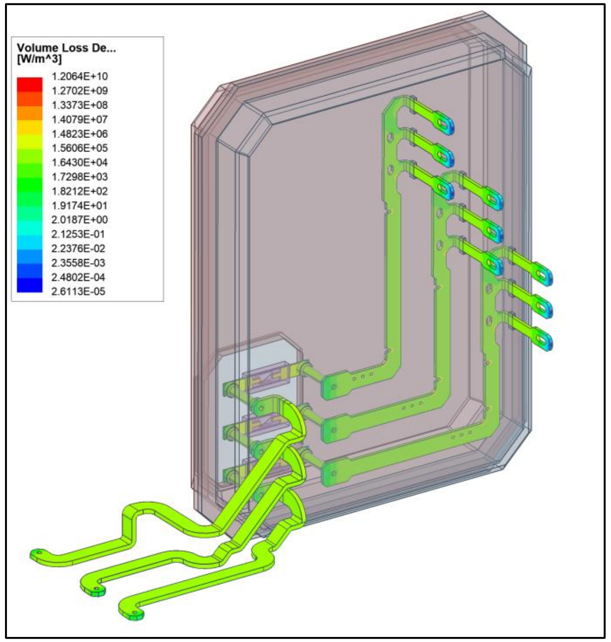
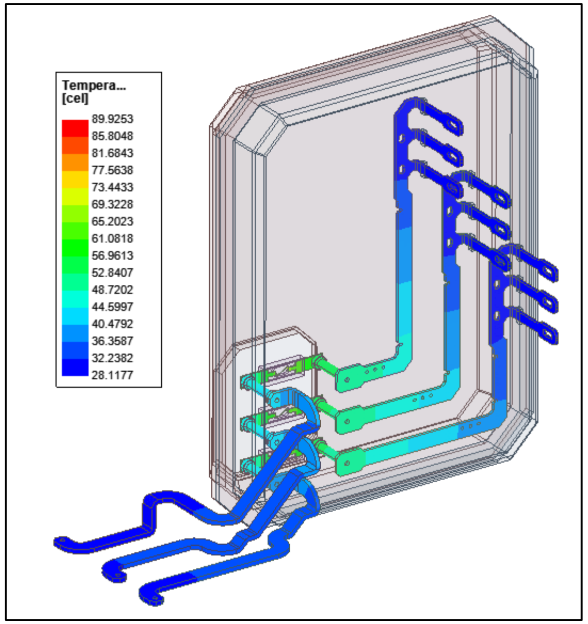
Module 4 – Workshop 3: Q3D and Icepak Multiphysics in AEDT

Release 2020 R1

Q3D Extractor and Icepak

- **ANSYS Q3D Extractor**
  - **ANSYS Q3D Extractor characterizes three-dimensional interconnect structures such as those found in connectors, Printed Circuit Boards (PCBs), Ball Grid Arrays (BGAs), and Multi-Chip Modules (MCMs).**

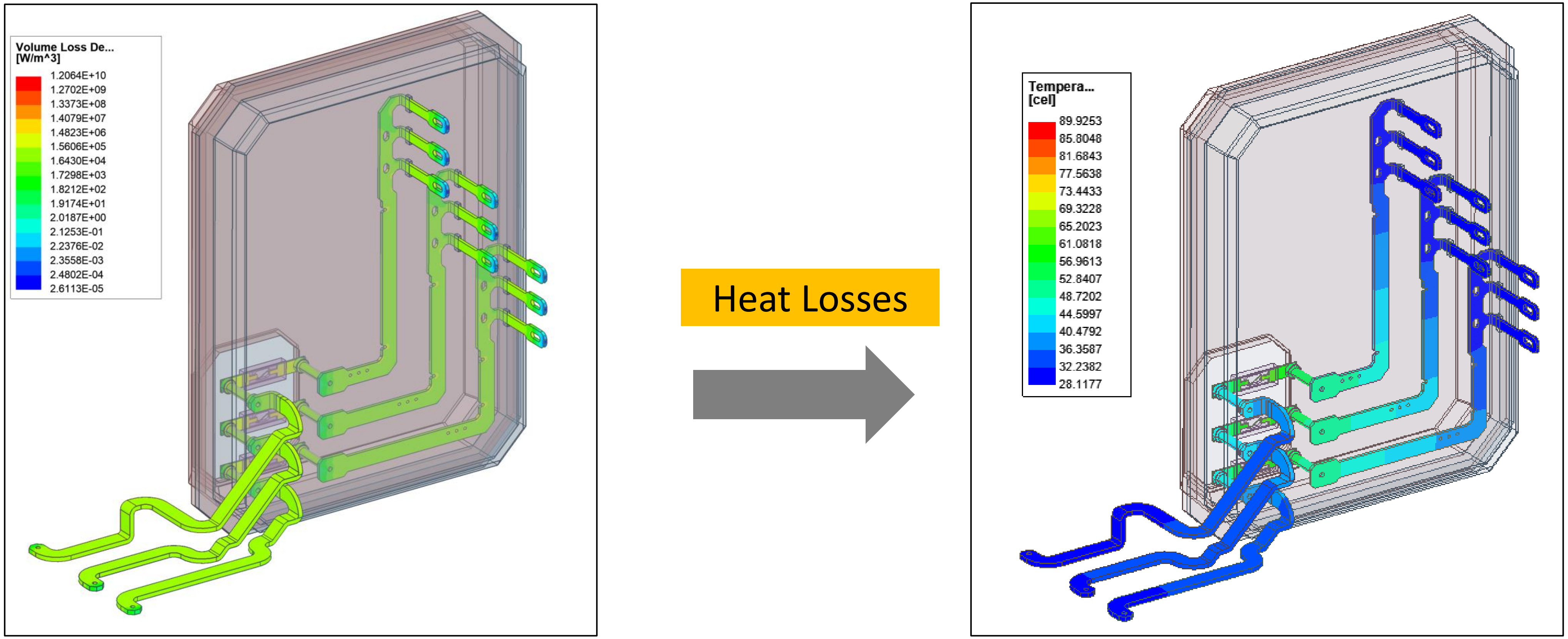




- **ANSYS Icepak**
  - **ANSYS Icepak is an integrated electronics cooling solution for IC packages, printed circuit boards and complete electronic systems.**

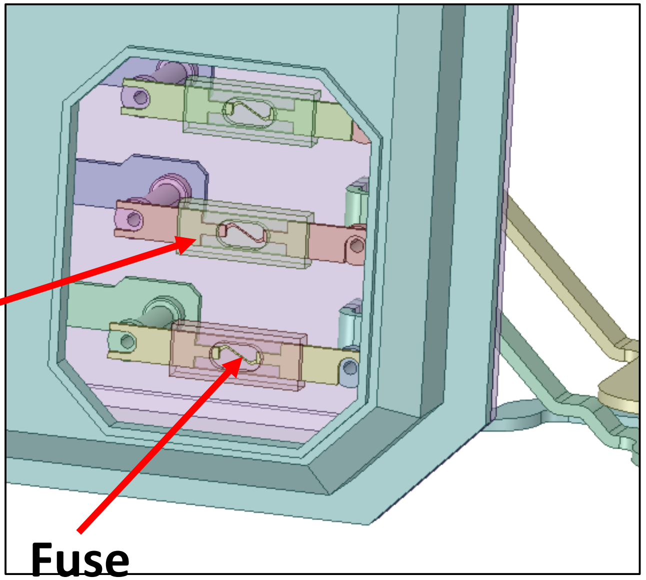
Q3D Icepak Coupling

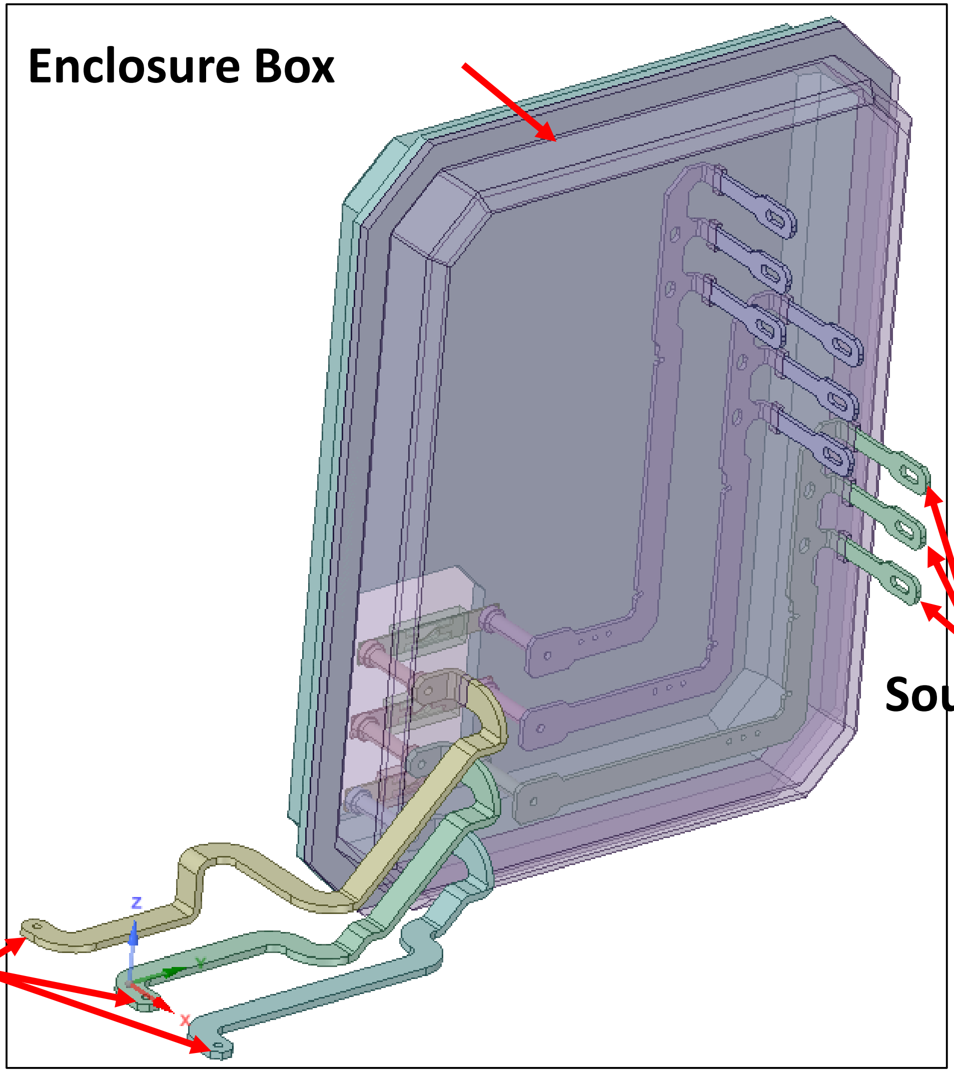
- **Q3D calculates both DC and AC Losses.**
- **The losses are mapped to Icepak. Both surface and volumetric mapping are supported.**



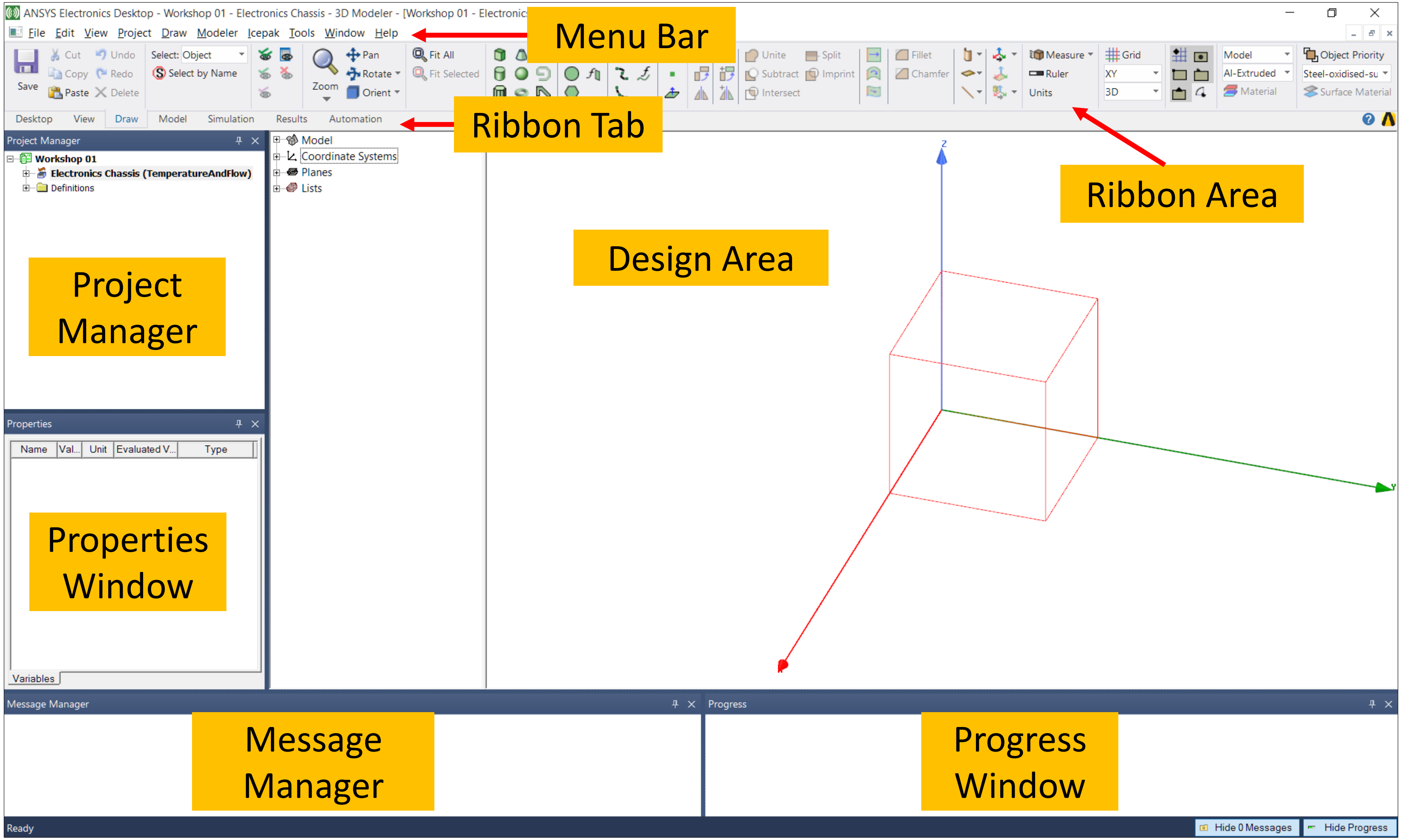
Busbar

- **Objective: Calculate the temperatures of a busbar and its enclosure box**
- **Analysis:**
  - A DC RL Analysis is performed in Q3D
  - Q3D is used to calculate the Ohmic losses in metallic parts
  - Thermal analysis with Natural Convection is performed in ANSYS Icepak



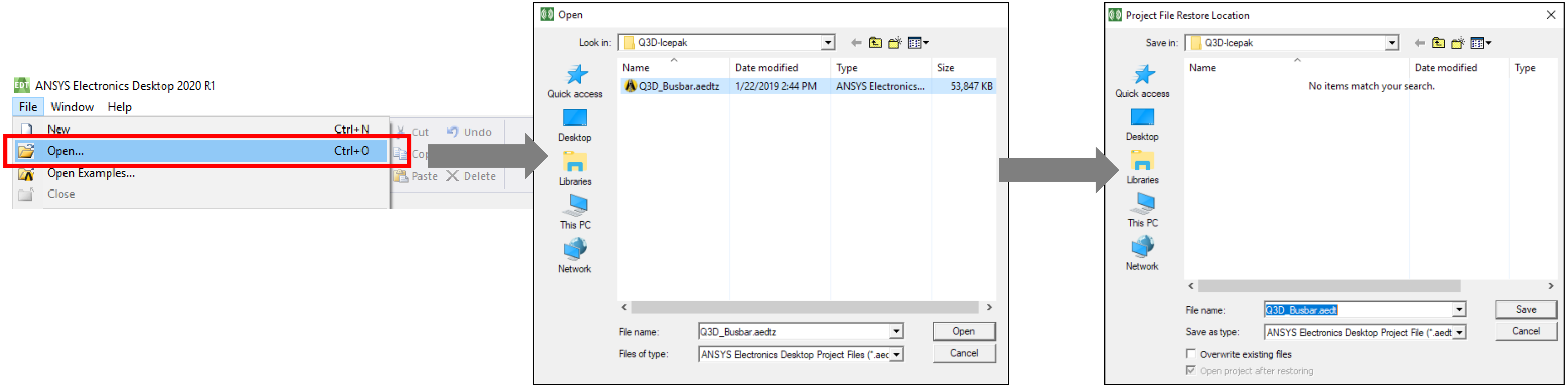


AEDT Interface



Import an Existing Q3D Project into AEDT 2020 R1

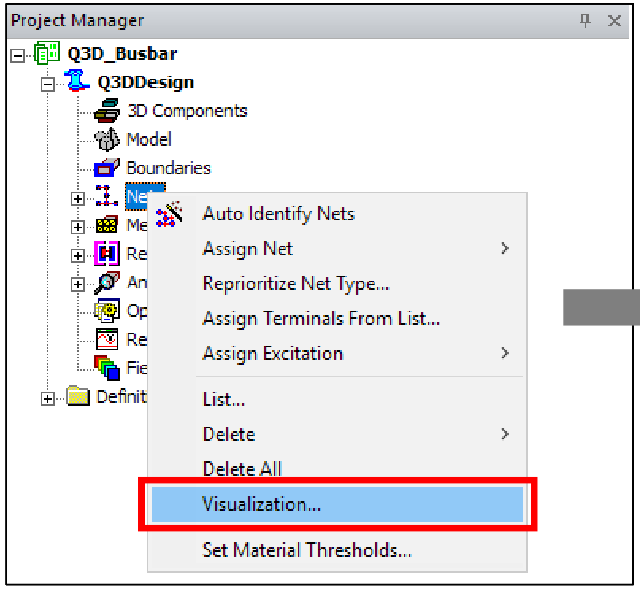
- **Import an Existing Q3D Project into AEDT 2020 R1**
  - **File/Open – Browse and select Q3D_Busbar.aedtz**
  - **Restore the Project files as Q3D_Busbar.aedt**

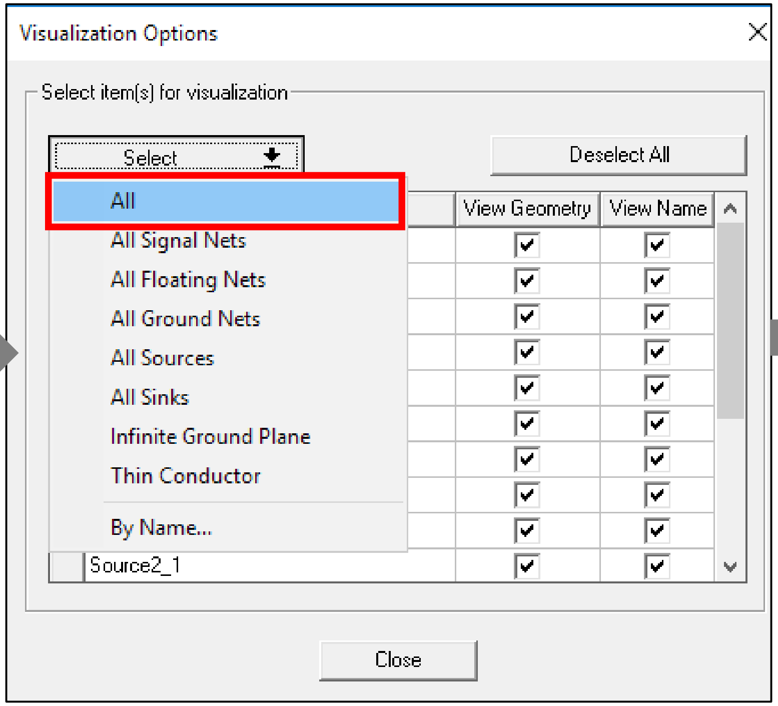


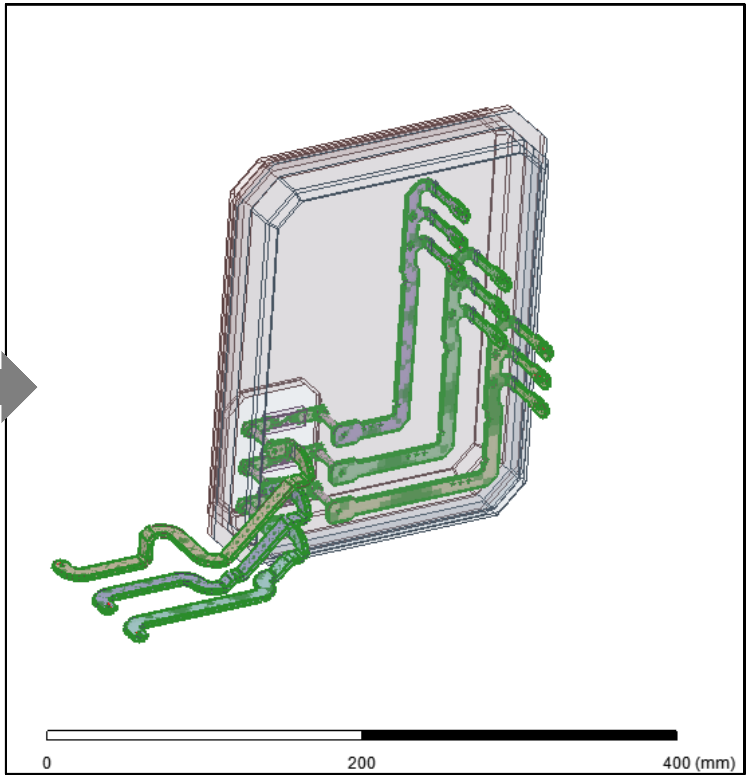
Ansys

Visualize the Nets

- **In the project Manager, under Q3DDesign, Right click Nets and select Visualization …**
- **Select “All” in the Group Visualization Options to view the Current Paths for the 4 nets in the model.**

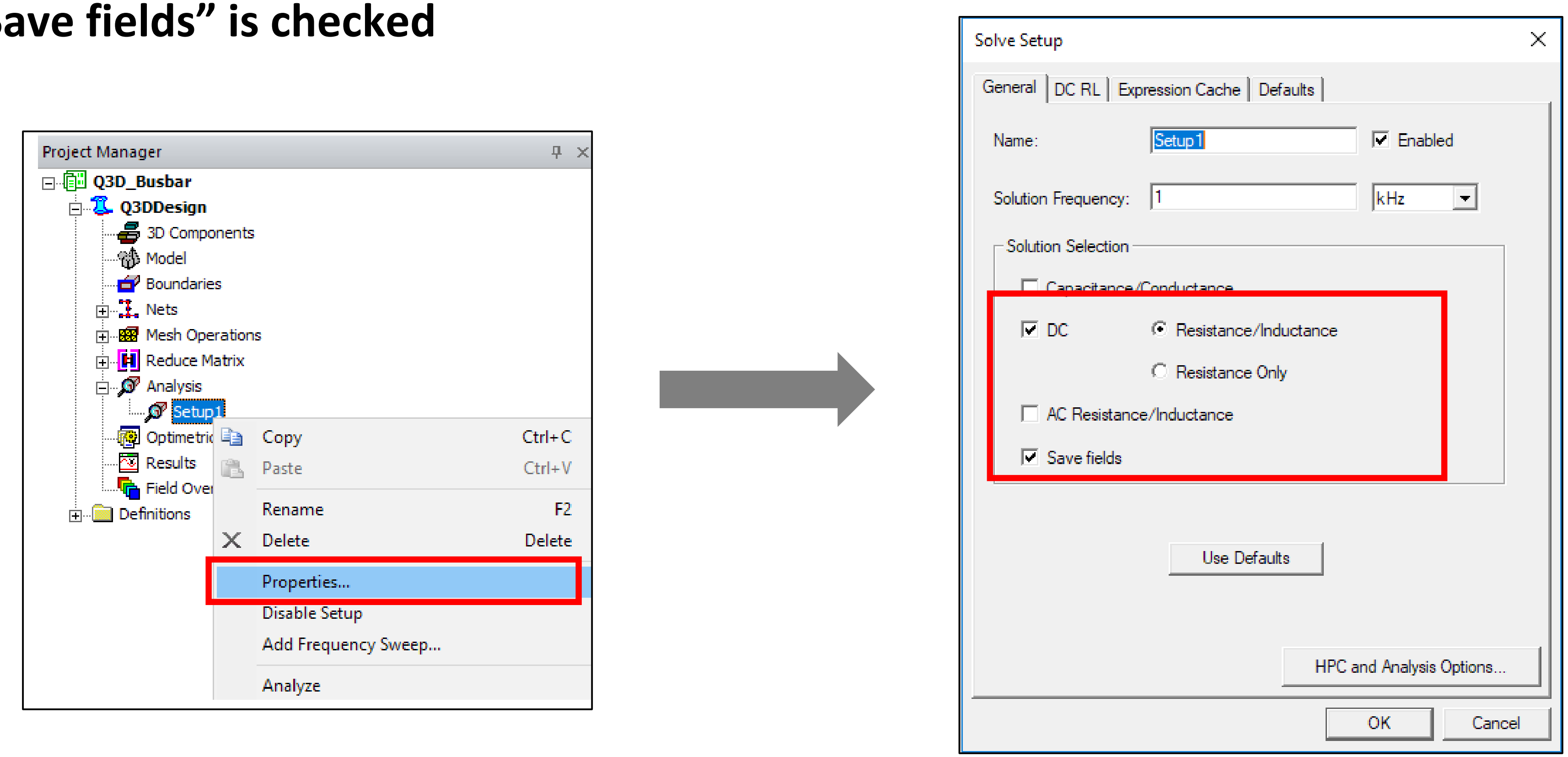






Enable DC RL Run

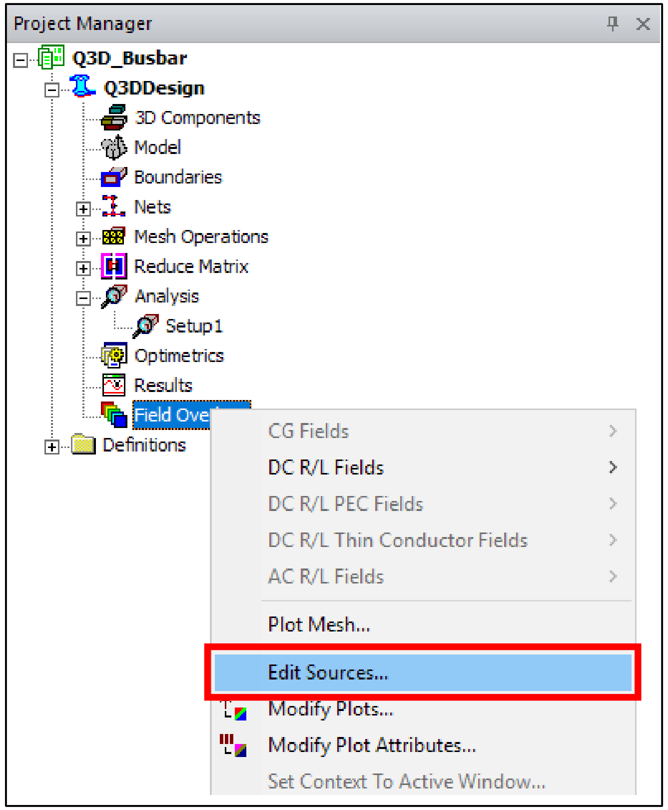
- **Enable DC RL Run in Solve Setup**
  - **In Project Manager, under Q3DDesign, right click Setup1 and select Properties**
  - **In Solve Setup panel, make sure under Solution Selection “DC” with “Resistance/Inductance” is selected and “Save fields” is checked**

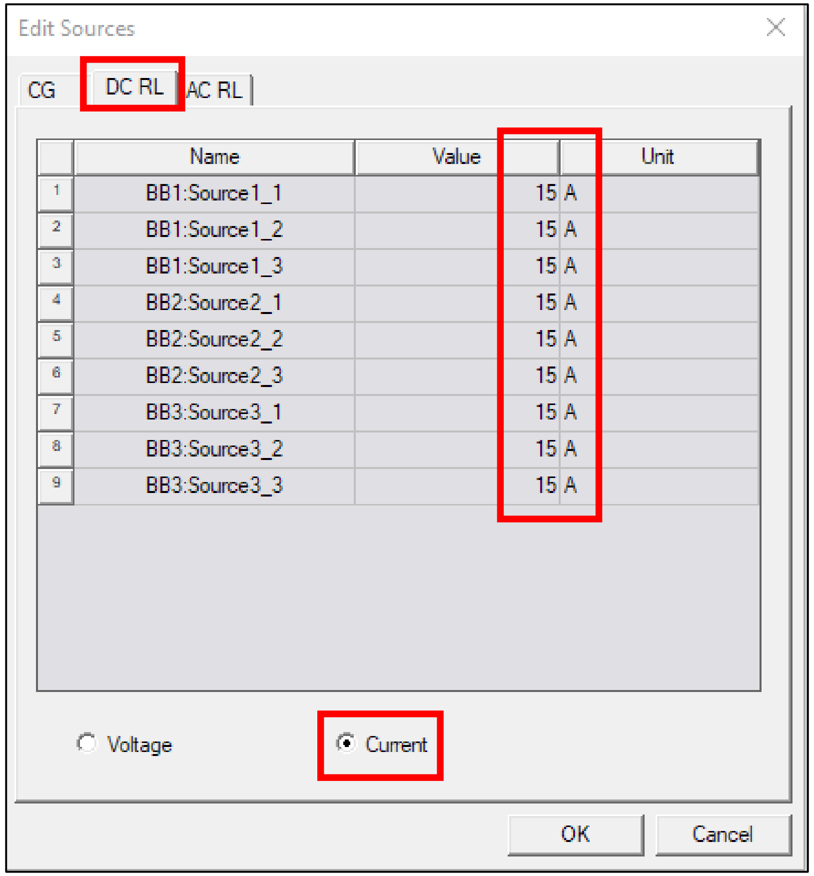


Ansys

Check Sources

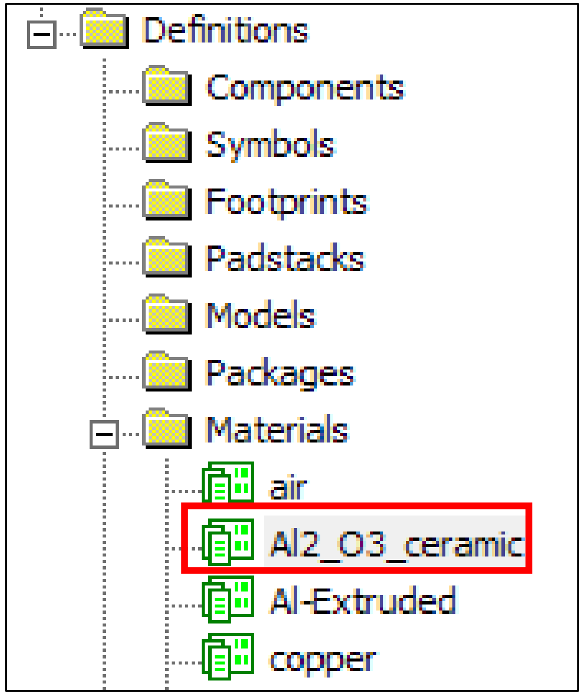
- **Check the edit sources dialog**
  - **Click on Q3D /Fields /Edit Sources…**
  - **Click on the DC RL tab, as we are going to perform DC RL Calculations :**
    - **Make sure each of the sources has a value of 15 A**

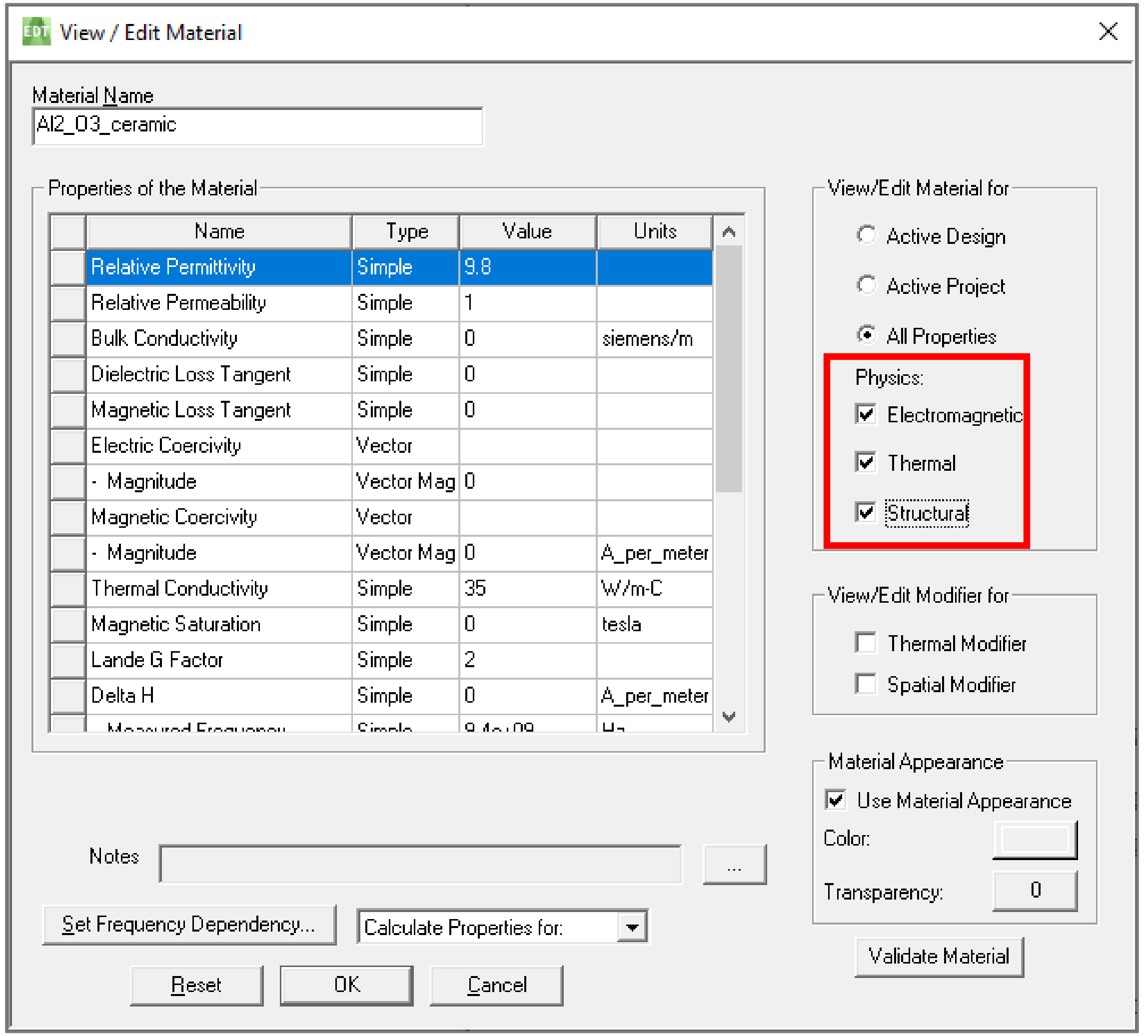




Material Properties

- **Explore the material properties by double-clicking on the appropriate entry in the Project Tree under Definitions > Materials**
  - **Enable All Properties to view the material properties used in electromagnetic, thermal, and structural simulations**

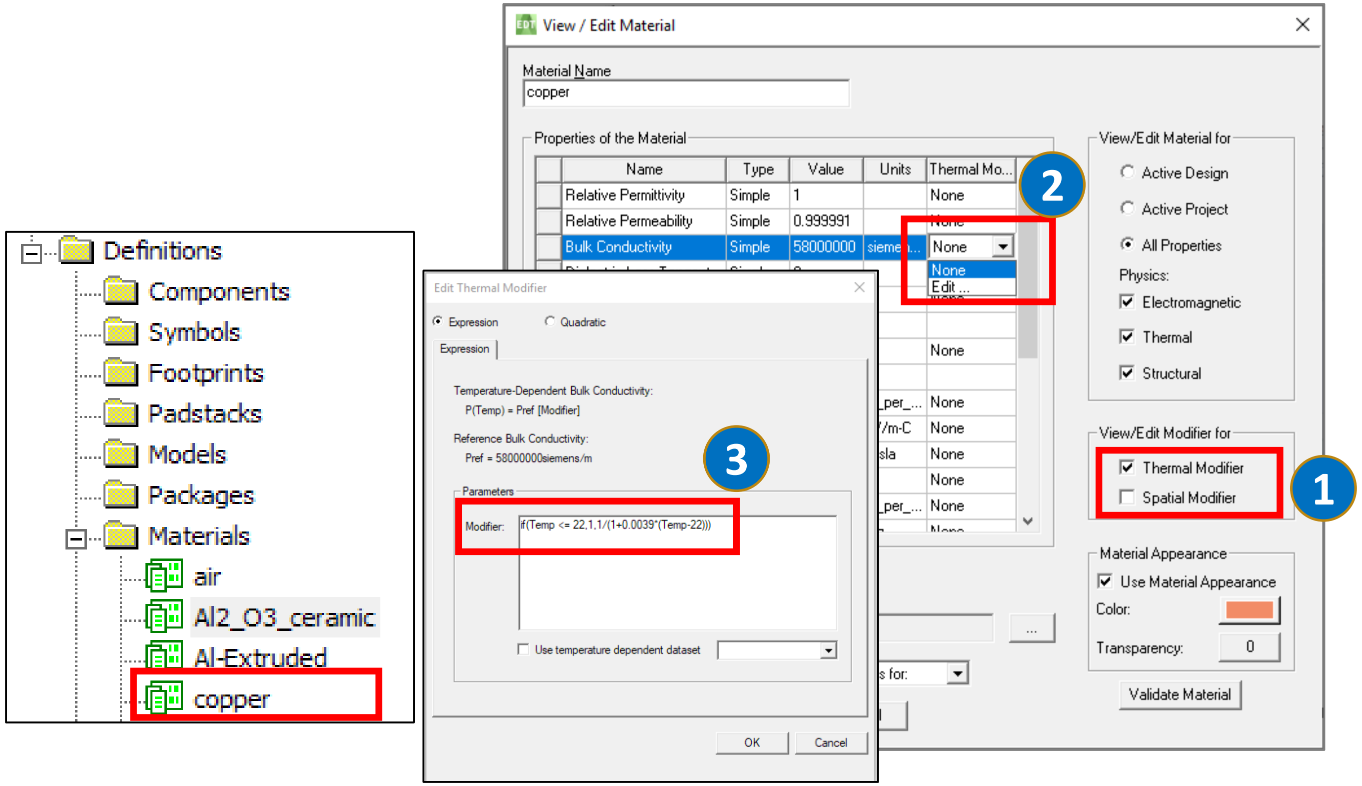




Ansys

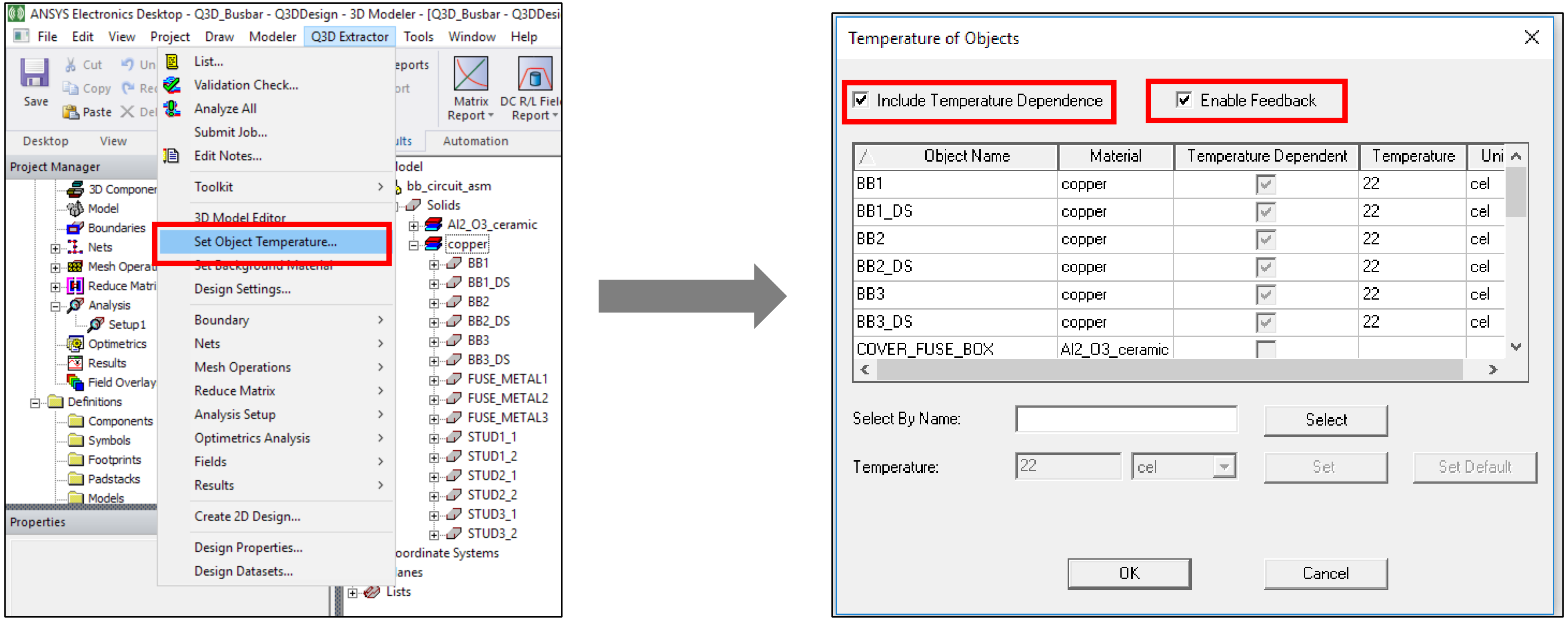
Temperature Dependent Material Settings for a 2-Way Coupling

- **In order to see the impact of temperature on results in Q3D after the Icepak simulation is completed later, material properties must be temperature dependent.**
- **Copper bulk conductivity (the reciprocal of electrical resistivity -- found under Electromagnetic category) varies with temperature.**
- **After opening the material panel for copper, check the "Thermal Modifier".**
- **Then, switch the thermal modifier of "Bulk conductivity" from "None" to "Edit" and use the following correlation:**
  - **if(Temp <=22,1,1/(1+0.0039*(Temp-22)))**



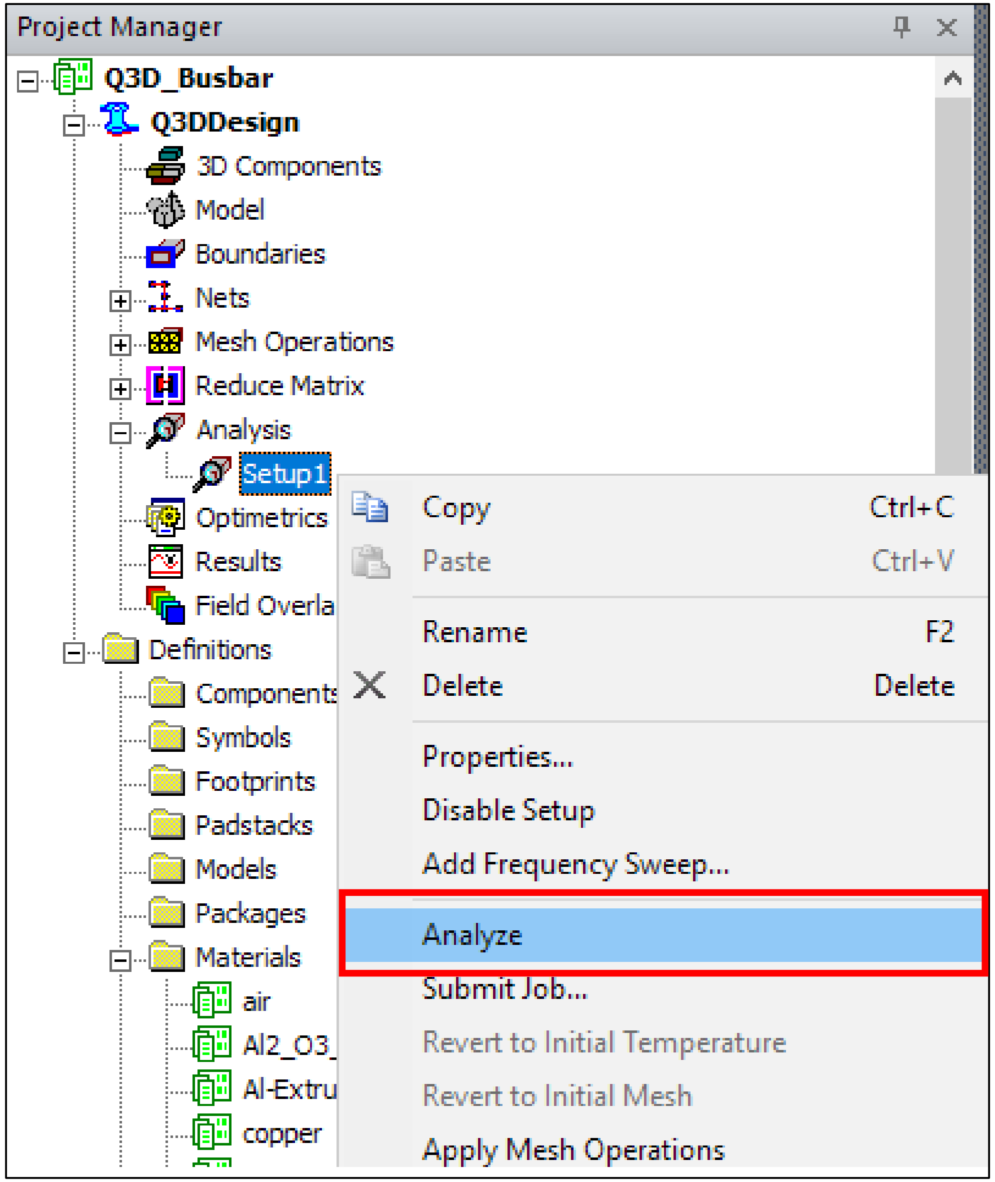
Enabling Feedback

- **One more step is needed to prepare the Q3D model for a bidirectional coupling to ANSYS Icepak, enabling feedback.**
- **From the main menu Q3D Extractor, click on “Set Object Temperature”**
  - **Check both “Include Temperature Dependence” and “Enable Feedback”**



Run Q3D Analysis

- **To Run Q3D Analysis:**
  - **Right click Setup1 and select Analyze**

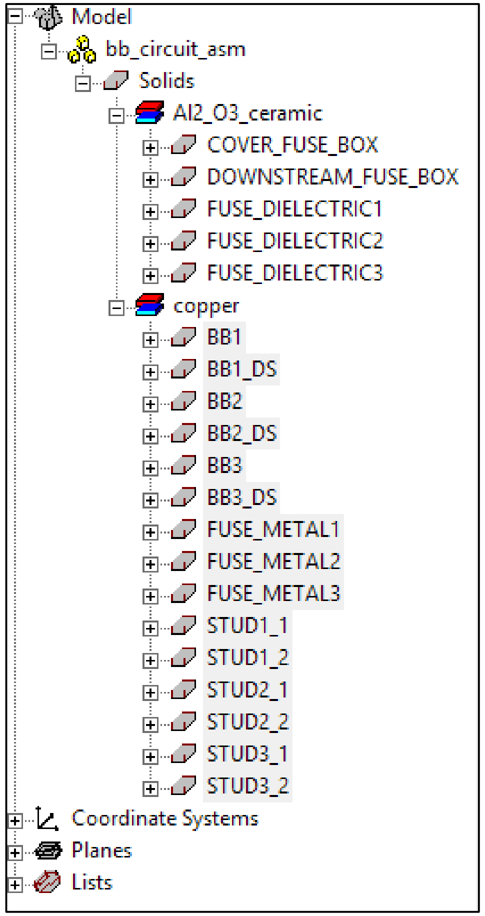


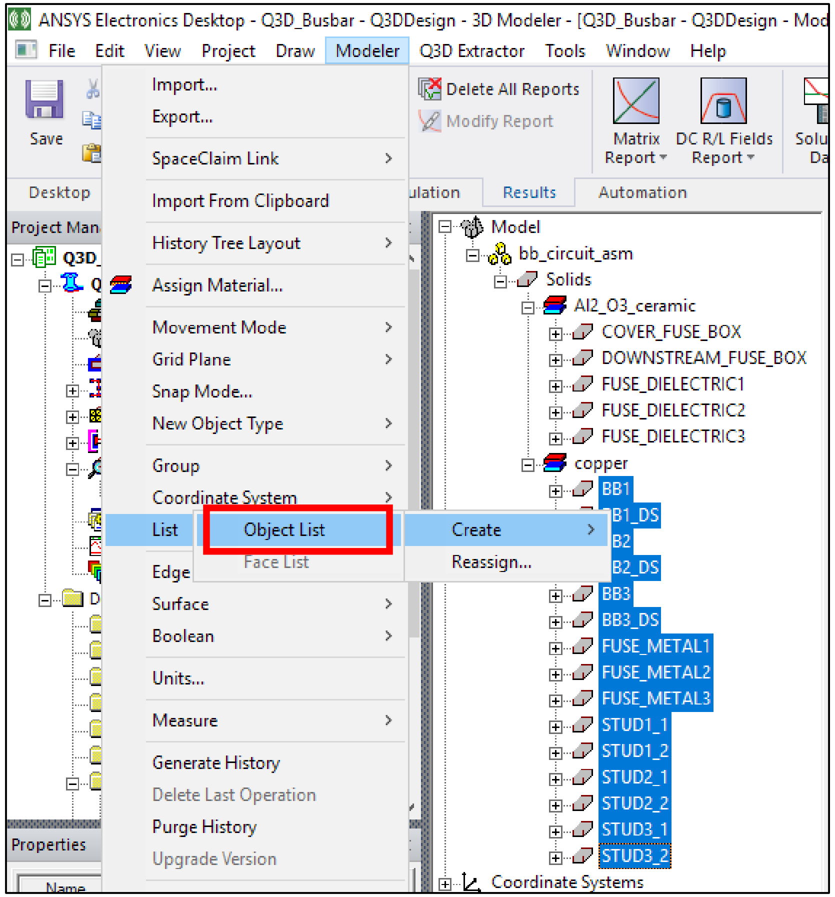
Create an Object List

- CTRL"+ "Left Click" to select all objects with the "copper" materials from the Model node.

- Use Main Menu/Modeler/List/Create/Object List to create ObjectList1 for the selected objects.

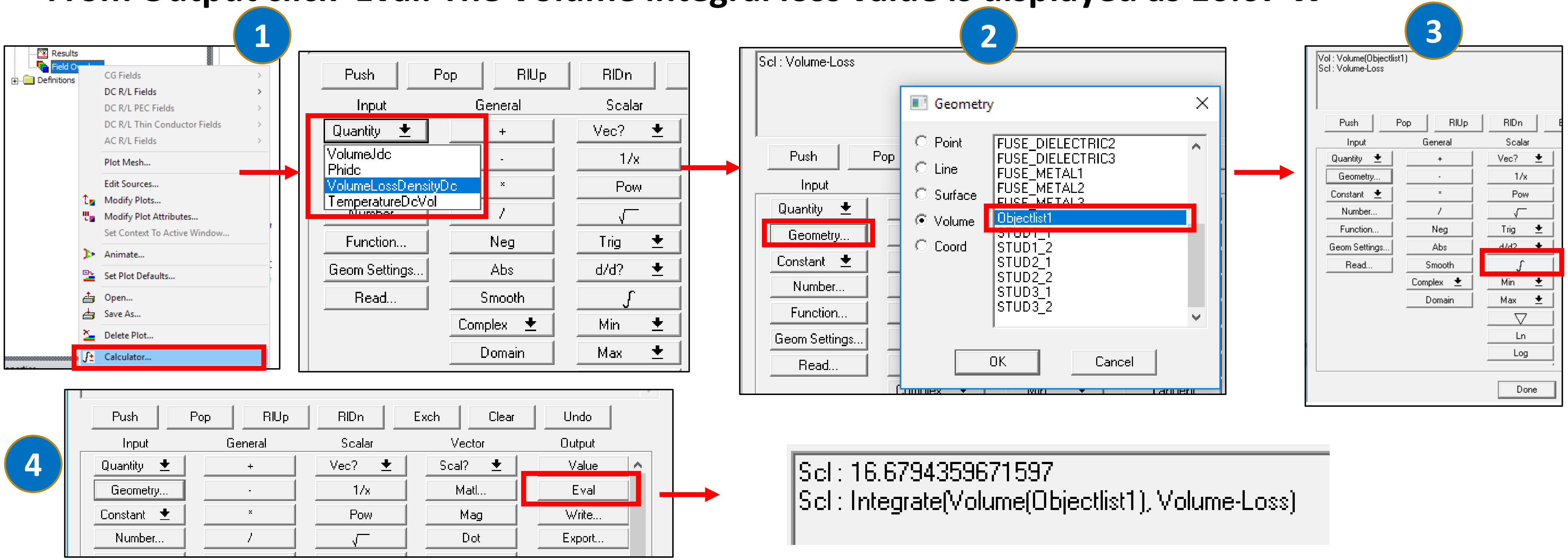
  – Note: The object list is treated as one volume when you plot and perform fields calculations





Report Volumetric Losses

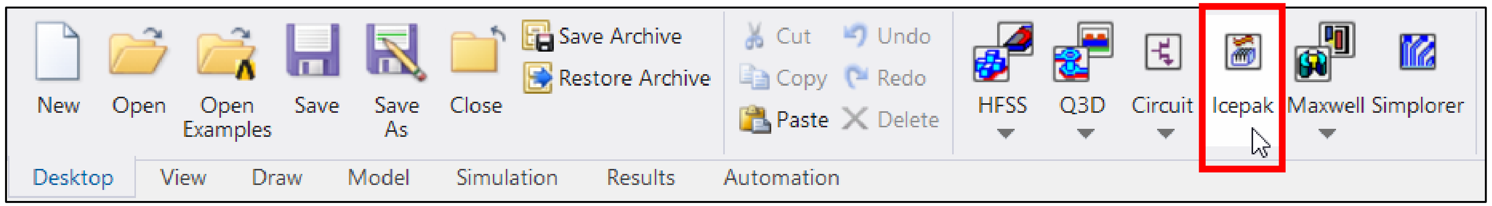
- **Use the Field Calculator in Project Manager/Q3D Design/Field Overlays to report the Volume Losses on ObjectList1.**
  - **From Input/Quantity add VolumeLossDensityDC.**
  - **From Input/Geometry Add Objectlist1. From Scalar add Integral.**
  - **From Output click Eval. The Volume integral loss value is displayed as 16.67 W**

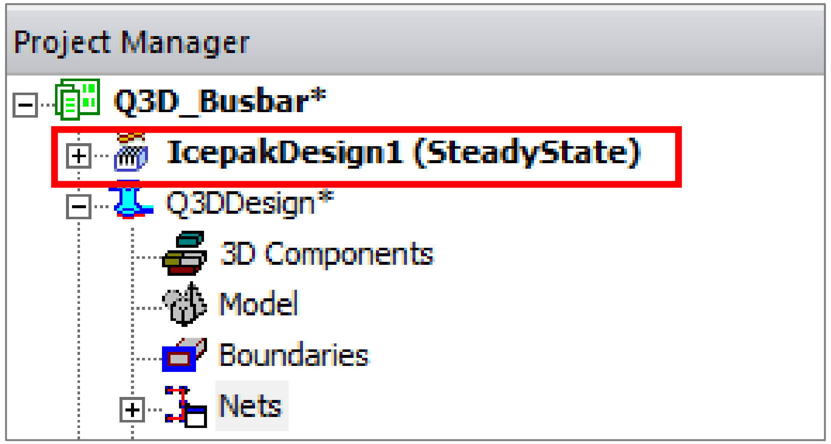


Ansys

Insert Icepak Design

- **From Ribbon Menu/Desktop/Insert Icepak Design.**

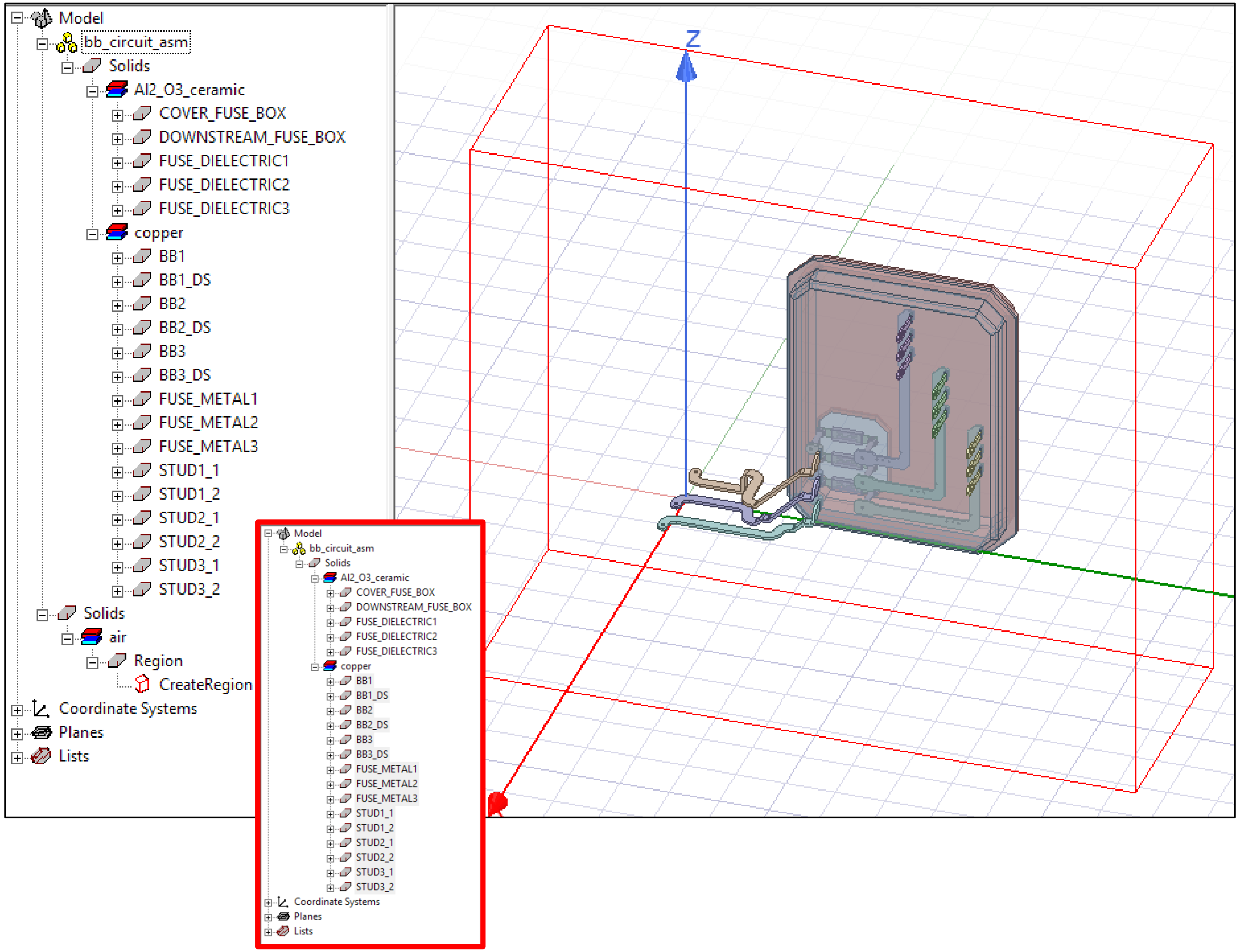




Ansys

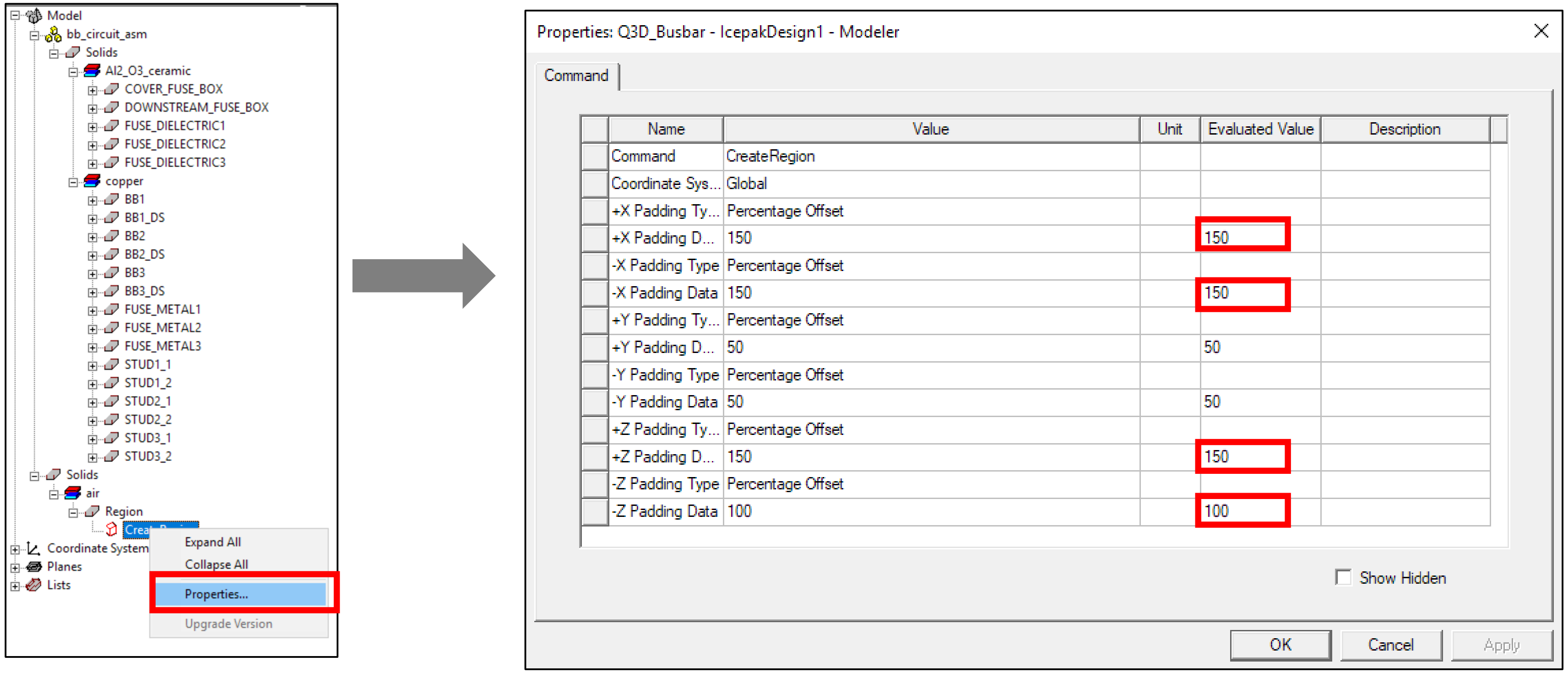
Bring Q3D Geometry and Material Setup into Icepak Design

- **In the Q3D Graphics window, use Ctrl+A, Ctrl C to copy the entire Q3D Geometry and Material information**
- **In the Icepak Graphics window, use Ctrl+V to paste the Geometry and Material information copied from Q3D**
- **Use Ctrl+D to zoom to fit**
  - **Note the automatic creation of the "Air Region" around the copied geometry**



Edit Region

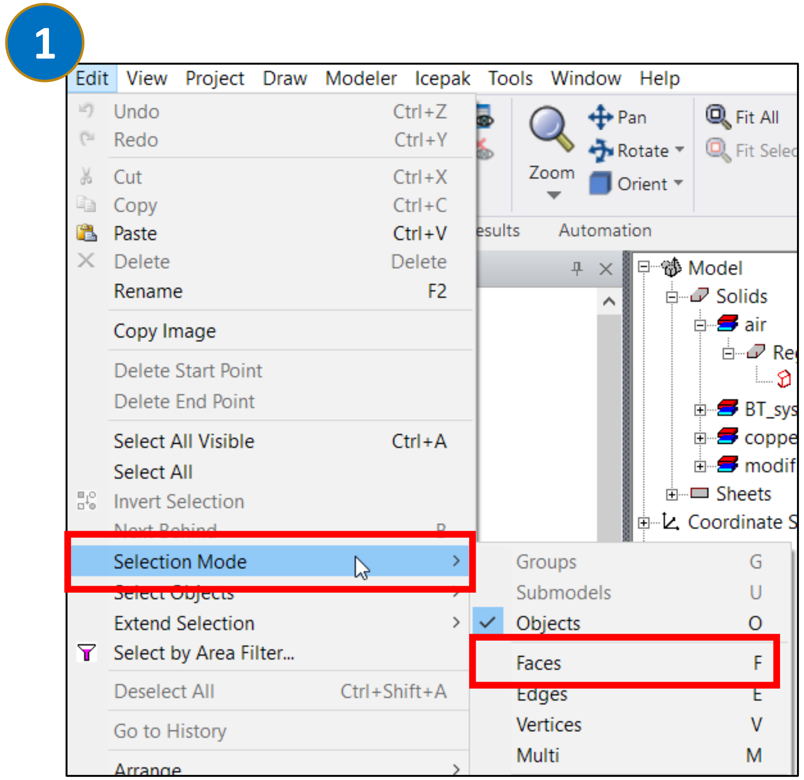
- **In the Model tree, Double Click on the Region > Create Region.**
- **Increase the padding in +Z and –Z direction to 150 and 100 Percentage, respectively.**
- **Increase the padding in +X and –X direction to 150 and 150 Percentage, respectively.**
  - **It is recommended to have larger padding in the direction opposite to gravity direction for natural convection problem**

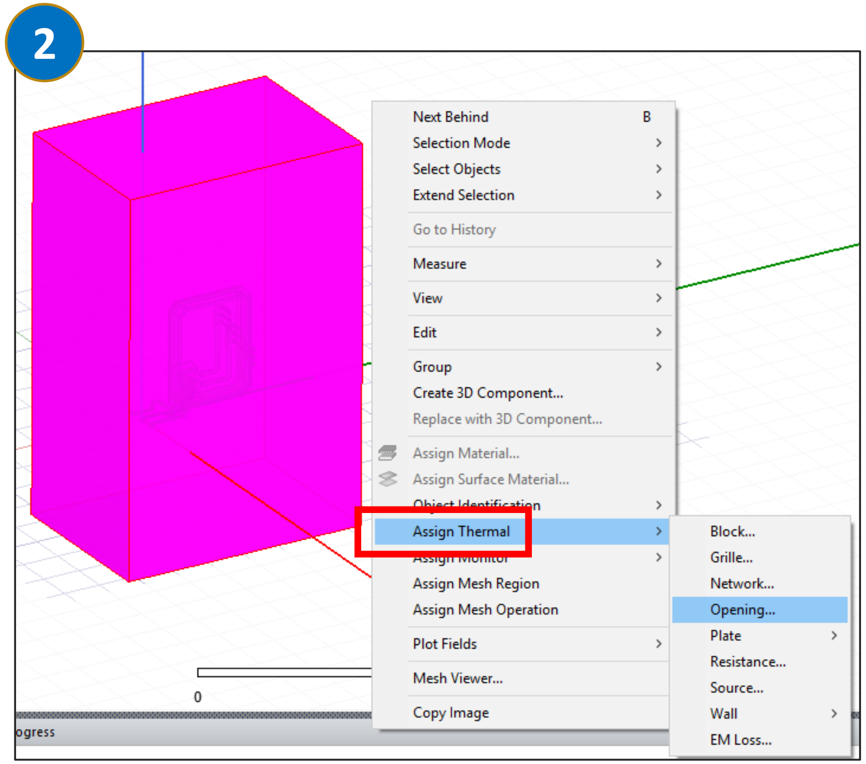


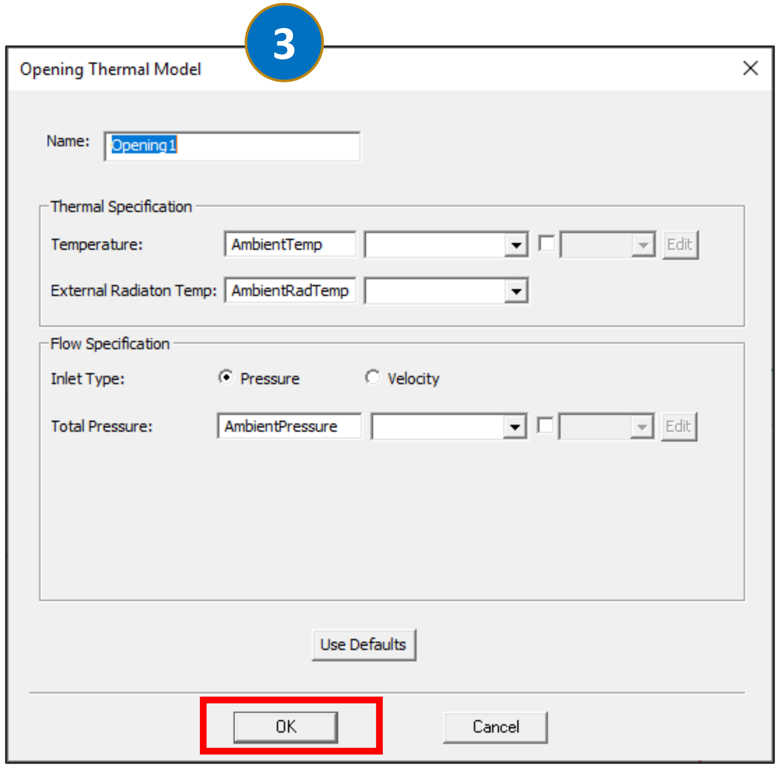
Ansys

Physics Setup – Flow Outlet

- **Change Selection mode to Faces**
  - **Main Menu/Edit/Selection Mode, Select faces (or) Use a shortcut key [f] in graphical window to select face in place of volume. Default is volume.**
  - **Select all the faces of the Region, Right click Assign Thermal > Opening.**

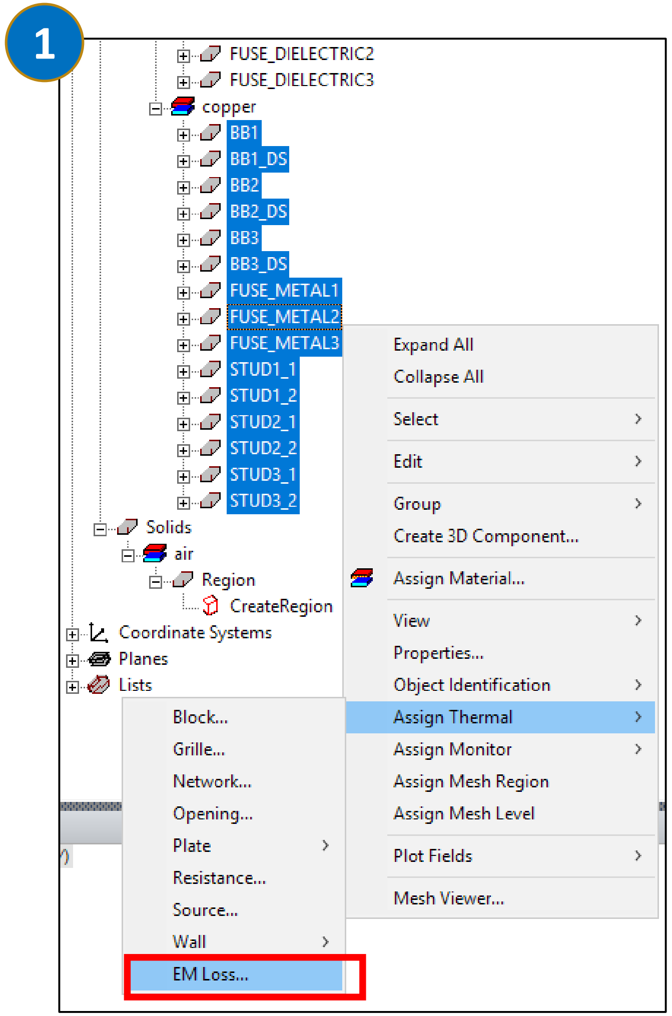


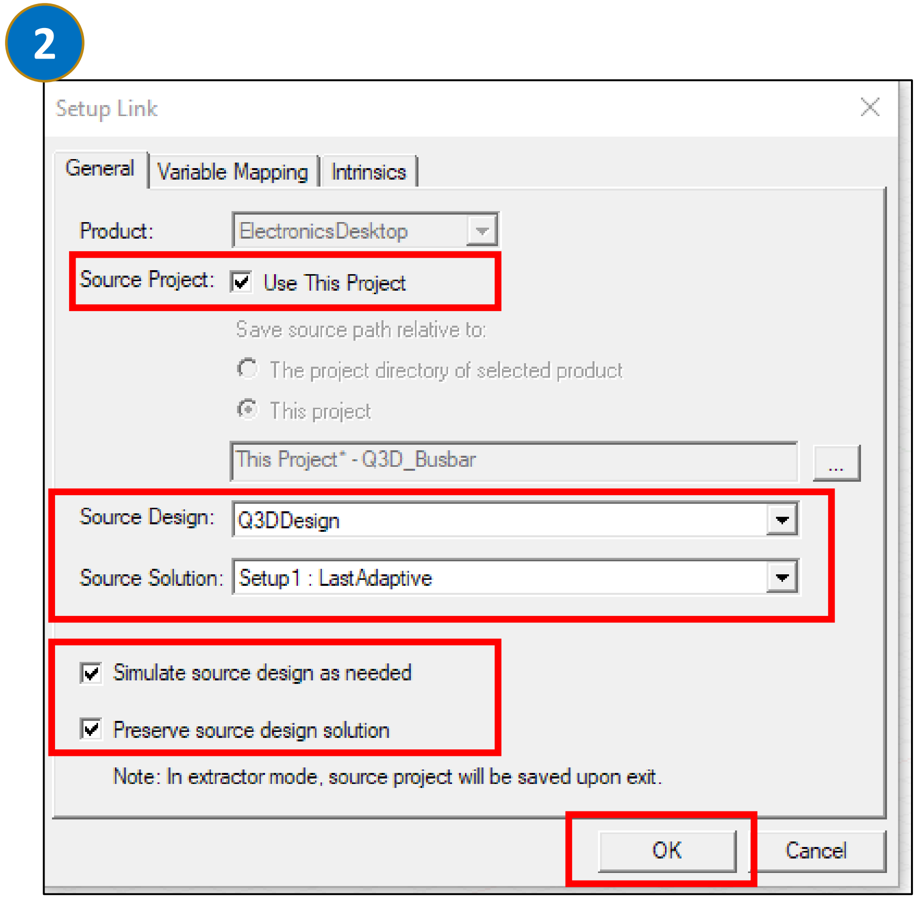


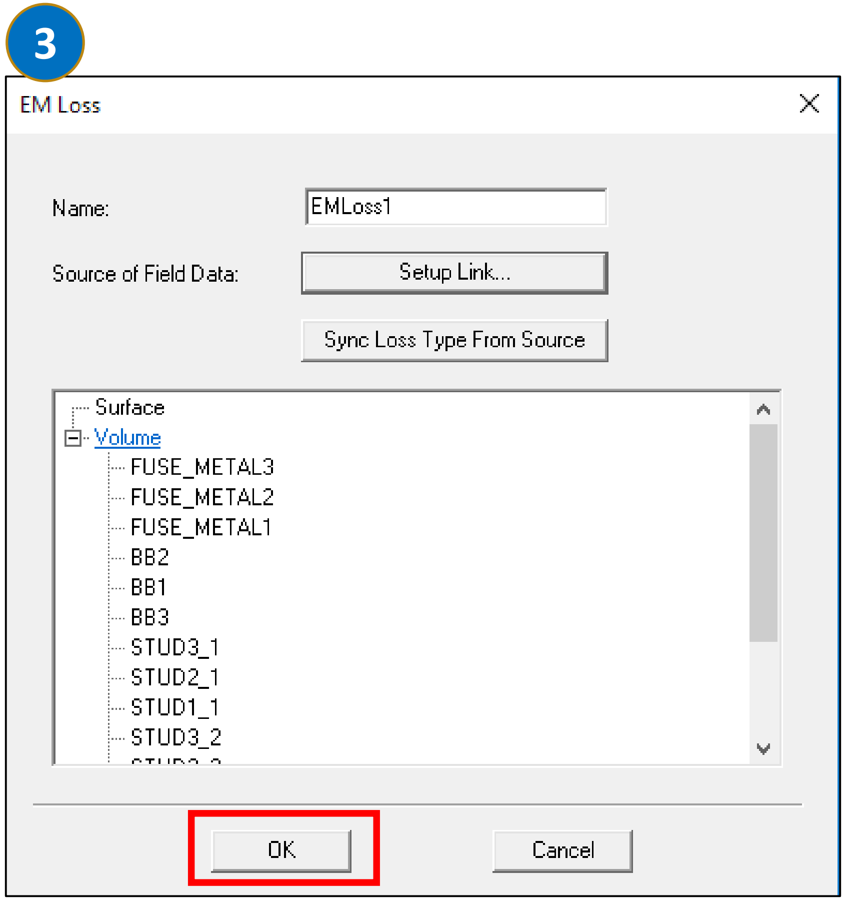


Physics Setup – EM Volume Loss

- **Set up all the copper objects for EM loss mapping.**



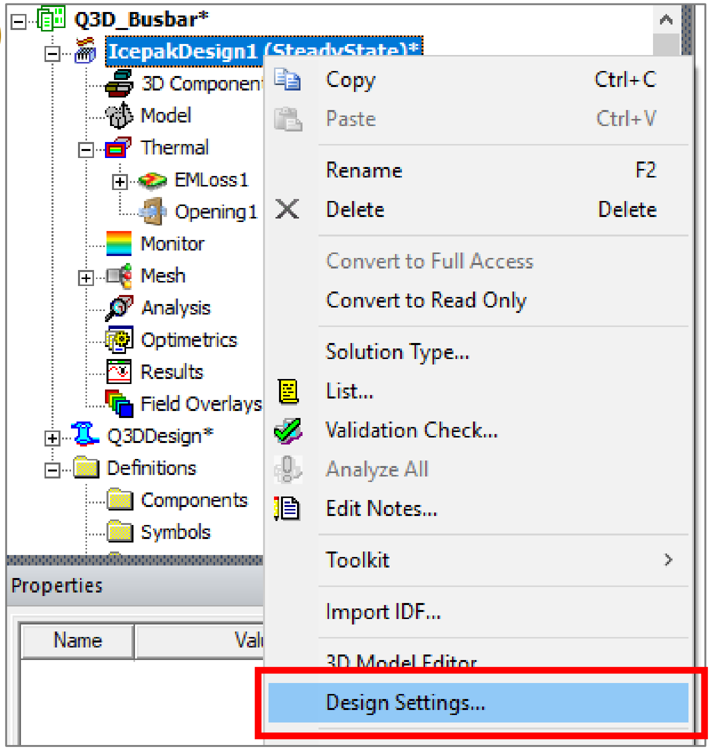




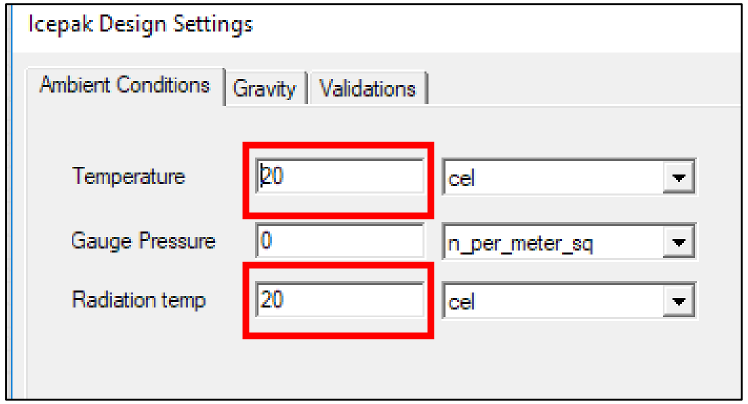
Icepak Design Settings

- **Right Click the Icepak Project and Select Design Settings…**
- **In the Ambient Conditions tab, keep the default values at 20°C**
- **In Gravity tab, select negative Z direction for gravity vector. (Gravity must also be enabled in Setup which will be showed in following slide.)**

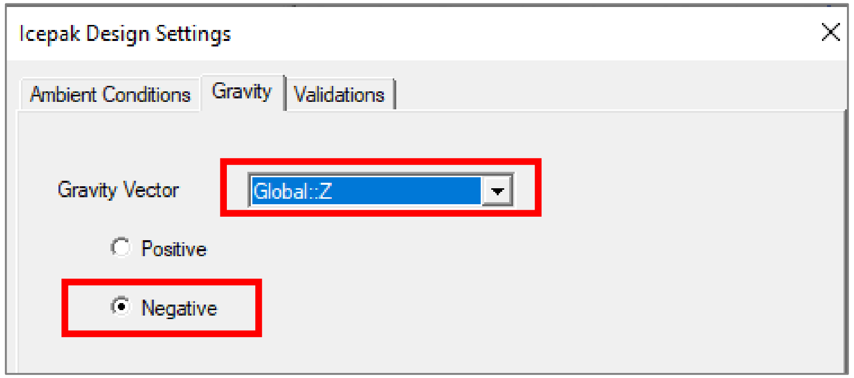
1



2



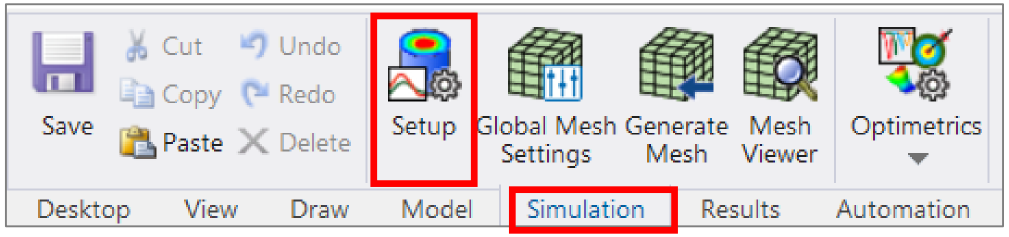
3



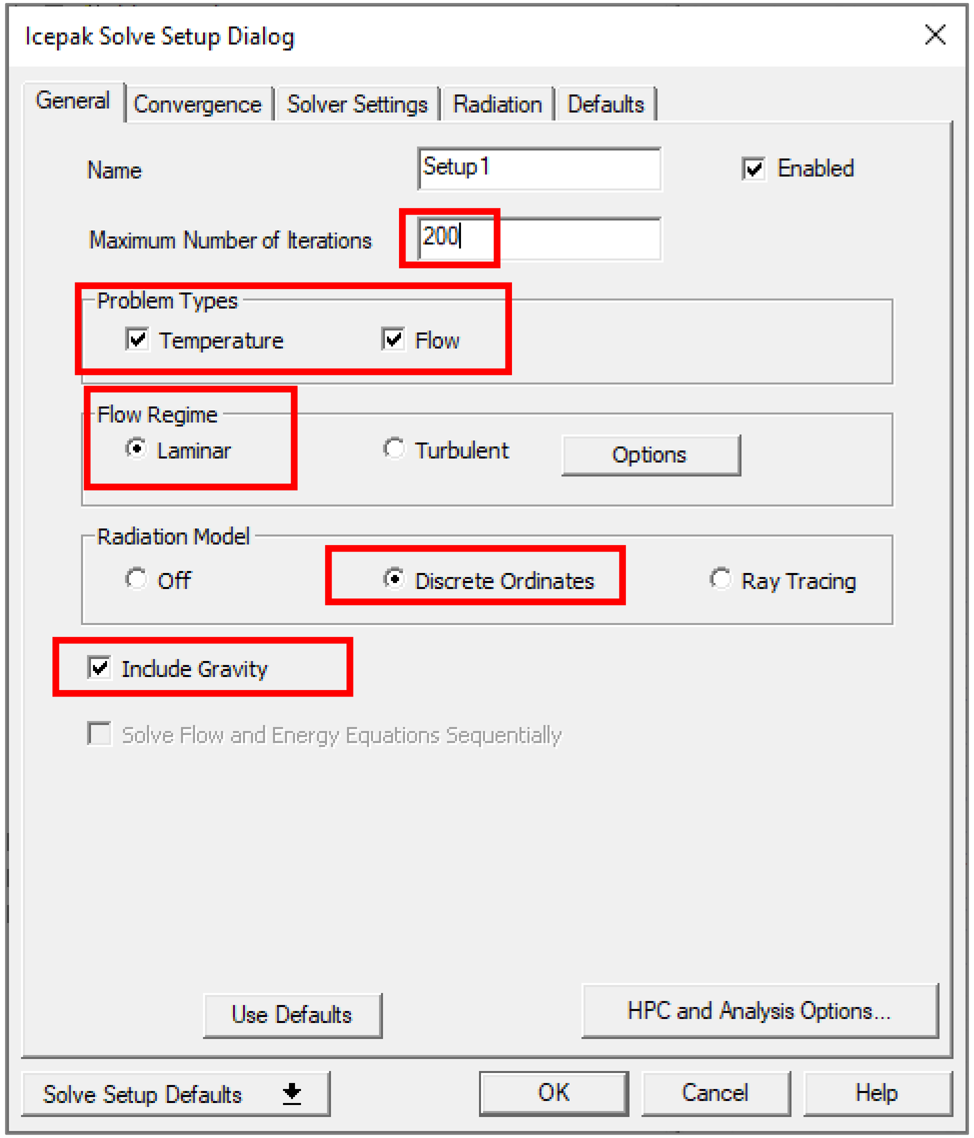
Ansys

Analysis Setup

- **In the Simulation tab of the ribbon Menu, Click Setup.**

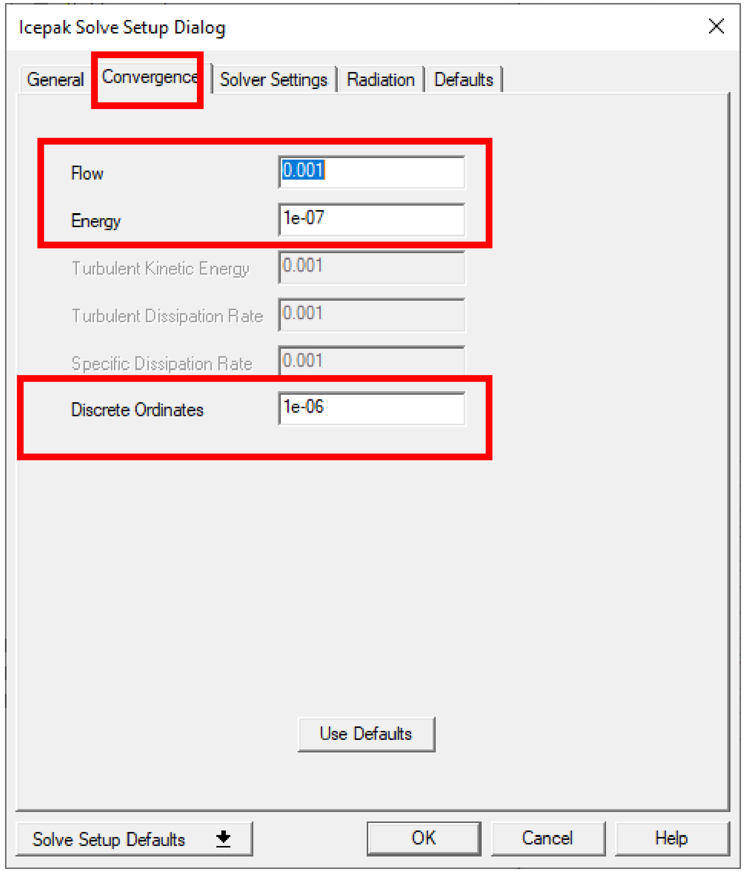


- **On the General Tab:**
  - **Set the number of iterations to 200**
  - **Make sure both Flow and Temperature are checked to solve**
  - **Include Radiation and set it to "Discrete Ordinates" – enabling radiation is highly recommended for natural convection**
  - **Check "Include Gravity"**

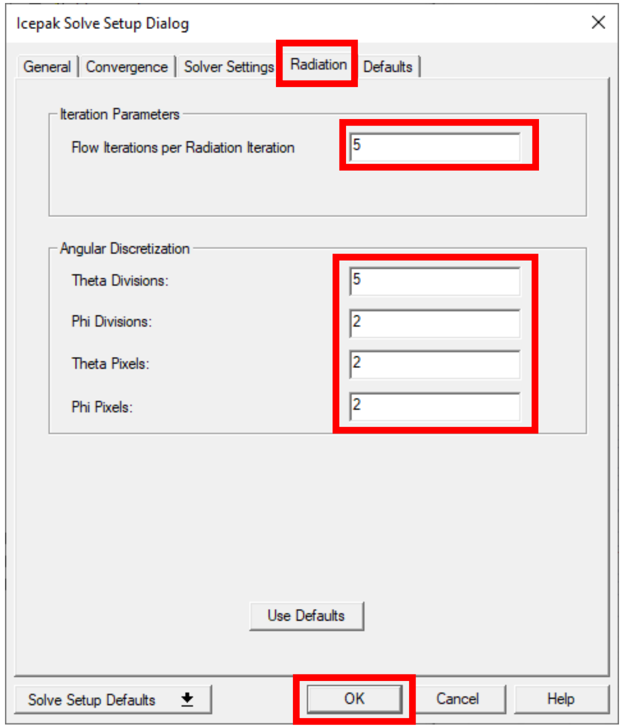


Analysis Setup

- **Setting convergence criteria:**
  - **On the Convergence tab, keep the default values for Flow and Energy and radiation**



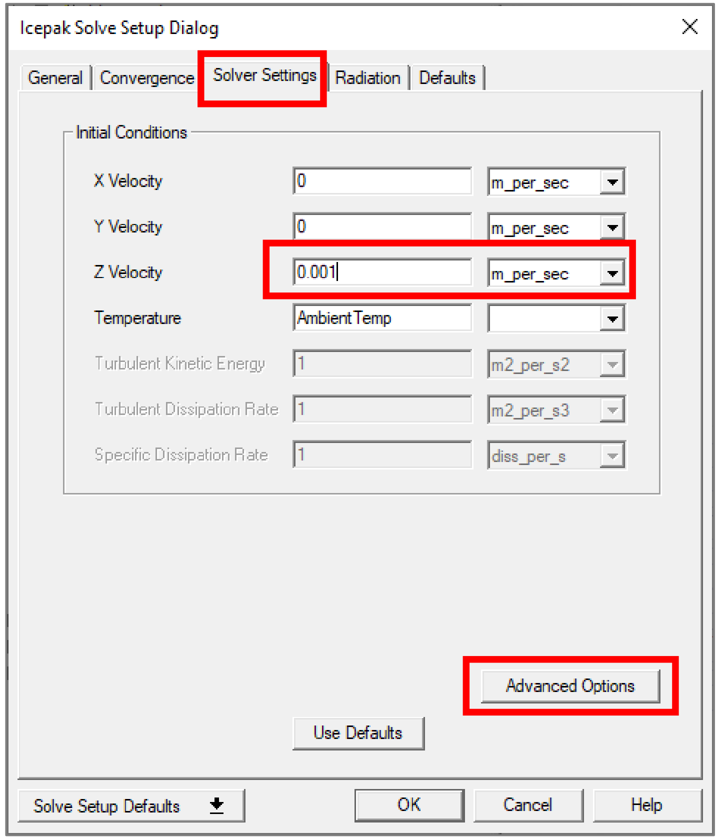
- **On the Radiation Tab:**
  - **Set the number of iterations per Radiation iteration to 5 – then radiation equation is solved at every 5 flow iteration**
  - **Set all the angular discretization to 2 for higher accuracy**
  - **Click "OK"**



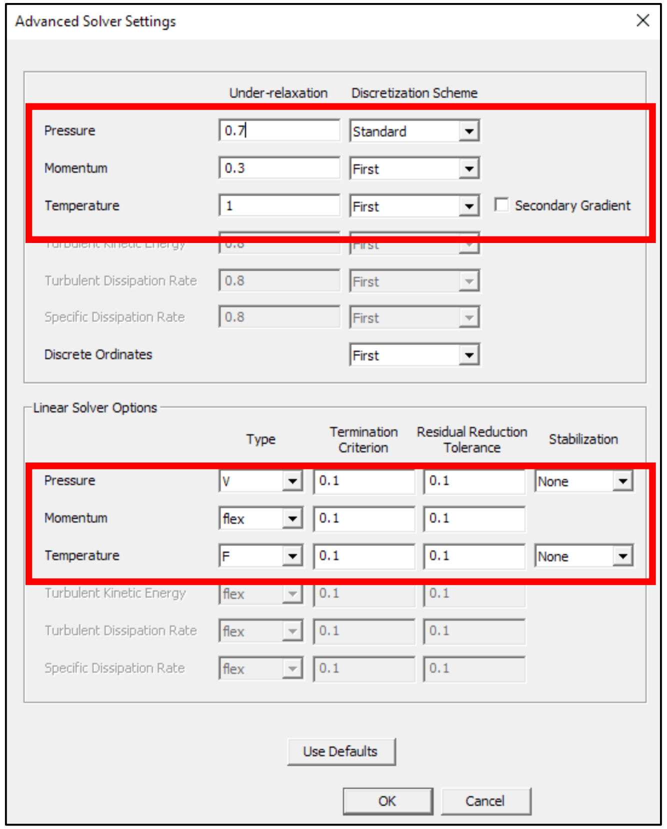
Ansys

Analysis Setup

- **Setting an initial condition:**
  - **On the Solver Settings Tab, set Z Velocity to 0.001**
  - **Note: Settings small velocity opposite to Gravity direction will help for faster convergence.**

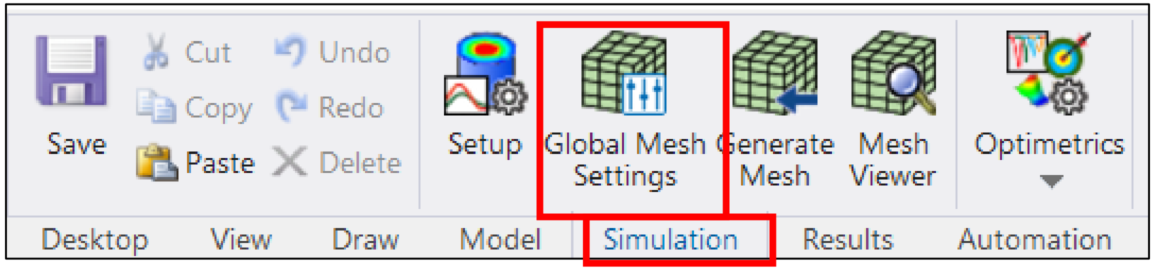


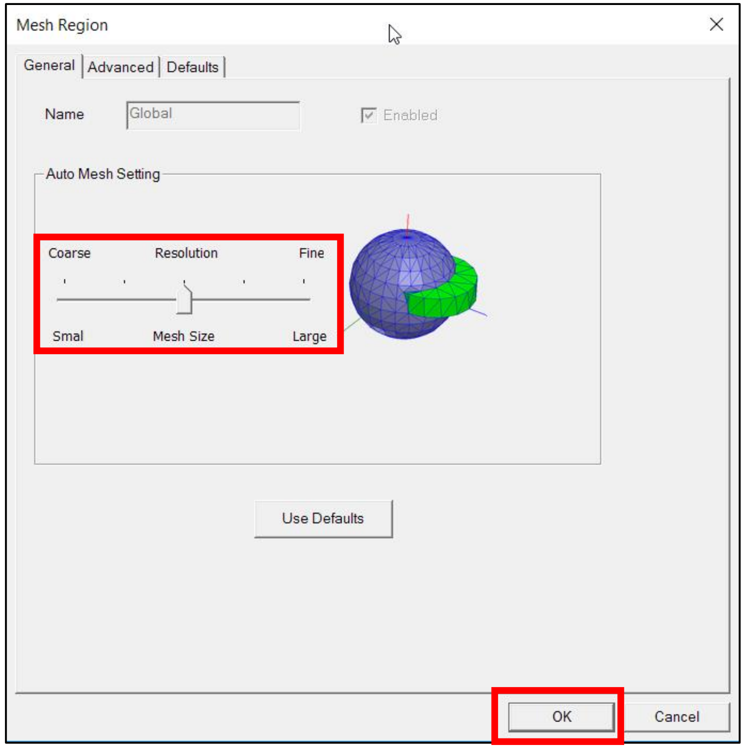
- **Advanced settings:**
  - **On the Solver Settings Tab, click on Advanced Options**
  - **Set the under-relaxation factors (URFs) of pressure and momentum to 0.7 and 0.3, respectively.**



Check Global Mesh Settings

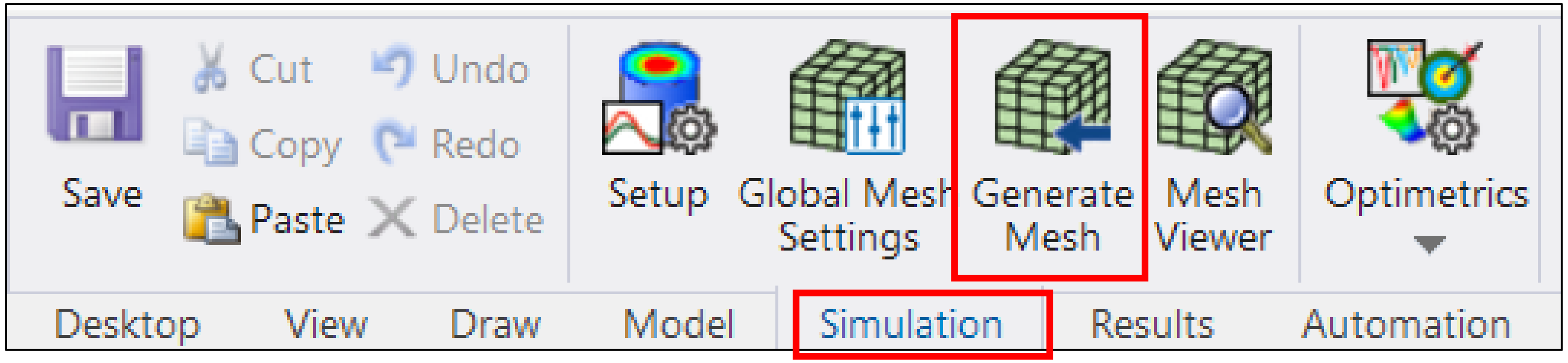
- **In the Simulation tab of the ribbon Menu, Click Global Mesh Settings.**
- **We will use the default Medium Mesh Resolution (Resolution 3).**





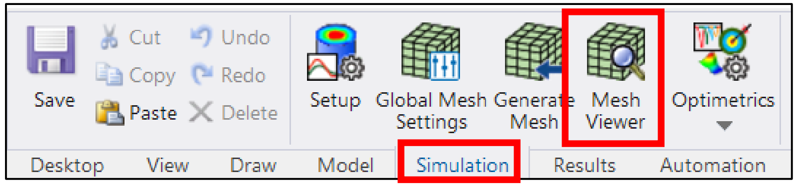
Mesh Generation

- **In the Simulation tab of the ribbon Menu, Click Generate Mesh.**

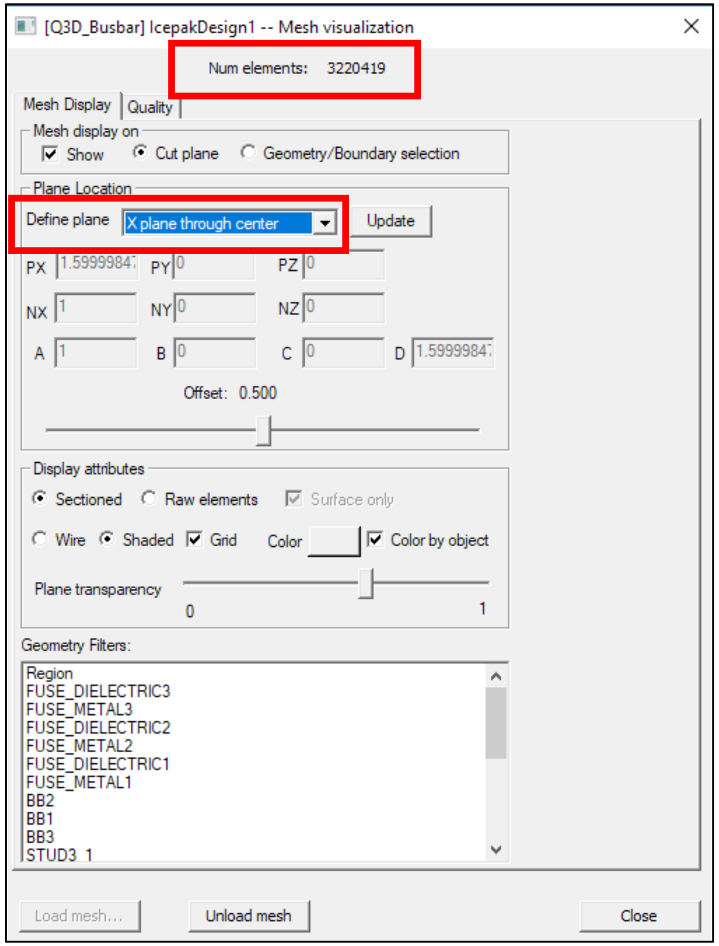


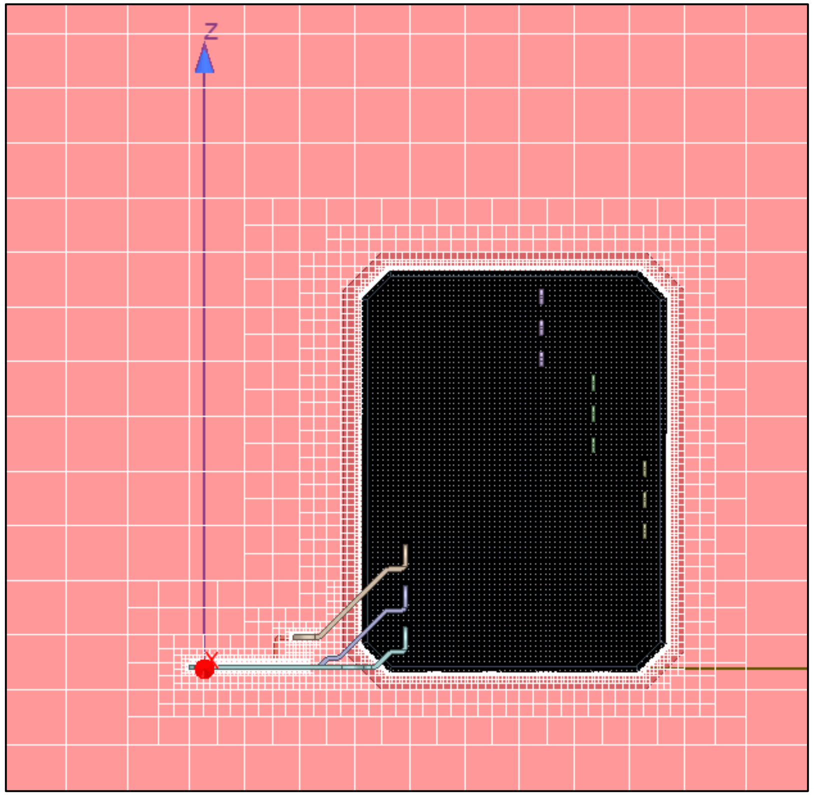
Display Mesh

- **In the Simulation tab of the ribbon Menu, Click Mesh Viewer.**



- **Display Mesh on X plane through center.**

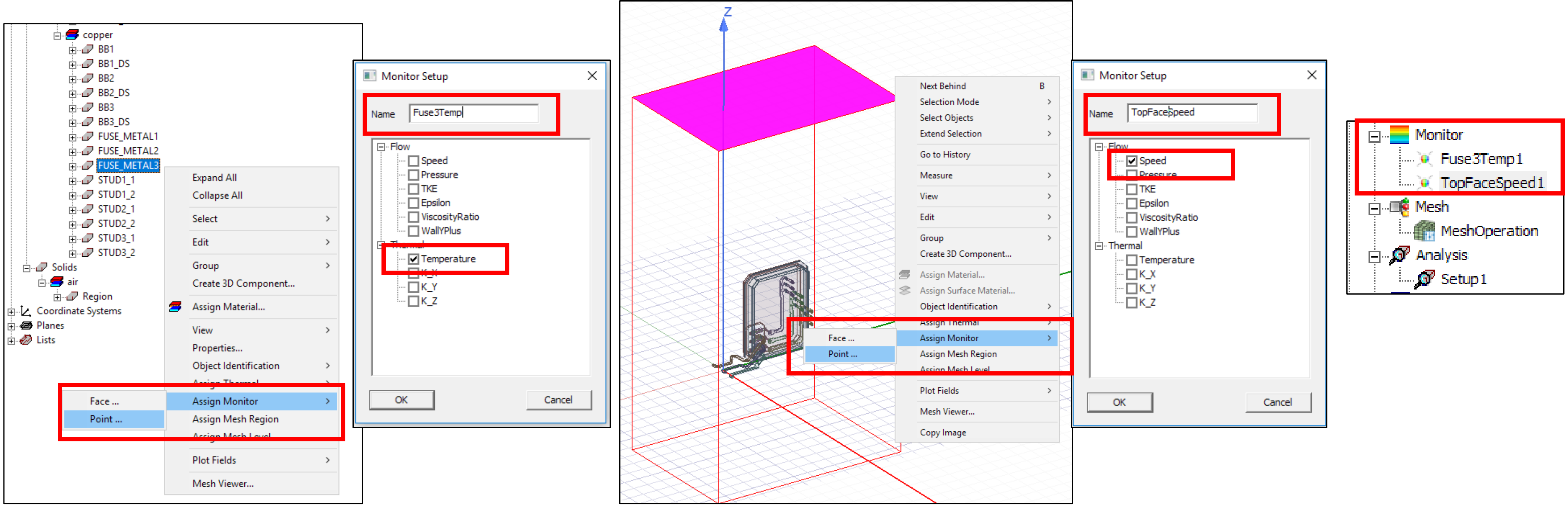




Ansys

Creating Monitor Points

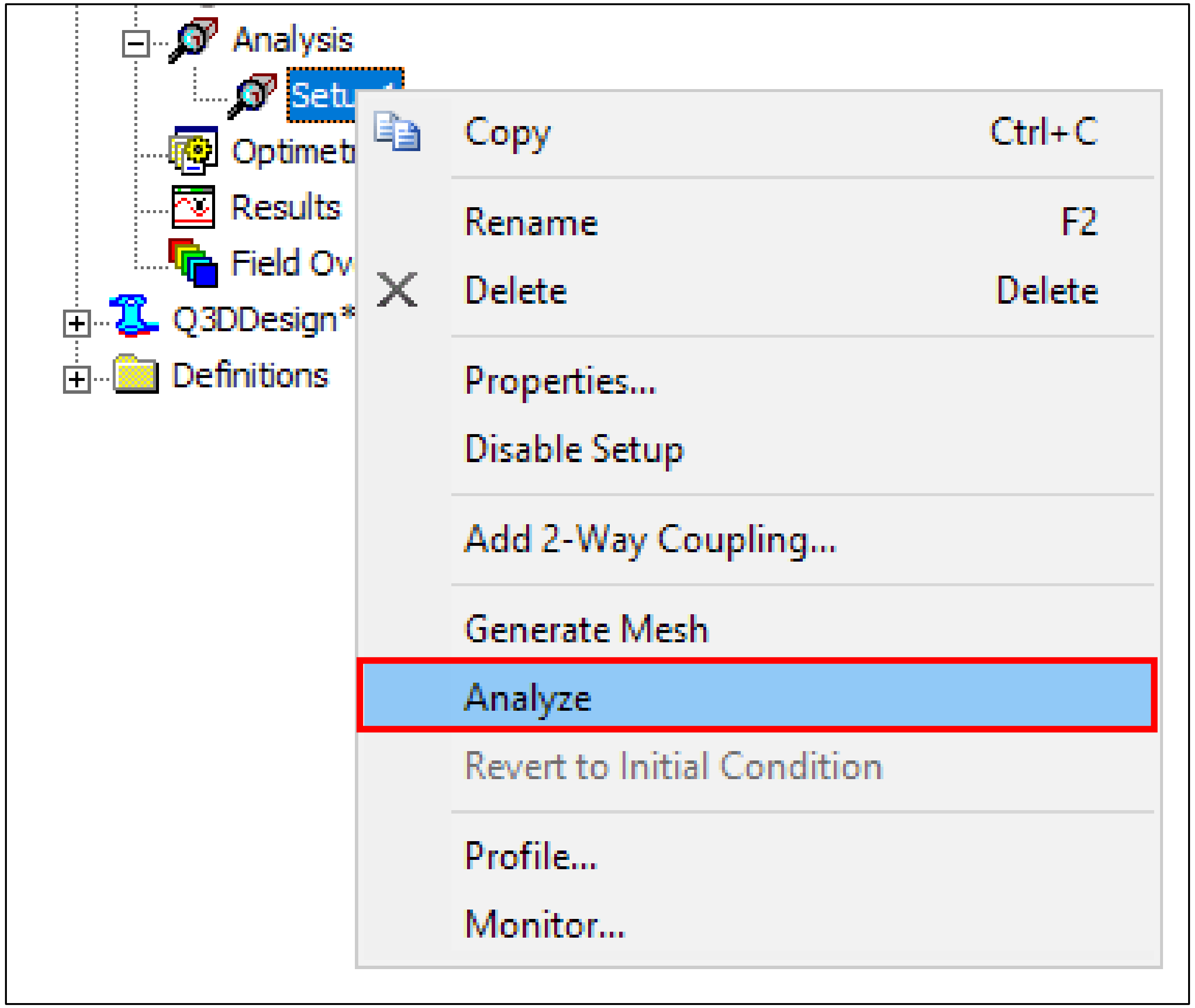
- **To create temperature and speed monitor points:**
  - **Right click on any desired number of solid objects (i.e. FUSE_METAL3), select Temperature in the new panel and name it**
  - **Switch to face mode and select the +Z face of the region in the GUI to create a speed monitor point**



Ansys

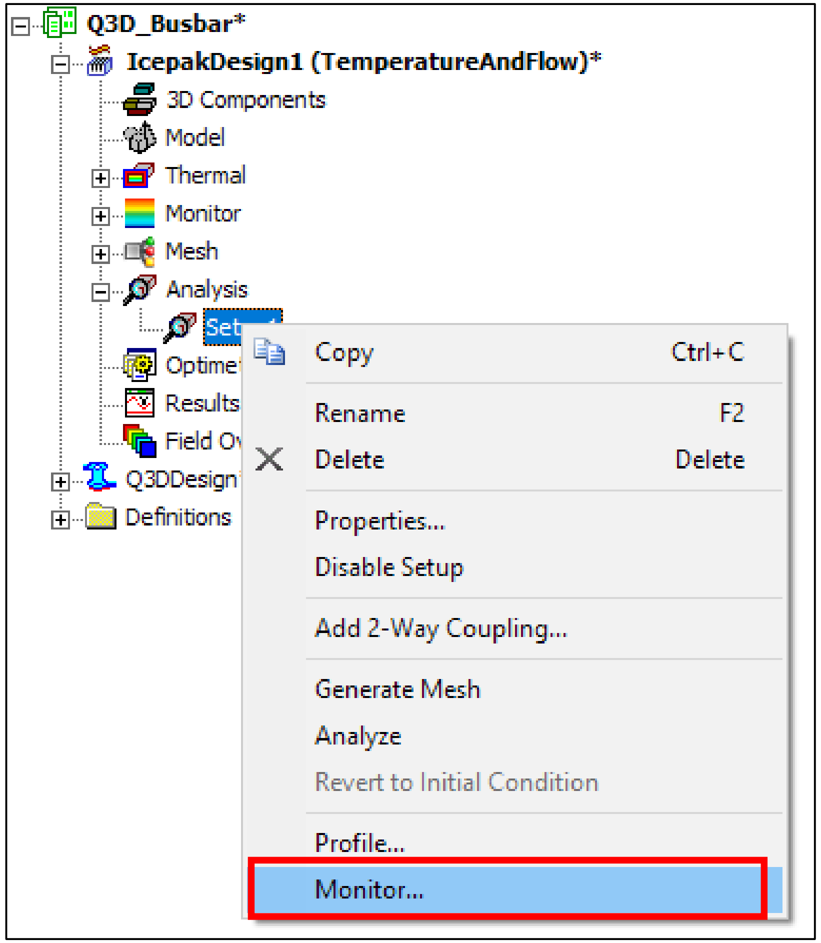
Run Analysis

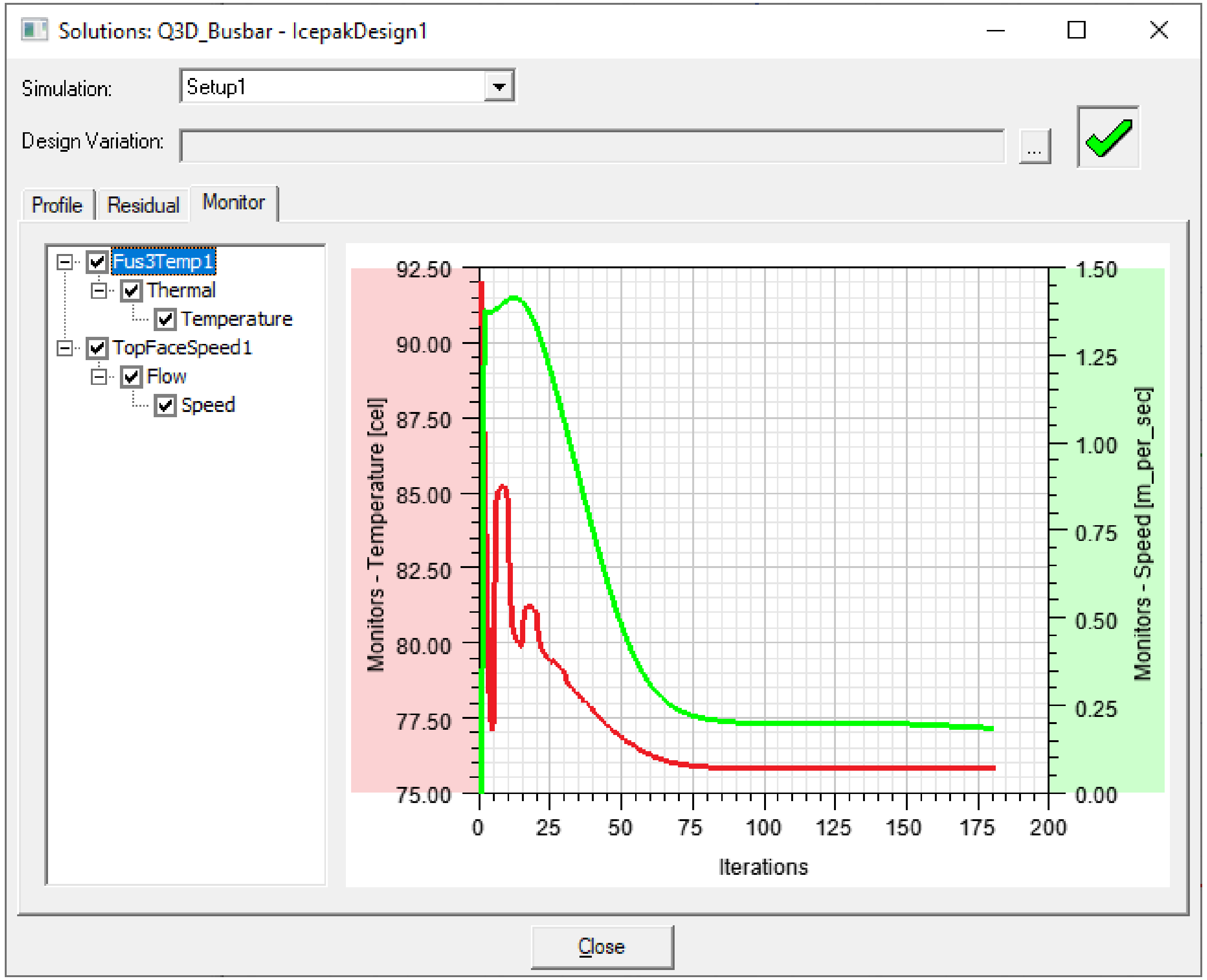
- **To run the Icepak analysis (one-way simulation with EM losses from Q3D without a temperature feedback to Q3D):**
  - **Right click Setup1 and select Analyze**



Review Solution Monitors (For Convergence)

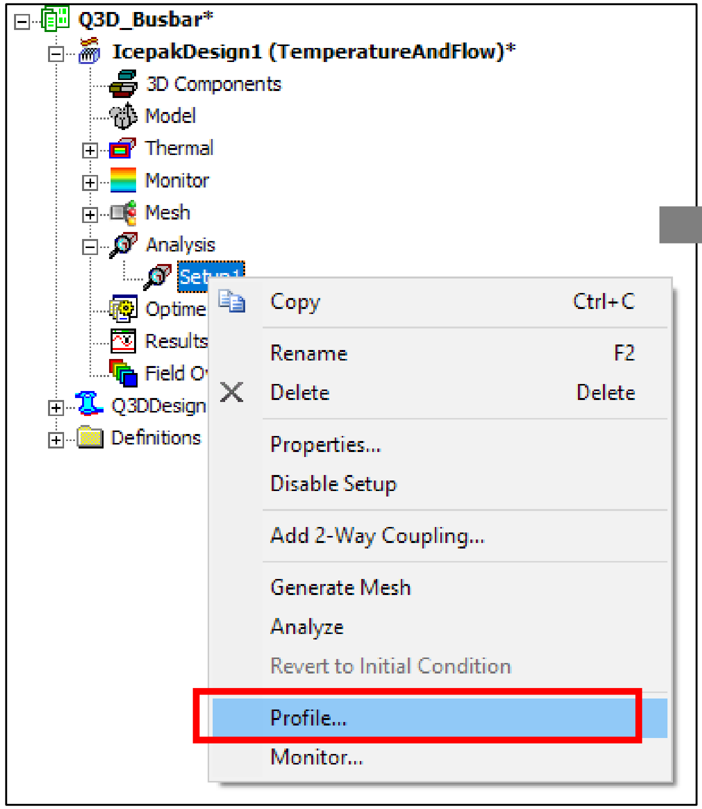
- **To review Solution Monitors and residual plots:**
  - **Right click Setup1 and select Monitor**

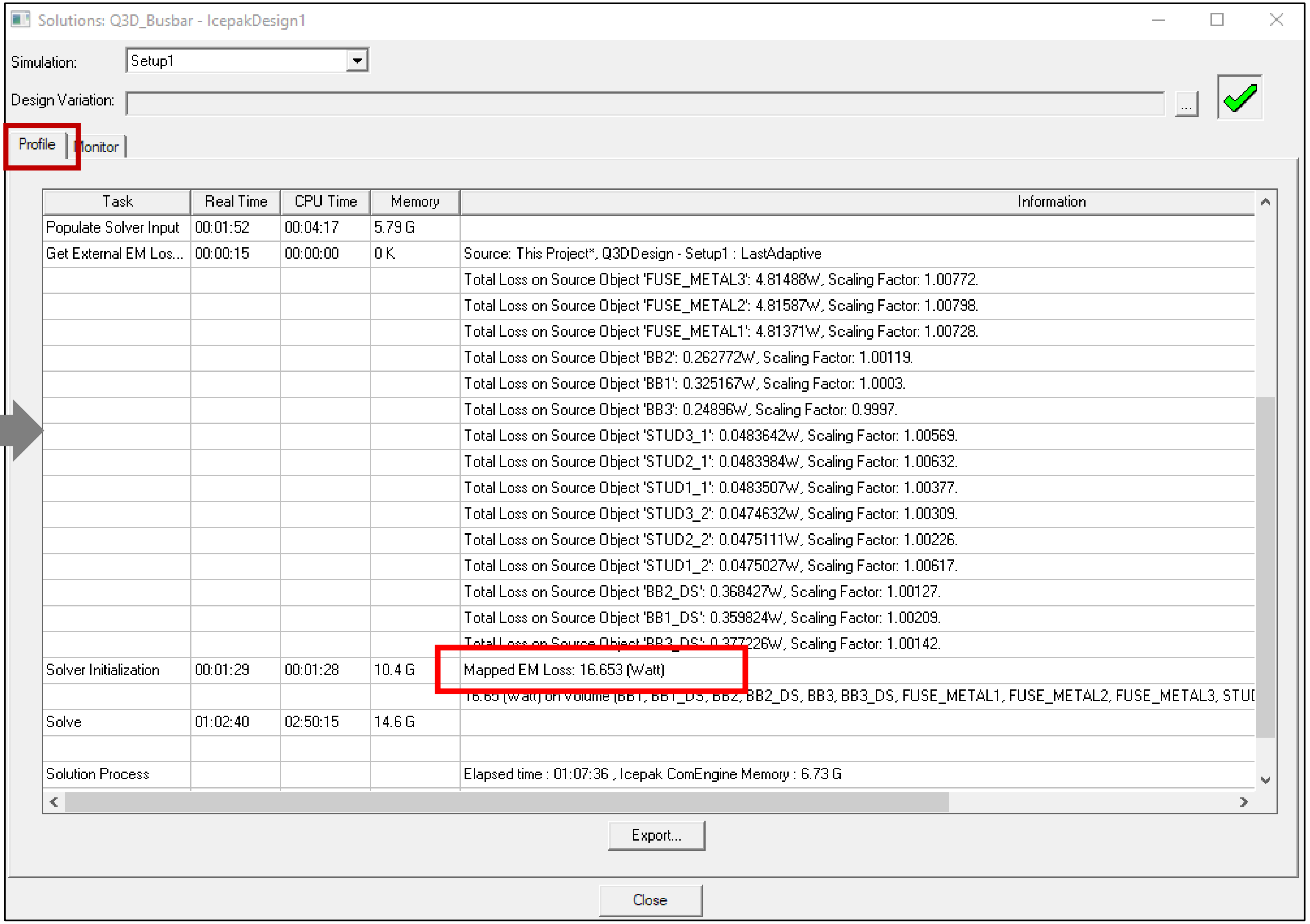




Review Solution Profile

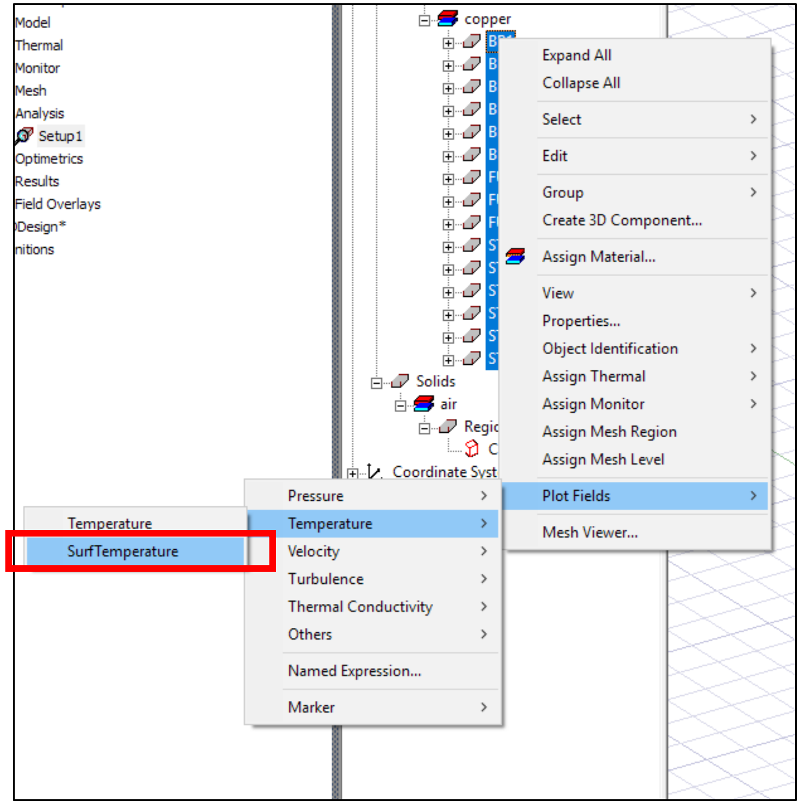
- **To review Solution Profile**
  - **Right click Setup1 and select Profile**
  - **Check the Mapped EM Loss**

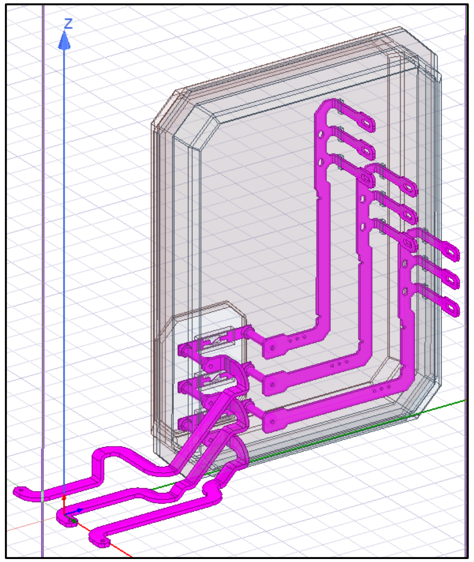


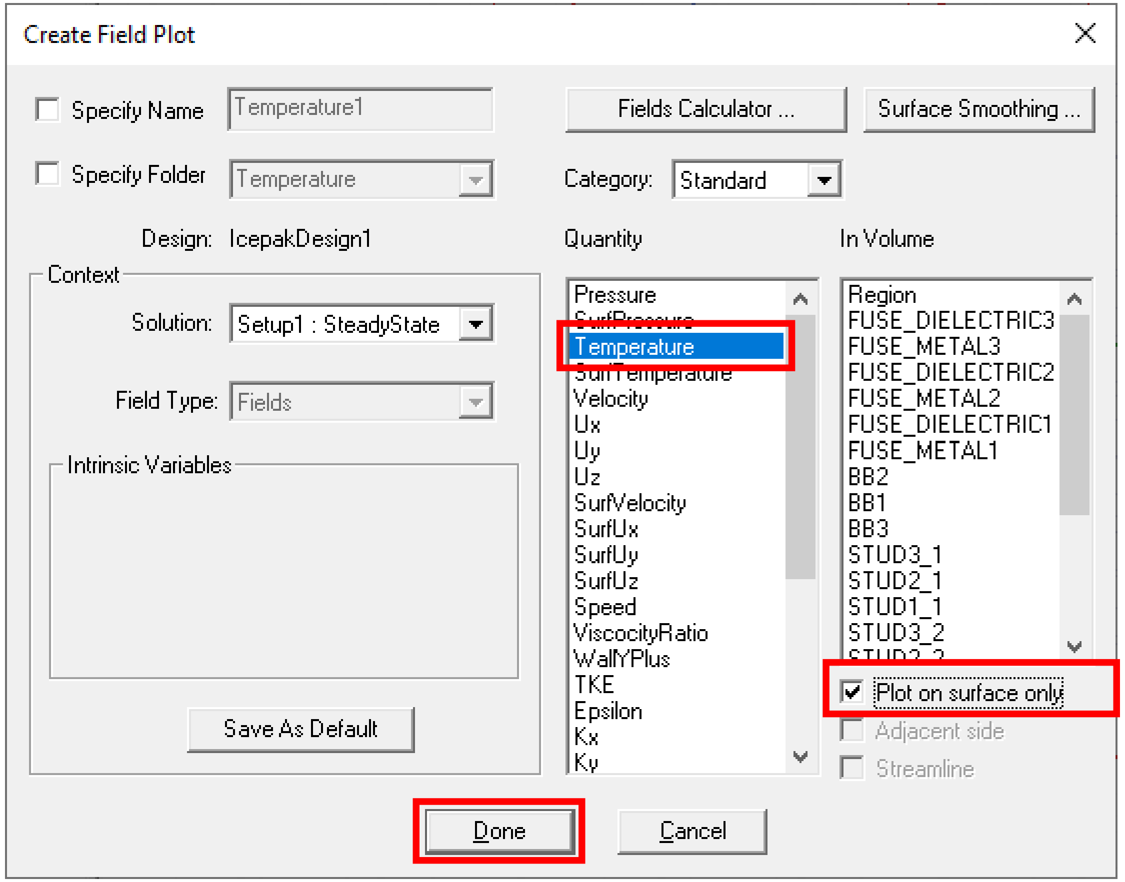


Post-Processing – Contours of Temperature

- **Select any desired Solids for post-processing: RMB click > Plot Fields > Temperature > Temperature**

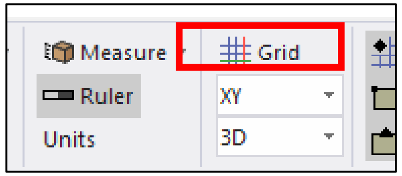


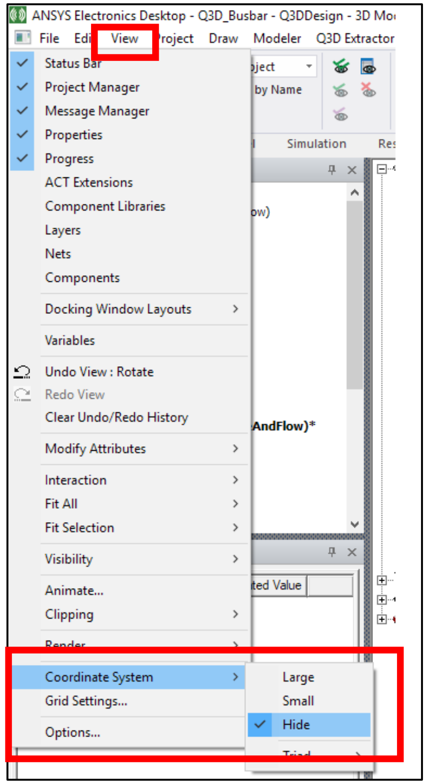


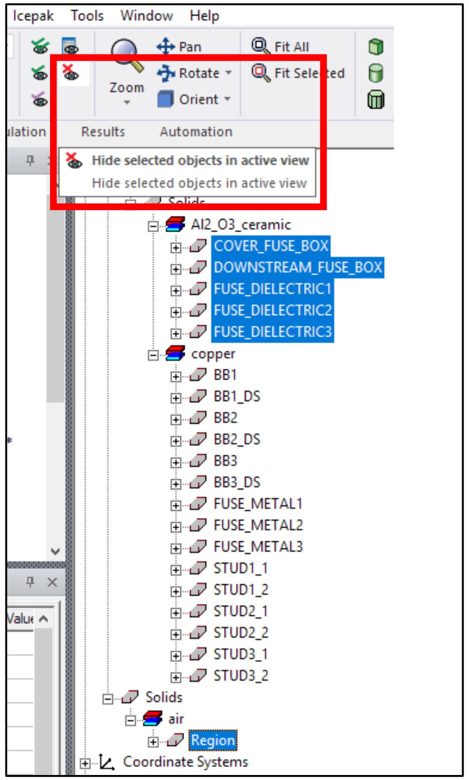


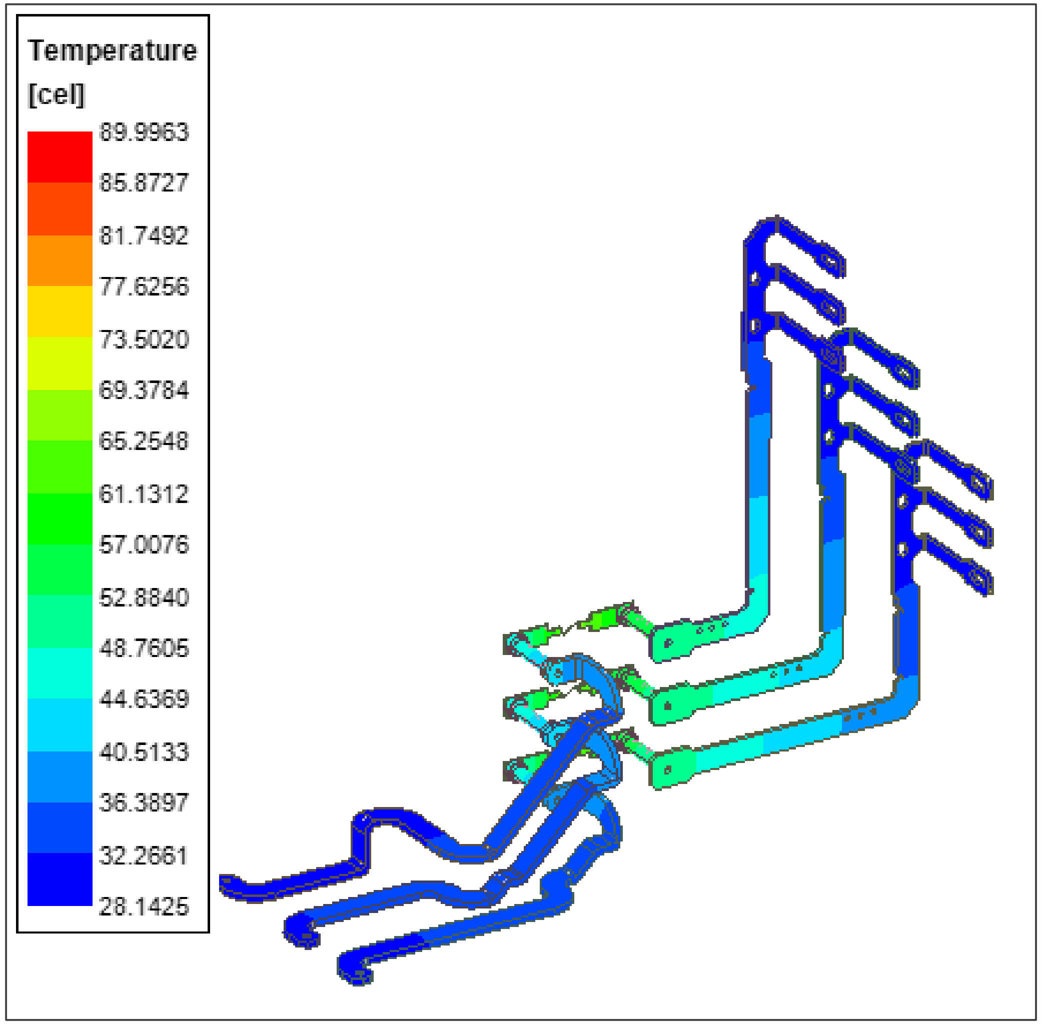
Postprocessing Temperature Contours

- **Optional: user can hide the coordinate system, grids, region, as well as any unused objects for a particular post-processing**





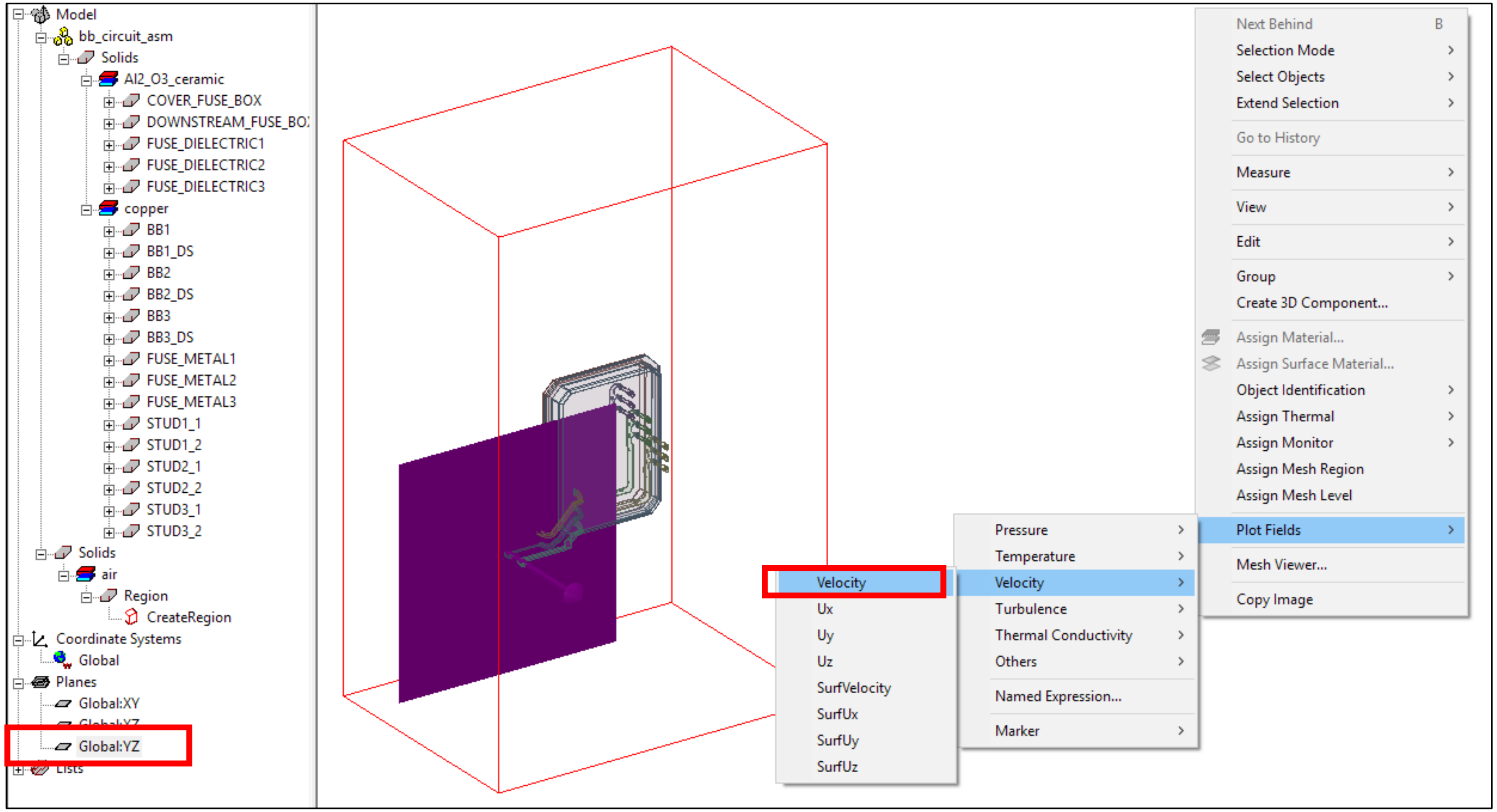




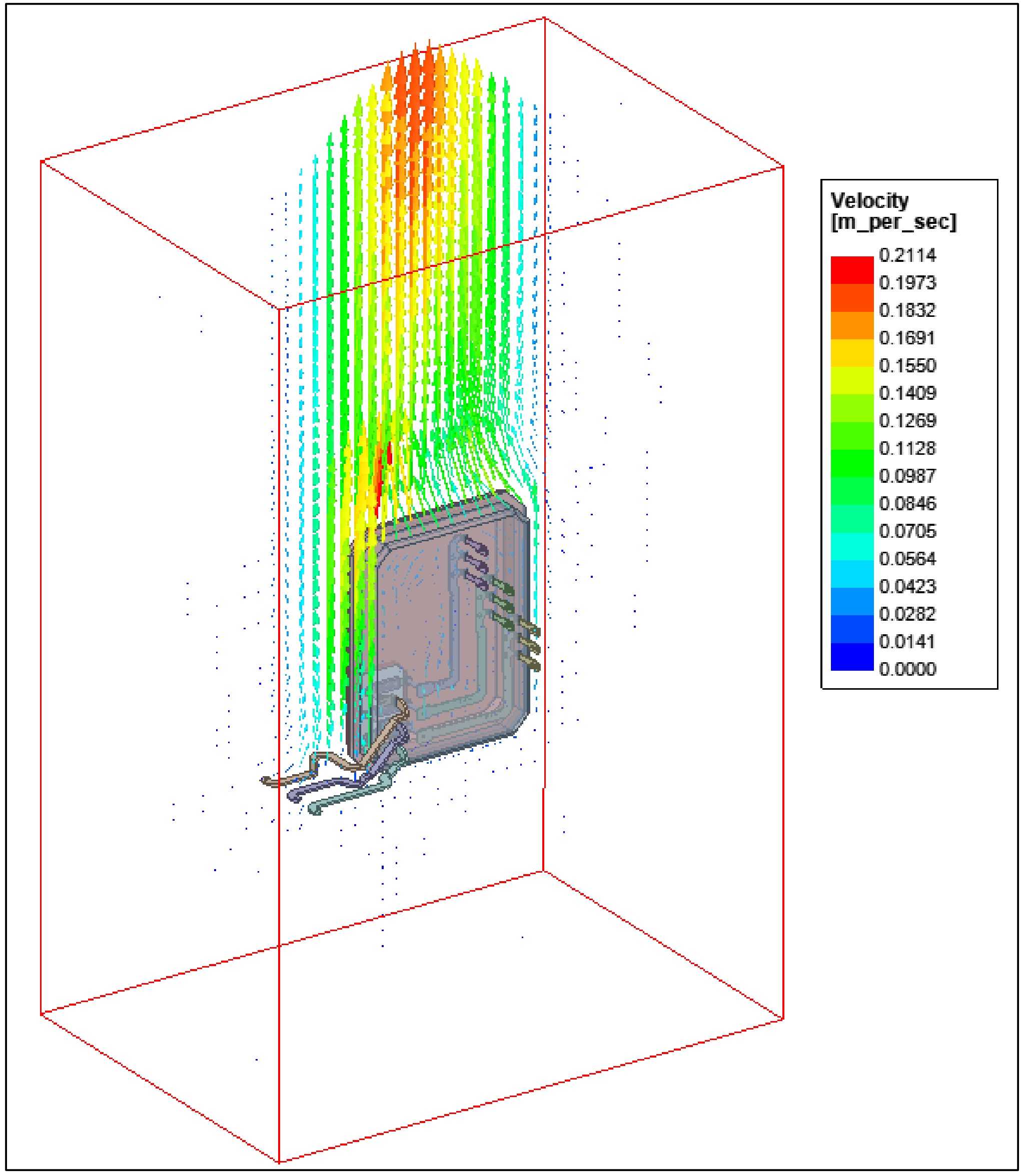
Ansys

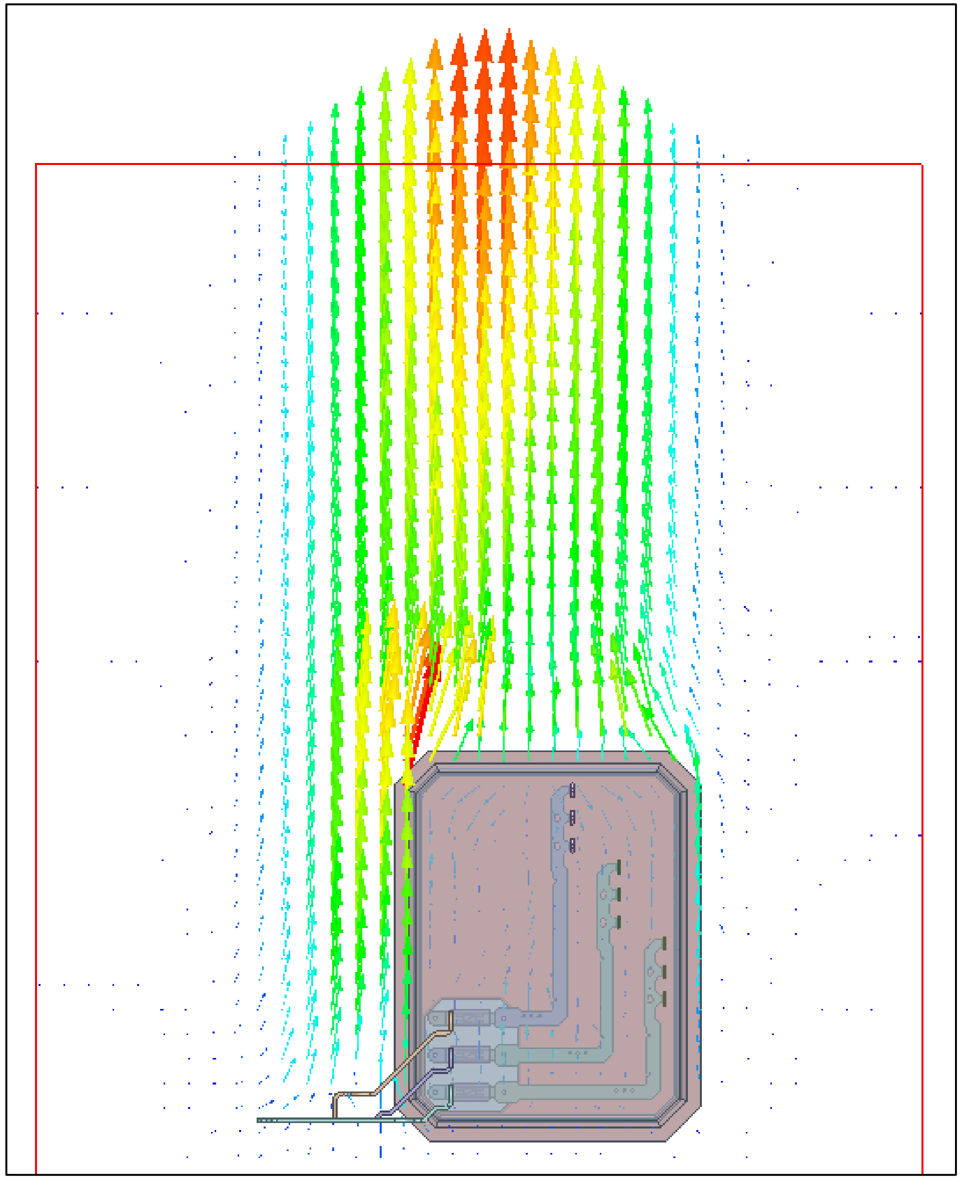
Post-Processing – Velocity Vectors

- **Select the "Global:YZ" Plane to plot the Velocity Vectors**
  - **RMB Click Field Overlays > Plot Fields > Velocity > Velocity**



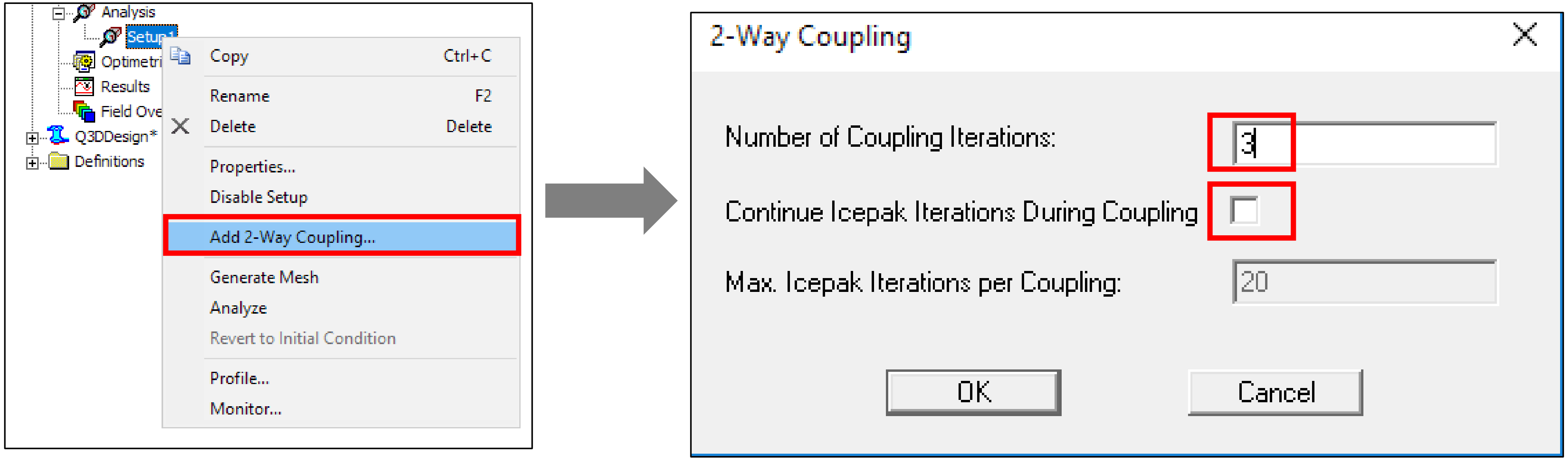
Postprocessing Velocity Vector





Run Analysis with a Two-way coupling to Q3D

- **To enable temperature feedback and run the Q3D design at the correct spatial temperature map, the Icepak simulation must be launched with bidirectional coupling.**
- **To run the Icepak analysis with a 2-way coupling to Q3D:**
  - **Right click Setup1 and select “Add 2-Way Coupling”.**
  - **In the new panel, set “Number of coupling iterations” to 3.**
  - **Keep the “Continue Icepak Iterations During Coupling” unchecked. This option offers full restart and accelerates convergence in the next rounds but it only works for forced convection at this stage.**



Comparison of One-Way and Two-Way Coupling – EM Losses

- **2-Way coupling clearly show a higher EM Loss generation and higher temperature values.**
- **For most accurate solution, a 2-way electro-thermal analyses until full convergence must be conducted.**
  - **In Q3D simulation at the ambient temperature, the total power was 16.65 W**
  - **With the two-way coupling, the total power will increase to 20.1 W, 20.9 W, and 21.0 W. Therefore, 3 iteration is enough to call this convergence as the difference between iteration 3 and 4 is negligible (<0.5%).**

Coupling Iteration 1

Solver Initialization	00:01:03	00:01:02	11.1 G	Mapped EM Loss: 16.653 (Watt)
				16.65 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FU
Solve	00:17:06	02:15:28	15 G	

Coupling Iteration 2

Solver Initialization	00:01:05	00:01:05	11.1 G	Mapped EM Loss: 20.115 (Watt)
				20.11 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FUSE_METAL2, F
Solve	00:16:49	02:13:27	0 K	

Coupling Iteration 3

Solver Initialization	00:01:03	00:01:03	11.1 G	Mapped EM Loss: 20.88 (Watt)
				20.88 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FUSE_META
Solve	00:17:10	02:15:58	14.9 G	

Coupling Iteration 4

Solver Initialization	00:01:03	00:01:03	11.1 G	Mapped EM Loss: 21.048 (Watt)
				21.05 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FUSE_ME

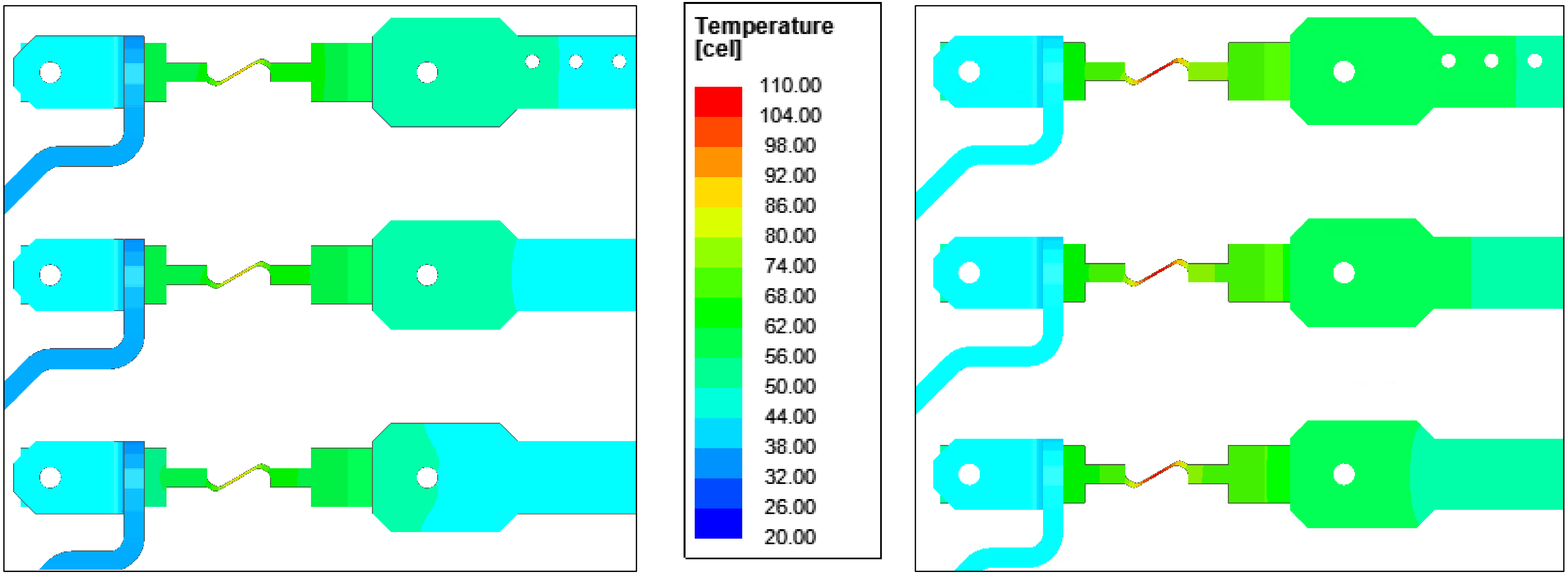
Ansys

One-Way versus Two-Way Coupling - Temperature

- **20^{o}C difference in the fuse maximum temperature (89.9^{o}C vs. 109.9^{o}C)**

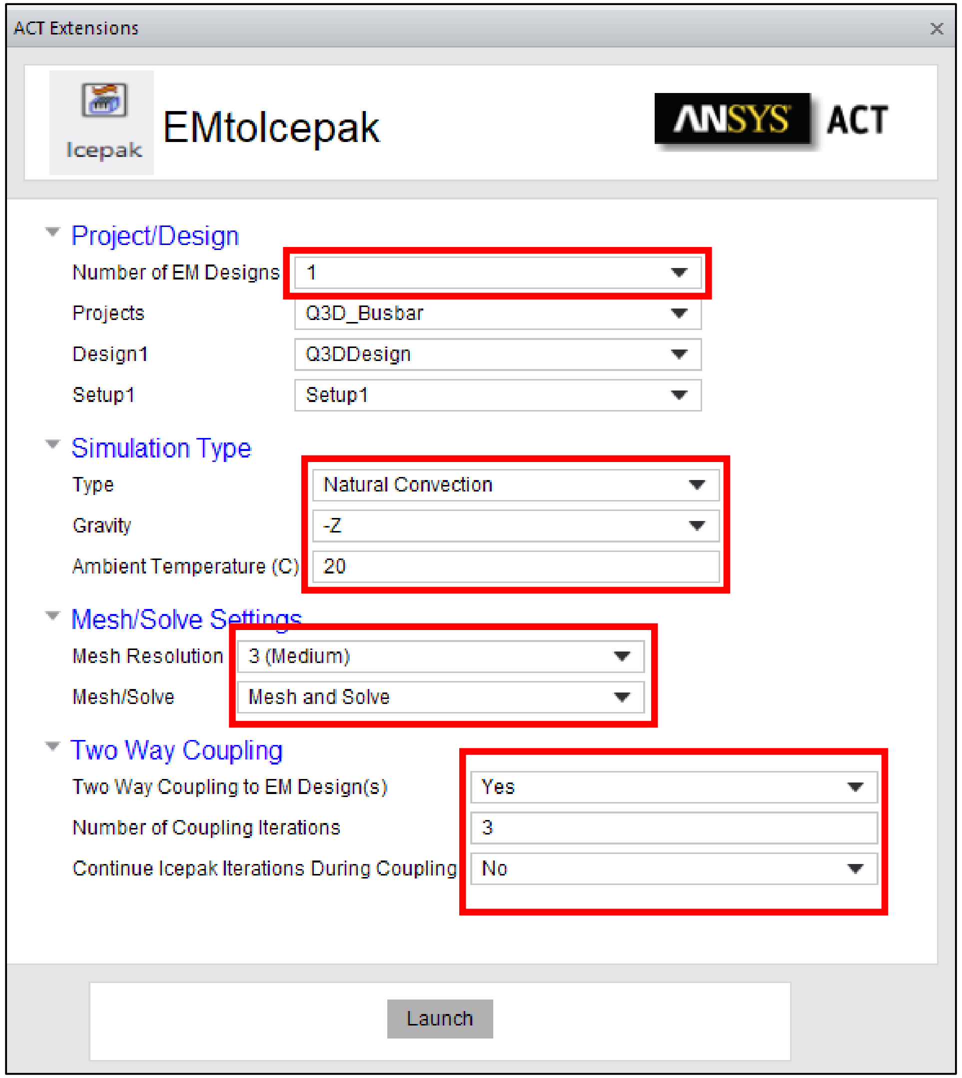
Coupling Iteration 1

Coupling Iteration 3



Running the Electro-Thermal ACT (Automated Workflow)

- **All the aforementioned steps to create a thermal solution can be automated using the electro-thermal ACT.**
- **After installation of the ACT (Refer to ANSYS App Store and its help document), it can be launched for a complete automated 2-way analyses from modeling building to post-processing.**
  - **Set number of designs to "1" ; Q3D design and setup will be automatically be populated**
  - **Choose "Natural Convection" under Simulation type**
  - **Set Gravity direction to "–Z"**
  - **Set ambient temperature to 20°C**
  - **Under Mesh/Solve select "Mesh and Solve"**
  - **Under Two-Way Coupling, select "yes" and set number of iterations to "3"**
  - **Set "Continue Icepak Iterations…" to "no"**



Summary

- In this Workshop, key learnings are as follows
  - Performing Q3D DC RL analysis.
  - Setting up temperature dependent material in Q3D.
  - Copying and Using Q3D Model and Material settings in to Icepak.
  - Transferring Volumetric losses from Q3D to Icepak.
  - Setting up natural convection case in Icepak with a one way and two-way coupling to Q3D.
  - Postprocessing the temperature and flow field in Icepak.
  - Running the Electro-thermal ACT for an automatic bidirectional analyses.

End of presentation