40 AWG (0.08 mm diameter), or equivalent. As shown in Figure 15, the Primary Coil has a circular shape and consists of multiple layers. All layers are stacked with the same polarity. Table 13 lists the dimensions of the Primary Coil.

**Figure 15. Primary Coil of Power Transmitter design A10**

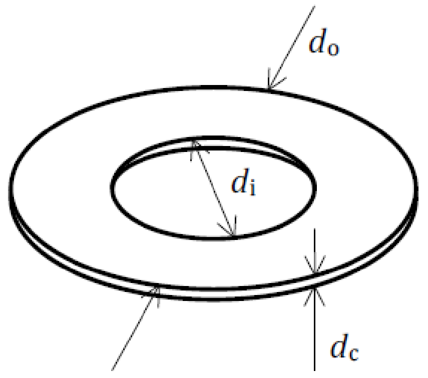


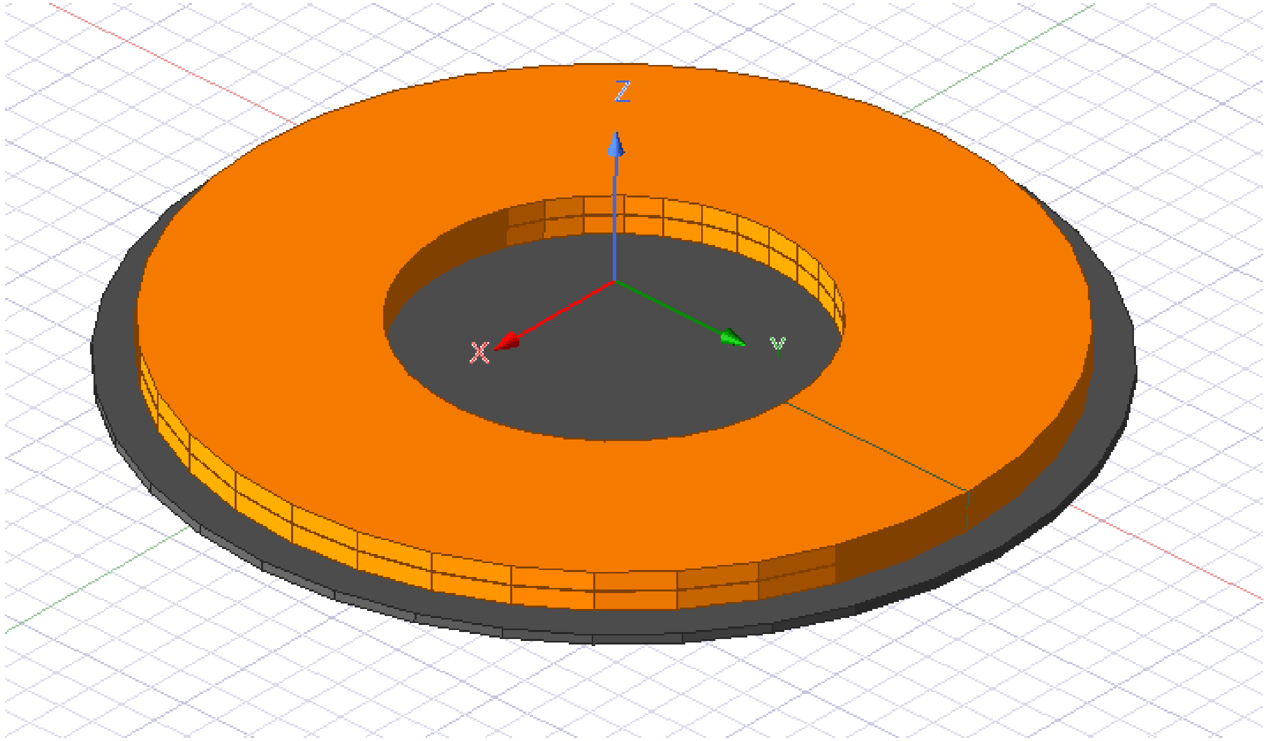

**Table 13: Primary Coil parameters of Power Transmitter design A10**

| Parameter | Symbol | Value |
|---|---|---|
| Outer diameter | $d_o$ | $43^{\pm 0.5}$ mm |
| Inner diameter | $d_i$ | $20.5^{\pm 0.5}$ mm |
| Thickness | $d_c$ | $2.1^{+0.5}$ mm |
| Number of turns per layer | $N$ | 10 |
| Number of layers | – | 2 |

# SIMULATION WORKFLOW: 3D MODEL

Make your 3D model in Ansys or choose a Qi design from the component library (doubleclick on the QI-Design -> it will be loaded into the modeler screen)

Example: Qi A10 Design from the component library (corresponding WE coil 760308101410):



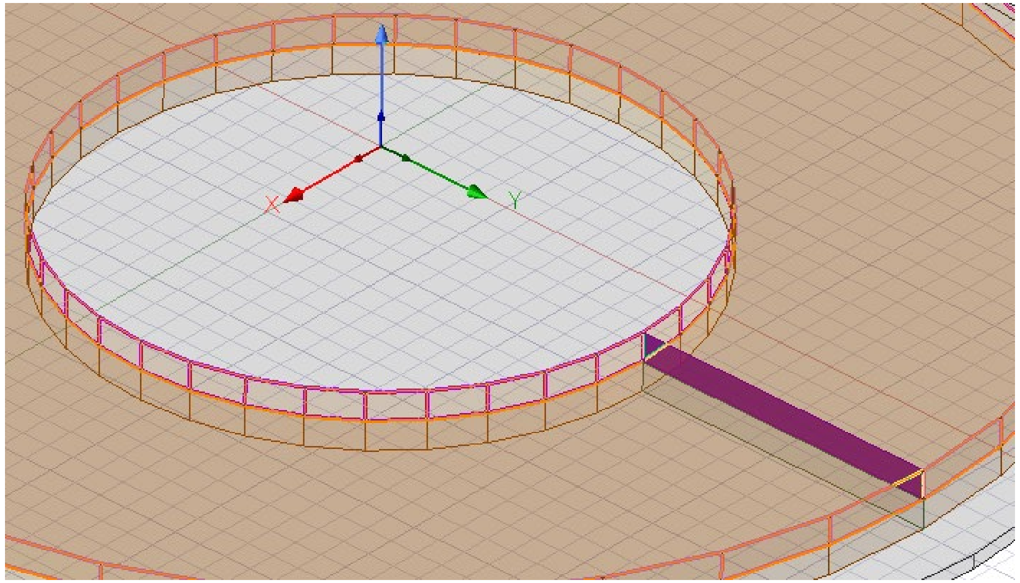

Consists of:

Grey cylinder (material: ferrite)
2 discs (1 for each layer, material copper)
two cross sections, 1 for each layer of the coil)

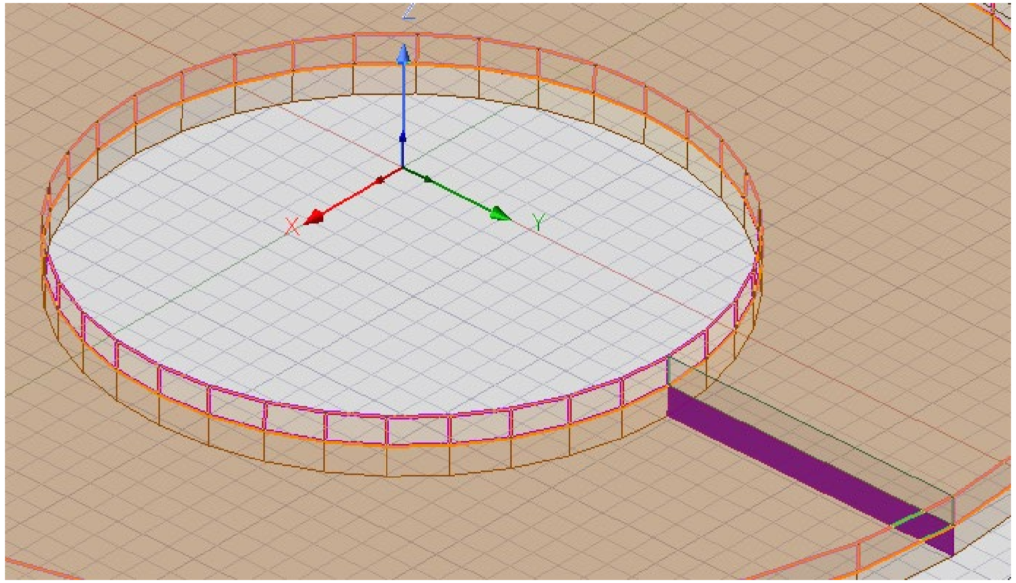
# 3D SIMULATION WORKFLOW: CROSS SECTION AND EXCITATION

If you made your own model, the cross sections have to be added and the materials have to be defined in the properties window (is already done if you choose the model from the library):

- Cross section layer 1:

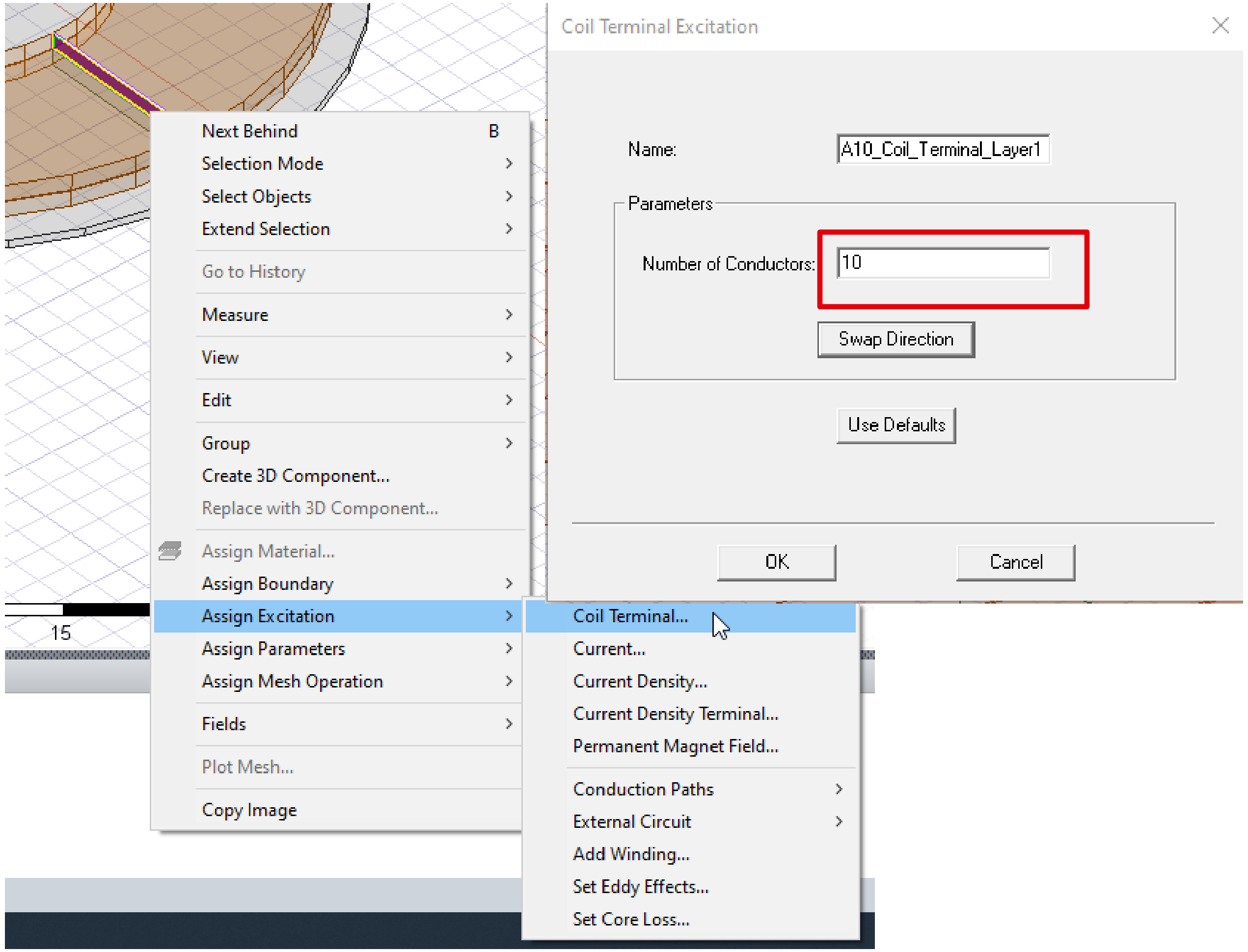


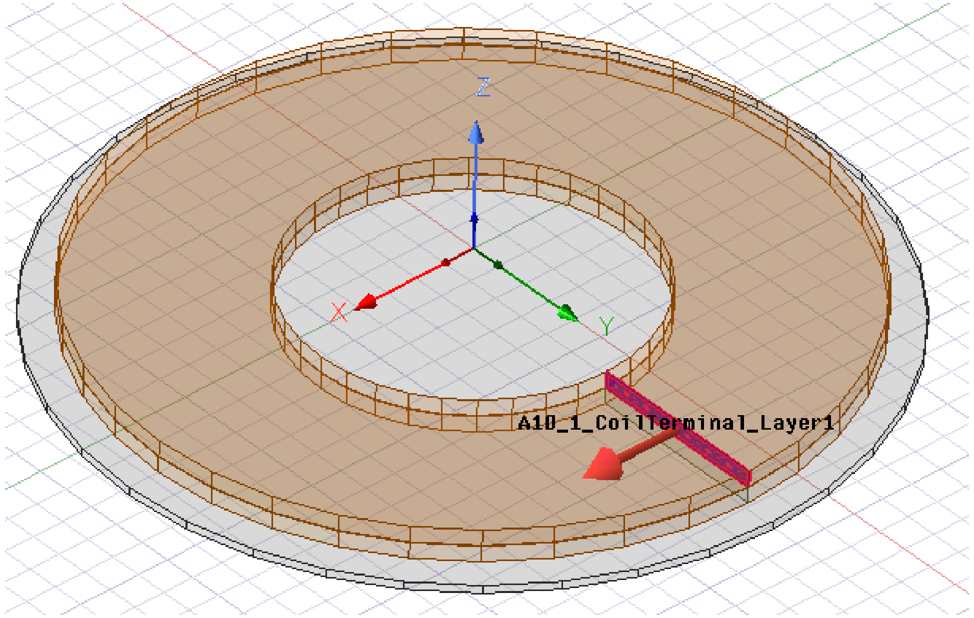

Cross section layer 2:



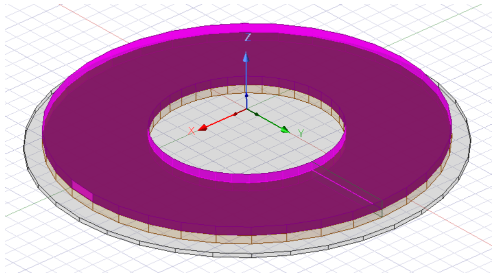
# 3D SIMULATION WORKFLOW: CROSS SECTION AND EXCITATION

Define Excitation on both cross sections: Right click on the cross section -> Assign Excitation -> Coil Terminal 1/2

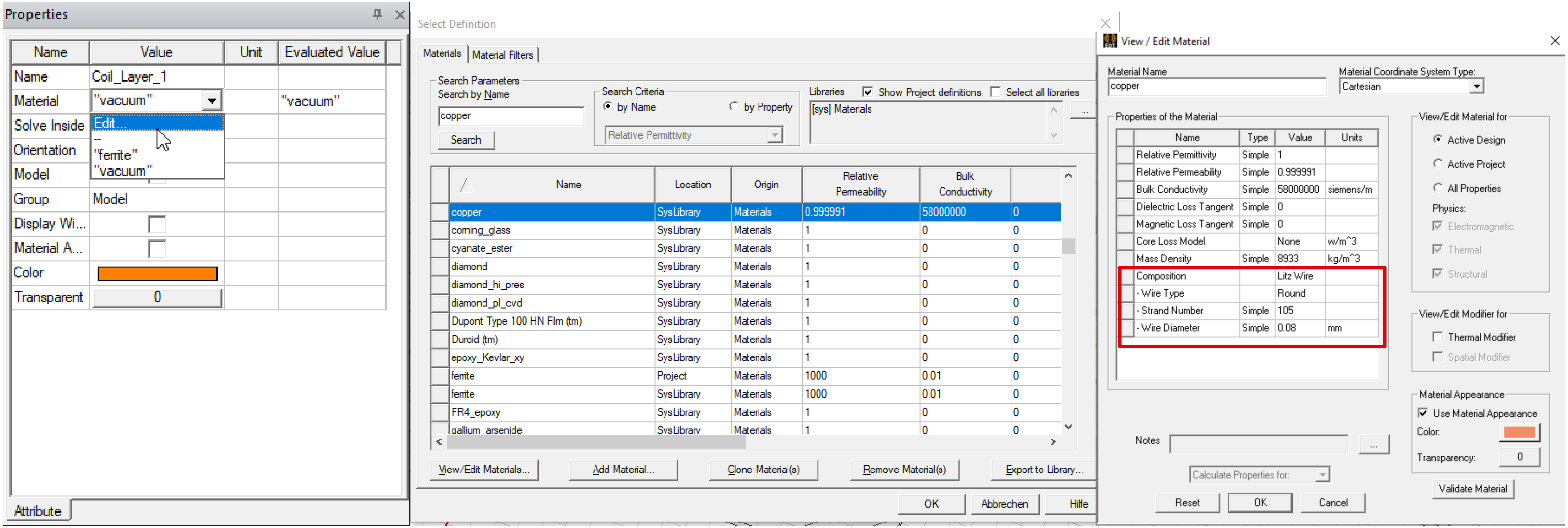


# 3D SIMULATION WORKFLOW: MATERIAL PROPERTIES (COPPER)

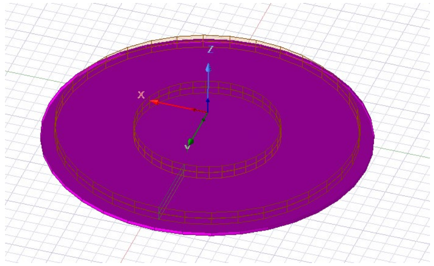


Define materials (and colors) at the properties window, first select the part, where the material has to be defined by clicking on it, go to Material -> Edit -> choose needed material (e.g. copper) and go to View/Edit materials:

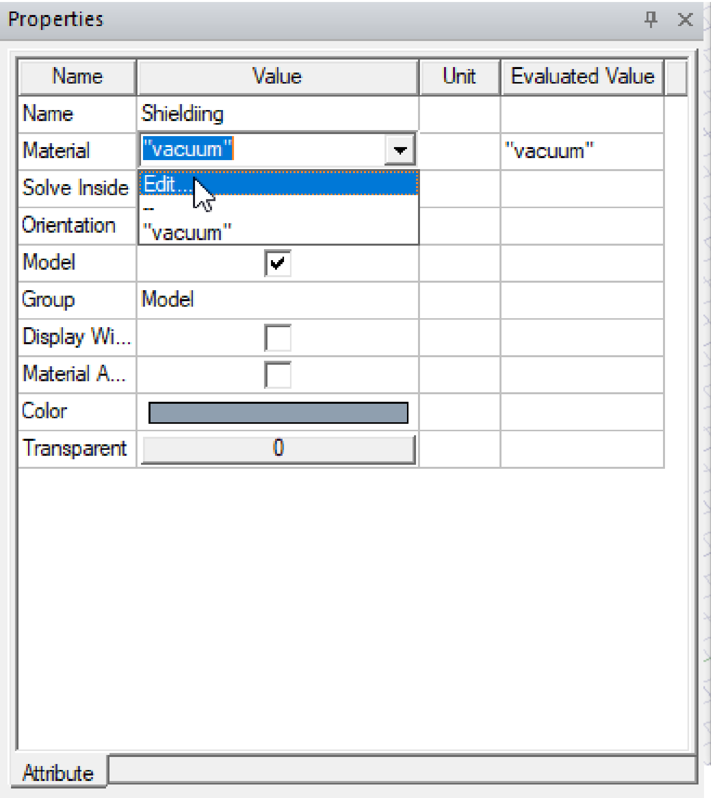


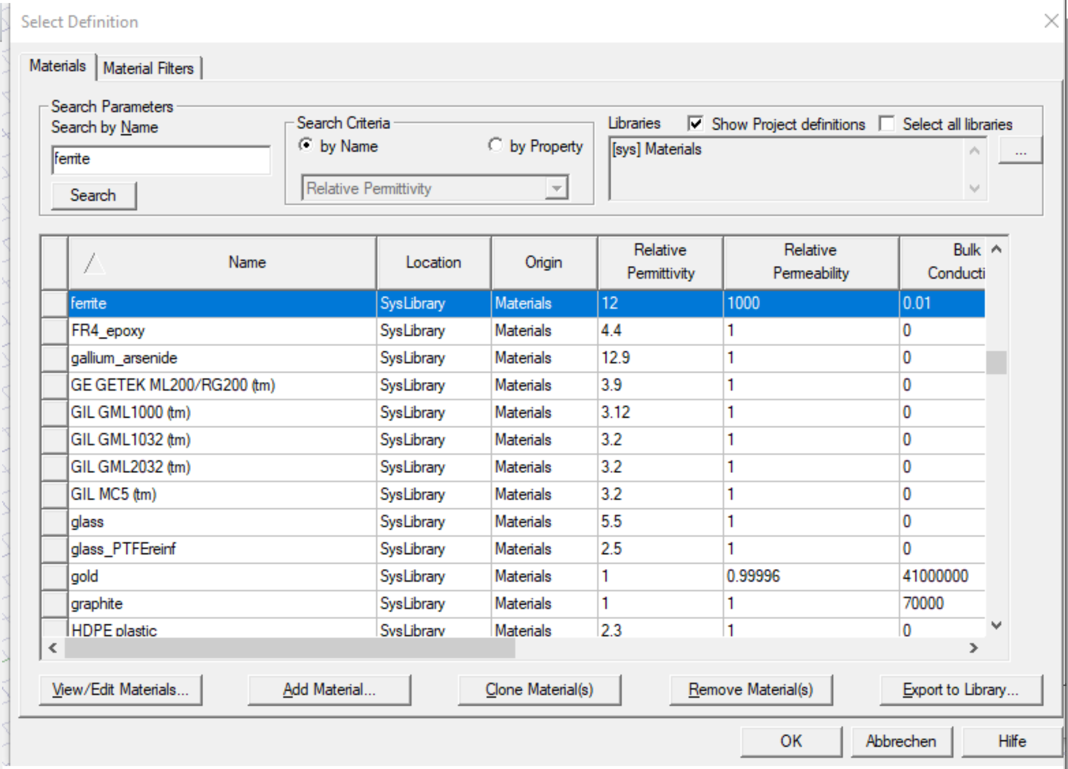

Define Composition: Litz wire, wire type, strand number and wire diameter

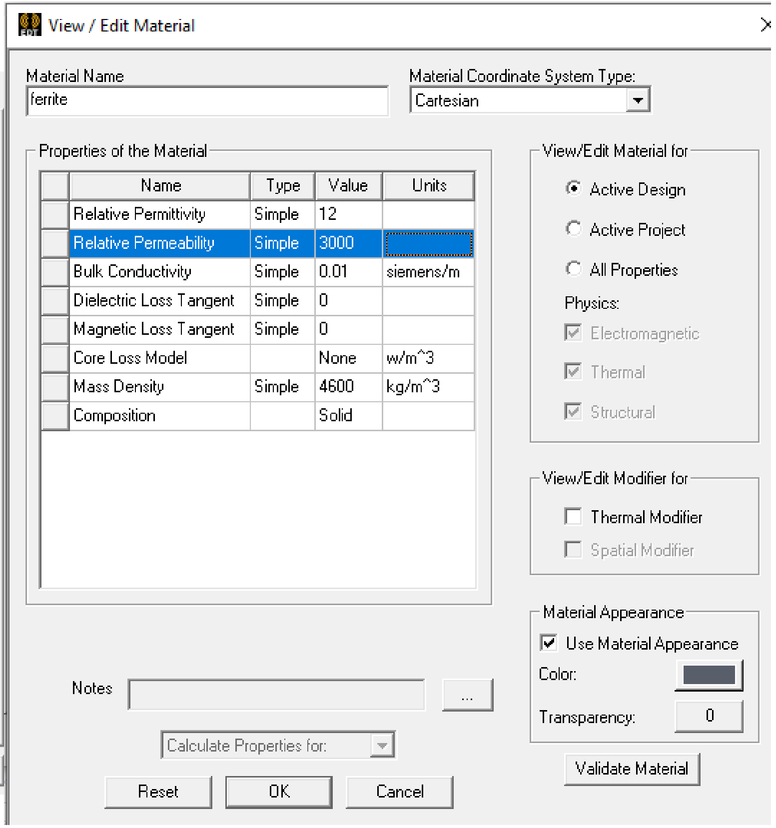
# 3D SIMULATION WORKFLOW: MATERIAL PROPERTIES (FERRITE)



Go to Material -> Edit -> choose needed material (e.g. ferrite) and go to View/Edit materials:



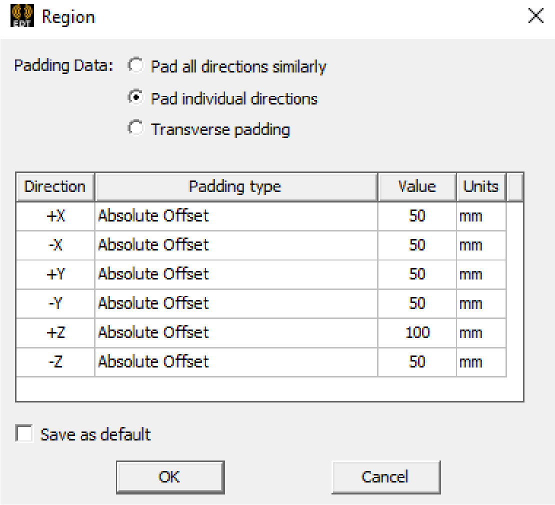




Relative permeability: ~ 3000

# SIMULATION WORKFLOW: REGION SIZING

- Rules of thumb recommended by Ansys:
- Top (against gravity): At least 2L
- Sides and below: at least L
- Where L is the largest dimension of the unit in 3 directions

In this case: L ~ 50 mm

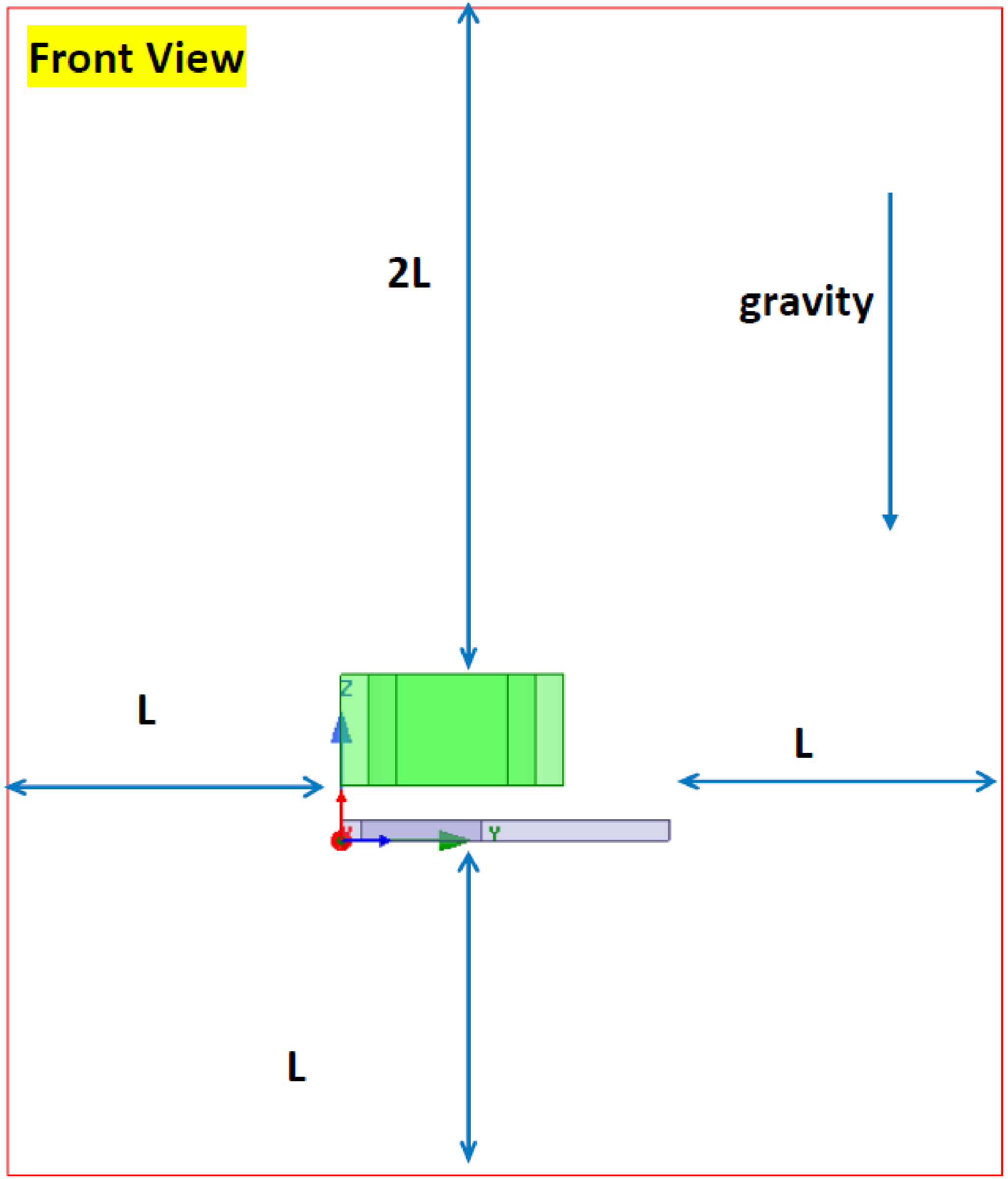

Region

Padding Data: ○ Pad all directions similarly
◉ Pad individual directions
○ Transverse padding

| Direction | Padding type | Value | Units |
|---|---|---|---|
| +X | Absolute Offset | 50 | mm |
| -X | Absolute Offset | 50 | mm |
| +Y | Absolute Offset | 50 | mm |
| -Y | Absolute Offset | 50 | mm |
| +Z | Absolute Offset | 100 | mm |
| -Z | Absolute Offset | 50 | mm |

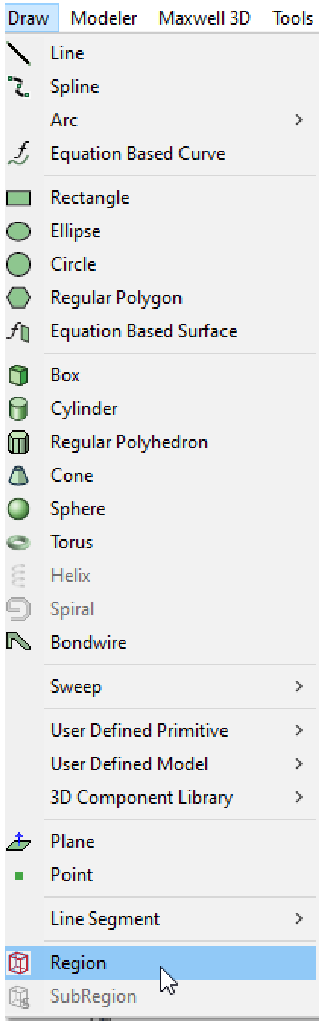
☐ Save as default

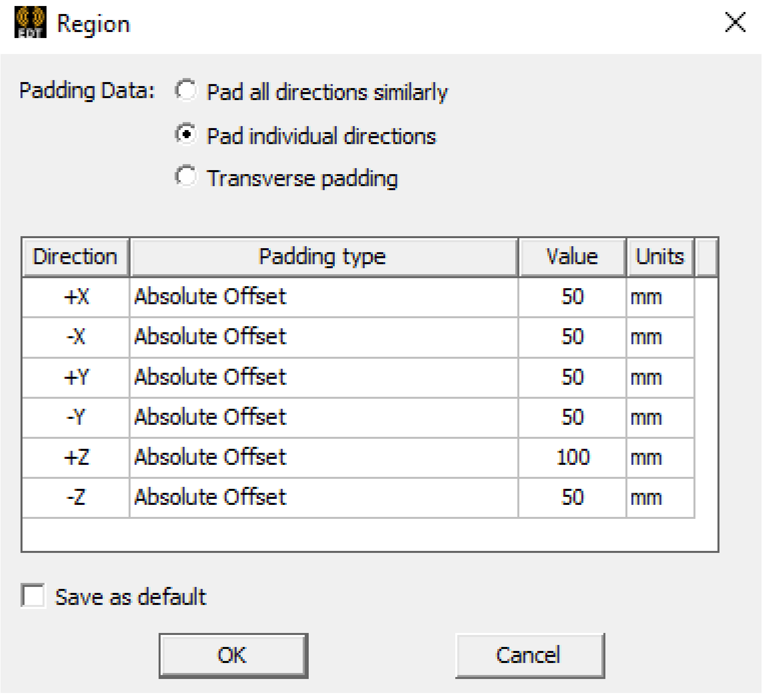
OK Cancel



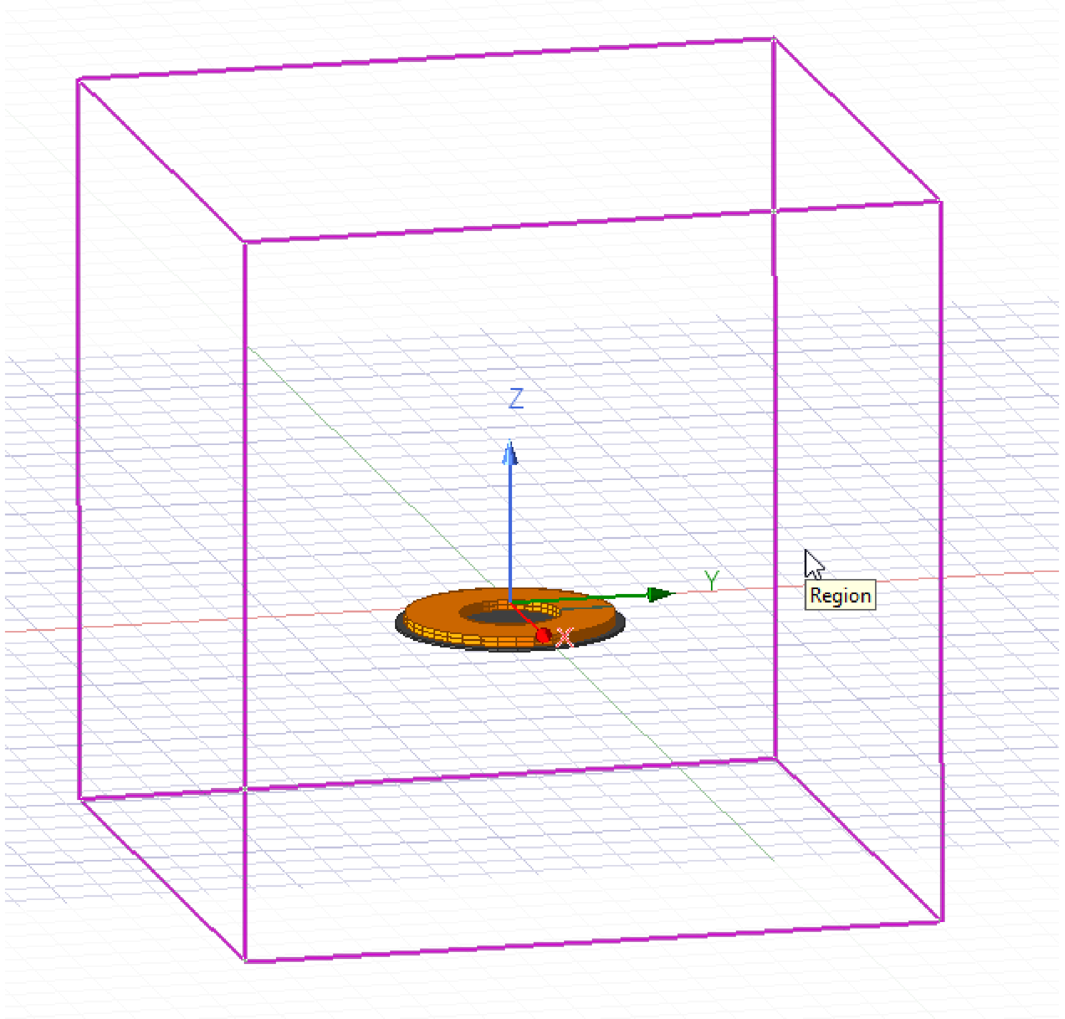
# 3D SIMULATION WORKFLOW: DEFINE THE REGION

The space, in which the electromagnetic field has to be calculated, is called the region, it is defined as default as vacuum



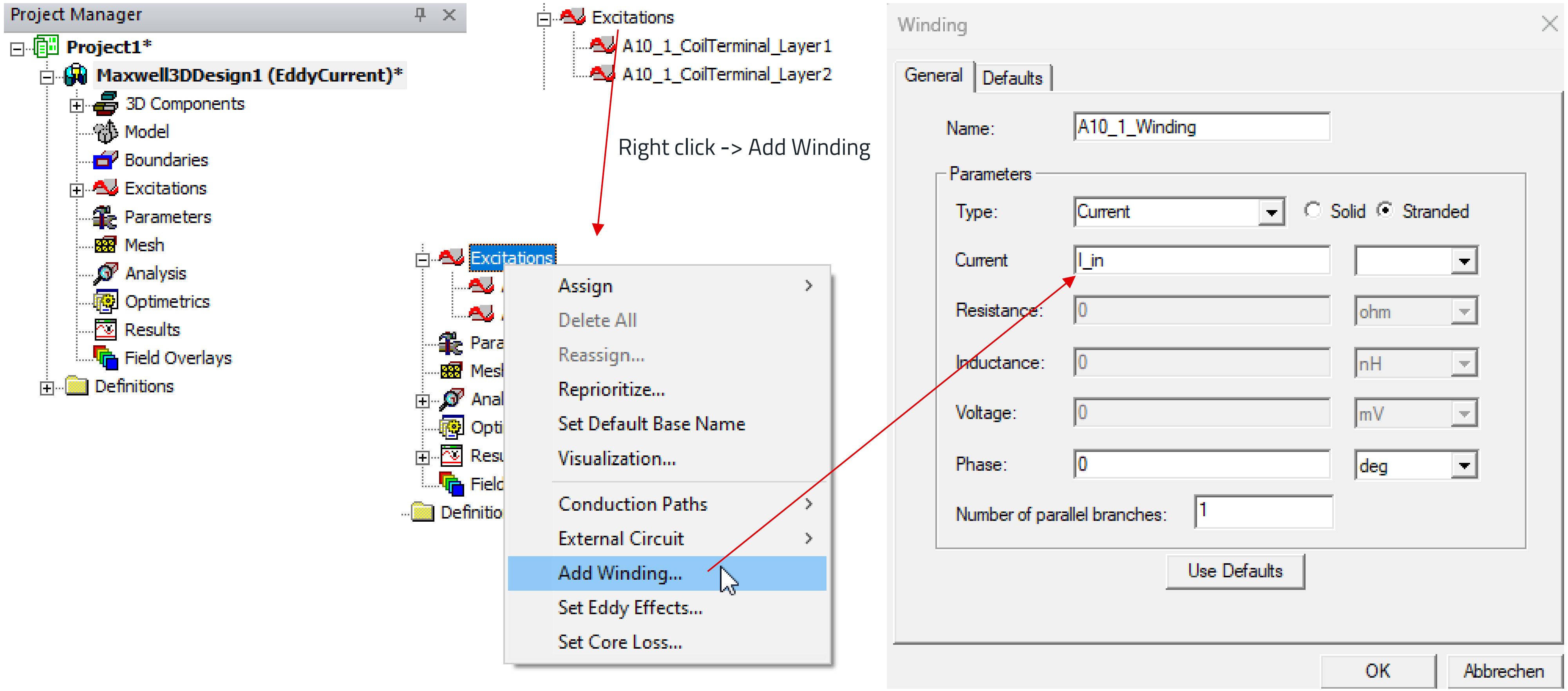




| Direction | Padding type | Value | Units |
|---|---|---|---|
| +X | Absolute Offset | 50 | mm |
| -X | Absolute Offset | 50 | mm |
| +Y | Absolute Offset | 50 | mm |
| -Y | Absolute Offset | 50 | mm |
| +Z | Absolute Offset | 100 | mm |
| -Z | Absolute Offset | 50 | mm |



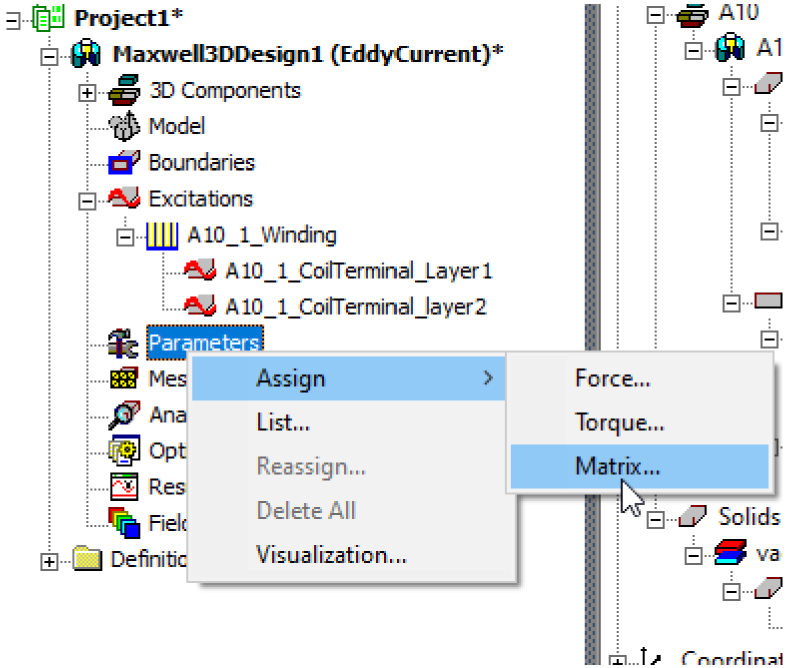
# 3D SIMULATION WORKFLOW: SETTINGS

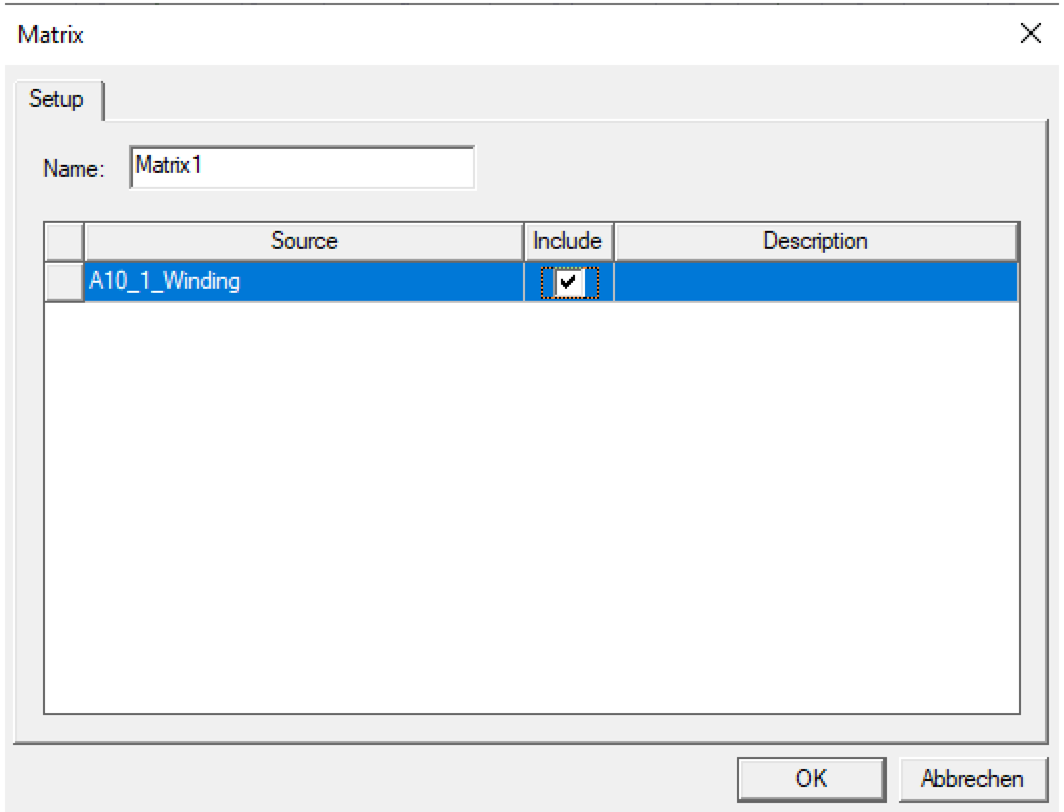
- Some settings have to be made at the Project Manager: Excitations, Parameters, Analysis and Results
- Excitations: Define the variable I_in for the AC amplitude variable



# 3D SIMULATION WORKFLOW: SETTINGS

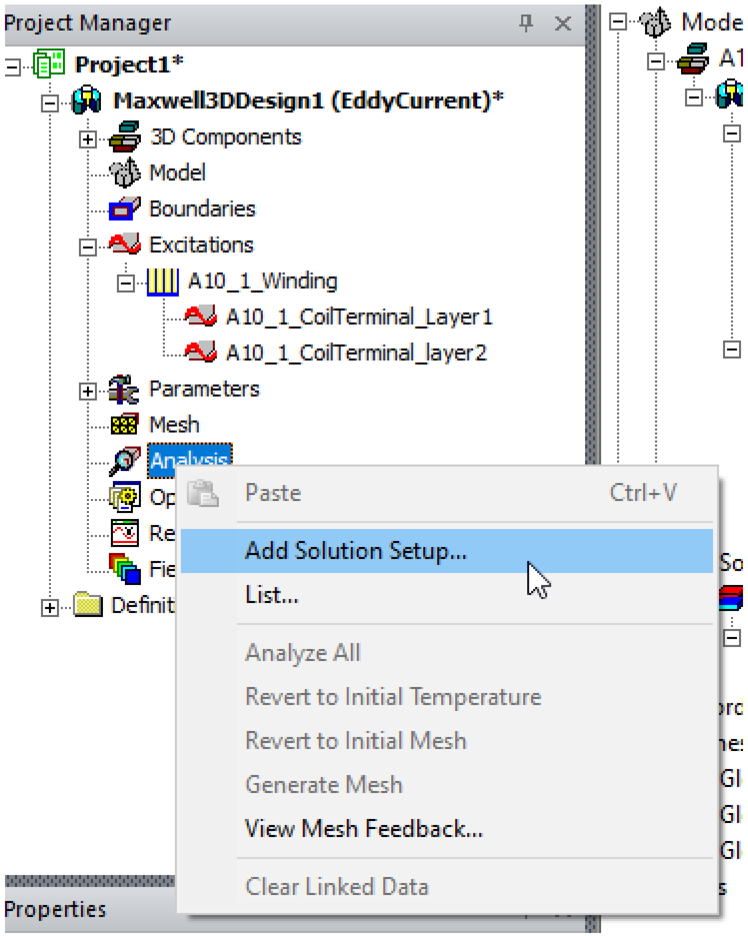
- Parameters:

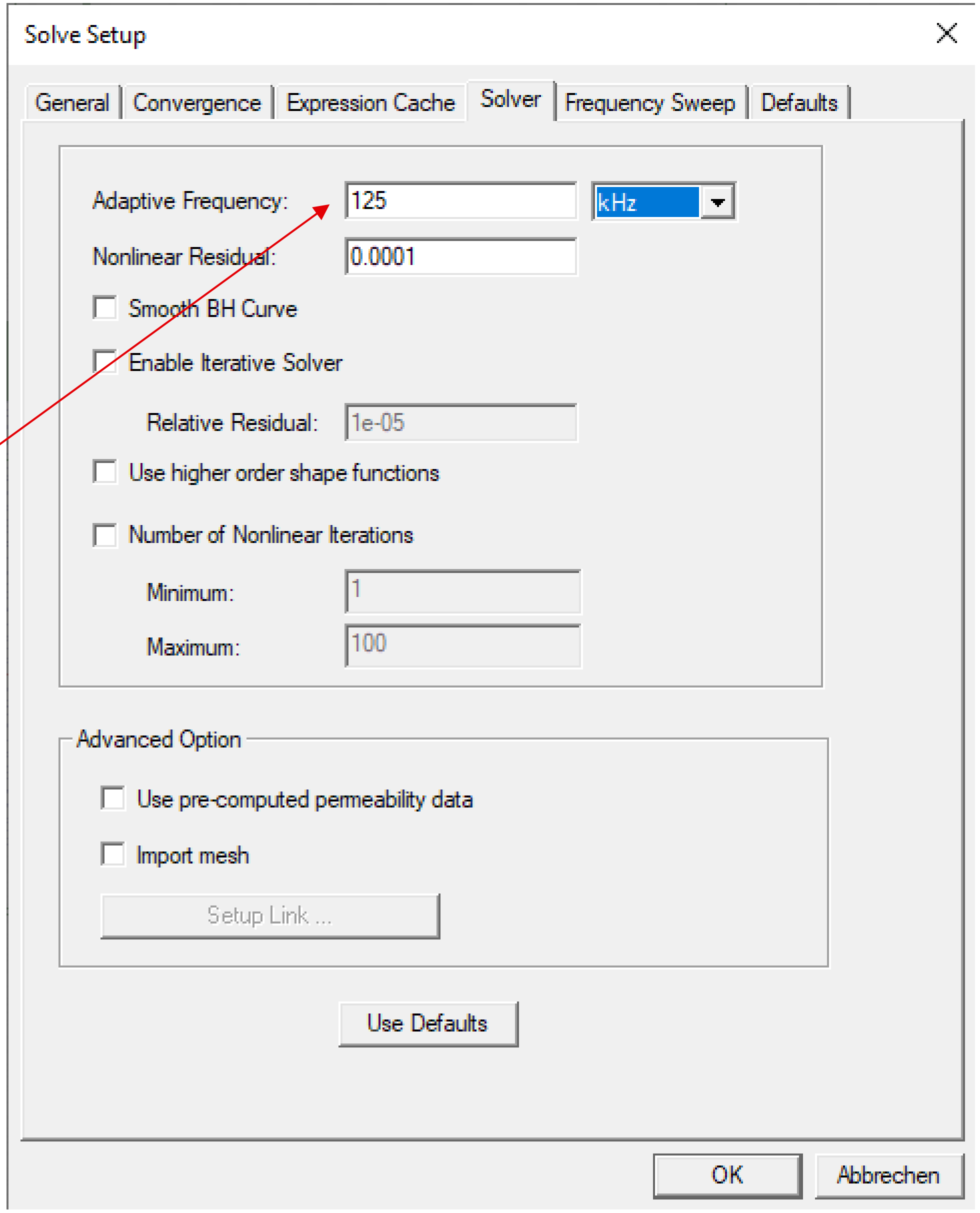


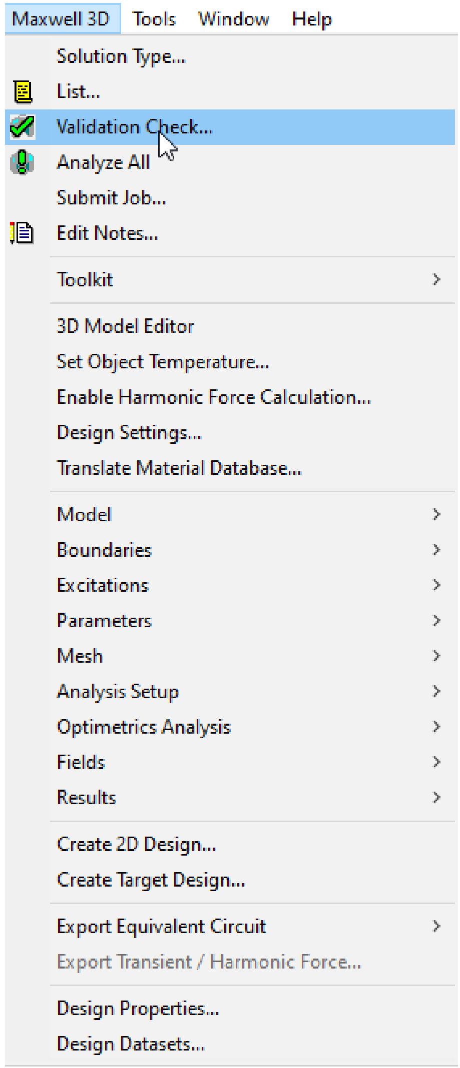
# 3D SIMULATION WORKFLOW: SETTINGS

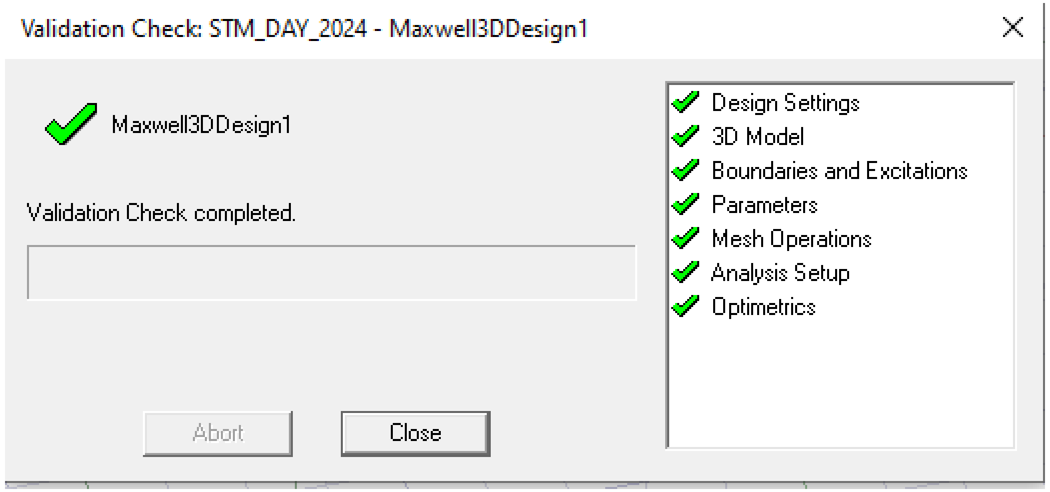
- Analysis:

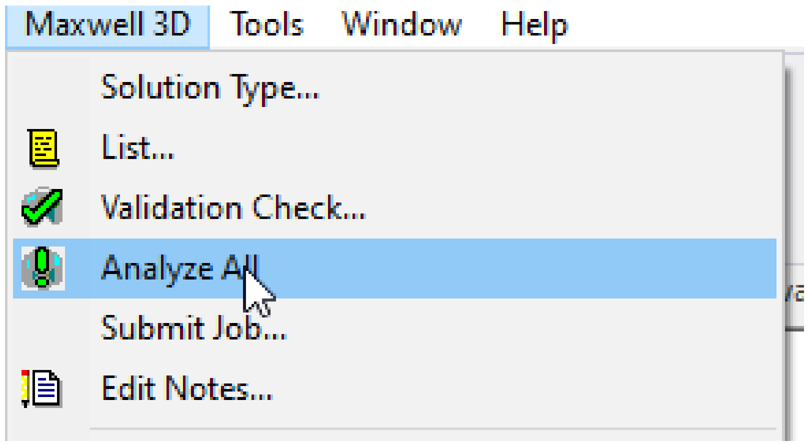


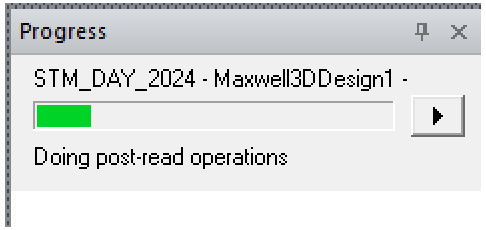



# 3D SIMULATION WORKFLOW: VALIDATION AND ANALYZATION

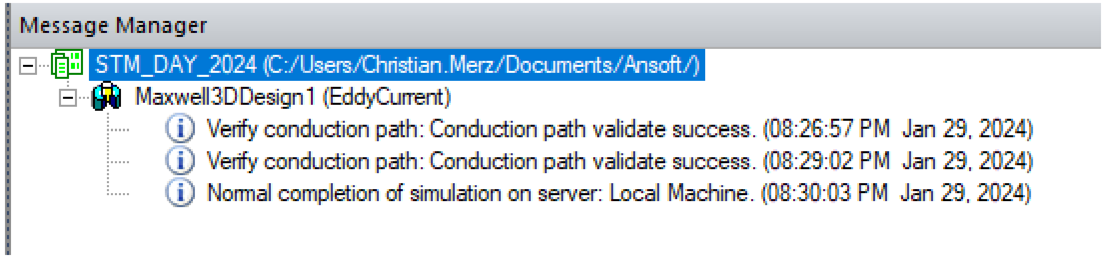


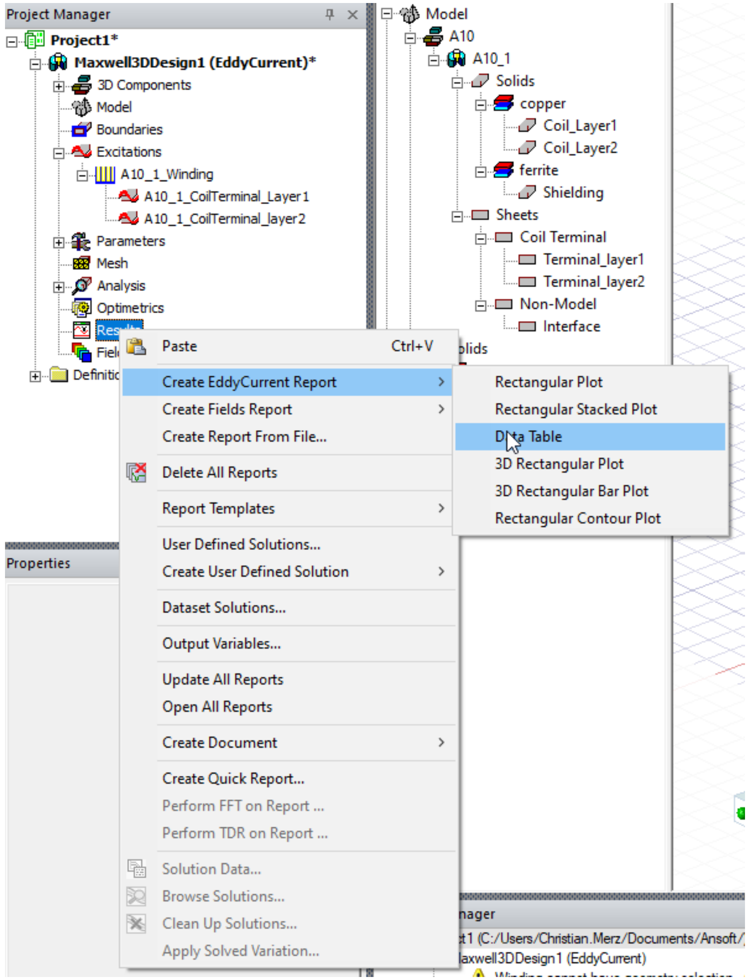



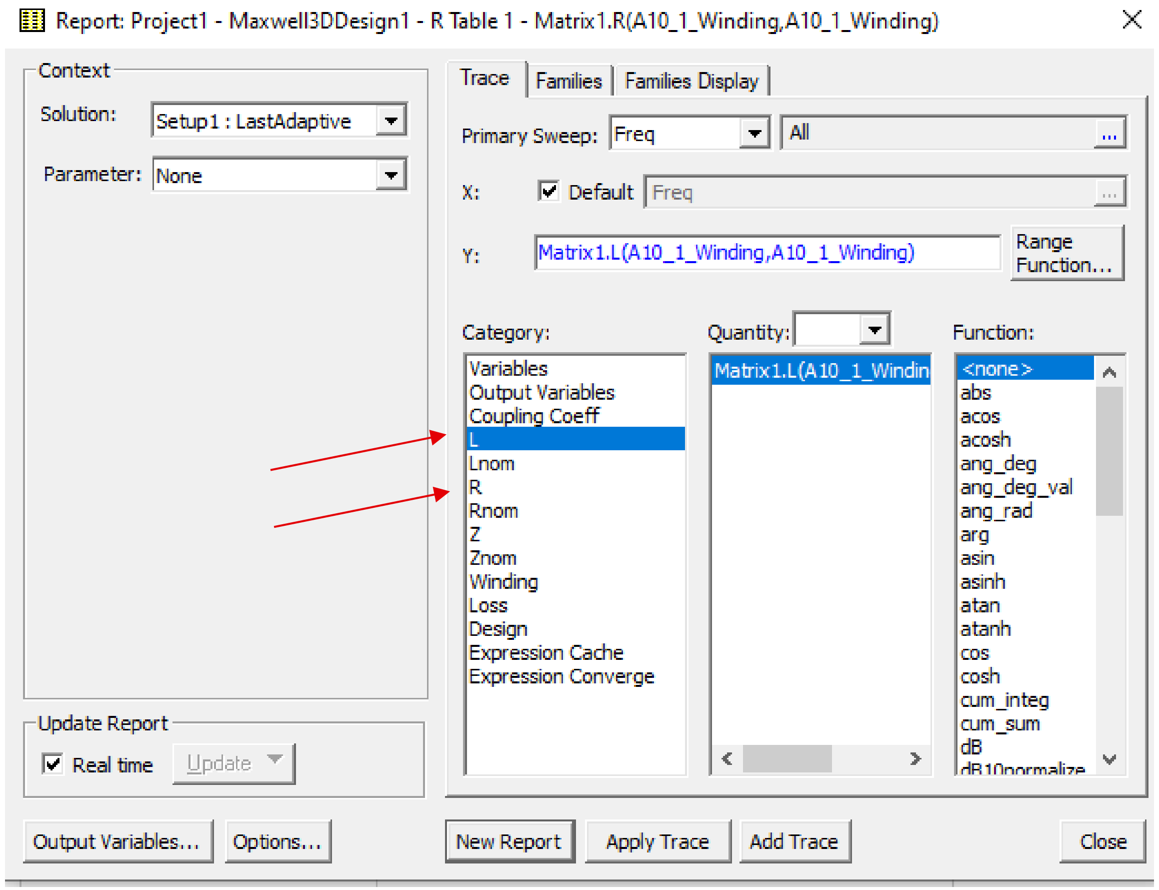

Simulation time about 3 mins



# 3D SIMULATION WORKFLOW: SETTINGS

- Results:



# 3D SIMULATION: INDUCTANCE AND R,AC

Inductance

| Freq [kHz] | Matrix1.L(A10_1_Winding,A10_1_Winding) [uH]<br>Setup1 : LastAdaptive |
|---|---|
| 125.000000 | 23.752755 |

A10 coil in the datasheet: 24 µH ± 10 % @ 125 kHz

**Deviation between simulated and nominal value: only 1%**

R,ac

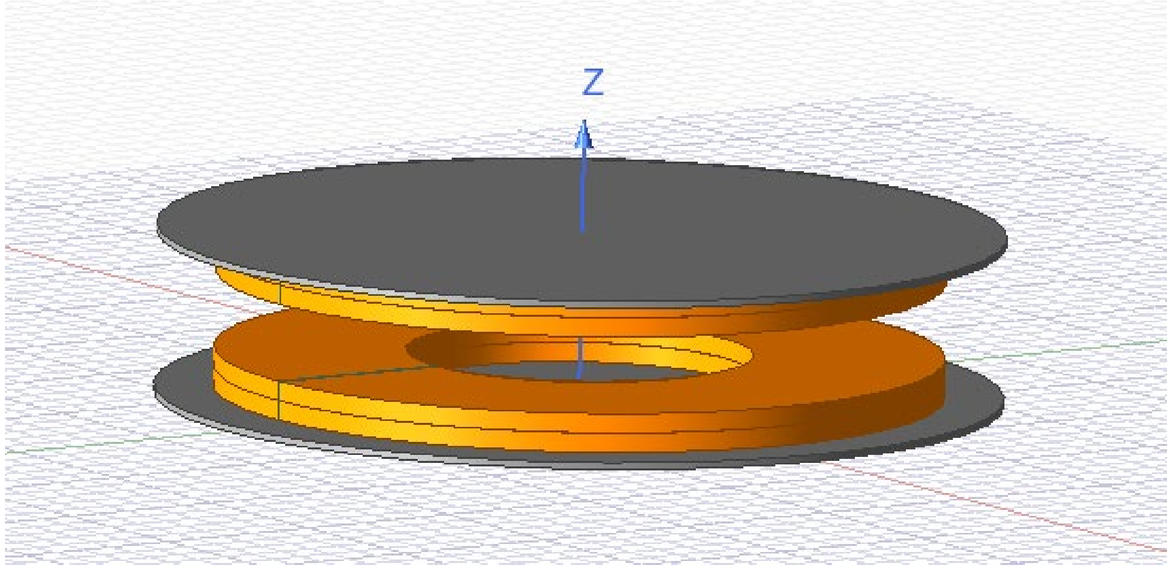
| | Freq [kHz] | Matrix1.R(Winding1,Winding1) [mOhm]<br>Setup1 : LastAdaptive |
|---|---|---|
| 1 | 125.000000 | 92.091459 |

Measured value with LCR-Meter @ 125 kHz: 96 mΩ

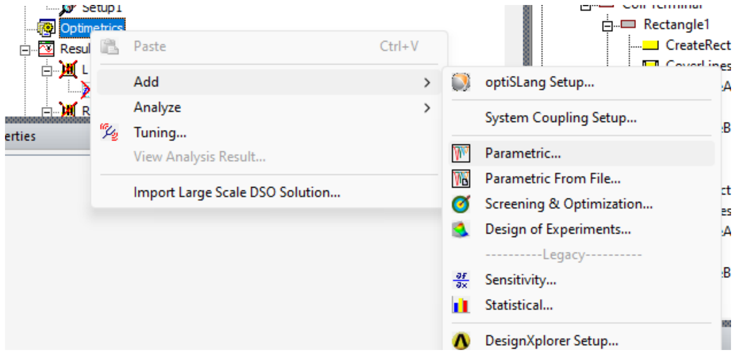
**Deviation between simulated and measured value: only 4%**

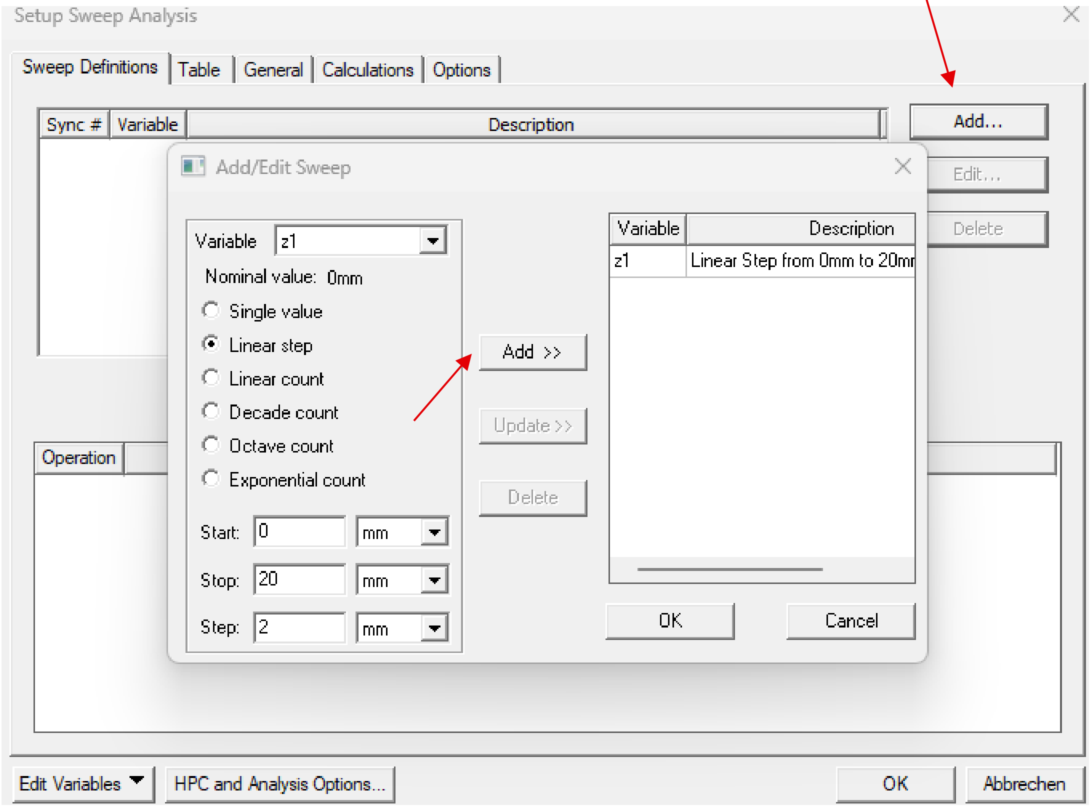
# SIMULATION: COUPLING BETWEEN TWO IDENTICAL COILS

The A10 Coil has been copied an flipped above the other coil, the coupling factor in dependence of the vertical misalignment has been simulated



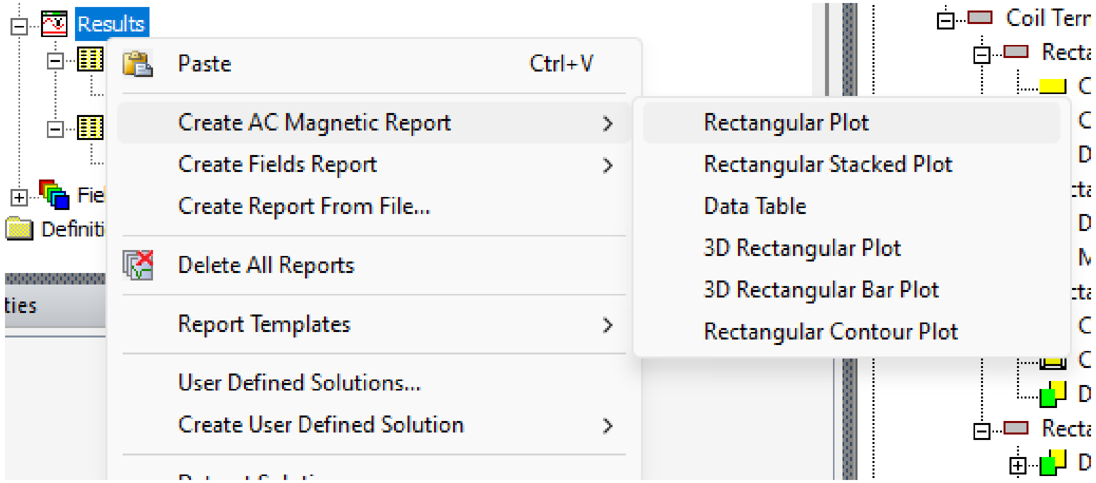

Vertical distance variable: z1

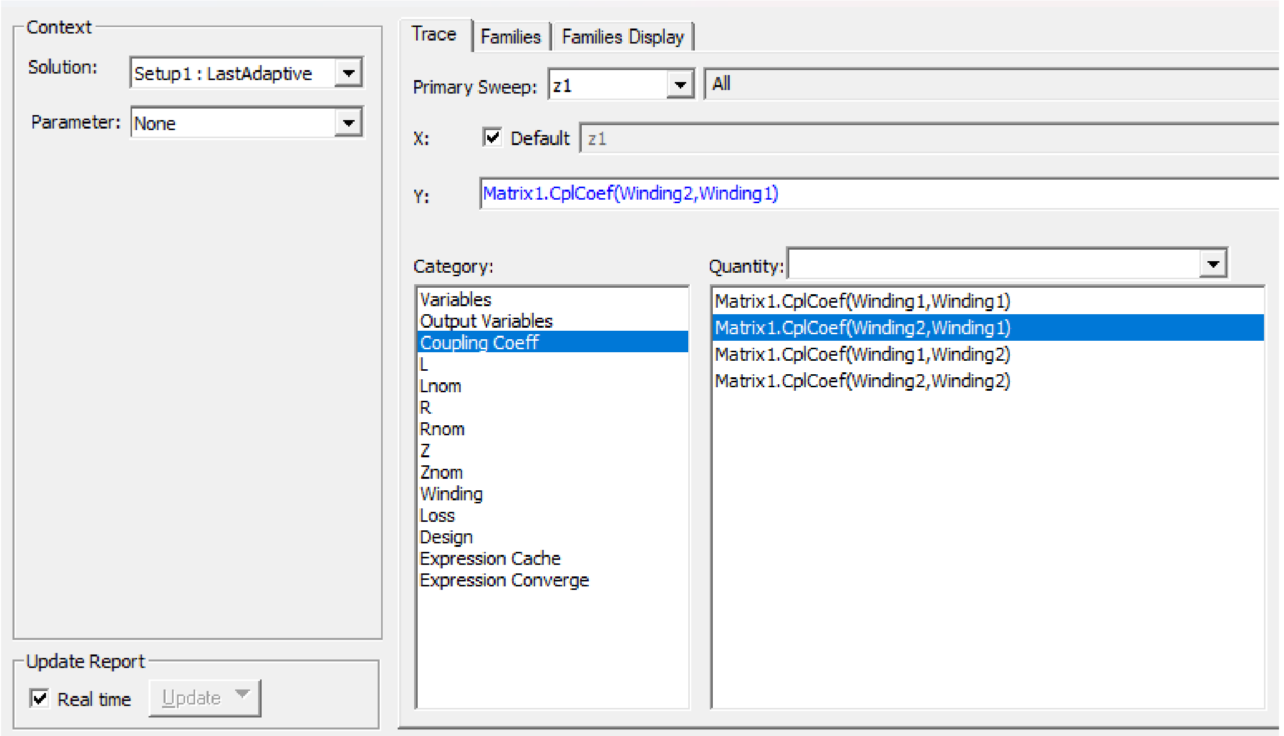


# SIMULATION: COUPLING BETWEEN TWO IDENTICAL COILS

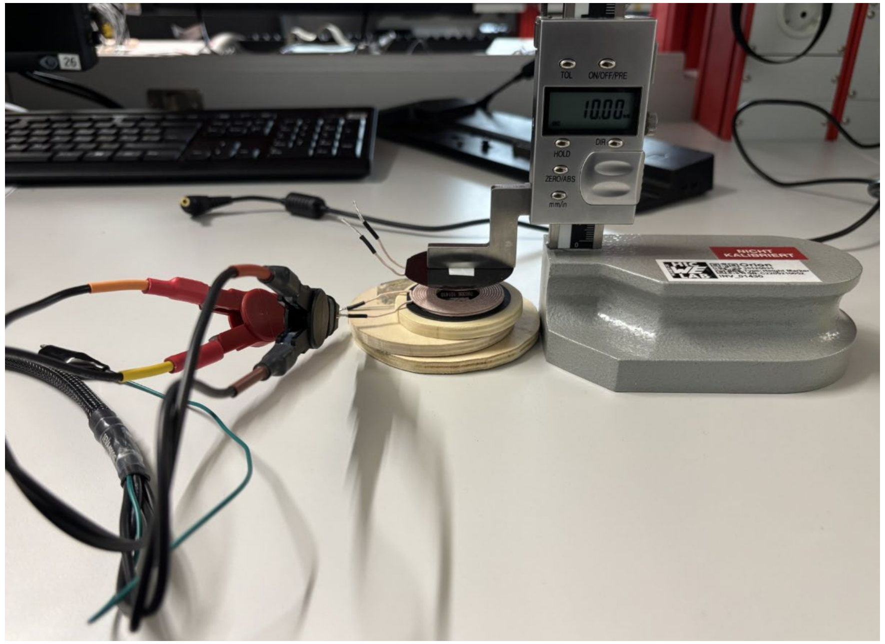
The A10 Coil has been copied an flipped above the other coil, the coupling factor in dependence of the vertical misalignment has been simulated

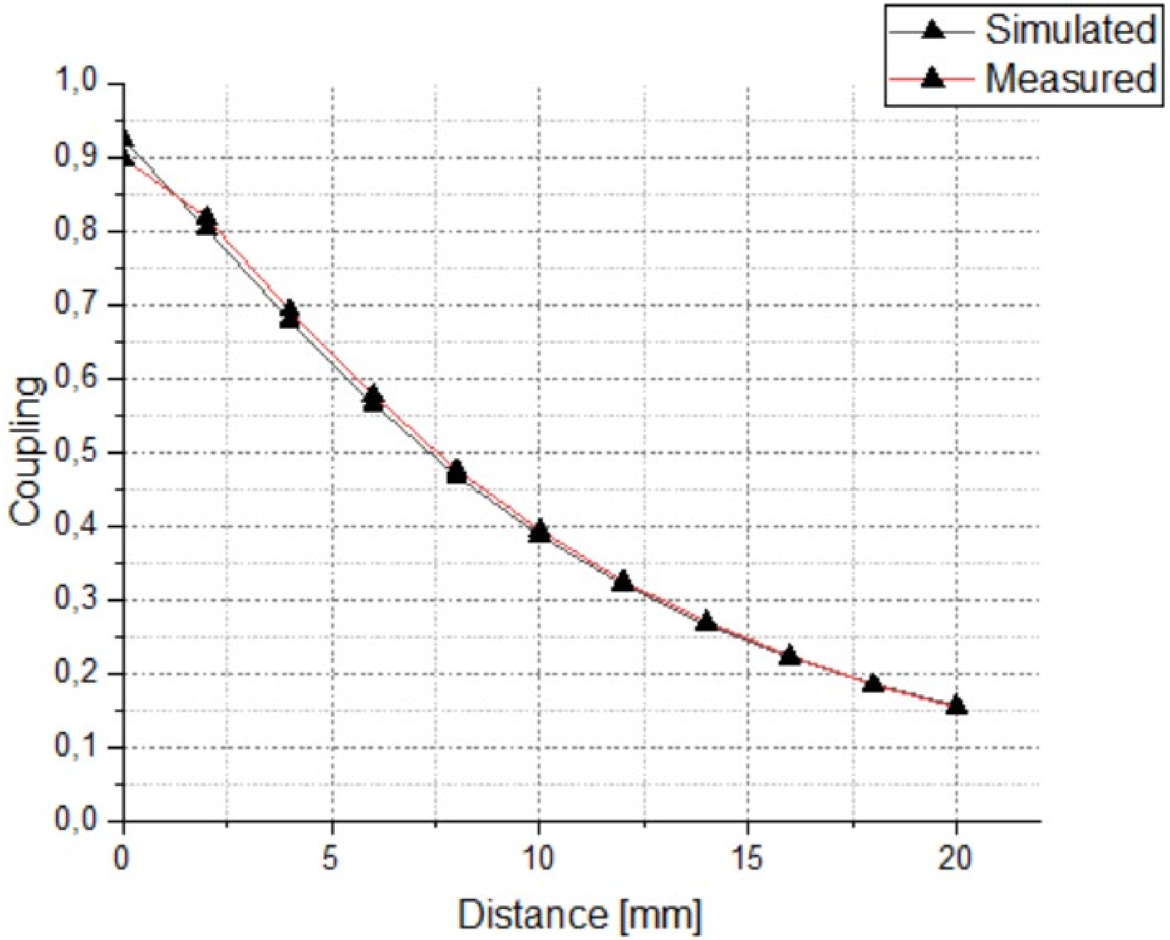


# SIMULATION: COUPLING BETWEEN TWO IDENTICAL COILS

The A10 Coil has been copied an flipped above the other coil, the coupling factor in dependence of the vertical misalignment has been simulated



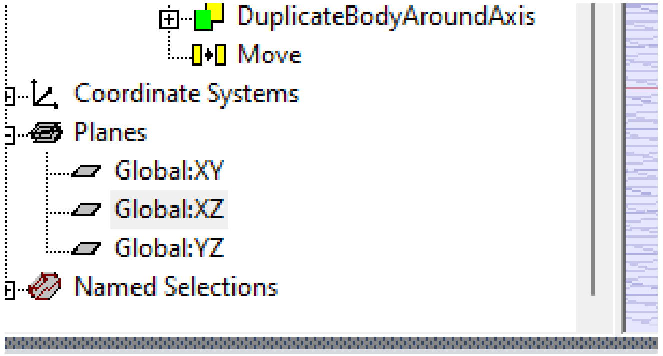

$$k = \sqrt{1 - \frac{L_1'}{L_1}}$$

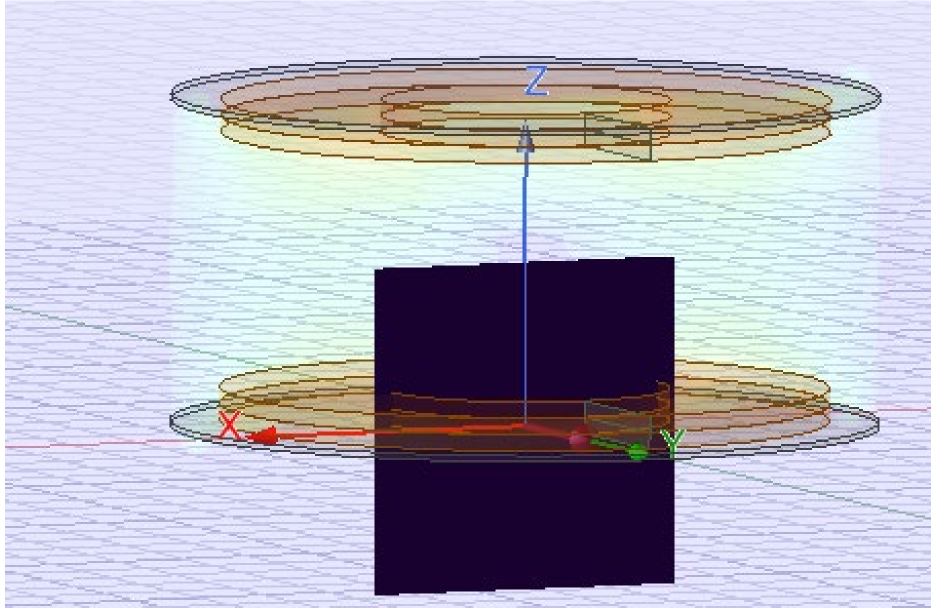
**Variation of 0.5%-3 %**

# SIMULATION: MAGNETIC FIELD BETWEEN TWO IDENTICAL COILS

Absolut Value of the magnetic field H [A/m] has been simulated at the distances between 0 – 20 mm

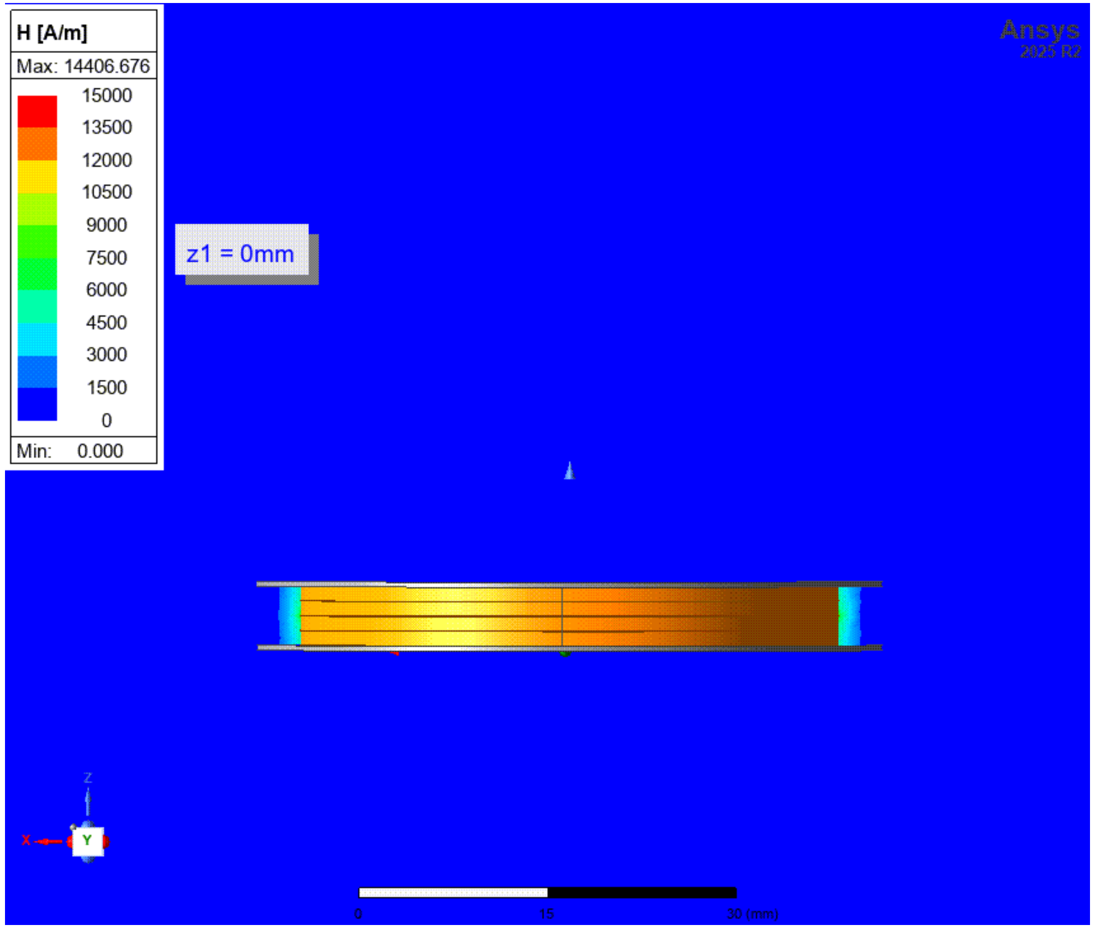
1. Select a geometry, in which the field distribution should be shown (e .g. XZ plane)



# SIMULATION: MAGNETIC FIELD BETWEEN TWO IDENTICAL COILS

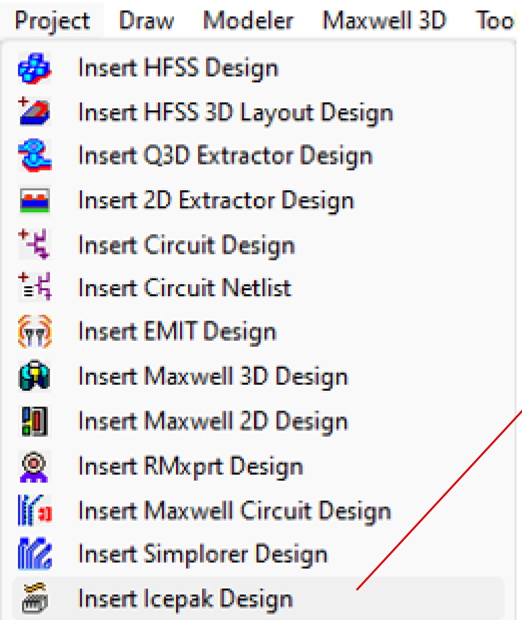
Absolut Value of the magnetic field H [A/m] has been simulated at the distances between 0 – 20 mm

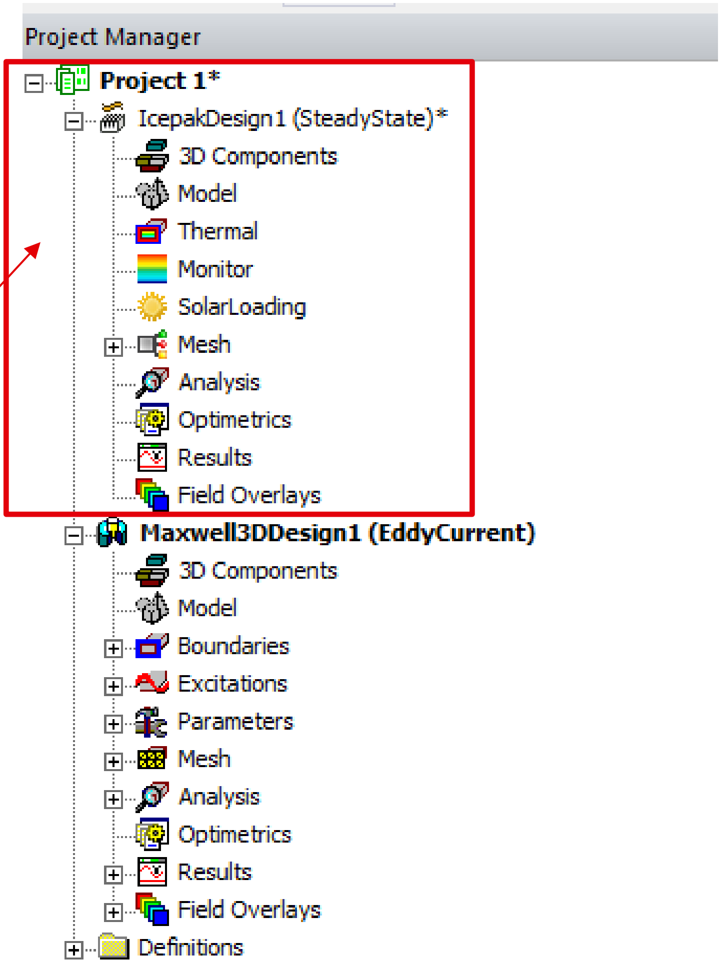


# ICEPAK MODULE

## Workflow of the ICEPAK Module

1. After the 3D design is done and simulated, the ICEPAK module has to be inserted
Projects -> Insert ICEPAK Design



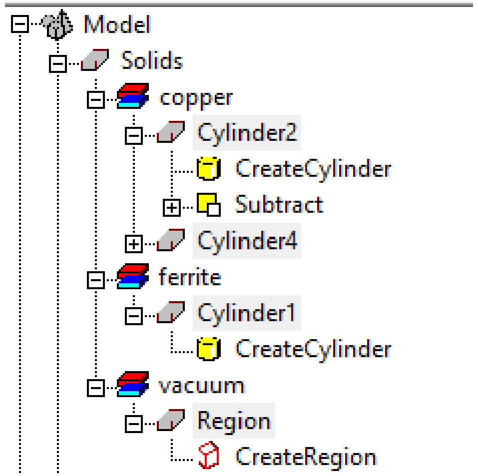



# ICEPAK MODULE

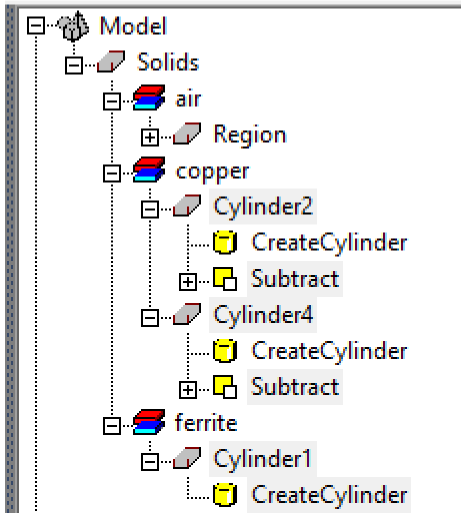
## Workflow of the ICEPAK Module

2. Copy the model of the 3D design into the ICEPAK history tree

### Maxwell 3D Design History Tree



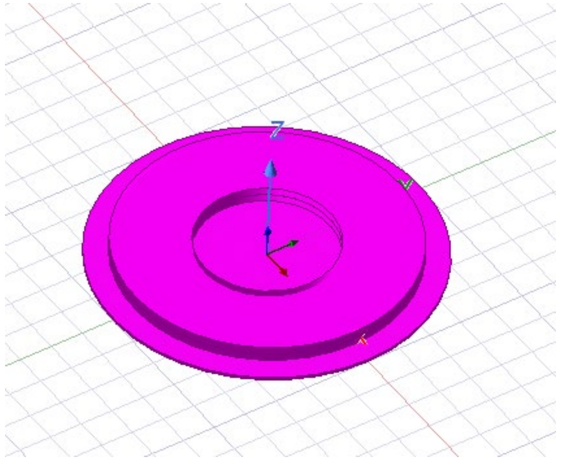

### ICEPAK History Tree

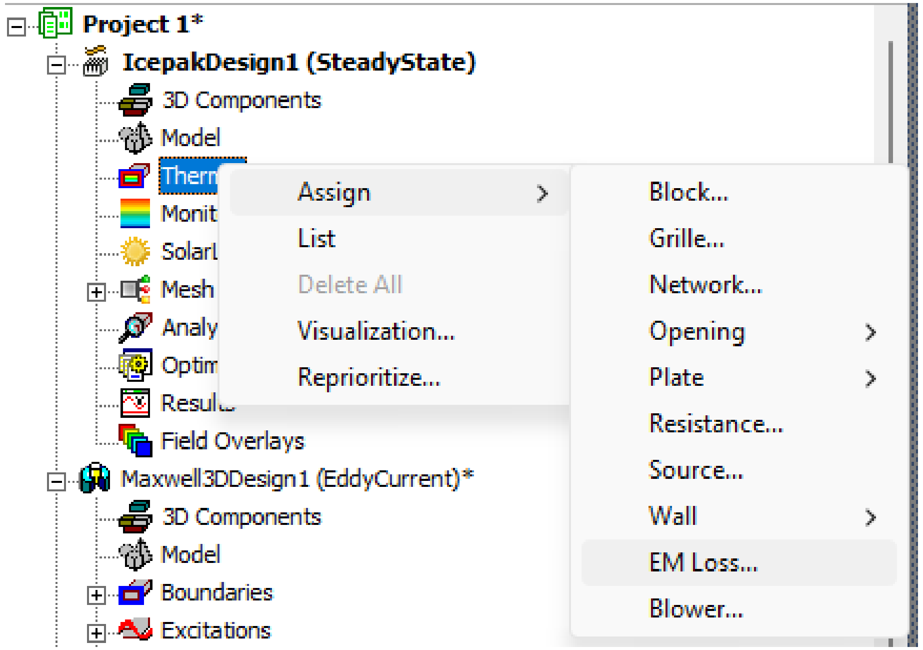


# ICEPAK MODULE

Workflow of the ICEPAK Module

3. Define EM Loss Settings -> Mark the 3d Model and go to Thermal -> Assign -> EM Loss

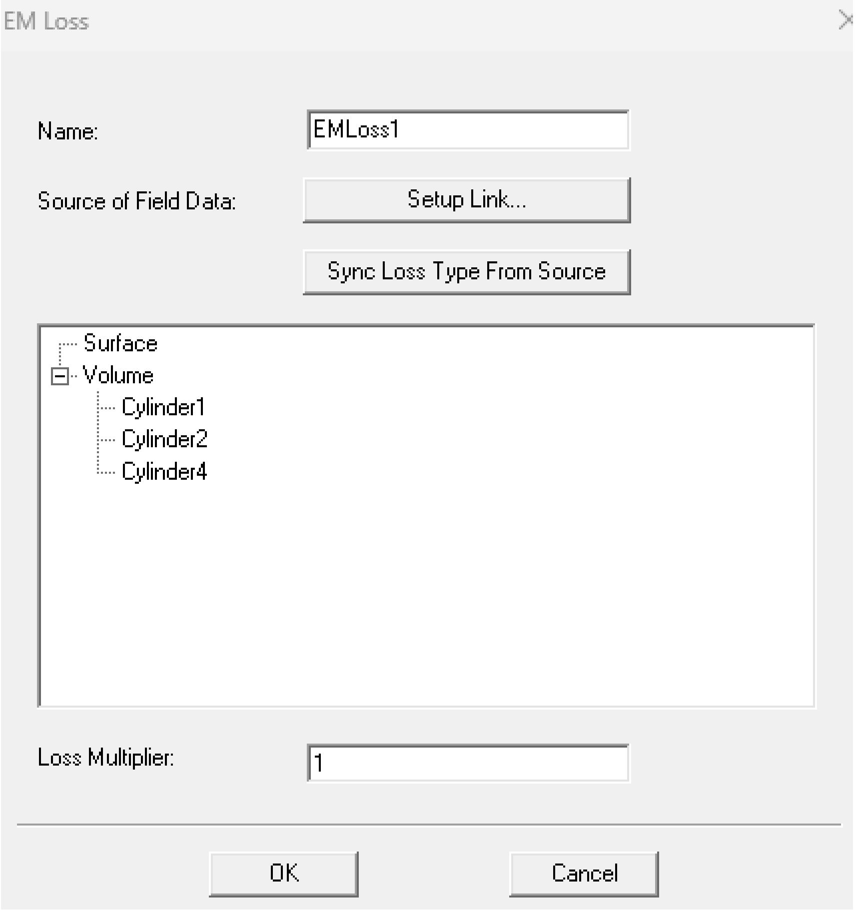


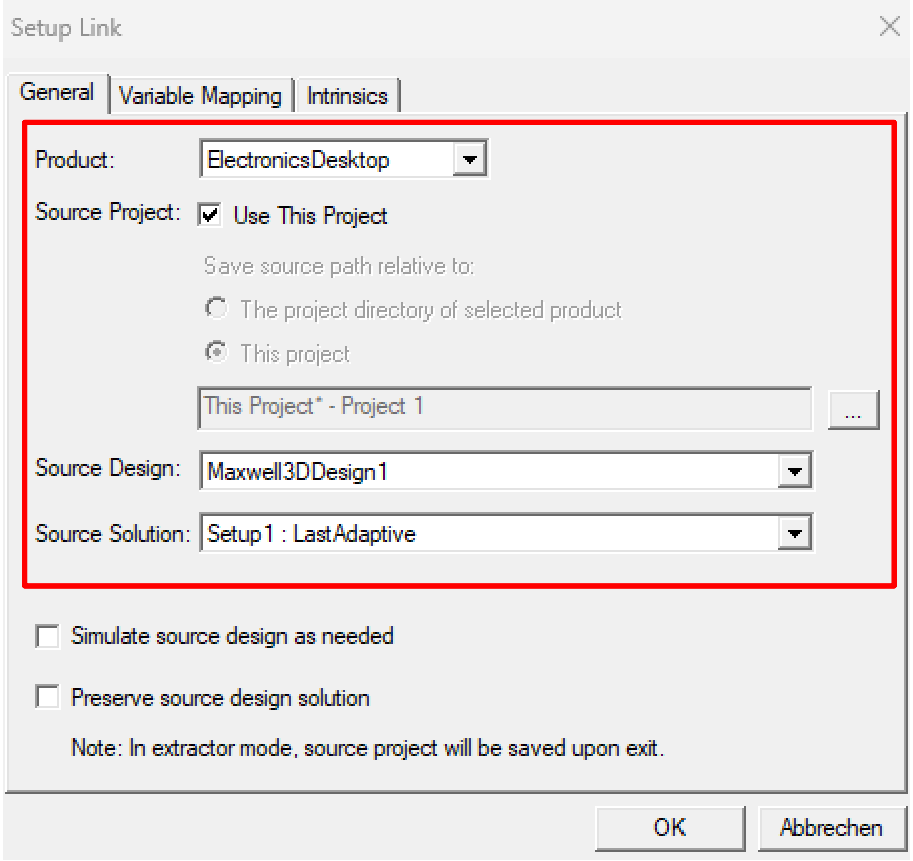


# ICEPAK MODULE

## Workflow of the ICEPAK Module

4. Setup the Link between ICEPAK and the 3D design

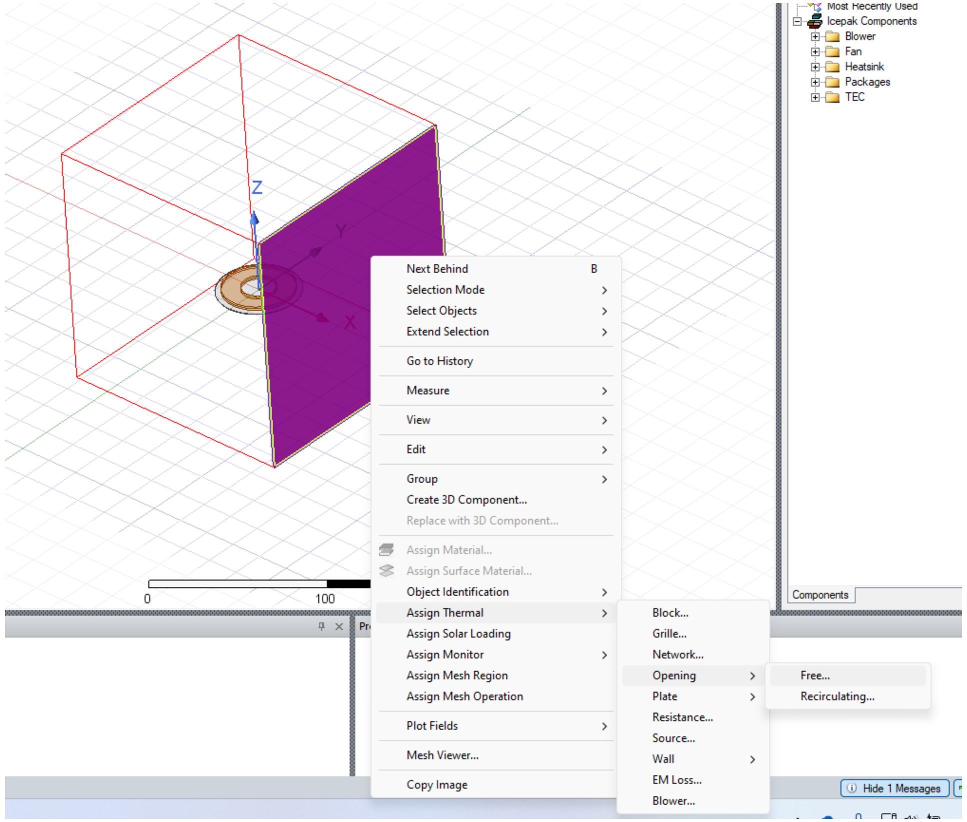


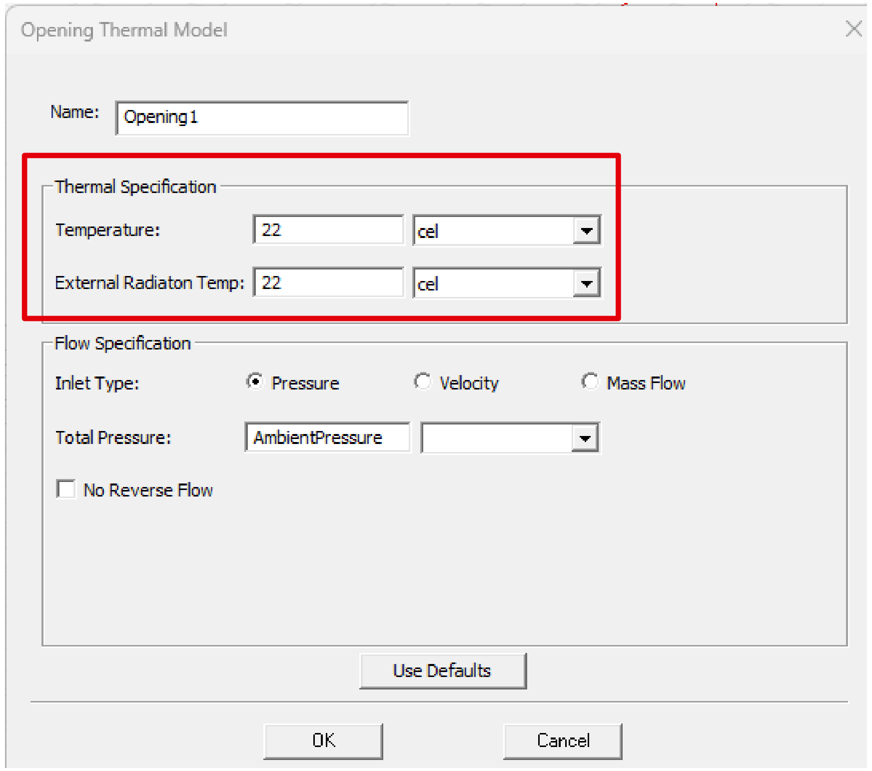



# ICEPAK MODULE

## Workflow of the ICEPAK Module

5. Define openings at all faces of the region except bottom) -> mark faces (push F)  and right click on assign Thermal-> opening -> free -> define ambient temperature

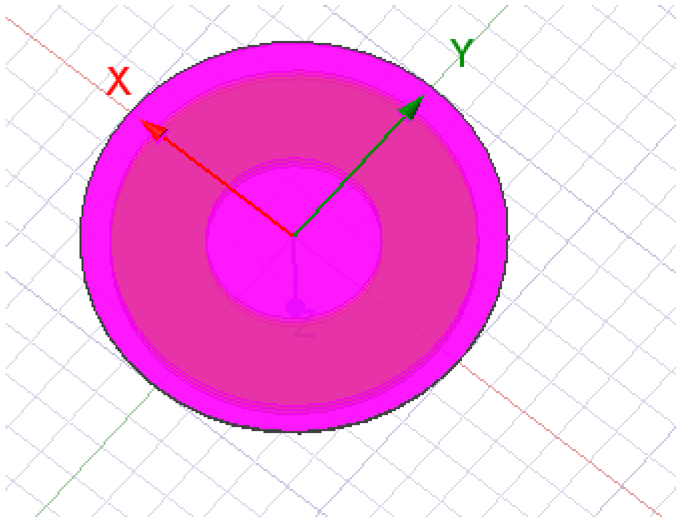


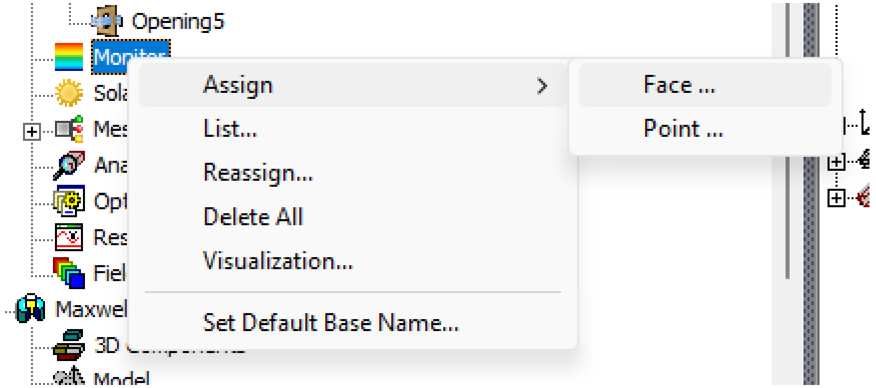



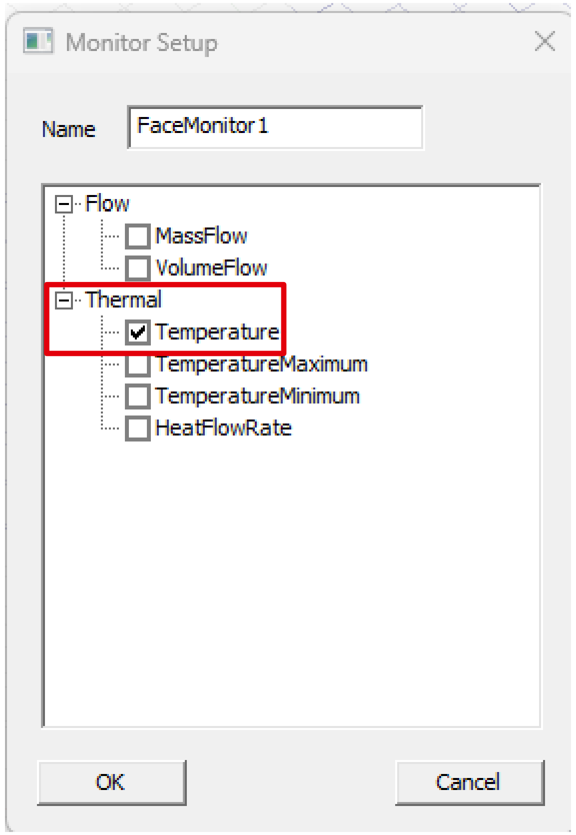
# ICEPAK MODULE

Workflow of the ICEPAK Module

6. Assign temperature monitor -> mark the surface of the ferrite -> click on Monitor -> Assign -> Face -> Monitor Setup Temperature



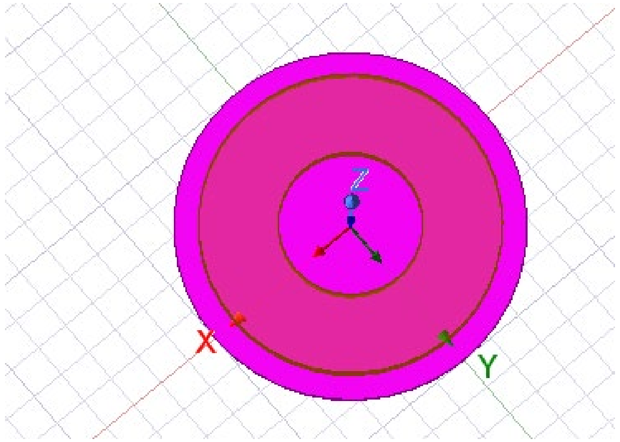



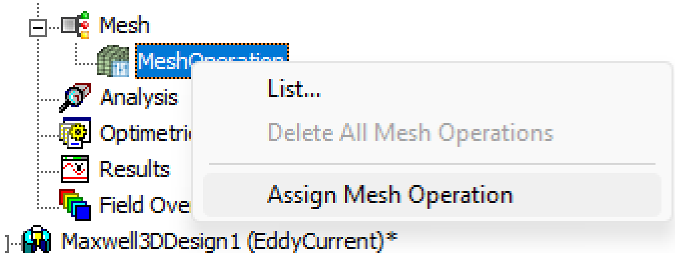



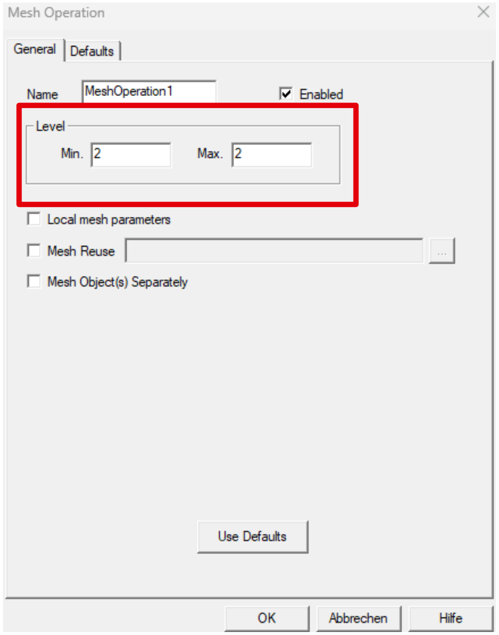
# ICEPAK MODULE

Workflow of the ICEPAK Module

7. Define Mesh -> Mark Coil -> Assign Mesh Operation -> Min, Max value 2




Purpose of the mesh:

Icepak is based on finite volume method

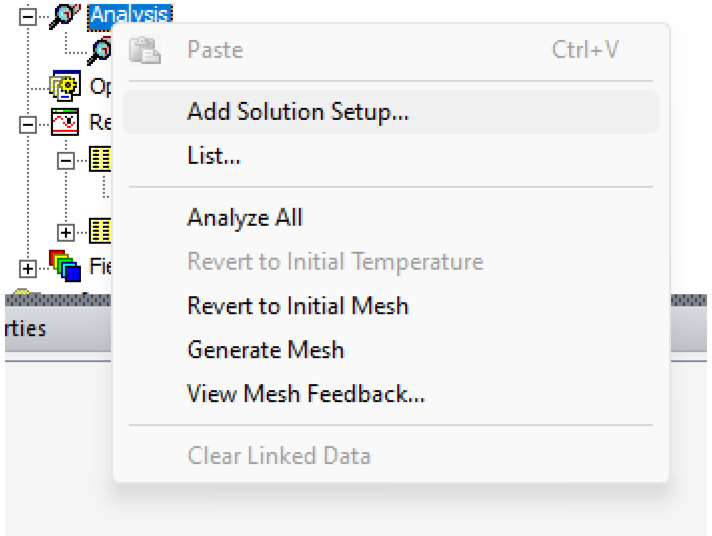
Domain is discretized into a finite set of volumes

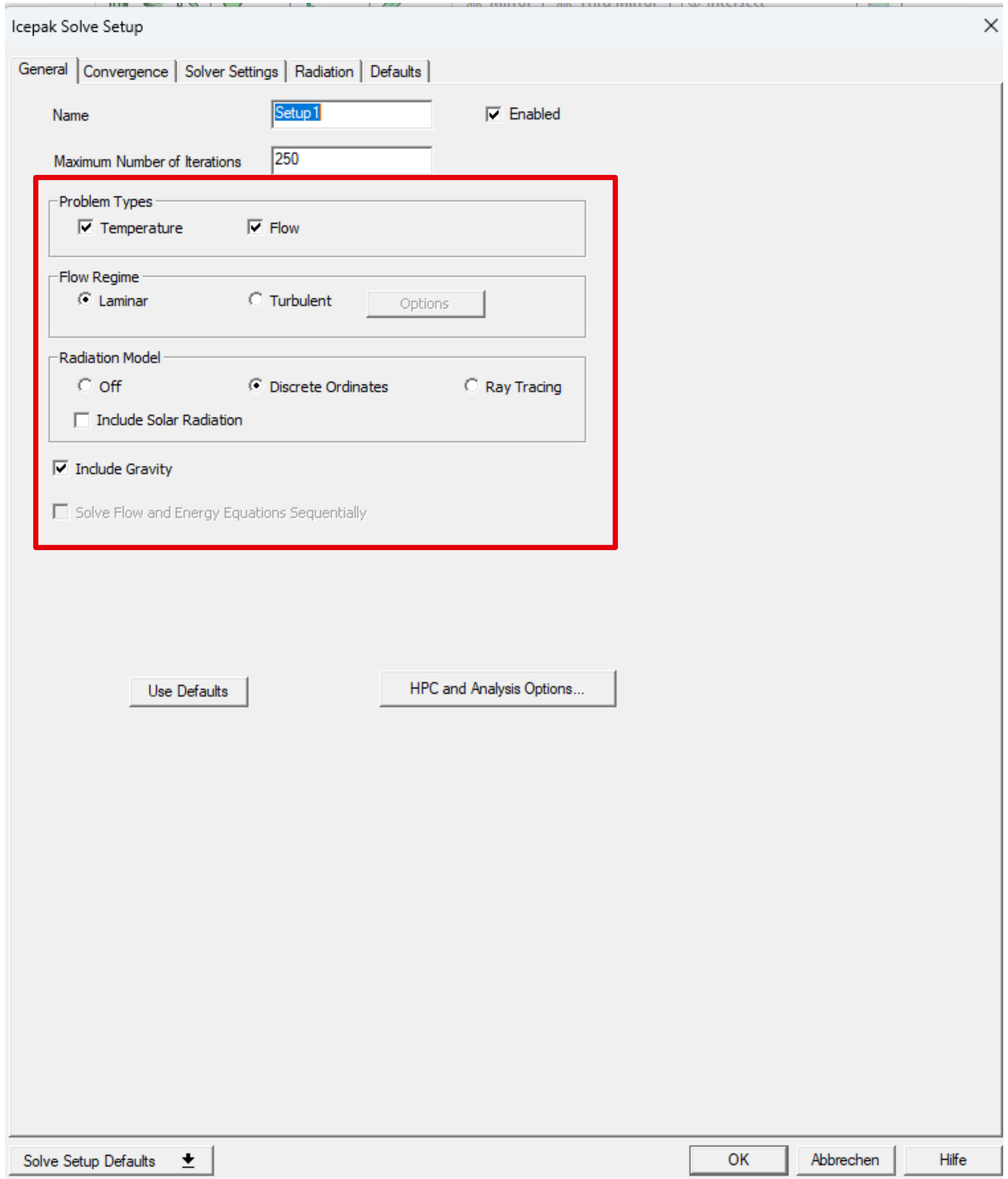
➔ The assignment of mesh operating levels improves the meshing near complex or bended geometry in comparison to the default meshing

# ICEPAK MODULE

## Workflow of the ICEPAK Module

7. Analysis -> Add solution setup



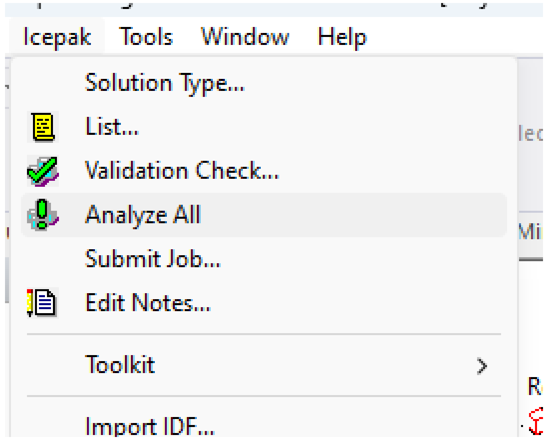
# ICEPAK MODULE

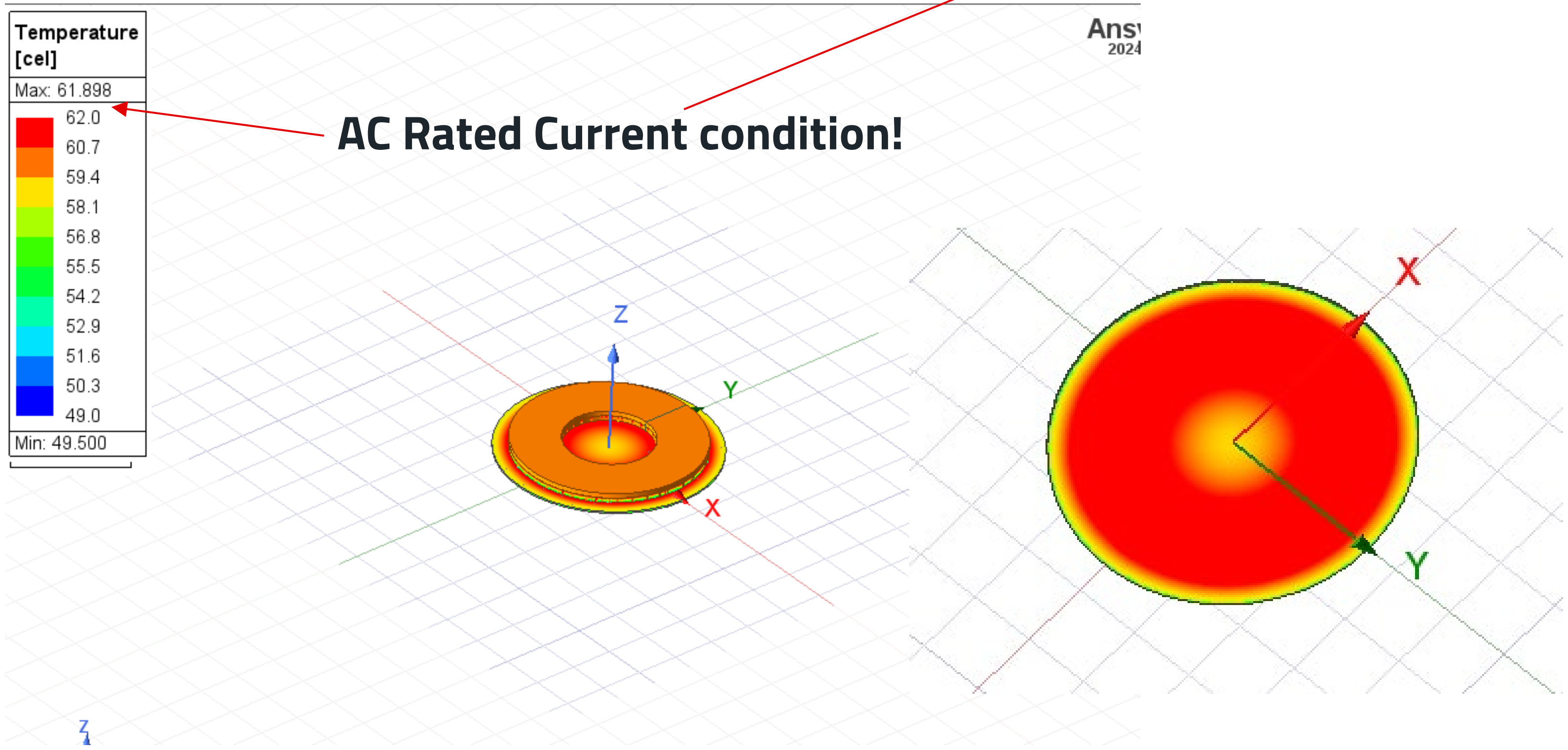
Workflow of the ICEPAK Module: Validation of the measurement result

8. Start simulation –> ICEPAK -> Analyze all

**Measured Value: 3.7 A RMS**

For 5A amplitude (Variable I_in) (~ **3.5 A RMS**) @ 125 kHz
Ambient Temperature 22 °C




**AC Rated Current condition!**

# RESULTS

Comparison of simulated and measured results: 3 Examples

| Coil | IR,AC (RMS) [A] simulated | IR,AC (RMS) [A] measured | **Deviation [%]** |
| --- | --- | --- | --- |
| 760308101410 | 3.5 | 3.7 | **5** |
| 760308101304 | 5.3 | 5.2 | **2** |
| 760308101141 | 5.7 | 5.8 | **2** |

# THANKS FOR YOUR ATTENTION

Questions?

Contact: wirelesspower@we-online.com