

## **Module 4 – Workshop 2: Maxwell and Icepak Multiphysics in AEDT**

Release 2020 R1

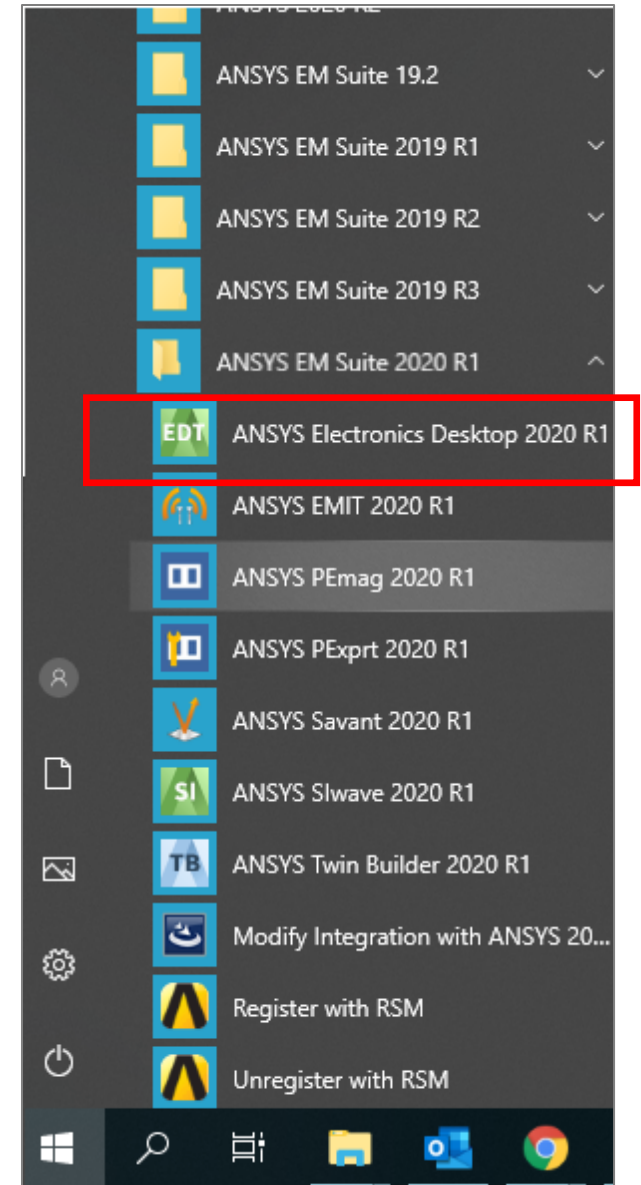


# Introduction

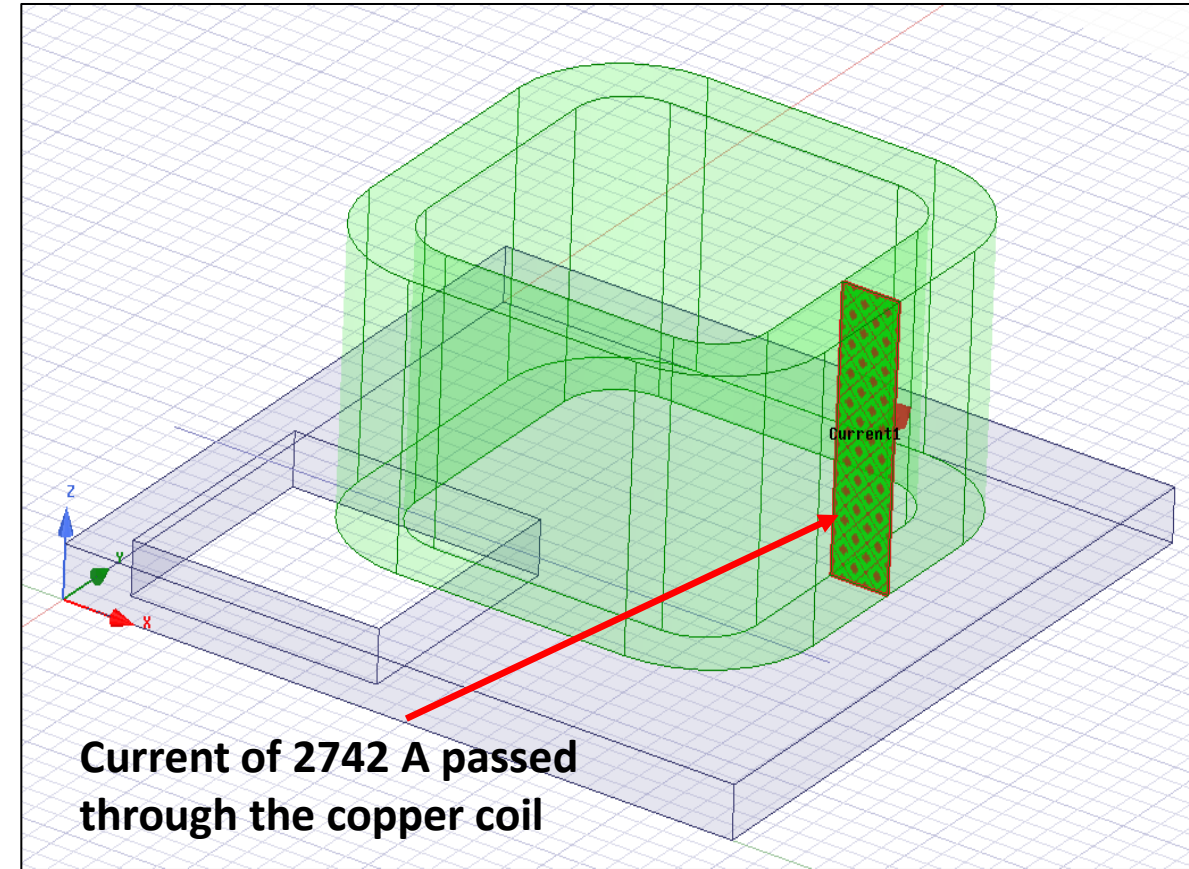
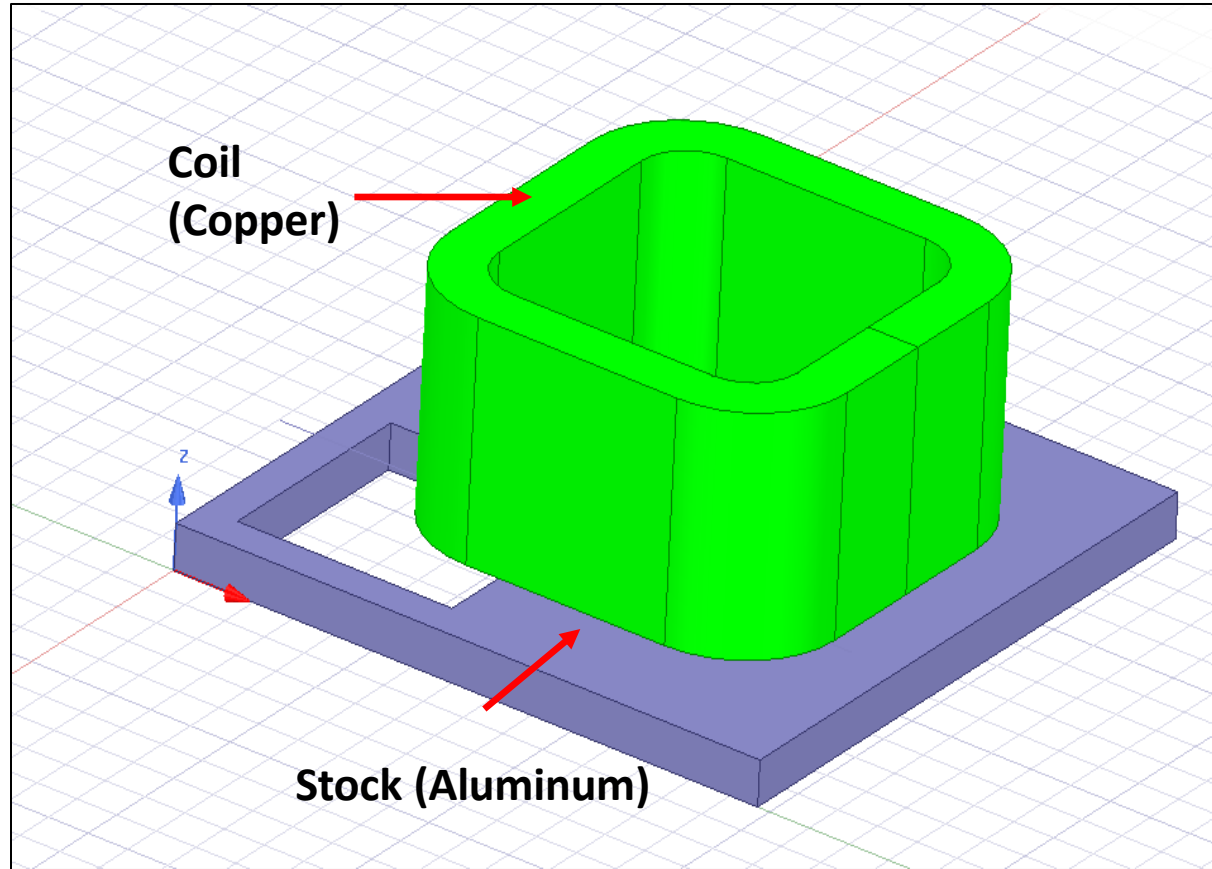
- This workshop demonstrates how Ohmic losses can be transferred from Maxwell to Icepak in ANSYS Electronics Desktop using 2-Way Coupling
- This workshop involves an eddy current solver of Maxwell to calculate heat losses which are then applied to Icepak to simulate natural convection.

# / Getting Started

- ANSYS Electronics Desktop is the electronics equivalent to ANSYS Workbench
- The tool is located under the ANSYS EM Suite 2020 R1

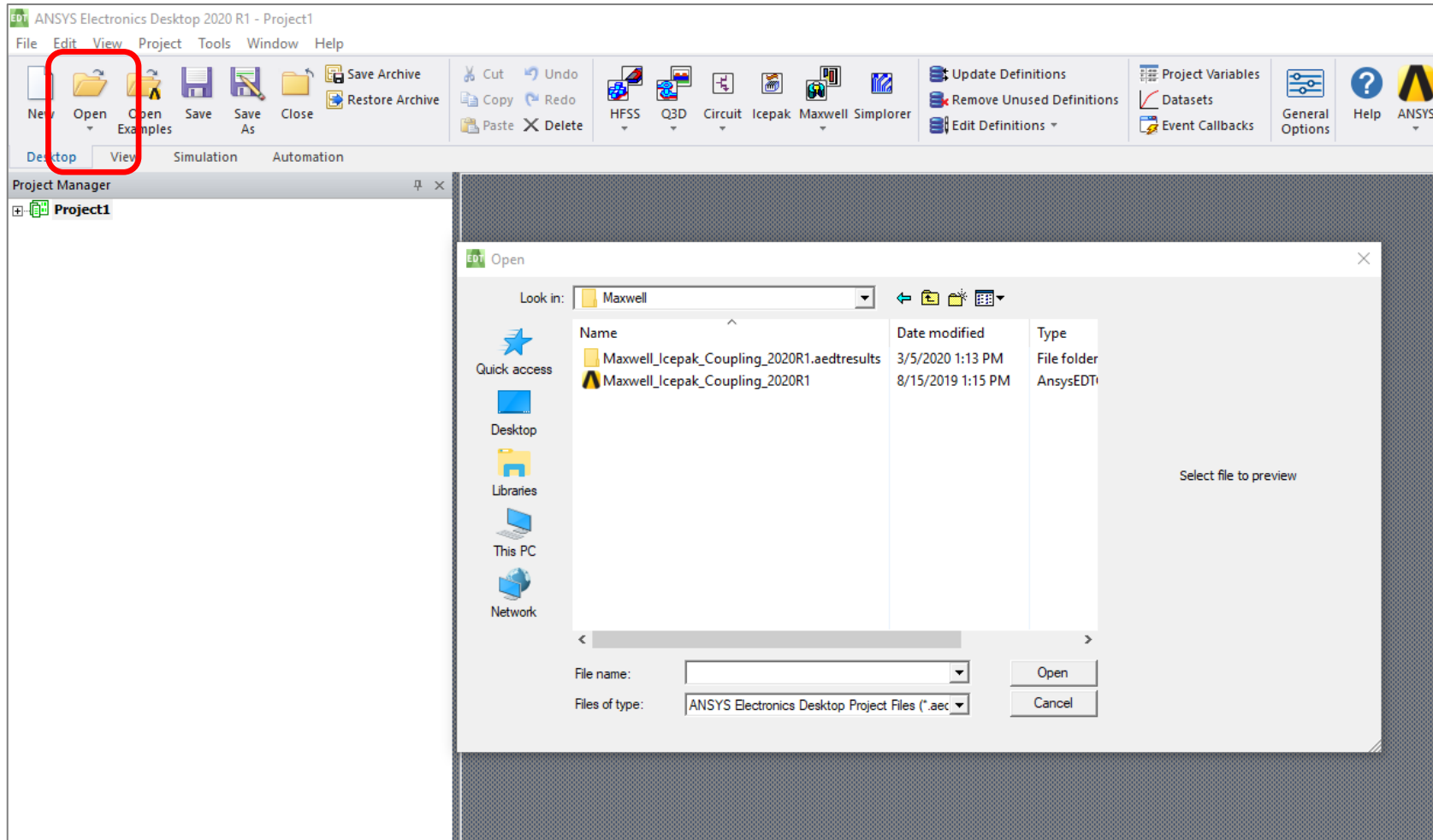


# Problem Description



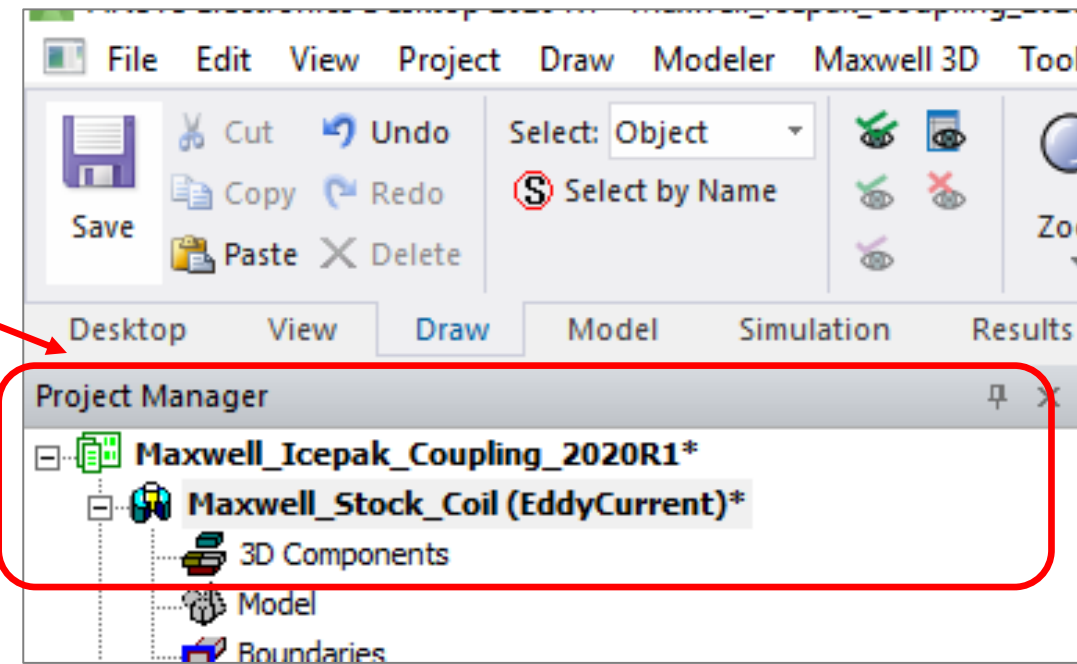
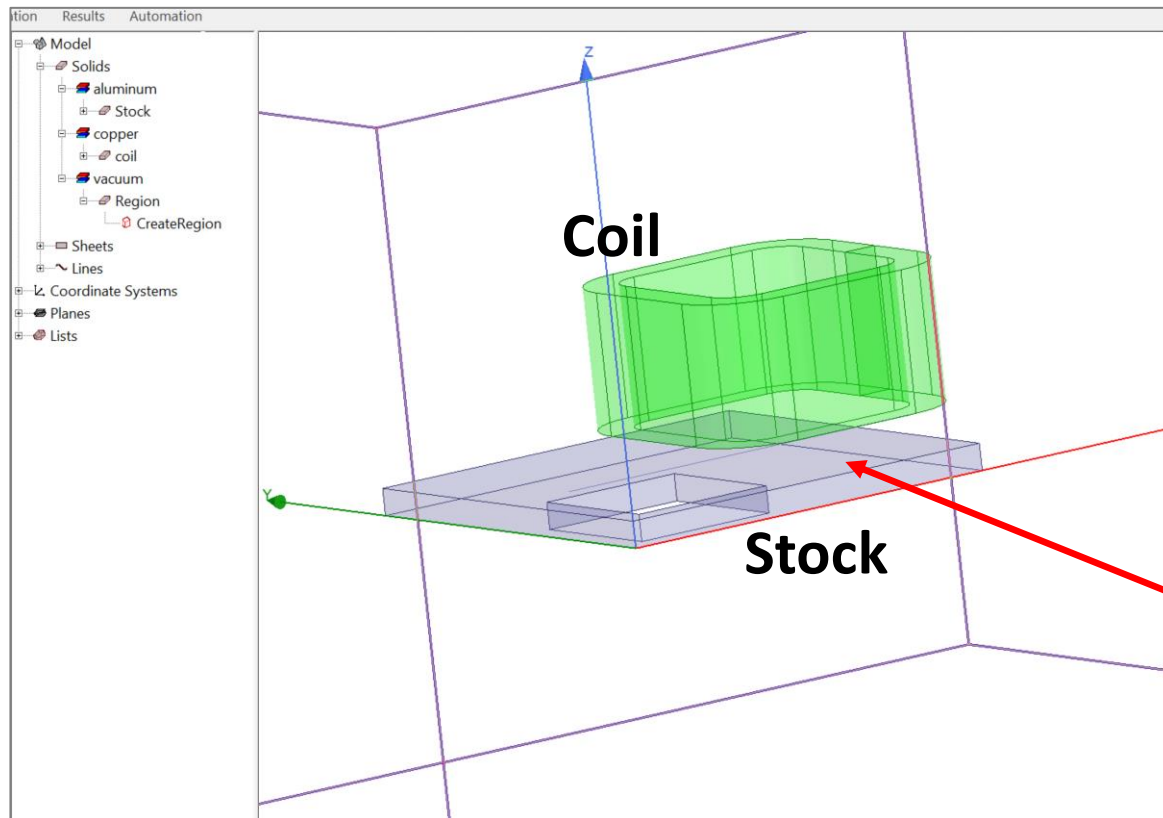
# Open Maxwell Project

- Open the Maxwell project file 'Maxwell\_Icepak\_Coupling\_2020R1' from the workshop folder



# Geometry in Maxwell GUI

Rename the design as 'Maxwell\_Stock\_Coil'

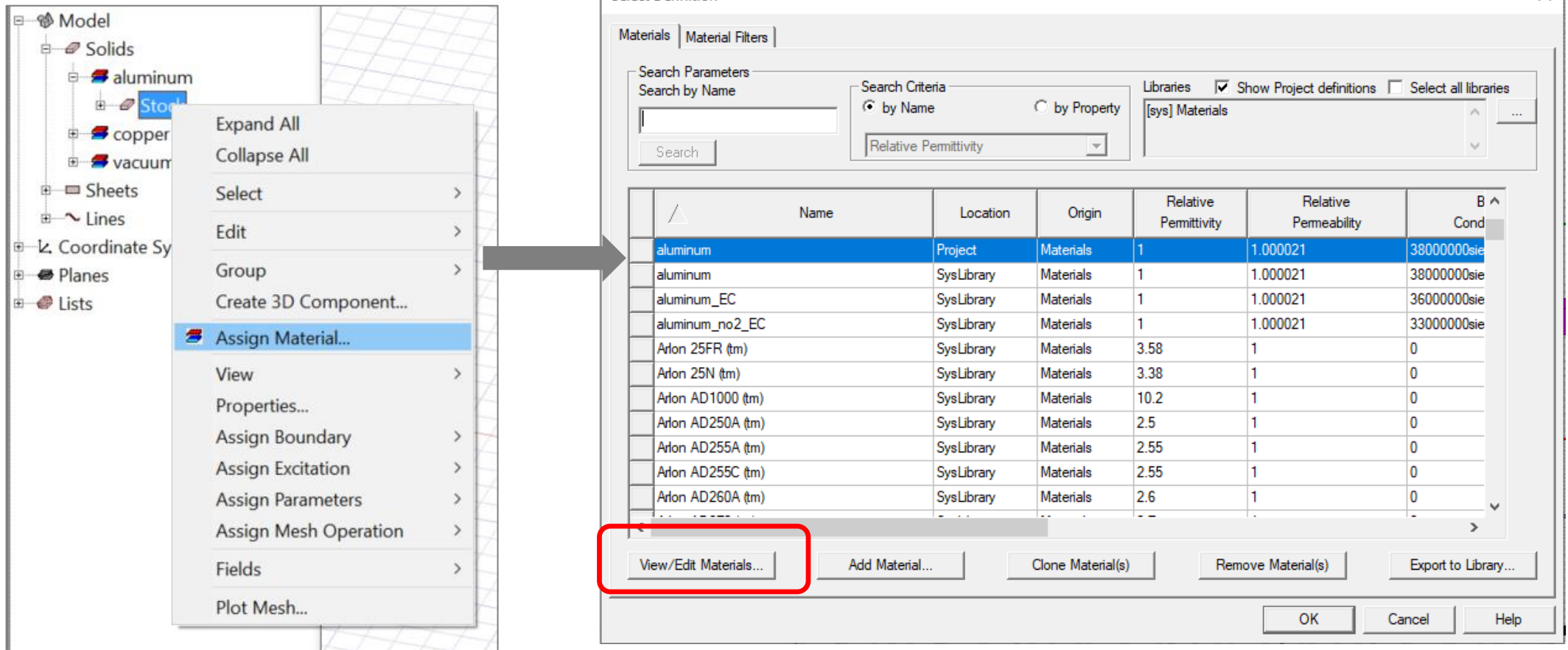


Coil and stock are electrically isolated (i.e. they are not touching each other)



# Maxwell: Assignment of Material Properties

- Select the Stock object from the history tree, right-click and select Assign Material.
- In the **Select Definition** panel, select **View/Edit Materials** to open **View/Edit Material Panel** ( shown in the next slide)



# Maxwell: Thermal Modifier

- Enable **Thermal Modifier**.
- Go to the **Thermal Modifier** column for bulk conductivity and select **Edit** from the drop-down menu.
- This will open 'Edit Thermal Modifier' panel ( shown in the next slide)

View / Edit Material

Material Name: aluminum

Material Coordinate System Type: Cartesian

Properties of the Material

Name	Type	Value	Units	Thermal Modifier
Relative Permittivity	Simple	1		None
Relative Permeability	Simple	1.000021		None
Bulk Conductivity	Simple	38000000	siemens/m	None
Dielectric Loss Tangent	Simple	0		None
Magnetic Loss Tangent	Simple	0		None
Core Loss Model		None	w/m^3	
Mass Density	Simple	2689	kg/m^3	None
Composition		Solid		

View/Edit Material for:

- ☒ Active Design
- ☐ Active Project
- ☐ All Properties

Physics:

- ☒ Electromagnetic
- ☒ Thermal
- ☒ Structural

View/Edit Modifier for:

- ☒ Thermal Modifier
- ☐ Spatial Modifier

Material Appearance

- ☐ Use Material Appearance

Color:

Transparency:

Notes:

Calculate Properties for:

Reset OK Cancel

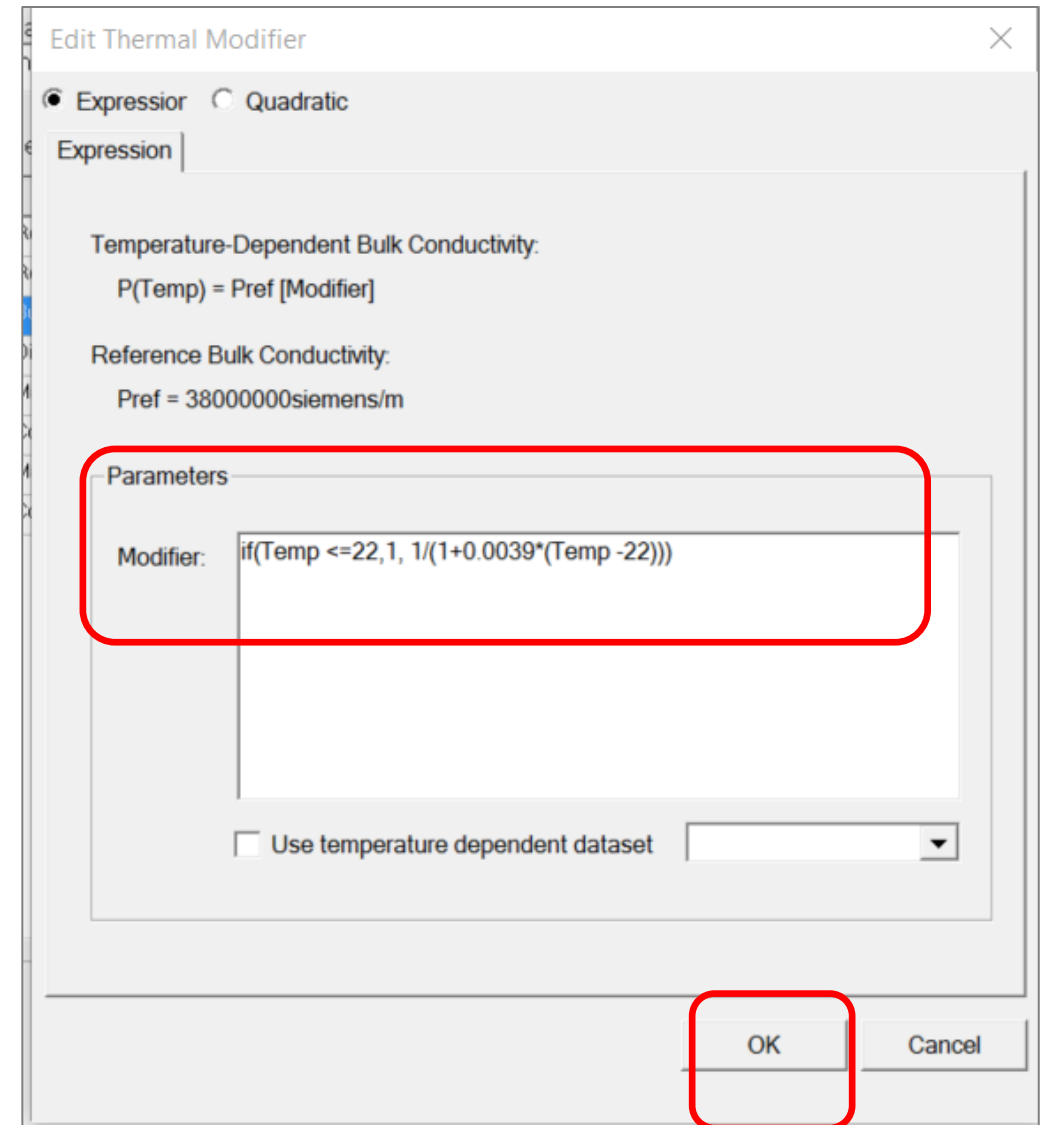
Validate Material



# / Maxwell: Temperature Dependent Bulk Conductivity

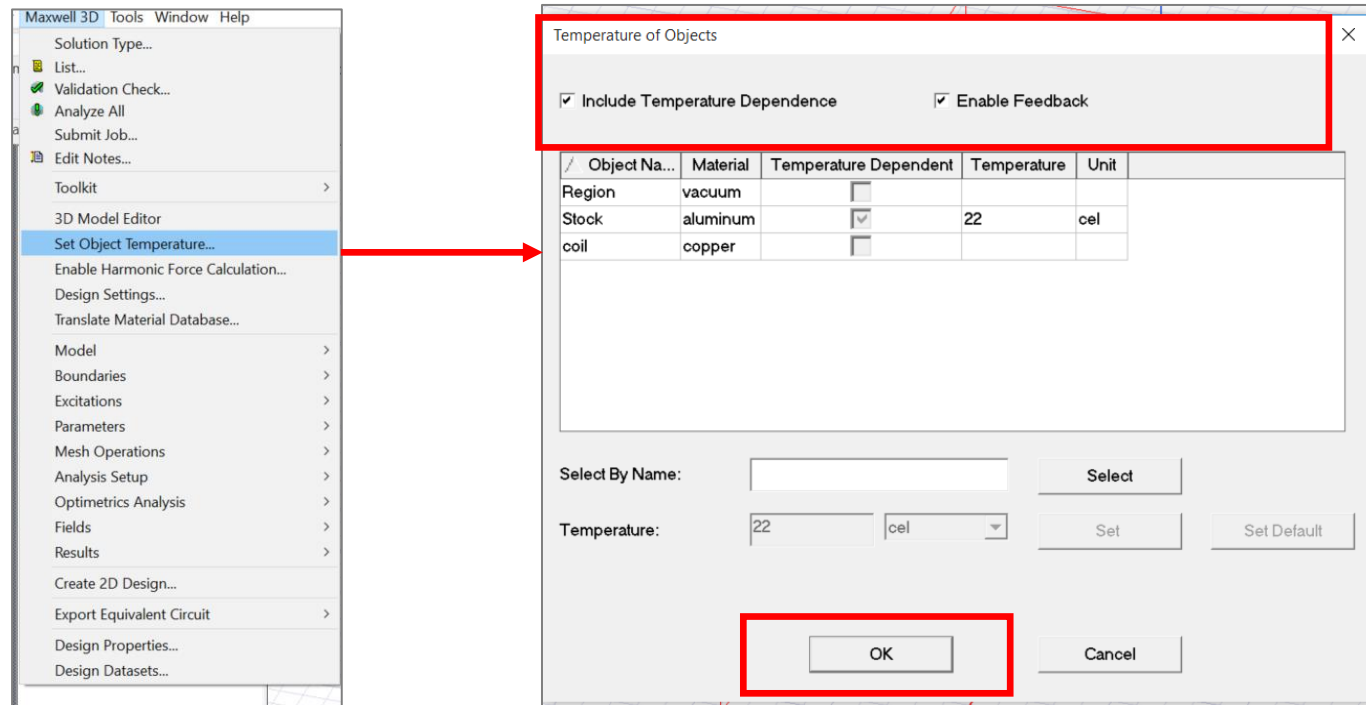
- In the **Edit Thermal Modifier** panel,  
Set modifier as:  
 $\text{if}(\text{Temp} \leq 22, 1, 1/(1+0.0039*(\text{Temp}-22)))$
- Click **OK**.

Note: In order to use 2-ways coupling  
Temperature dependent properties are required  
for at least one component



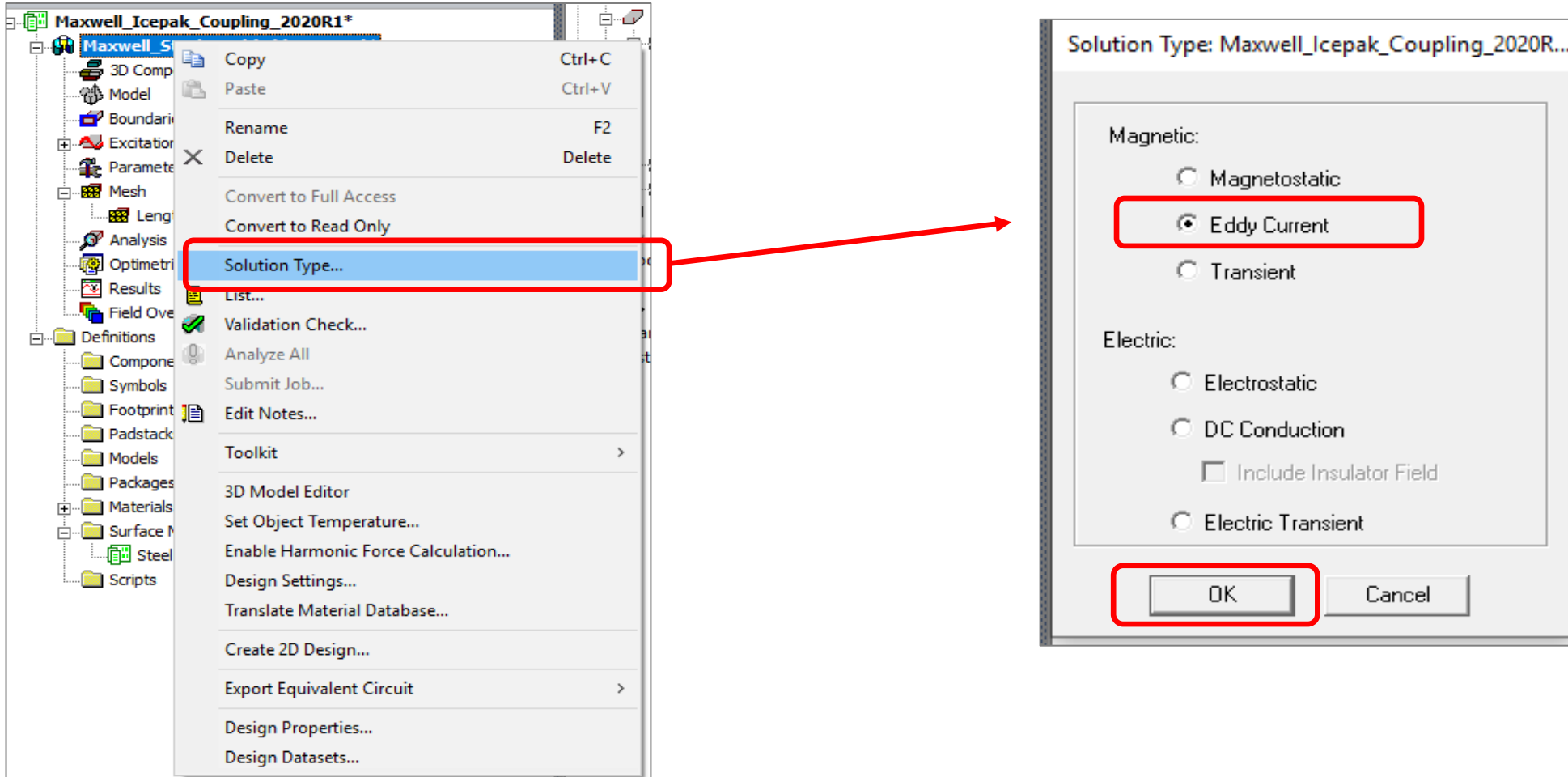
# Maxwell: Enable Temperature Feedback

1. From the Main Menu use **Maxwell 3D → Set Object Temperature**.
  - This feature can also be accessed from the Project manager → Maxwell\_Stock\_Coil → RMB → Set Object Temperature
2. In the **Temperature of Objects** panel, enable **Include Temperature Dependence** and **Enable Feedback**.
3. Click **OK**.



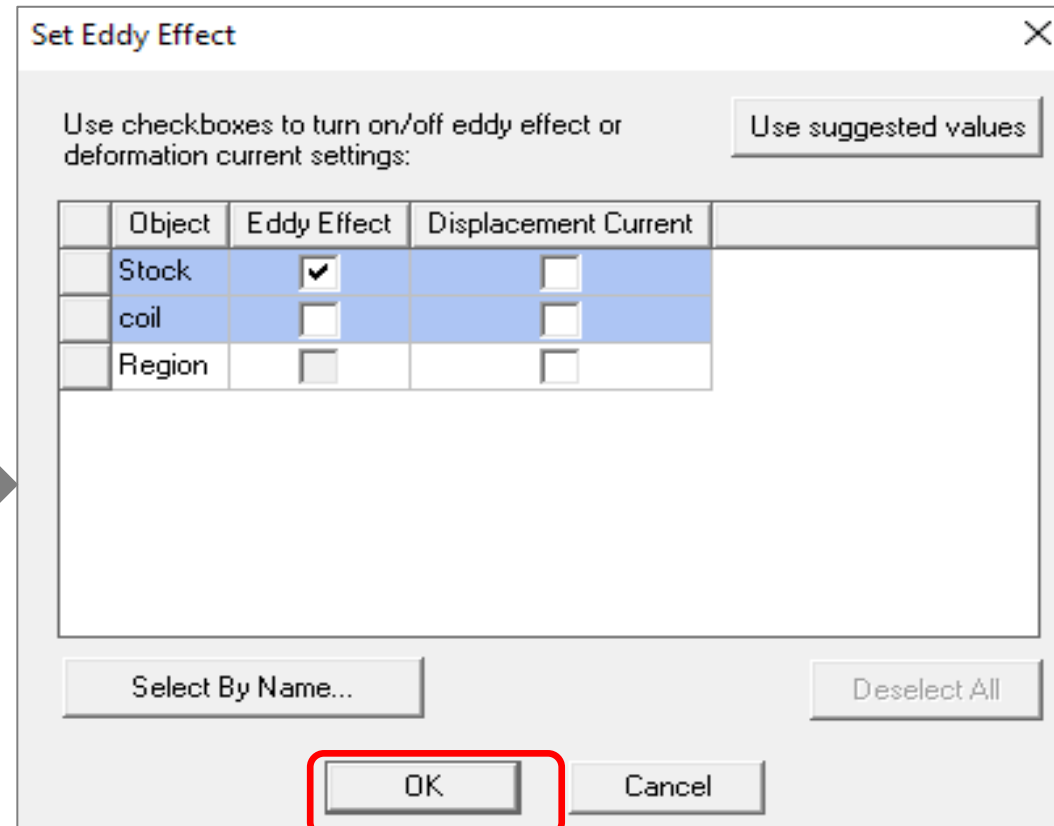
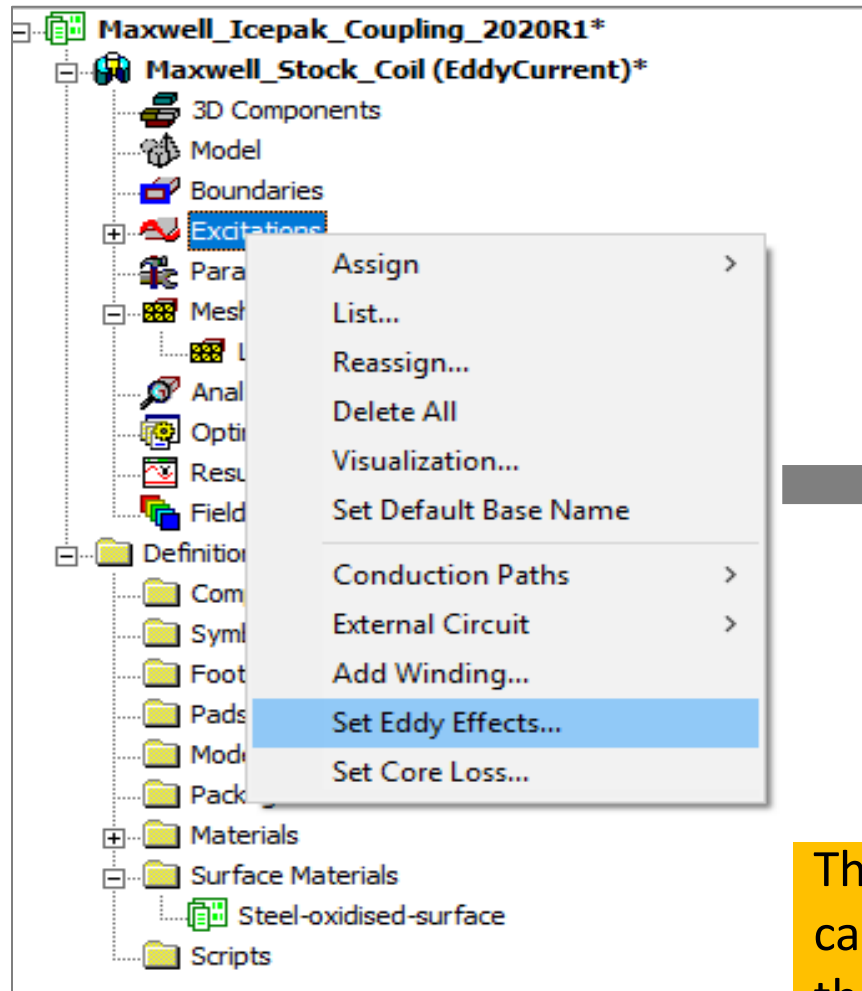
# Maxwell : Solution Type

- Right click on the Project Maxwell\_Stock\_Coil → Solution Type and define 'Eddy Current' under Magnetic solution type.



# Maxwell : Eddy Effects...

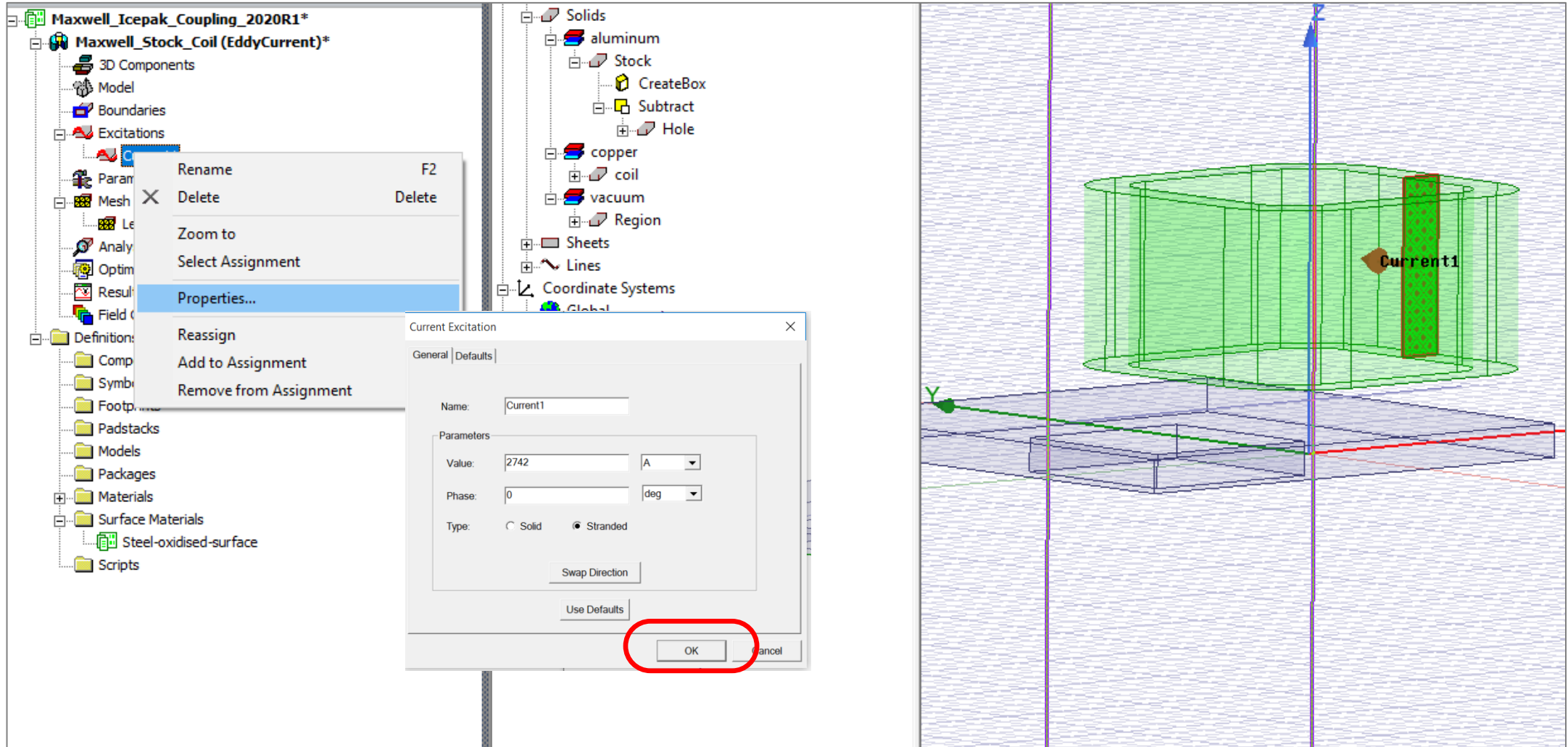
- Right click on 'Excitations' → Set Eddy Effects and select Eddy Effect for the object Stock.



This example uses a stranded "bulk" coil. For such cases, the eddy effect is turned OFF for coil and only the DC winding losses are calculated.

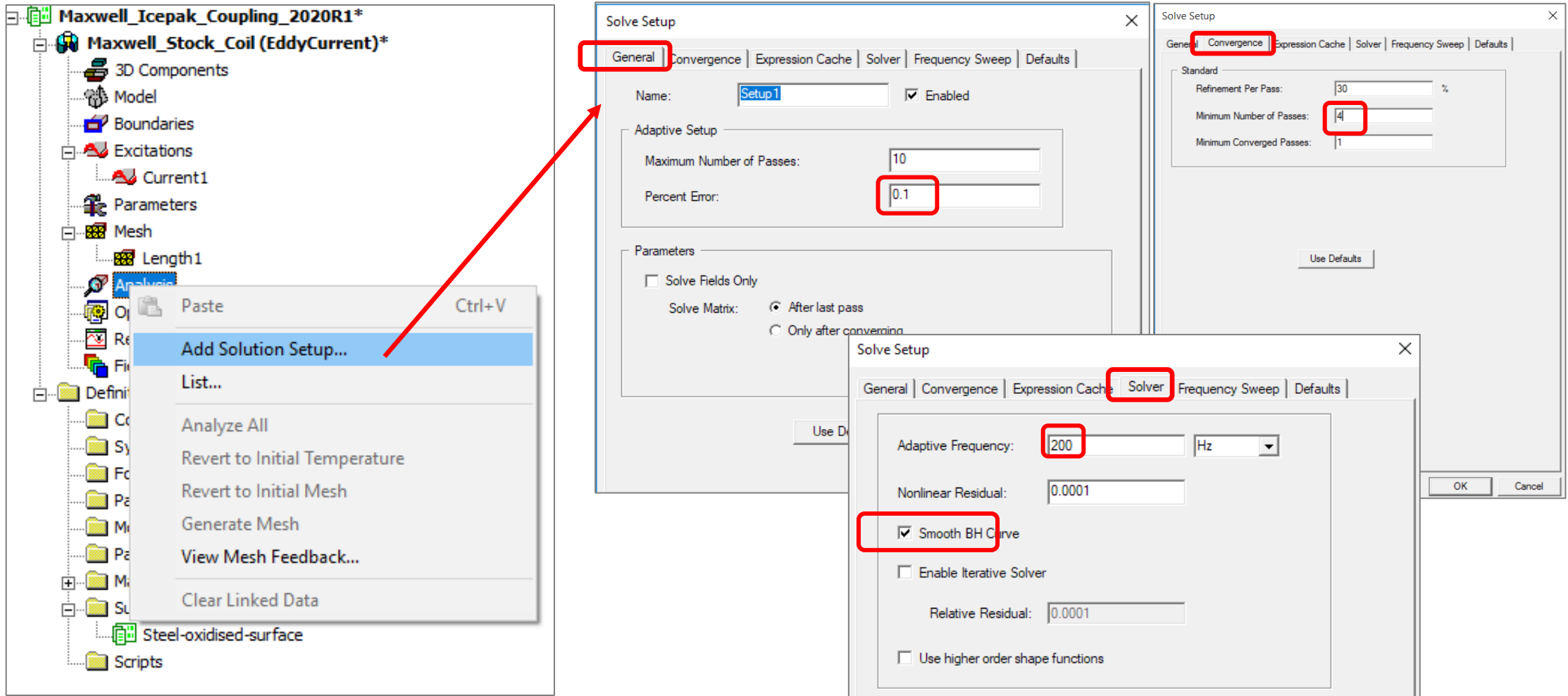
# Maxwell : Current Excitation

- Define the Current properties as shown in the Current Excitation panel.



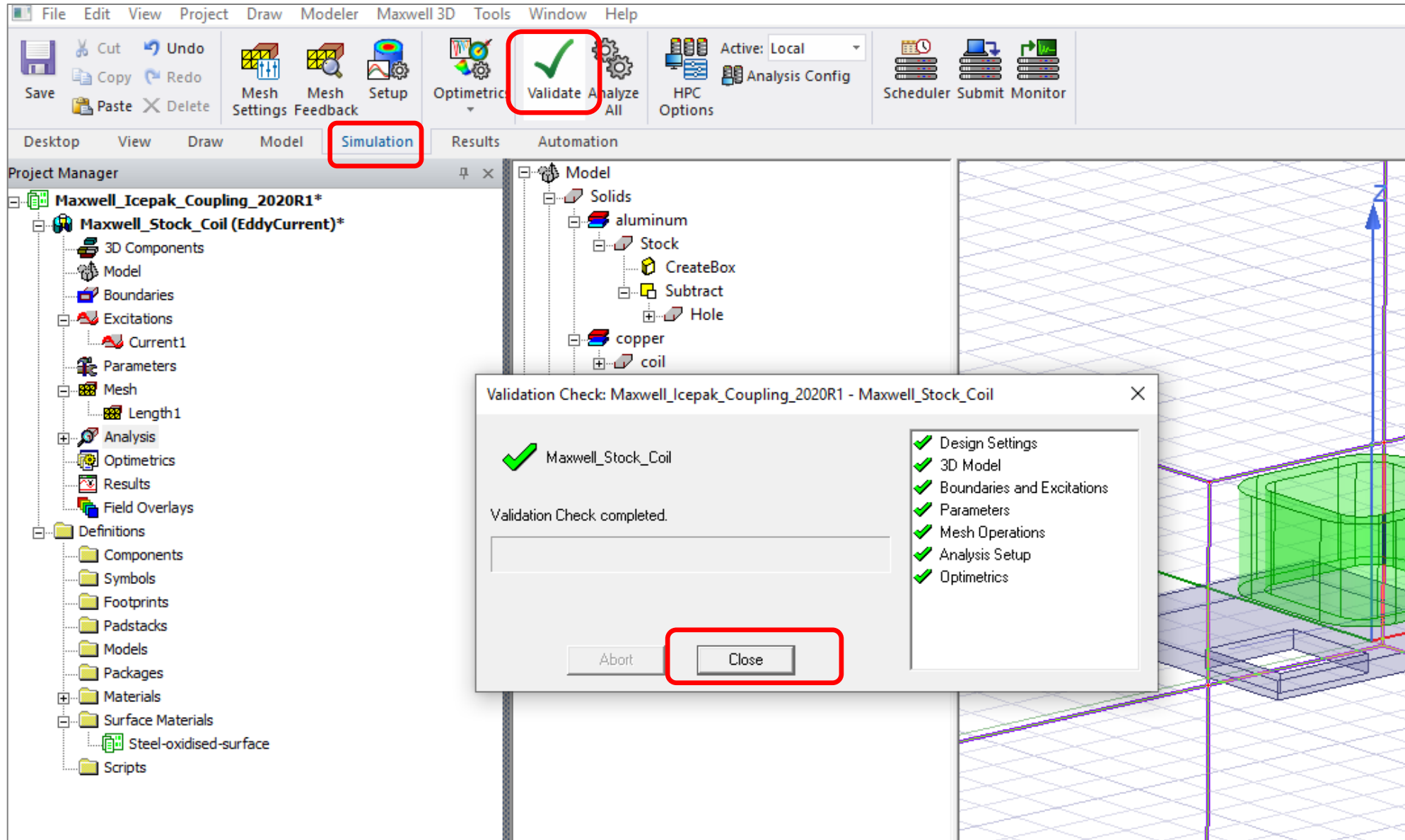
# Maxwell : Add Solution Setup

- In the Solve Setup Panel go through the General, Convergence and Solver panel as shown below:

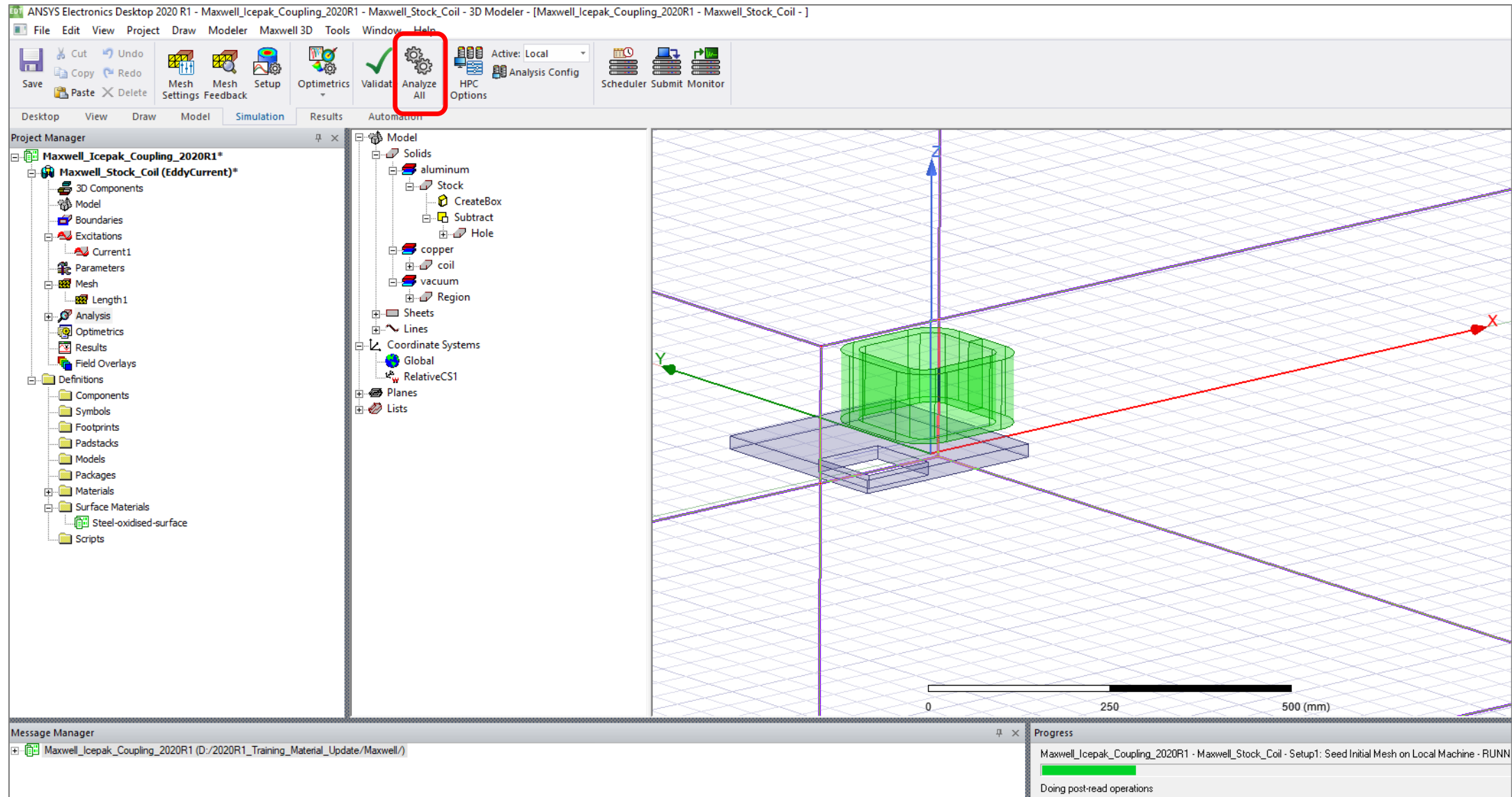




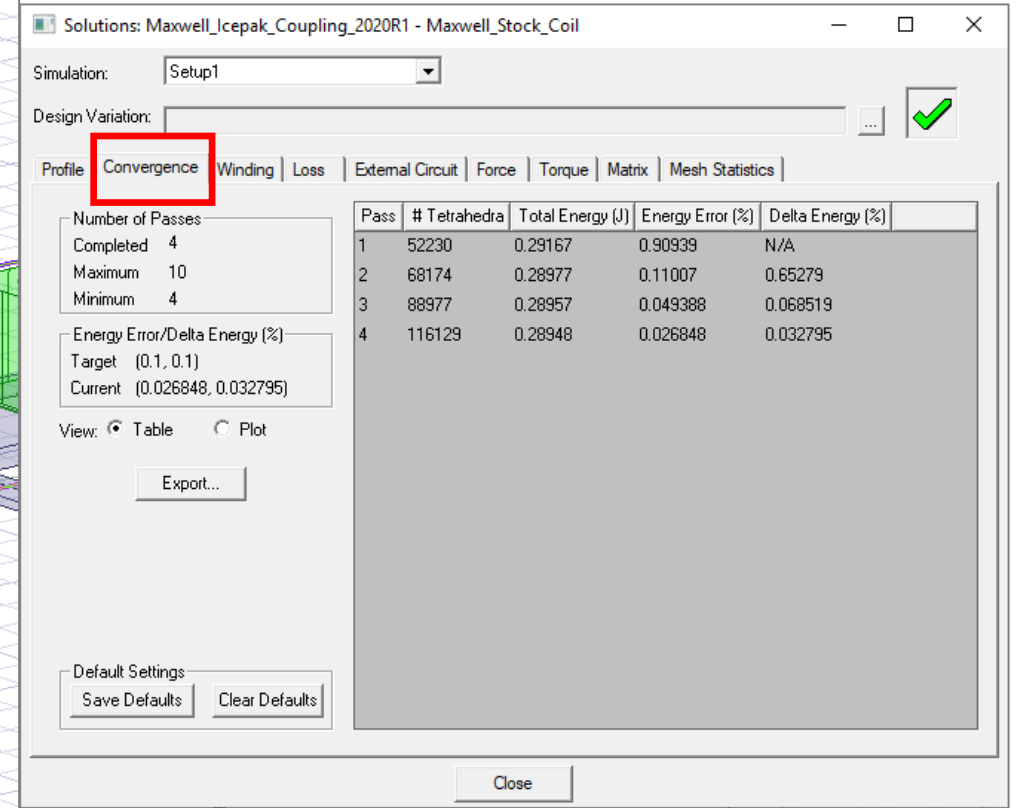
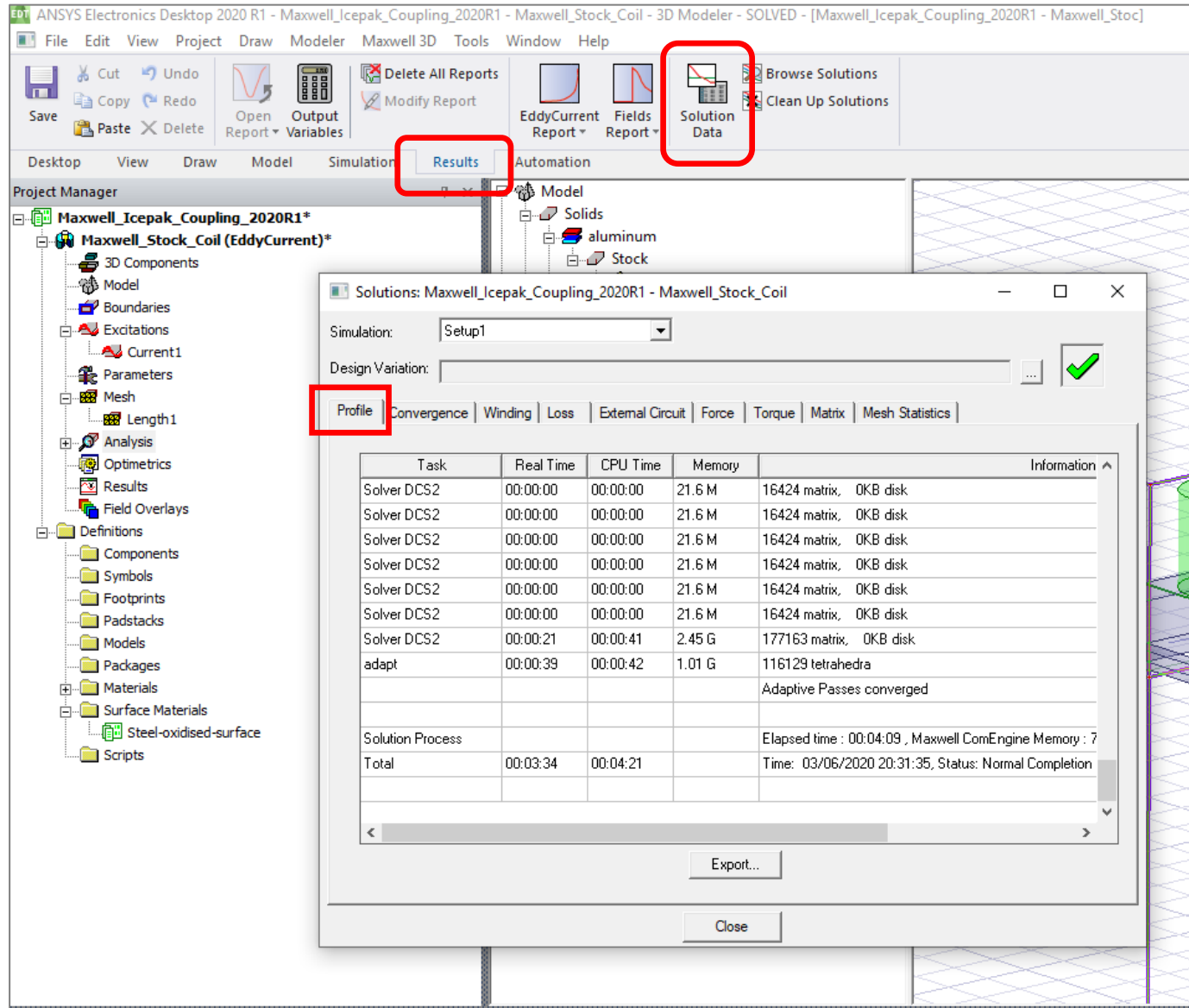
# Maxwell : Validate



# Maxwell : Solve



# Maxwell : Convergence



# Maxwell : Calculate Ohmic Loss for Stock using Field Calculator

- From Fields Calculator panel select:

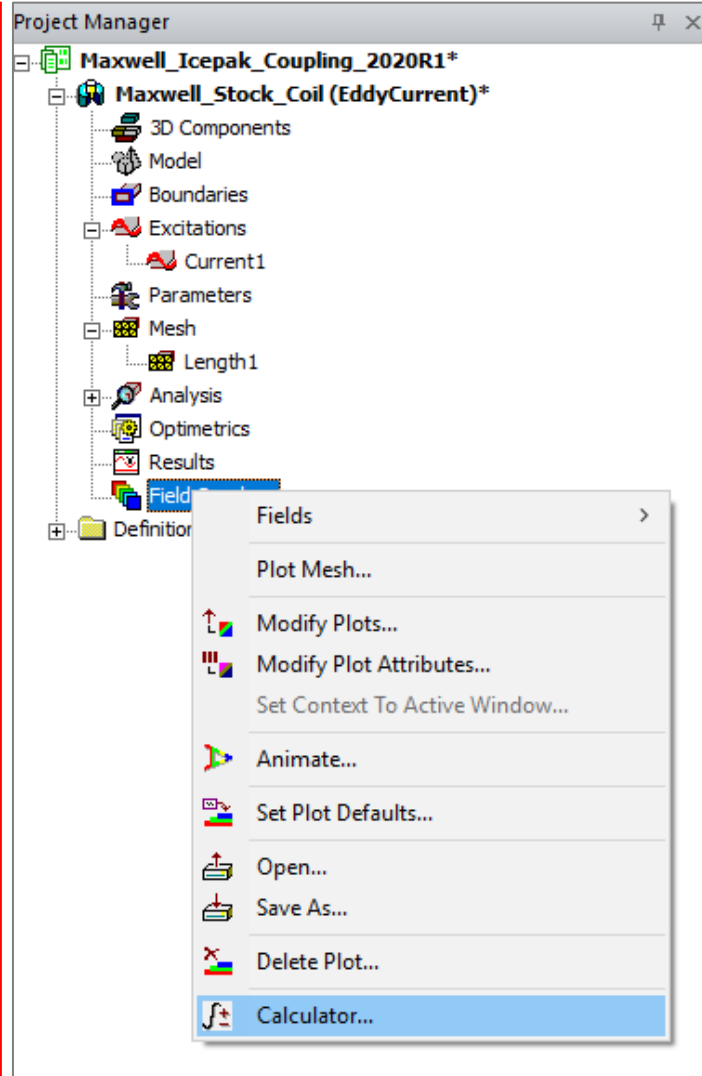
1. Input → Quantity → OhmicLoss
2. Input → Geometry → Volume → Stock
3. Scalar →  $\int$  (Integrate)

Click Add... and enter Name as - *loss\_stock*

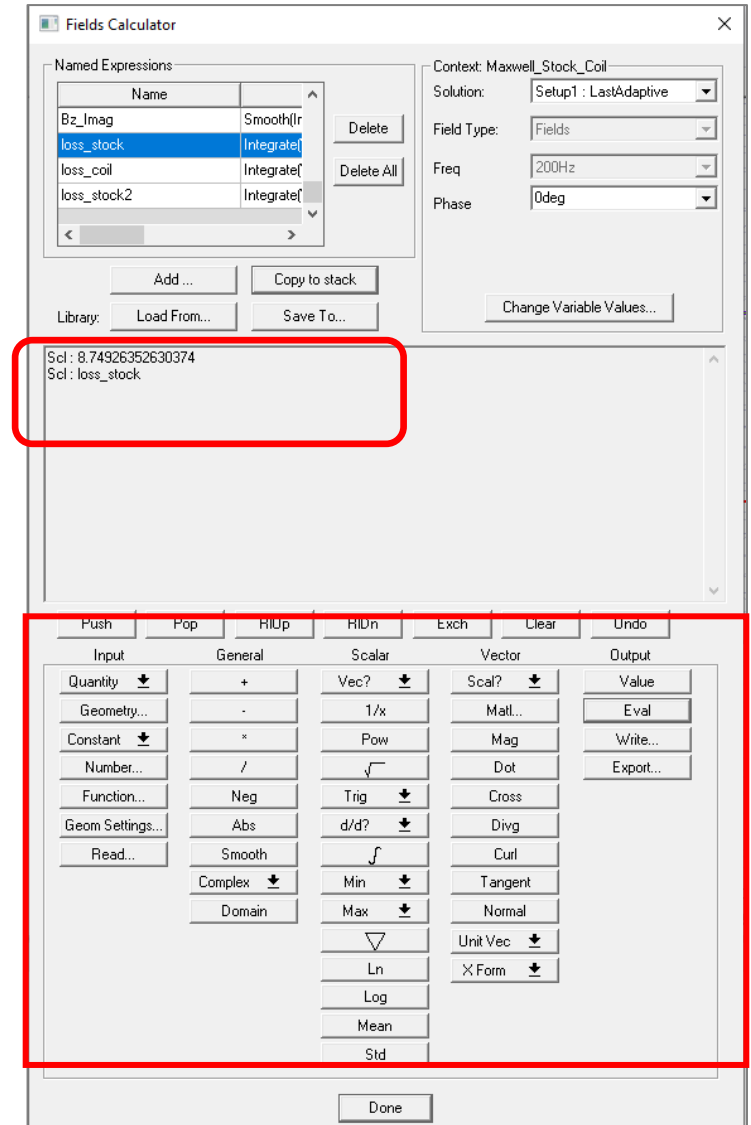
Select “loss\_stock” from Named expressions list and Select “Copy to stack”

- Select “Eval” under output option

The Ohmic losses in Stock volume are around 8.749 W. These losses are the induced AC losses in the plate.

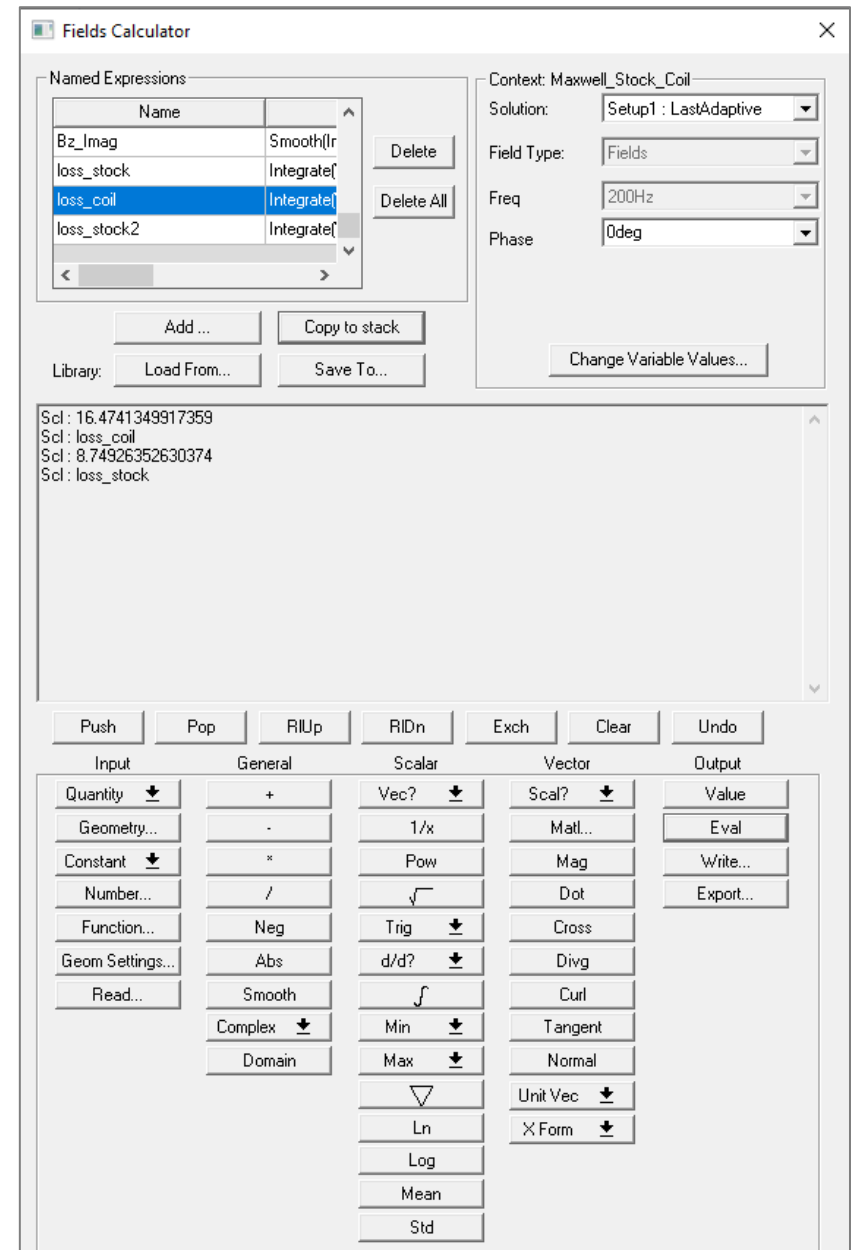


Has been created



# Maxwell : Ohmic Loss for Coil

- Create similar Named Expression for coil
- The Ohmic losses in Coil volume are ~16.47 Watts.  
*Note: Since the coil consists of many small conductors, it was modelled as a bulk “stranded” source which neglects all AC skin and proximity effects.*
- The reported loss is due to the DC resistance of the coil.



# Maxwell : Field Report - Data Table

The screenshot displays the ANSYS Electronics Desktop 2020 R1 interface. The top menu bar includes File, Edit, View, Project, Draw, Modeler, Maxwell 3D, Tools, Window, and Help. The toolbar contains icons for Save, Cut, Copy, Paste, Undo, Redo, Delete, Open Report, Output Variables, Delete All Reports, Modify Report, EddyCurrent Report, Fields Report, Solution Data, Browse Solutions, and Clean Up Solutions. The Project Manager on the left shows the hierarchy: Maxwell\_Icepak\_Coupling\_2020R1\* > Maxwell\_Stock\_Coil (EddyCurrent)\* > 3D Components > Model > Boundaries > Excitations > Current1. The Results tab is selected, showing various report types: 2D, Stacked, Data Table, and 3D. The Data Table report is highlighted with a red box. A red arrow points from the Data Table report to the 'New Report' button in the 'Report: Maxwell\_Icepak\_Coupling\_2020R1 - Maxwell\_Stock\_Coil - New Report - New Trace(s)' dialog. The dialog shows the 'Trace' tab with 'Primary Sweep' set to 'Freq' and 'All'. The 'X' axis is 'Default Freq' and the 'Y' axis is 'loss\_stock'. The 'Quantity' list includes 'loss\_stock', 'loss\_coil', and 'loss\_stock2'. The 'Function' list includes '<none>', 'abs', 'acos', 'acosh', 'ang\_deg', 'ang\_deg\_val', 'ang\_rad', 'arg', 'asin', 'asinh', and 'atan'. The 'New Report' button is highlighted with a red box. A red arrow points from the 'New Report' button to the 'Calculate' button in the 'Calculate' dialog. The 'Calculate' dialog shows a table with the following data:

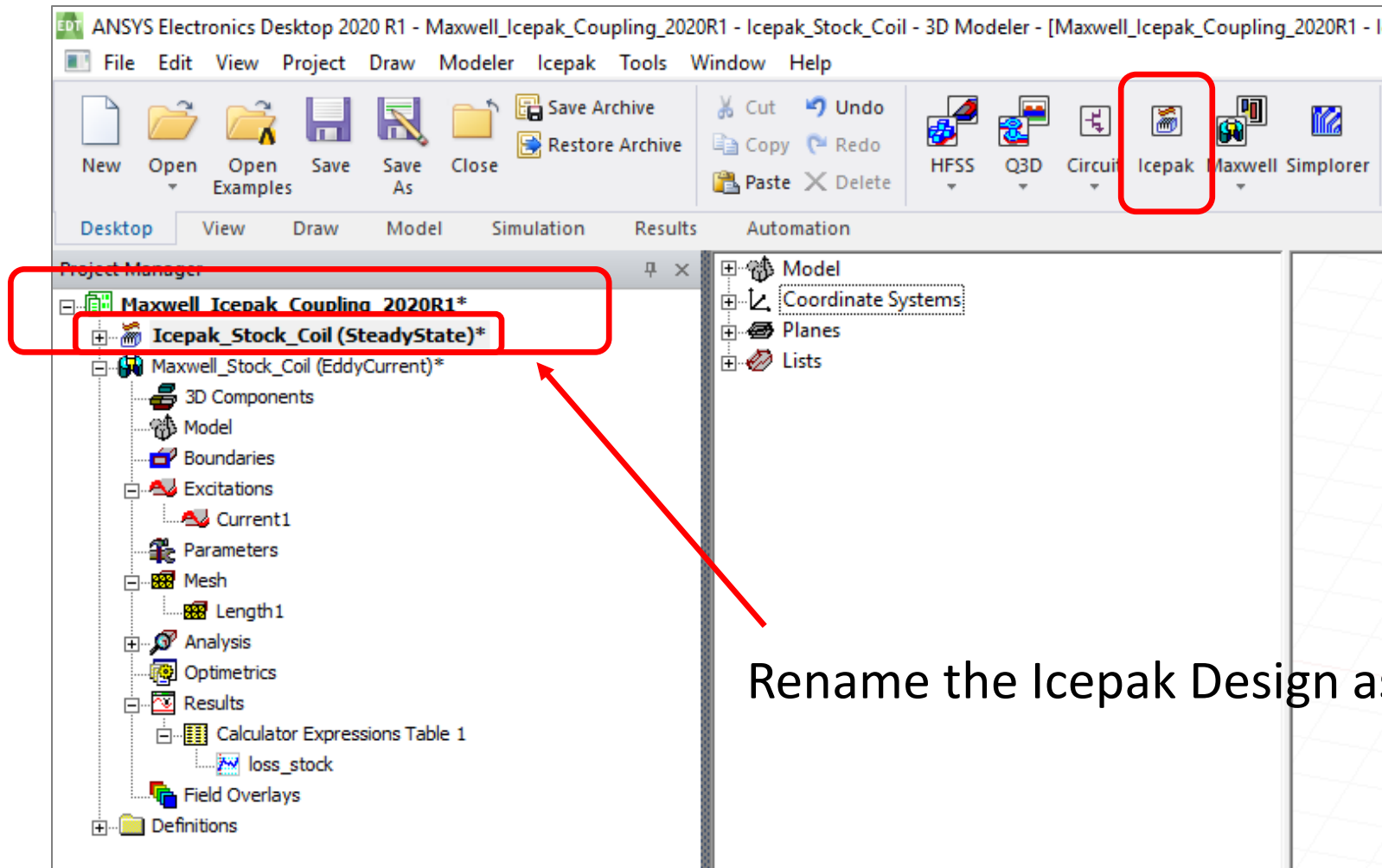
	Freq [Hz]	loss_stock Setup1 : LastAdaptive Phase='0deg'
1	200.000000	8.749264





# Insert Icepak Design

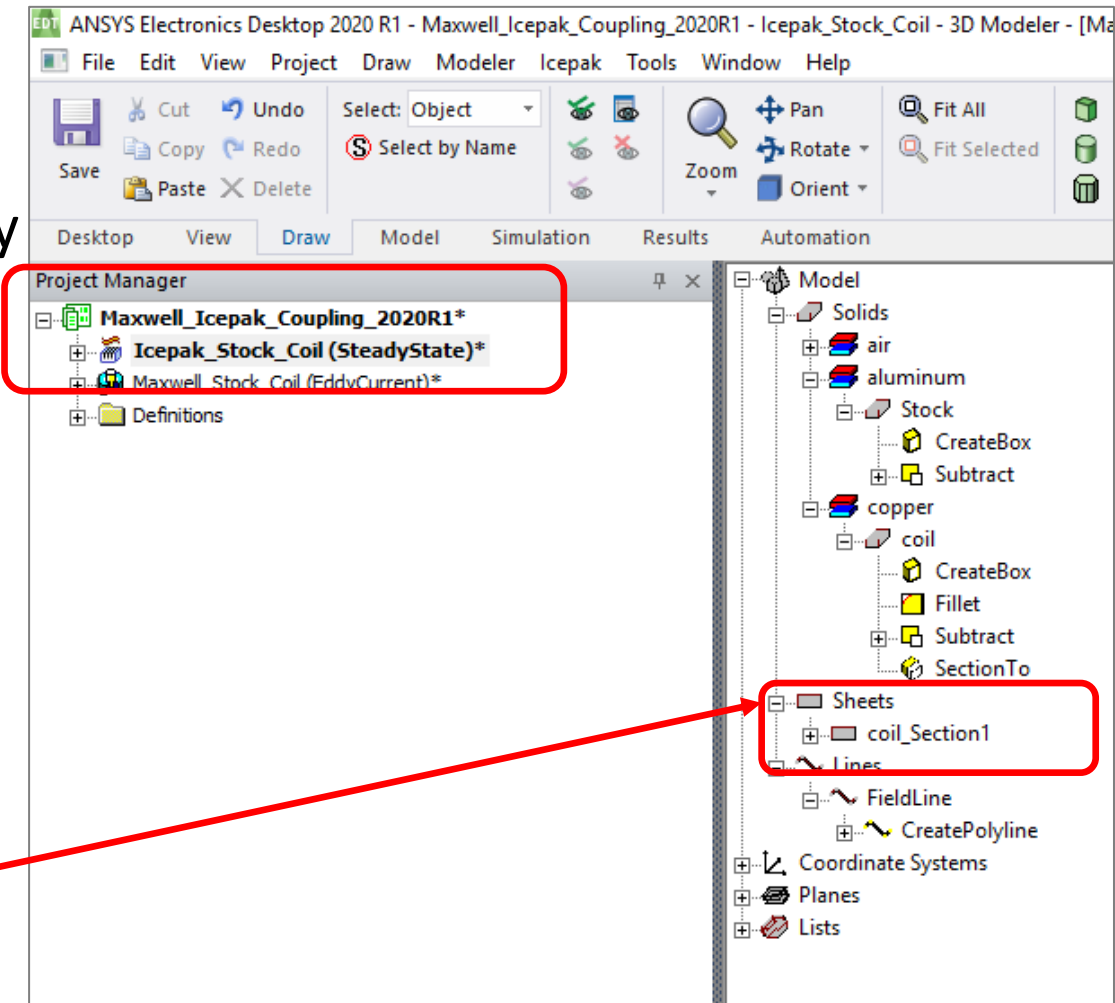
- Insert an Icepak Design from Desktop menu



Rename the Icepak Design as "Icepak\_Stock\_Coil"

# / Copy Maxwell model to Icepak

- Double click on 'Maxwell\_Stock\_Coil' design from Project Manager window.
- Use Ctrl A to select the entire Maxwell Geometry and Material information
- Use Ctrl C to copy the above information
- Double click on " Icepak\_Stock\_Coil"
- Use Ctrl V to paste the information in Icepak design and use Fit All option to see the model.
  - Note the automatic creation of the "air" region around the copied geometry
- Select unwanted bodies like Sheets, Lines which are not required for Thermal Analysis and right click to delete it.



# / Icepak : Solution Type & Design Settings..

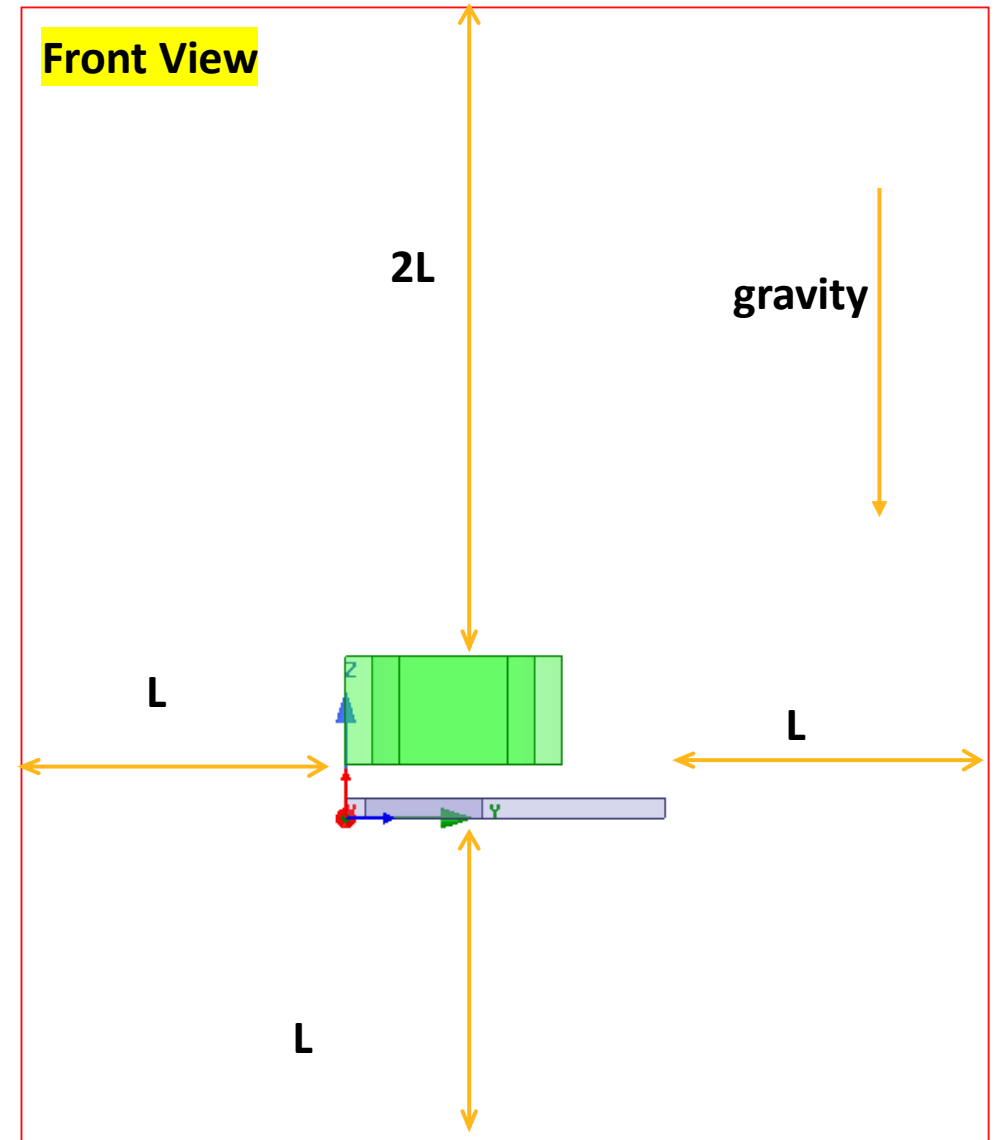
The image displays the ANSYS Icepak interface with several key components highlighted:

- Project Manager:** A context menu is open for the 'Icepak\_Stock' component. The 'Solution Type...' option is highlighted with a red box. The 'Design Settings...' option is also highlighted with a red box.
- Solution Type: Maxwell\_Icepak\_Coupling\_2...:** This dialog box shows the 'Solution types' section with 'Steady State' selected. The 'Problem types' section has 'Temperature and Flow' selected. A red box highlights the 'Solution types' and 'Problem types' sections.
- Icepak Design Settings:** The 'Ambient Conditions' tab is active. It shows fields for 'Temperature' (20 cel), 'Gauge Pressure' (0 n\_per\_meter\_sq), and 'Radiation temp' (20 cel). A red box highlights the 'Ambient Conditions' tab.
- Icepak Design Settings (Gravity):** The 'Gravity' tab is active. It shows the 'Gravity Vector' set to 'Global::Z' and the 'Negative' radio button selected. A red box highlights the 'Gravity' tab.
- 3D Model:** A 3D visualization of a component with a green wireframe mesh is shown on the right.

Note: Setting Gravity direction alone will not include the effect of gravity. The gravity needs to be included in Setup panel which is shown later.

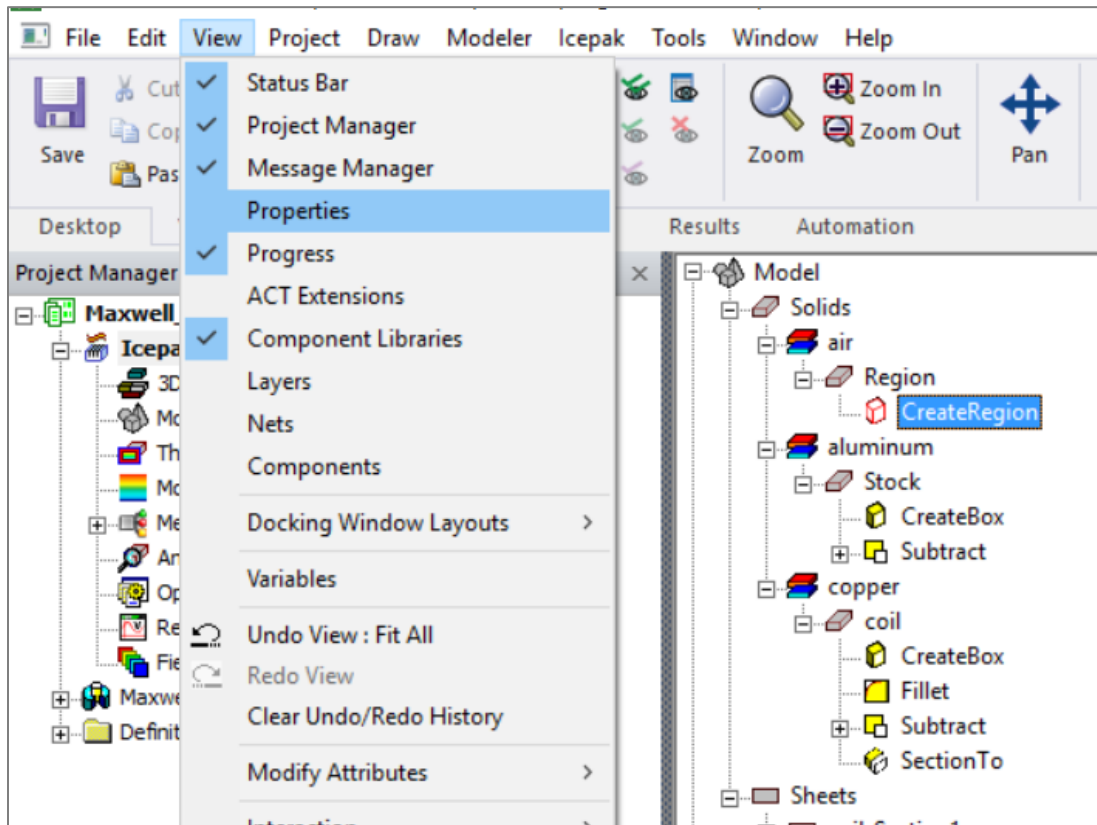
# / Icepak : Region Resizing

- The region needs to be sufficiently large so the boundary conditions do not affect the results – gradients of variables normal to the boundaries should be small.
- Rules of thumb :
  - Top (against gravity): at least  $2L$  to capture the plume accurately
  - Sides: at least  $\frac{1}{2} L$
  - Below: at least  $L$
  - Where  $L$  is the largest dimension of the unit in 3 directions.



# / Icepak : Region sizing

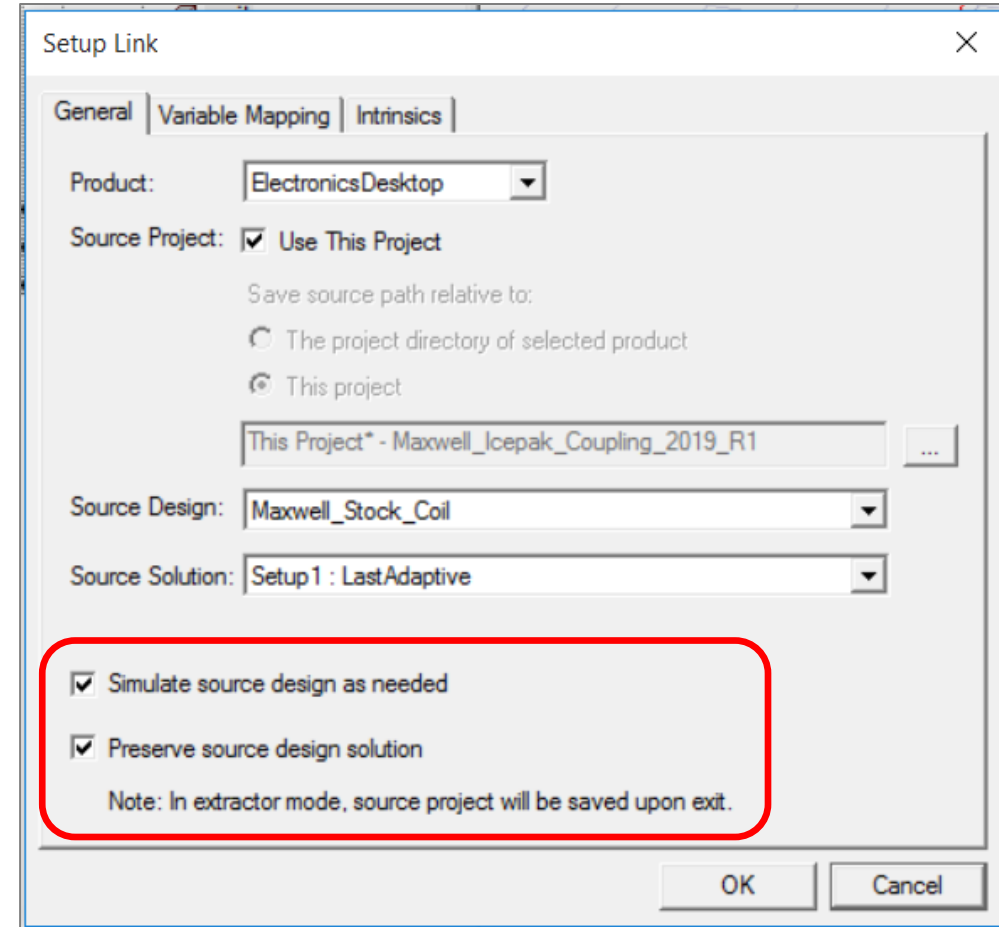
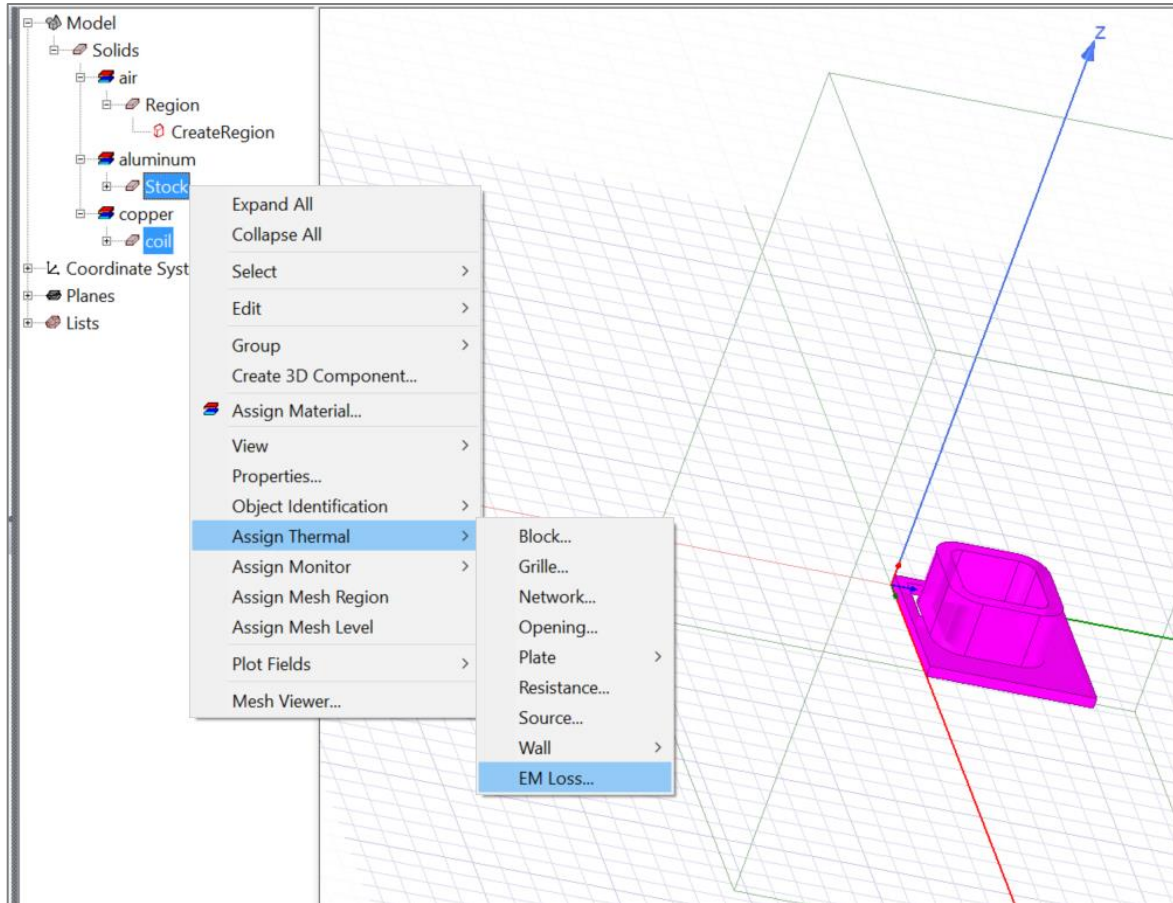
1. Select Create Region and use Properties option from View menu to open Properties panel
2. Change the Padding Type to Absolute Offset and use 600 mm for positive Z and 300 mm for other directions.



Properties			
Name	Value	Unit	Evaluated Value
Command	CreateRegion		
Coordinate...	Global		
+X Paddin...	Absolute Offset		
+X Paddin...	300	mm	300mm
-X Paddin...	Absolute Offset		
-X Paddin...	300	mm	300mm
+Y Paddin...	Absolute Offset		
+Y Paddin...	300	mm	300mm
-Y Paddin...	Absolute Offset		
-Y Paddin...	300	mm	300mm
+Z Paddin...	Absolute Offset		
+Z Paddin...	600	mm	600mm
-Z Paddin...	Absolute Offset		
-Z Paddin...	300	mm	300mm

# / Icepak : EM Mapping

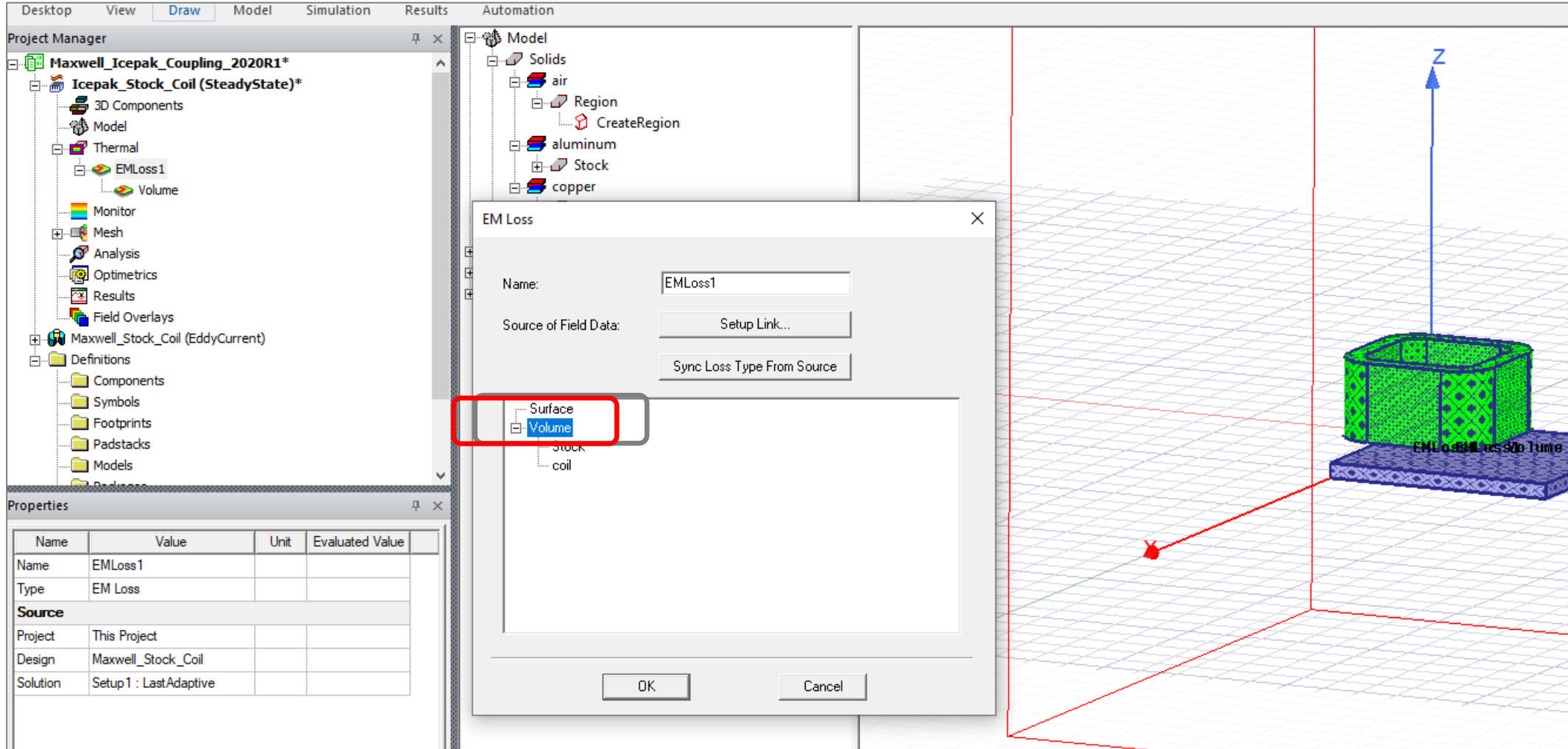
- Select Coil and Stock from GUI and right click to Assign Thermal → EM Loss
- Select 'Use This Project' and 'Simulate source design as needed' in setup link panel





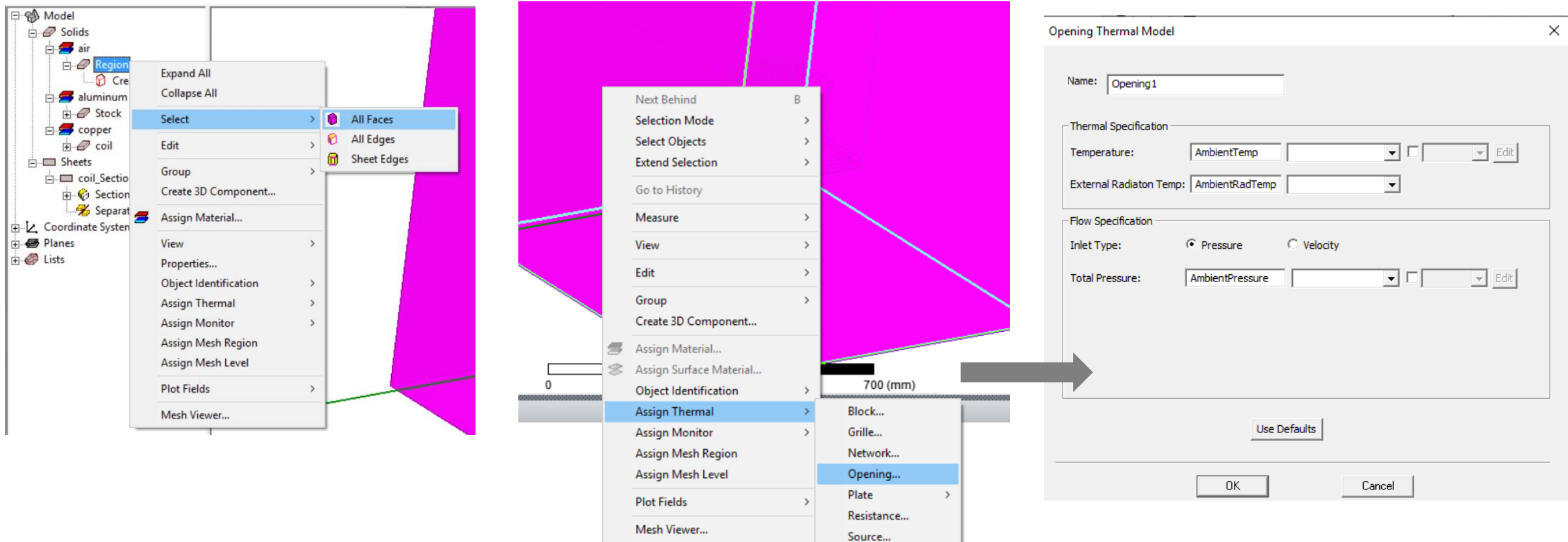
# / Icepak : EM Mapping

## Set up the Electro-Thermal link to map Volume EM losses on the Stock and Coil

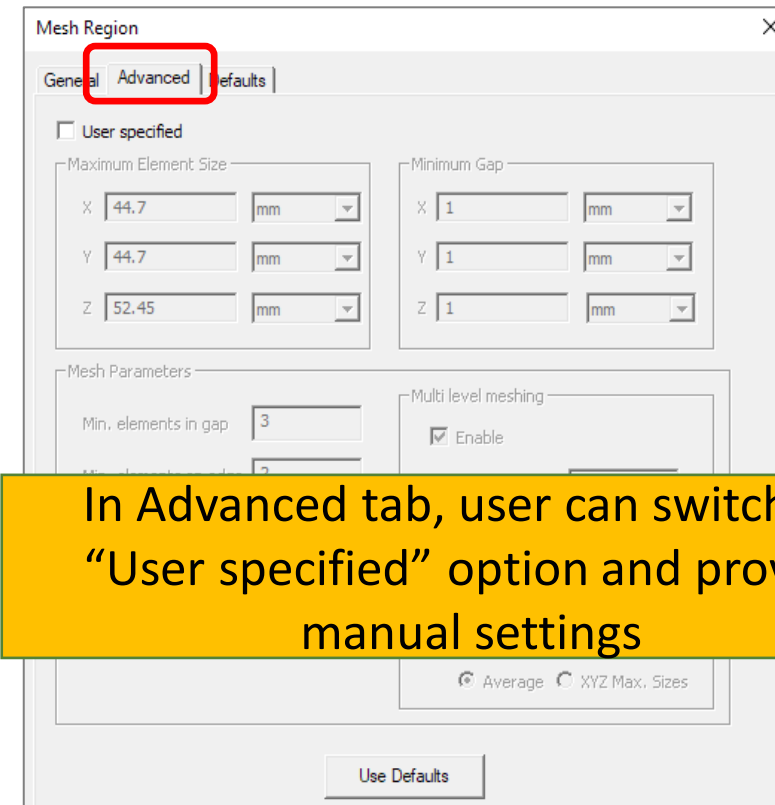
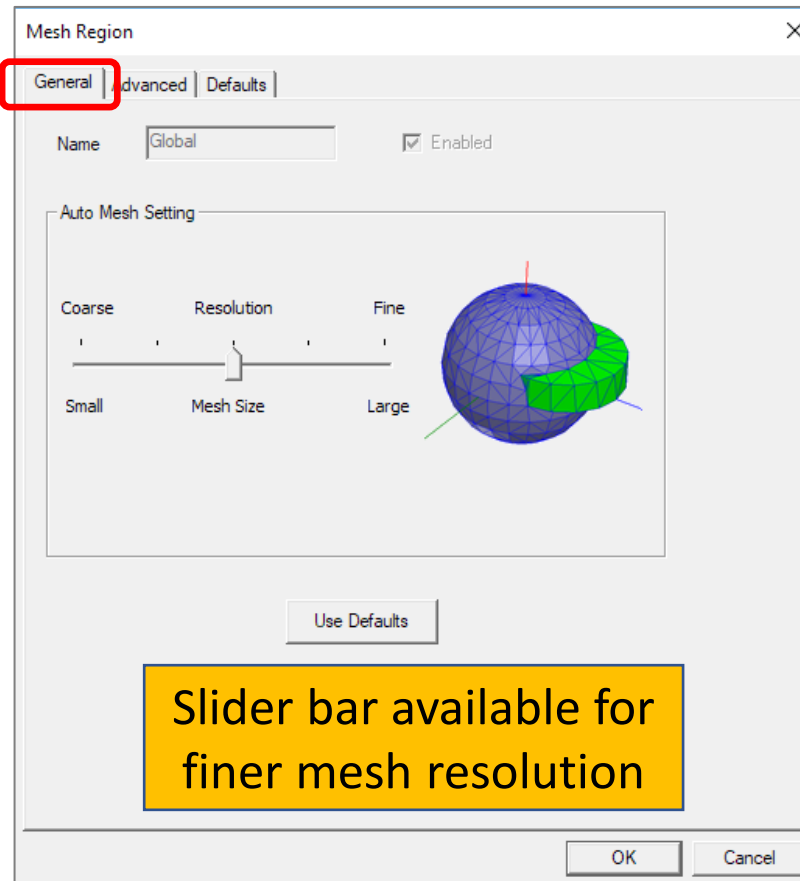
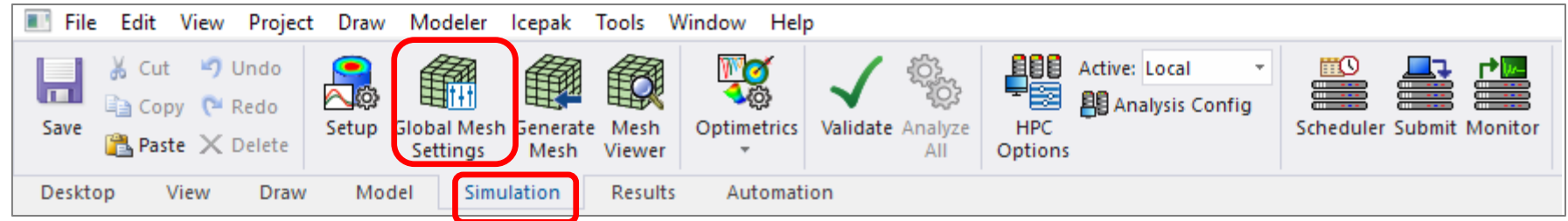


# / Icepak : Create Openings for the Region object

- Select object “Region” and right click and select → All Faces
- Right click on region face in GUI → Assign Thermal → Opening
- Keep default boundary conditions

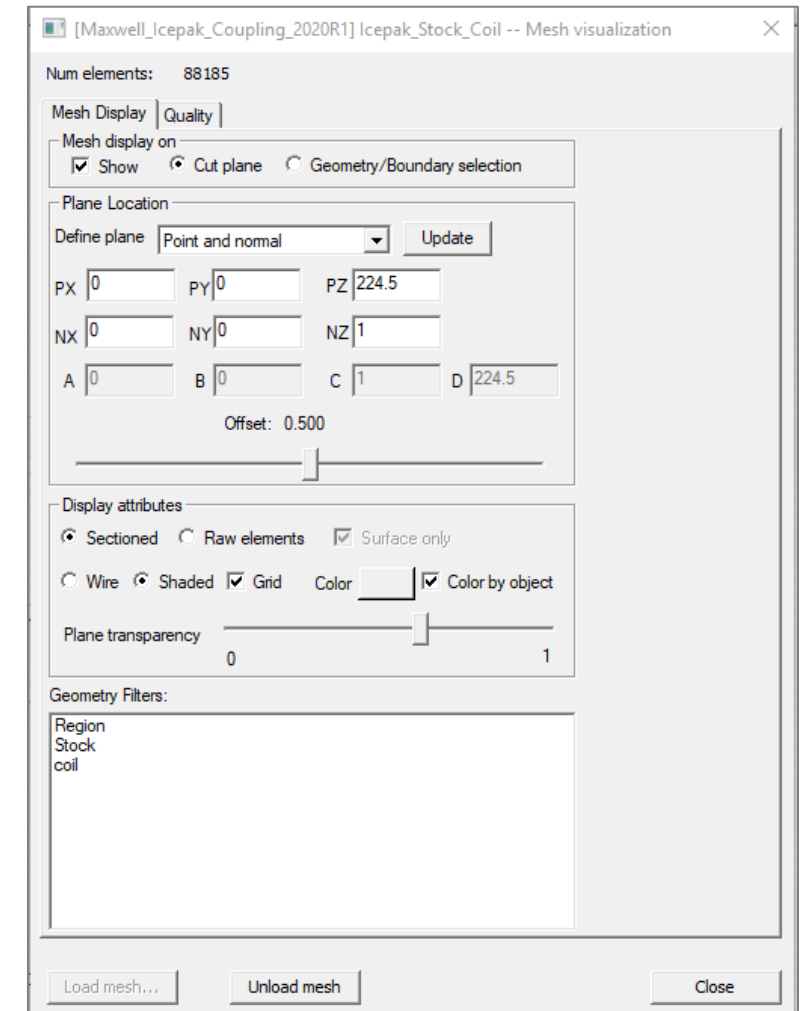
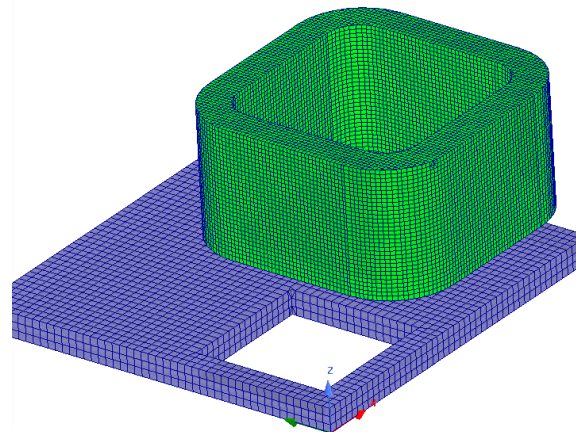
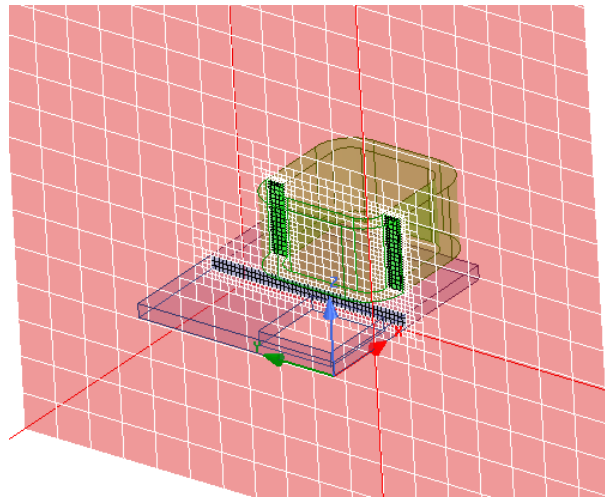
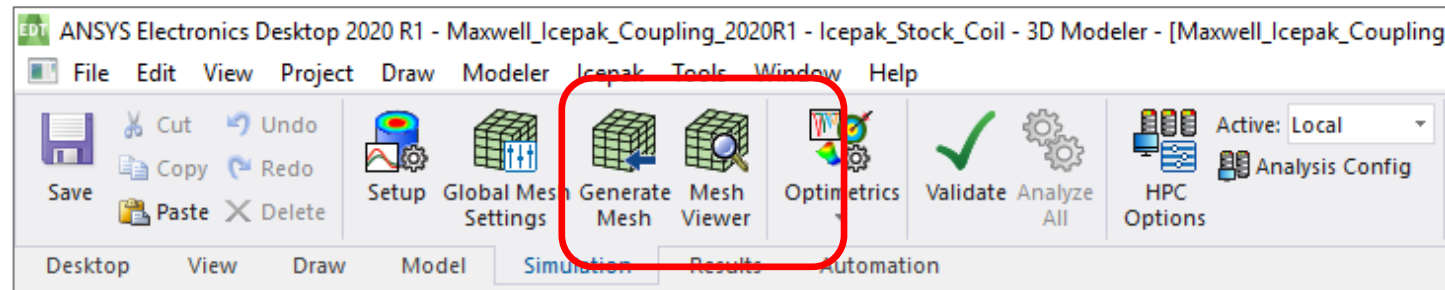


# / Icepak : Mesh



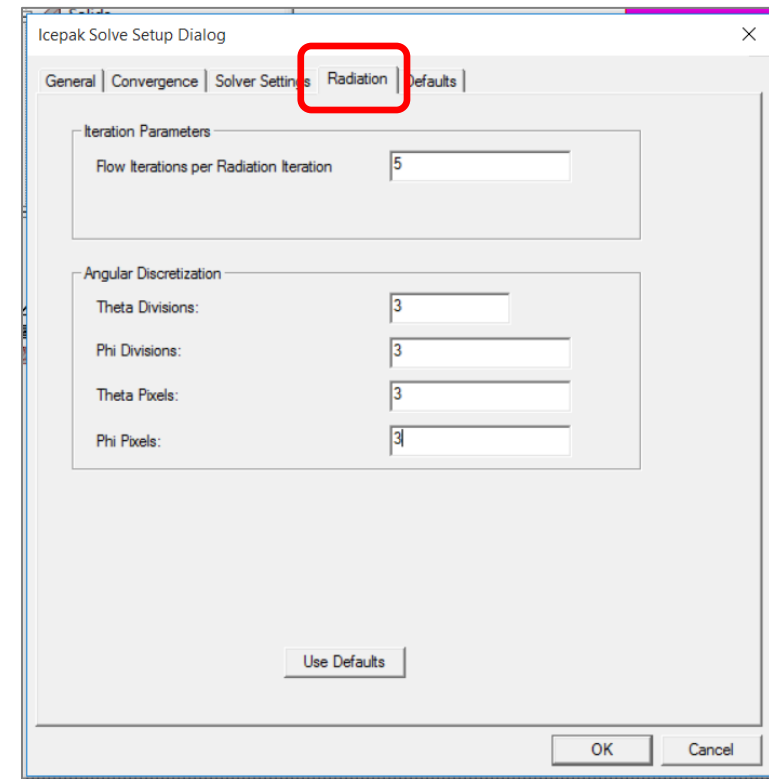
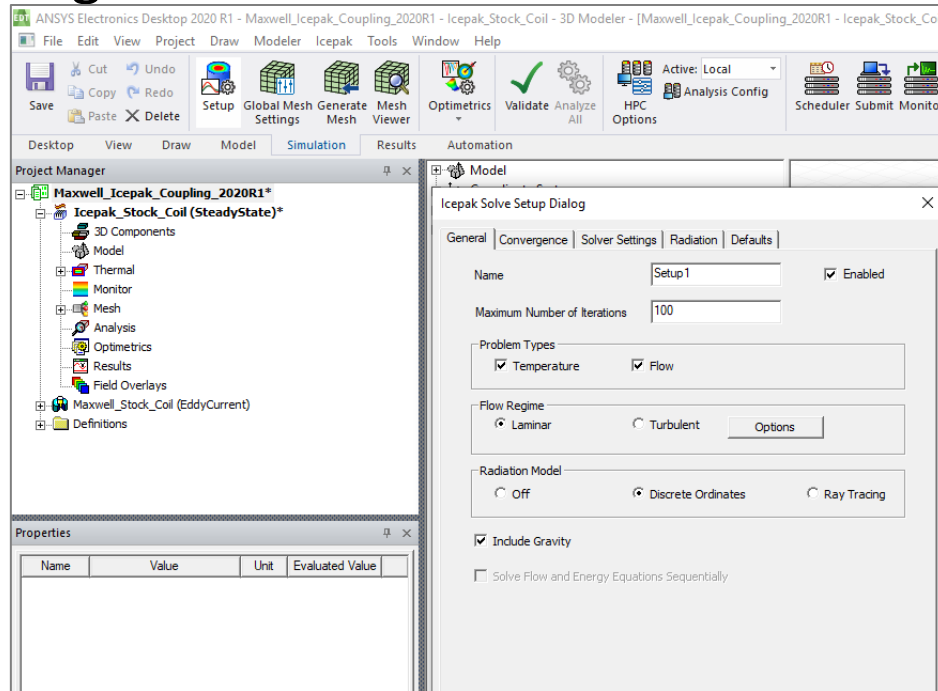
# / Icepak : Generate and View Mesh

Generate Mesh and Click Mesh Viewer to review the mesh on planes and individual bodies



# / Icepak : Setup the Model for Thermal simulation

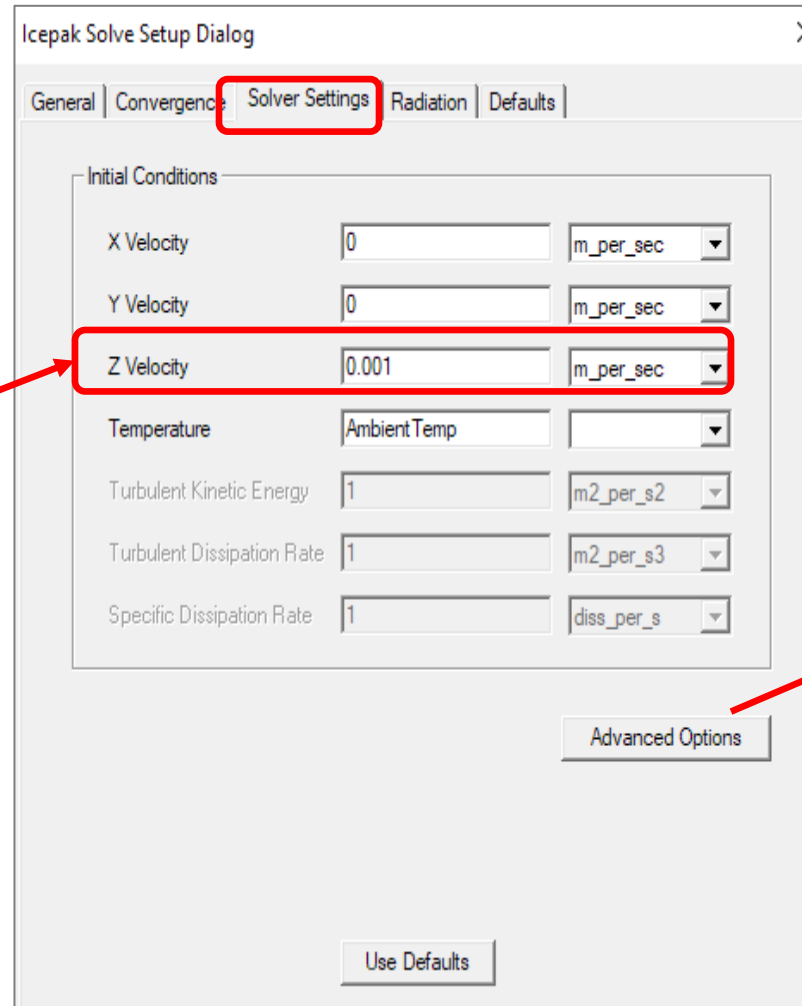
1. Click on 'Setup' to open Solve setup Panel.
2. In 'General' Tab:
  - Select Problem Types, Flow Regime and
  - Turn on DO Radiation Model and 'Include Gravity' option
3. Use finer settings for DO radiation model



# / Icepak : Additional Setup

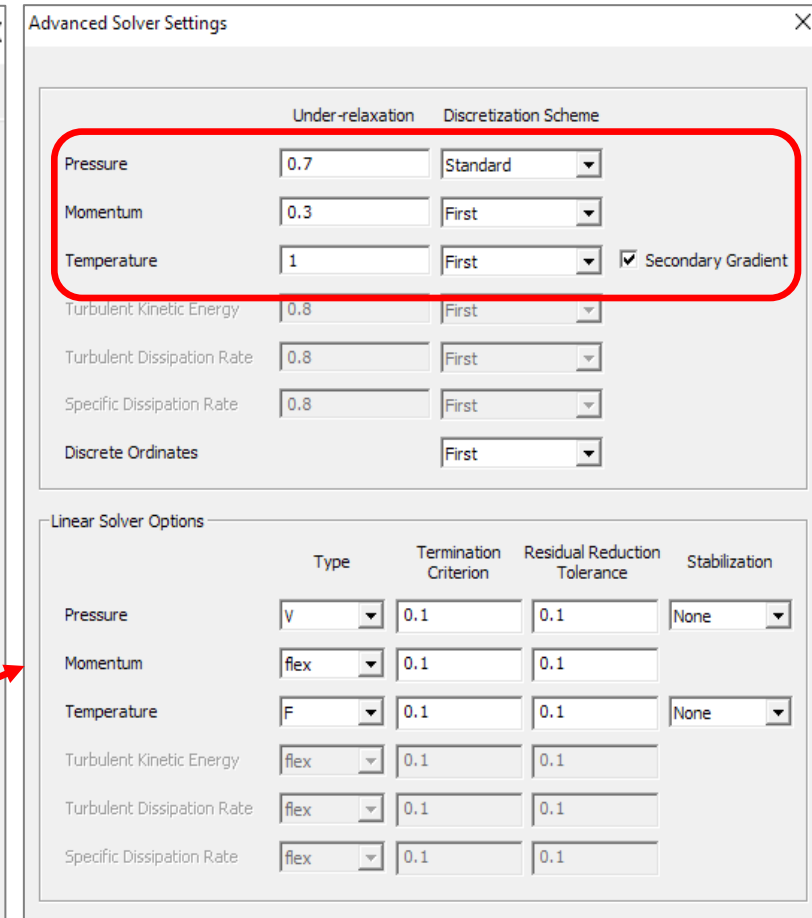
- Some additional settings for helping solver converge natural convection problems

Recommended to provide small velocity in flow direction (due to natural convection plumes will rise opposite to gravity )



The 'Icepak Solve Setup Dialog' is shown with the 'Solver Settings' tab selected. The 'Initial Conditions' section contains several input fields. A red box highlights the 'Z Velocity' field, which is set to '0.001' with units of 'm\_per\_sec'. A red arrow points from the yellow text box to this field. Below the 'Initial Conditions' section is an 'Advanced Options' button, also highlighted with a red arrow. At the bottom of the dialog is a 'Use Defaults' button.

Initial Conditions		
X Velocity	0	m_per_sec
Y Velocity	0	m_per_sec
Z Velocity	0.001	m_per_sec
Temperature	Ambient Temp	
Turbulent Kinetic Energy	1	m2_per_s2
Turbulent Dissipation Rate	1	m2_per_s3
Specific Dissipation Rate	1	diss_per_s



The 'Advanced Solver Settings' dialog is shown. A red box highlights the 'Under-relaxation' section, which includes settings for Pressure (0.7), Momentum (0.3), and Temperature (1). The 'Discretization Scheme' for Temperature is set to 'First', and the 'Secondary Gradient' checkbox is checked. Below this is the 'Linear Solver Options' section, which includes settings for various variables like Pressure, Momentum, Temperature, and Turbulent Kinetic Energy, with columns for Type, Termination Criterion, Residual Reduction Tolerance, and Stabilization.

	Under-relaxation	Discretization Scheme
Pressure	0.7	Standard
Momentum	0.3	First
Temperature	1	First
Turbulent Kinetic Energy	0.8	First
Turbulent Dissipation Rate	0.8	First
Specific Dissipation Rate	0.8	First
Discrete Ordinates		First

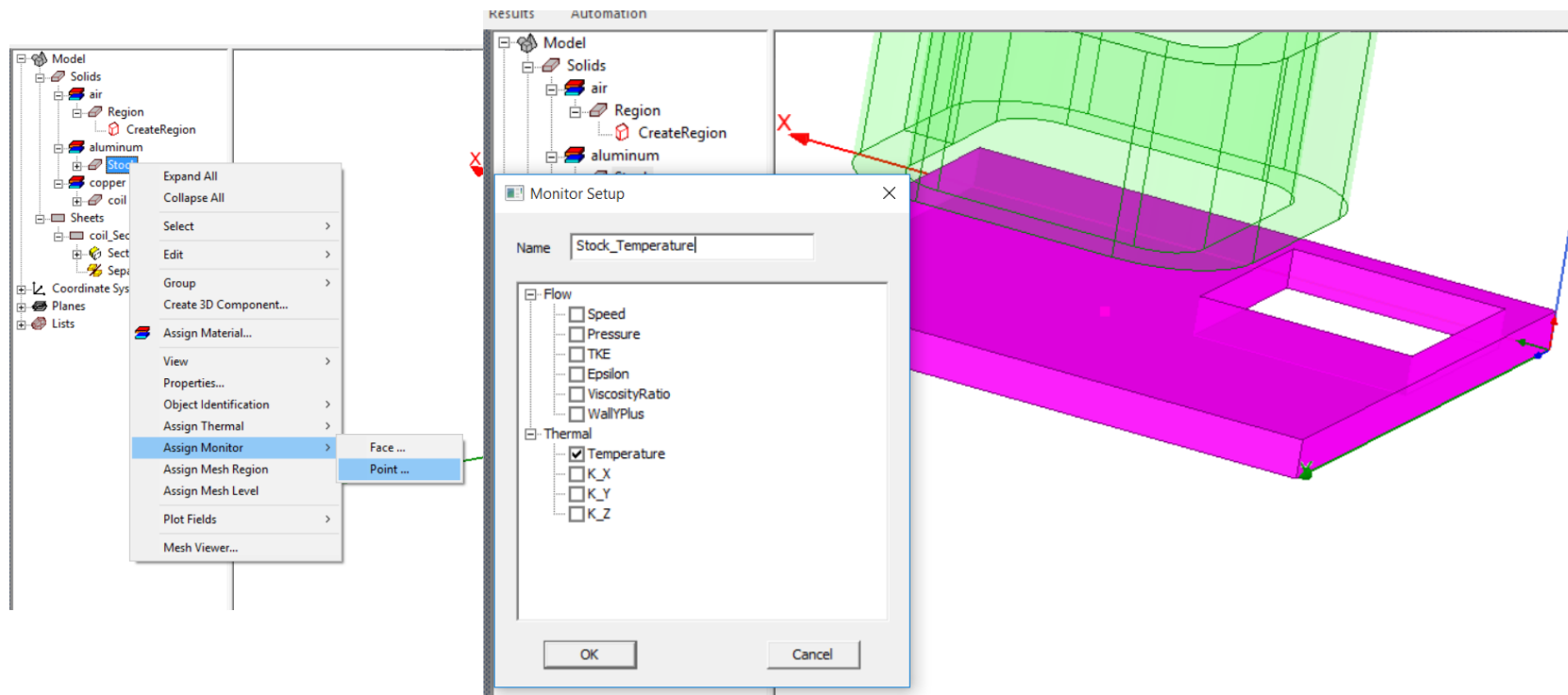
  

	Type	Termination Criterion	Residual Reduction Tolerance	Stabilization
Pressure	V	0.1	0.1	None
Momentum	flex	0.1	0.1	
Temperature	F	0.1	0.1	None
Turbulent Kinetic Energy	flex	0.1	0.1	
Turbulent Dissipation Rate	flex	0.1	0.1	
Specific Dissipation Rate	flex	0.1	0.1	



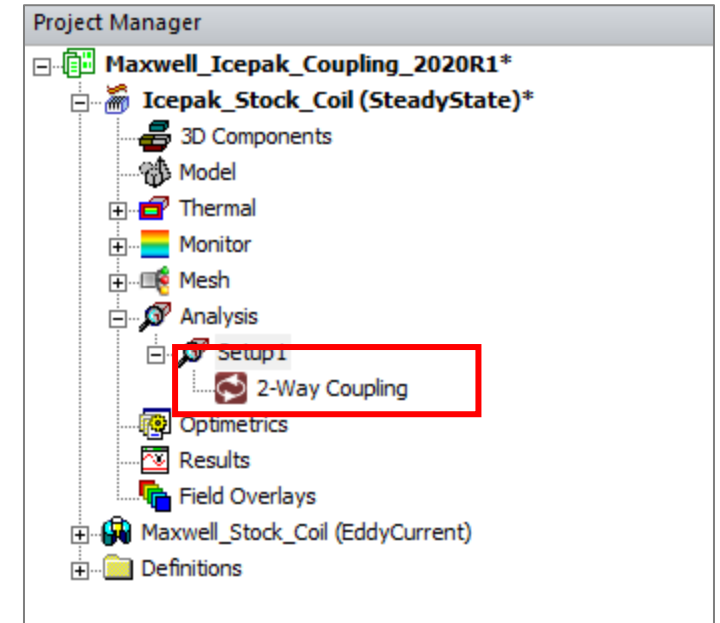
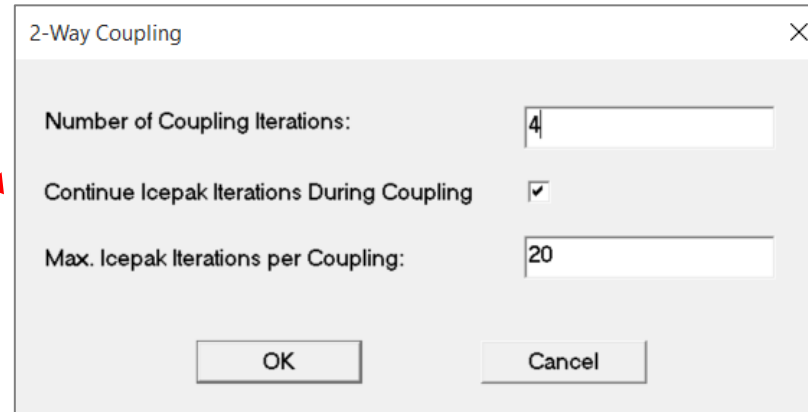
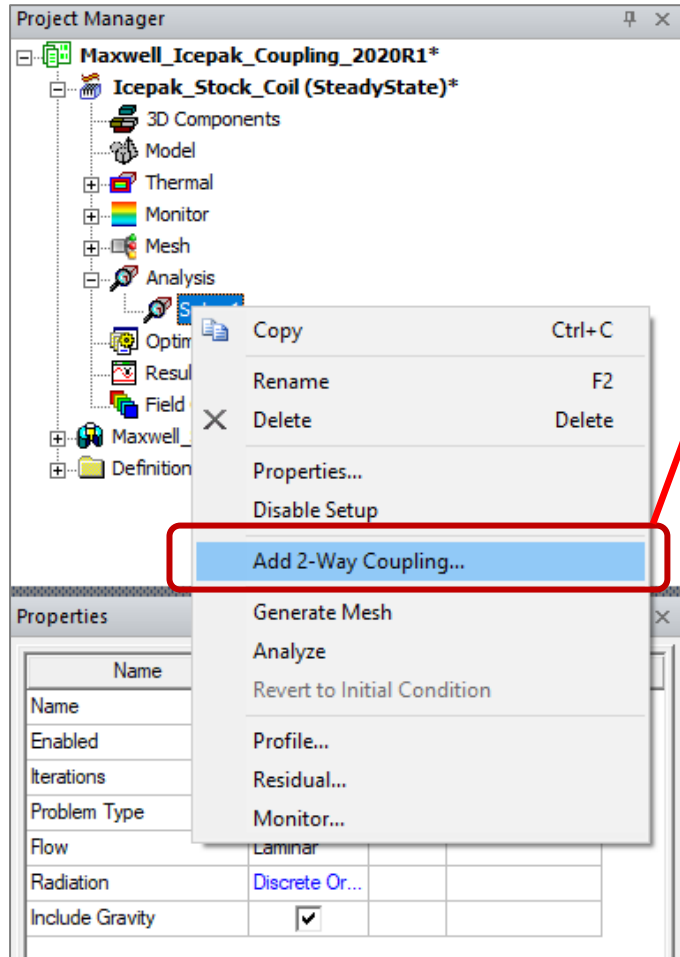
# / Icepak : Assign Monitor Point

- Select Stock and right click Assign Monitor → Point...
- Rename monitor name as Stock\_Temperature
- Select variables of interest Thermal > Temperature
- Note the monitor point location at the centre of stock



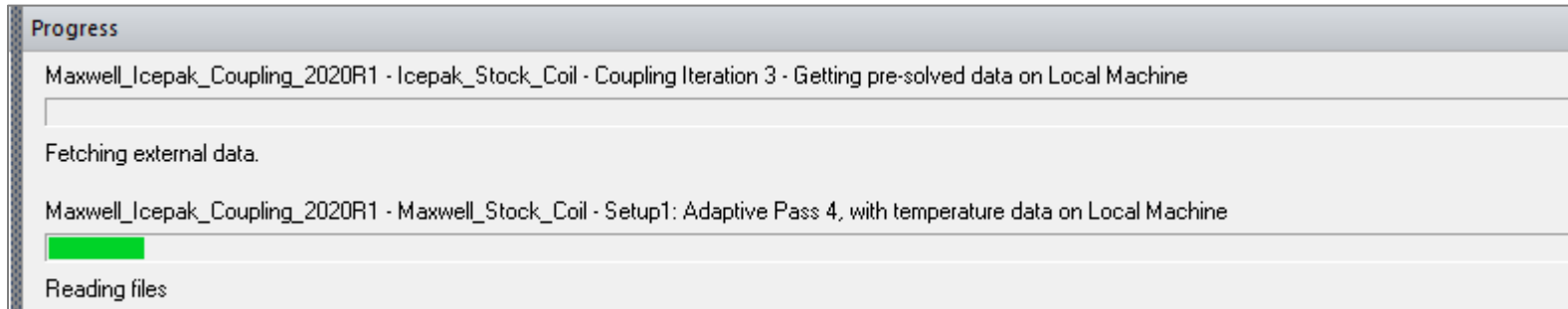
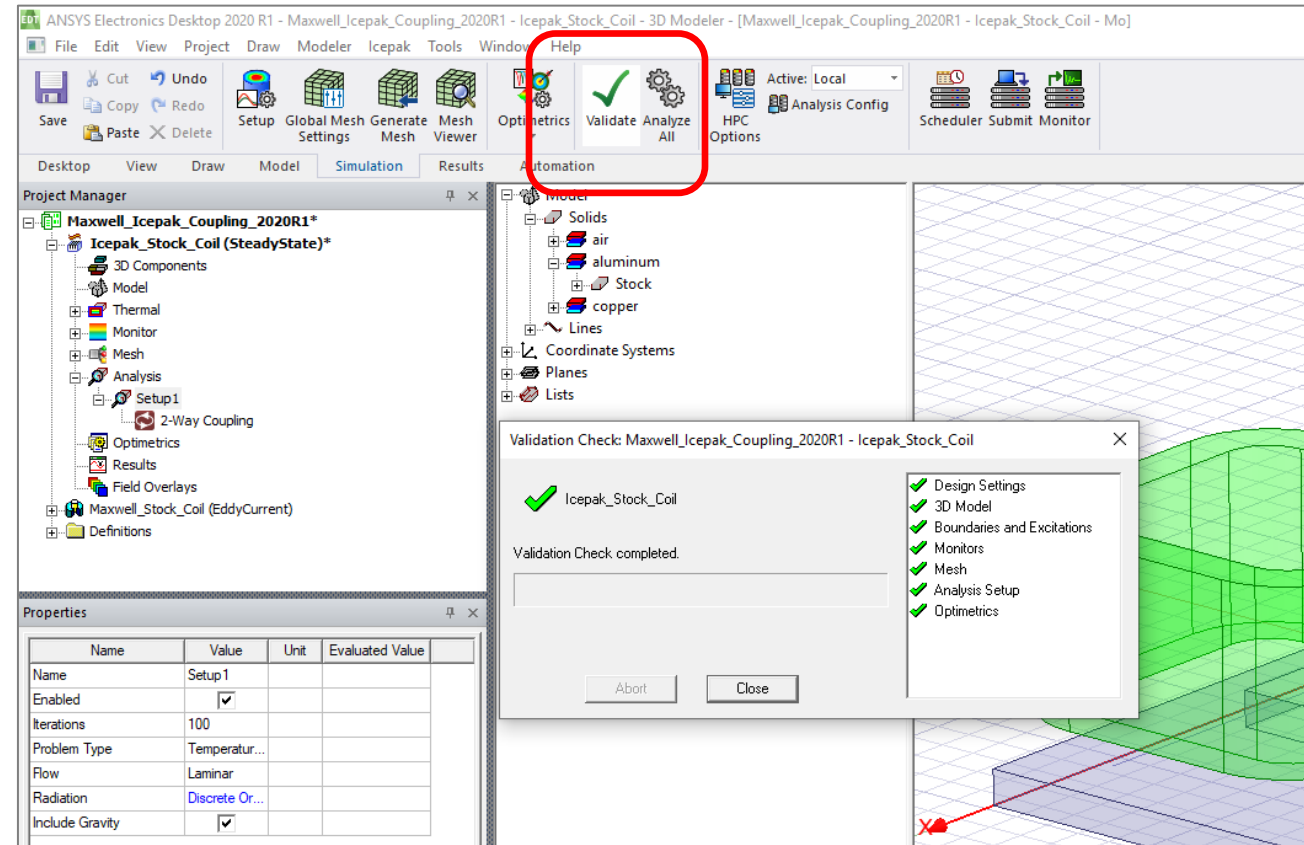
# 2-Way Coupling

Right click on Setup1 and select 'Add 2-Way Coupling'

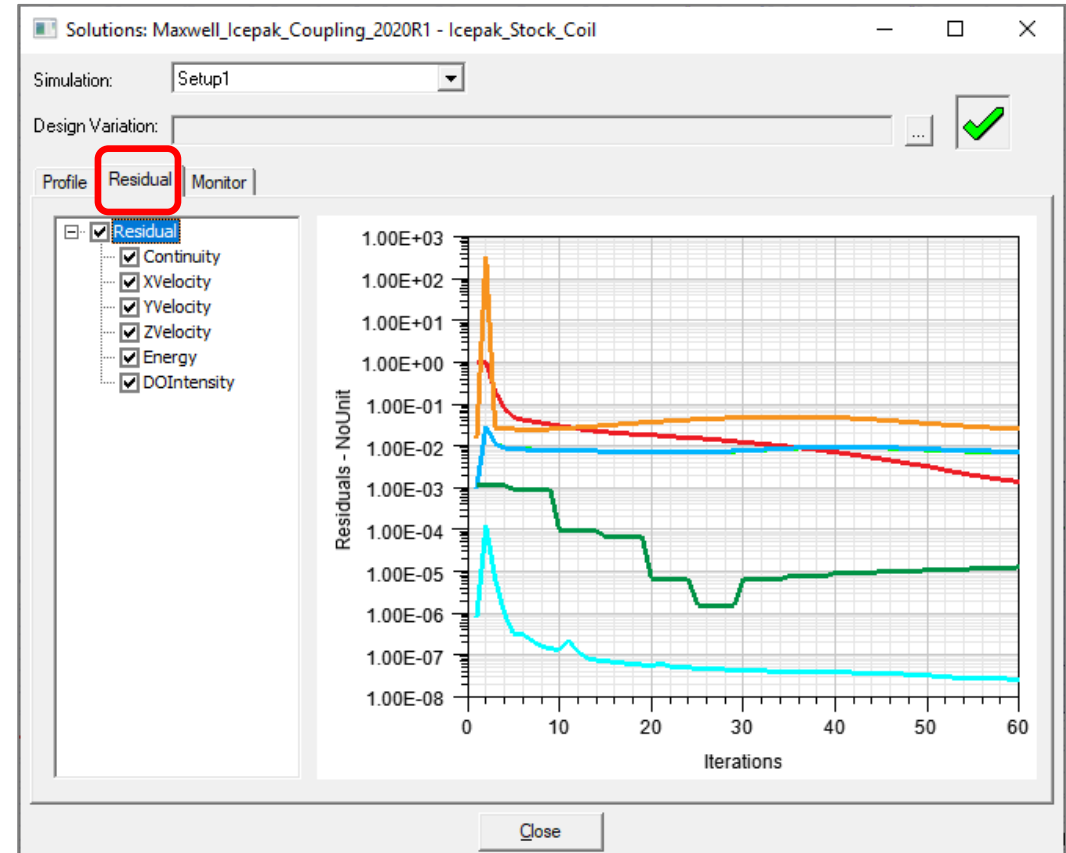
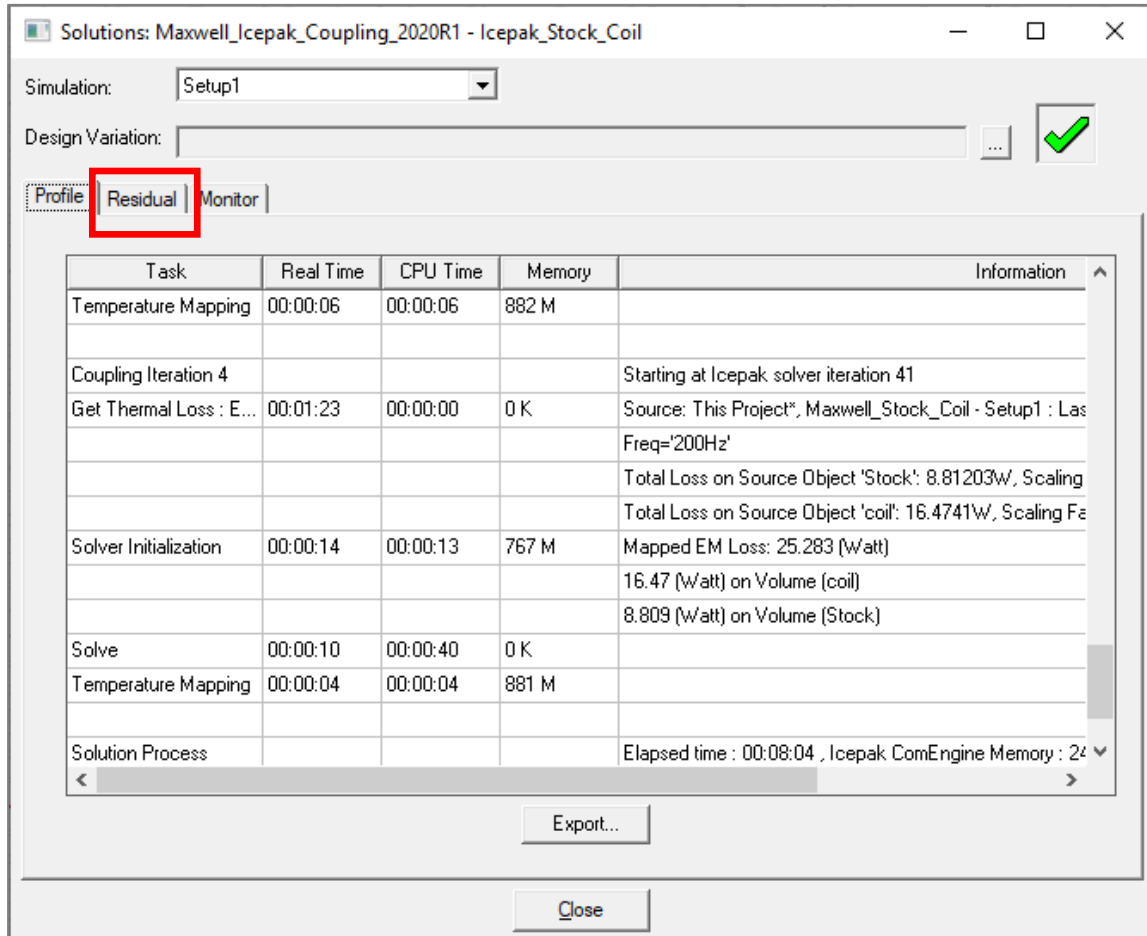
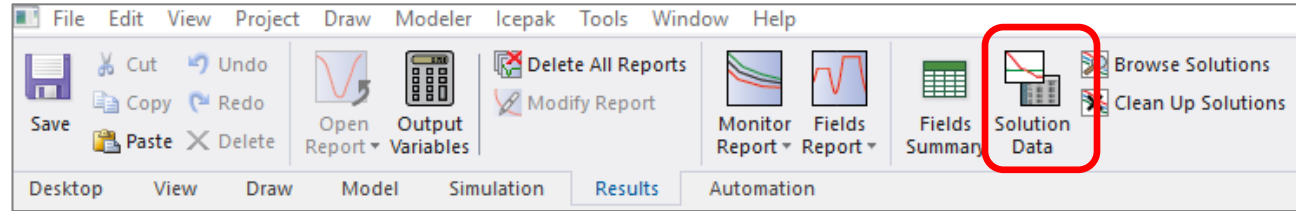


# / Icepak : Validation and Analyze

- Validation ensures all the elements of the setup are ready and good to solve!
- Click *Analyze All* to start the solver for 2-way coupling



# Icepak : Results



# / Icepak : Results – Heat Balance Check

## Heat Flow Rate through Opening1 Boundary and Coil and Stock

The screenshot shows the ANSYS Icepak interface with the following components:

- Fields Summary Window:** Displays a table of heat flow rates for three entities: Opening1, coil, and stock. The table has columns for Entity Type, Geometry Type, Entity, Quantity, Side, Normal, Area/Volume, and Total.
- Setup Calculation Window (Left):** Shows the selection of 'Adjacent side' geometry type and the 'HeatFlowRate' quantity.
- Setup Calculation Window (Right):** Shows the selection of 'Adjacent side' geometry type and the 'HeatFlowRate' quantity.

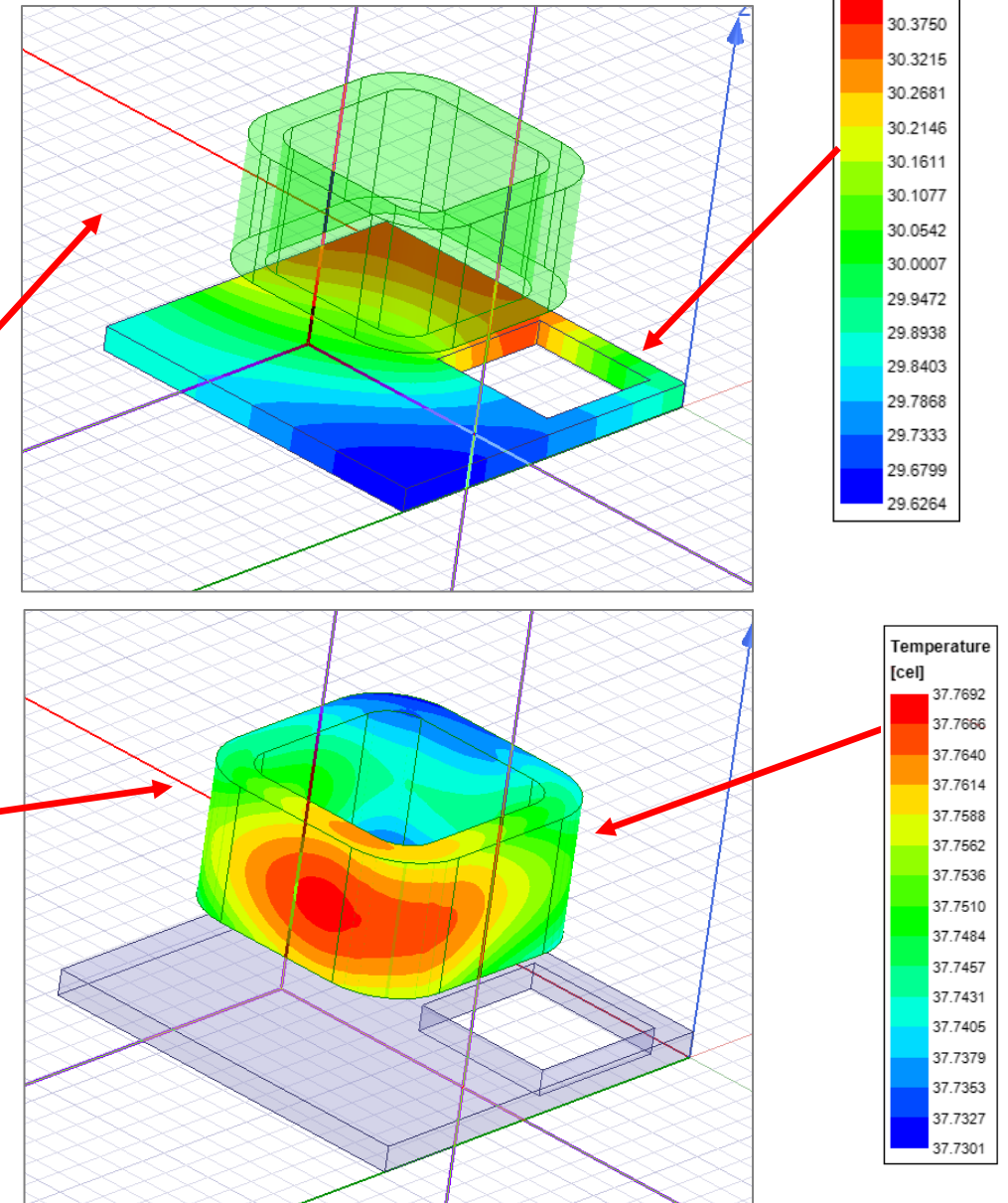
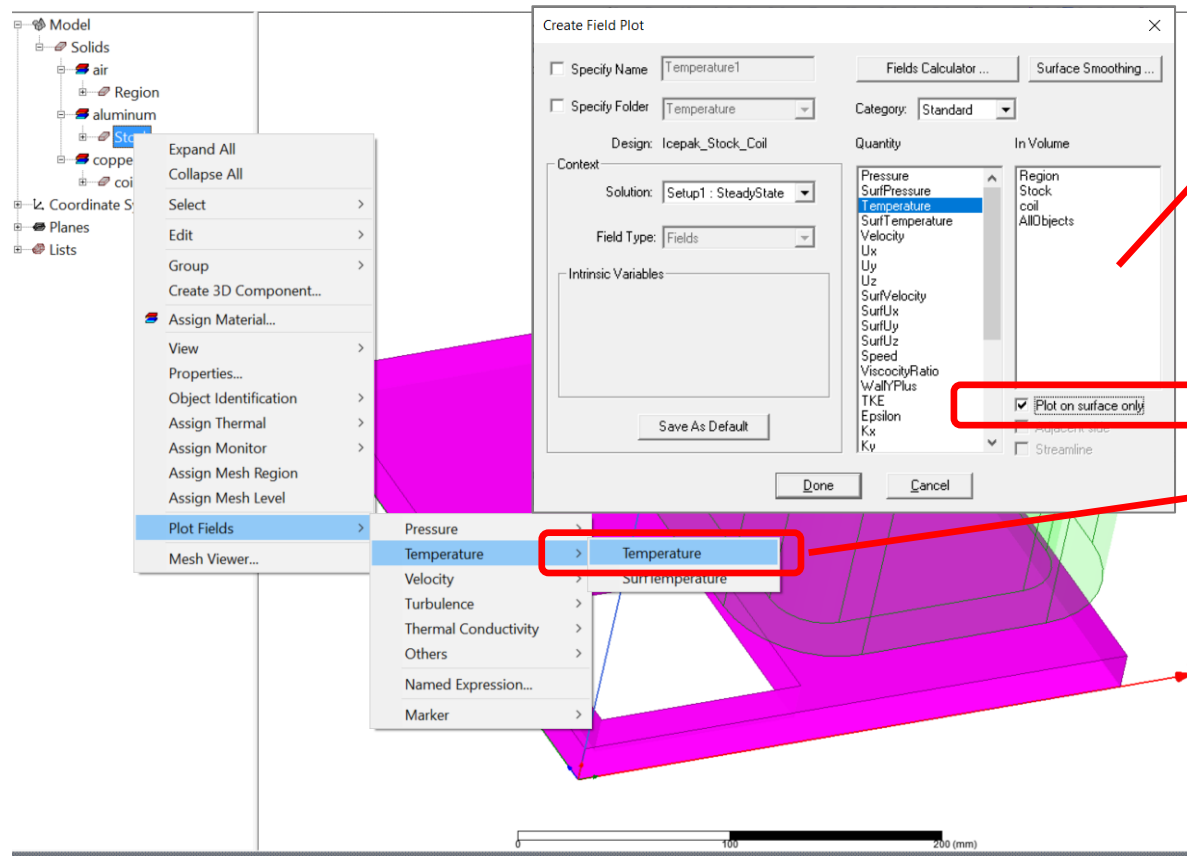
Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Area/Volume	Total
Boundary	Surface	Opening1	HeatFlowRate[W]	Adjacent		5.3497 m <sup>2</sup>	-25.2904
Object	Surface	coil	HeatFlowRate[W]	Adjacent		0.158689 m <sup>2</sup>	-16.4592
Object	Surface	Stock	HeatFlowRate[W]	Adjacent		0.180096 m <sup>2</sup>	-8.82914

Red boxes highlight the 'Fields Summary' button in the top toolbar, the 'Adjacent side' checkbox in both Setup Calculation windows, and the 'Add' button in the left Setup Calculation window. A red arrow points from the 'Total' column of the table to the yellow text box below.

Total heat dissipated out from the openings should match the Ohmic loss generated by Coil & Stock

# / Icepak : Results - Contour Field

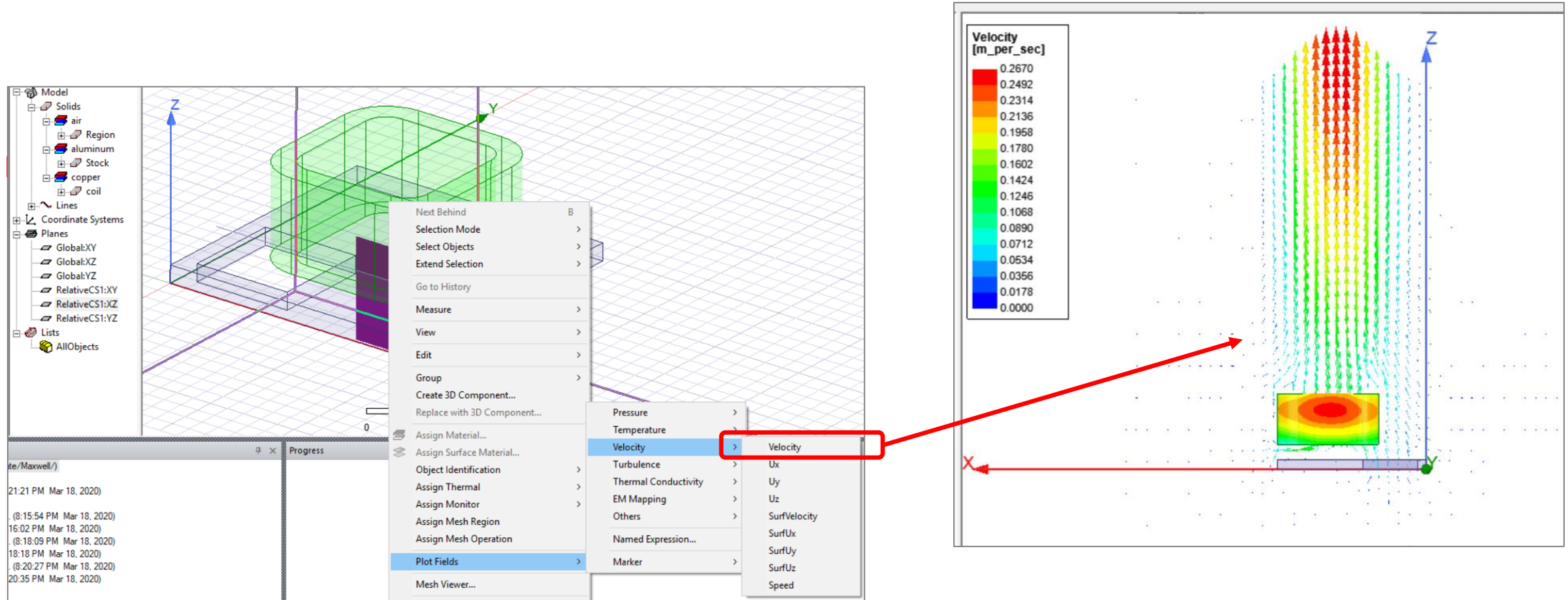
1. Select Coil from Model tree and plot temperature, make sure that check **Plot on surface only**
2. Similarly plot temperature for Stock





# / Icepak : Results Vector Field

- Select Plane RelativeCS1:XZ and right click to Plot Velocity Field







**End of presentation**