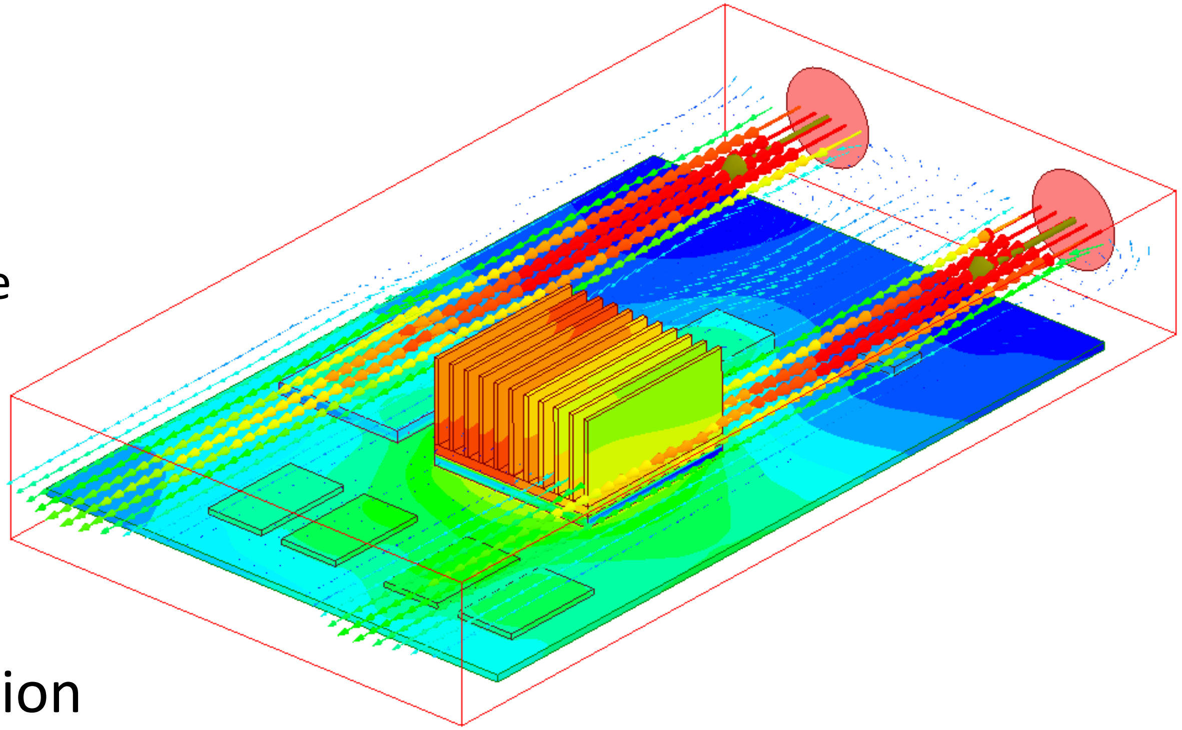
Module 1 – Workshop 1: Model Building in AEDT Icepak

Release 2020 R1

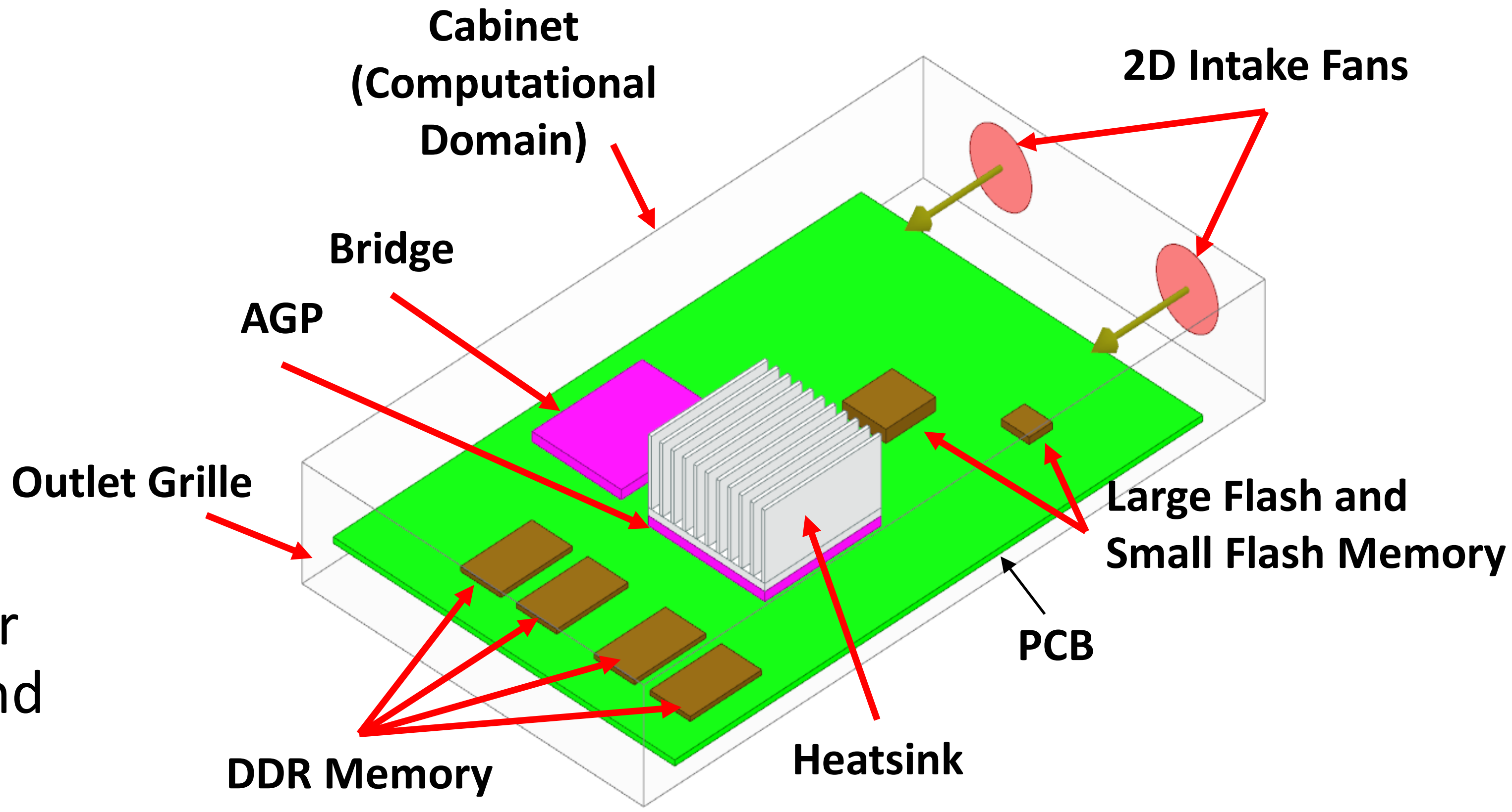
Objectives

- Build a simple model in AEDT Icepak
  - Create heat sinks and fans
  - Assign boundary conditions
- Generate and review the mesh
  - Use the automatic slider-bar mesher to generate the mesh
  - Review the mesh for resolution and quality
- Setup the model
- Analyze the simulation
- Use post processing tools to analyze the solution



Model Overview

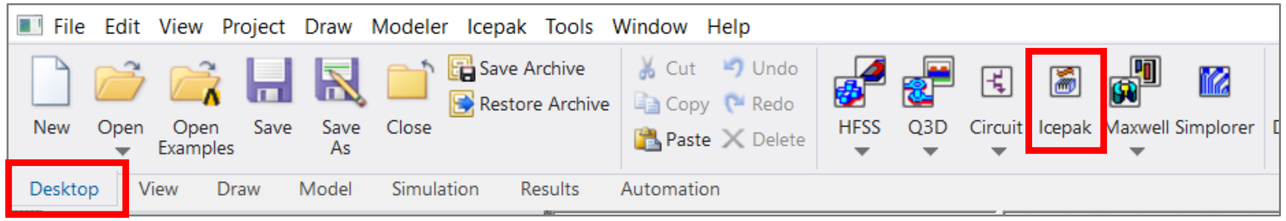
- The model has the following generic components
  - Intake fans
  - Outlet grille
  - Board
  - Heatsink
  - GPU
  - DDR memory
  - Flash memory
- A STEP file will be imported for the model and the heatsink and fans will be added



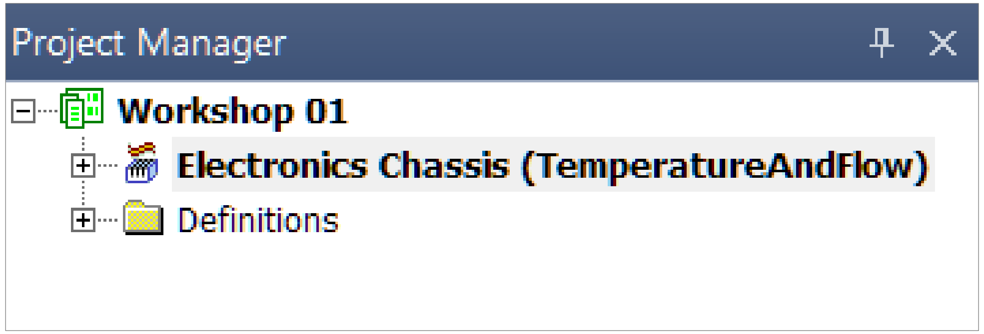
Ansys

Starting AEDT Icepak

- Create a new Icepak project in AEDT as follows:
  - Launch ANSYS Electronics Desktop (AEDT) from Windows start menu or the desktop shortcut
    - Start → All Programs → ANSYS EM Suite 2020 R1 → ANSYS Electronics Desktop 2020 R1
  - Insert a new Icepak project from the ribbon or the dropdown menu
    - Project → Insert Icepak Design
- Use Workshop 01 as the project name
- You can rename Icepak Design to Electronics Chassis
  - Right click on IcepakDesign1 and click Rename

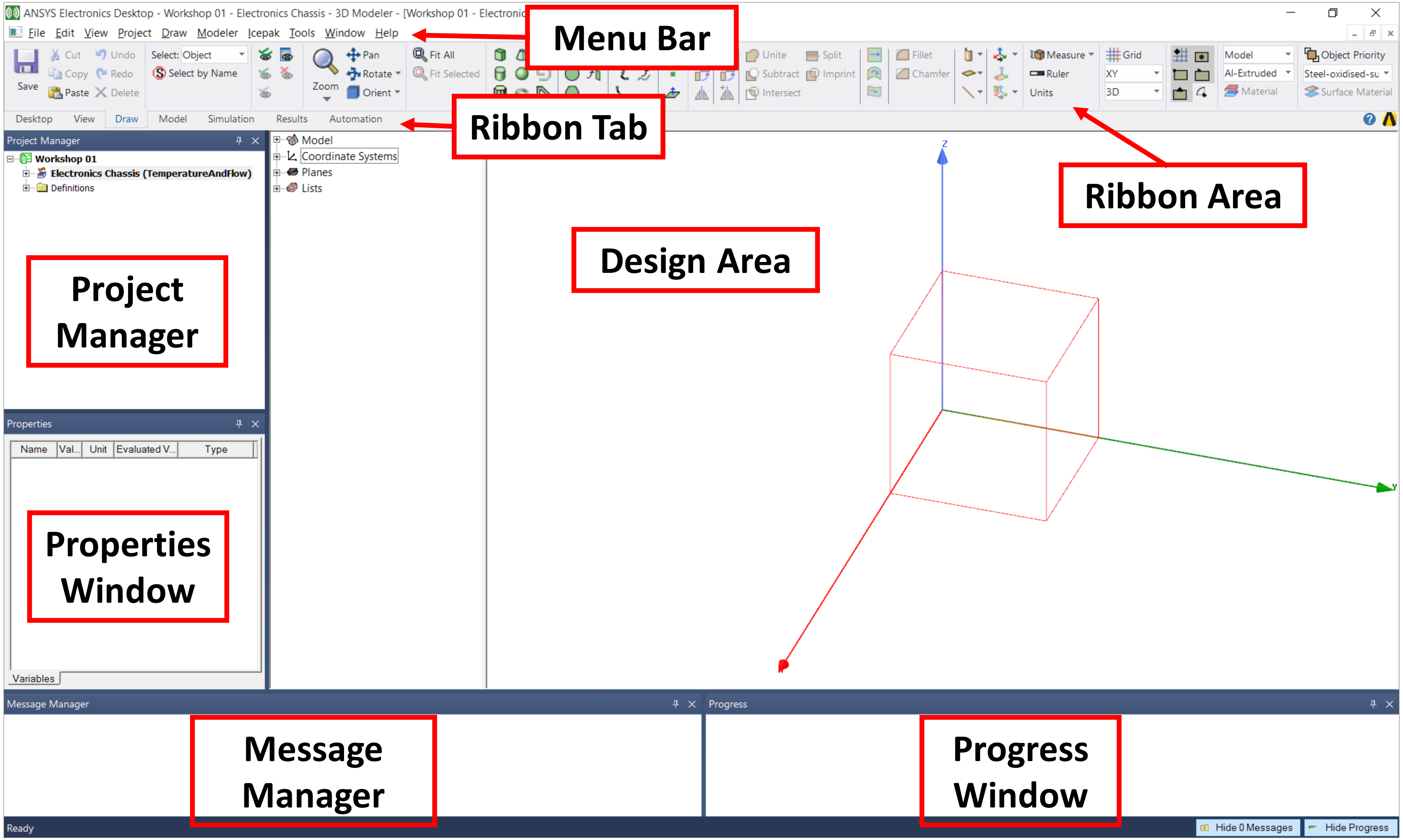


Select Icepak project under the Desktop tab



Change the project name and design name

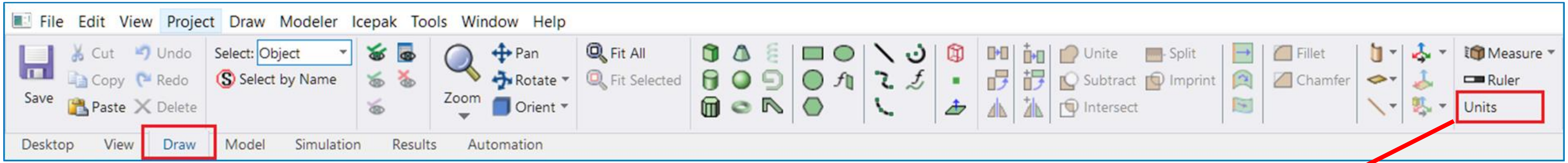
ANSYS Electronics Desktop Interface



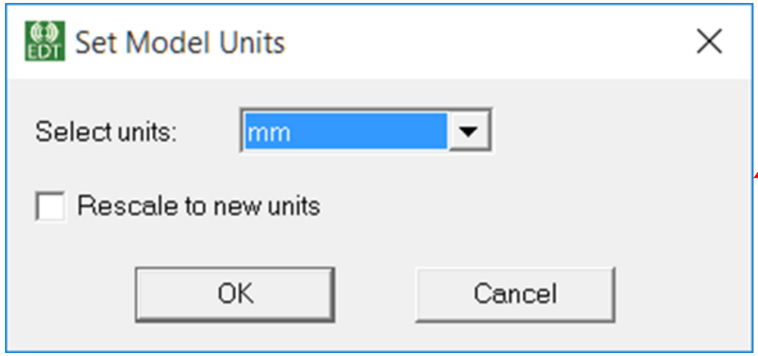
Set Default Units for Icepak Project

- Set Model Units panel can also be accessed from Modeler → Units

Navigate to Draw tab and click on Units



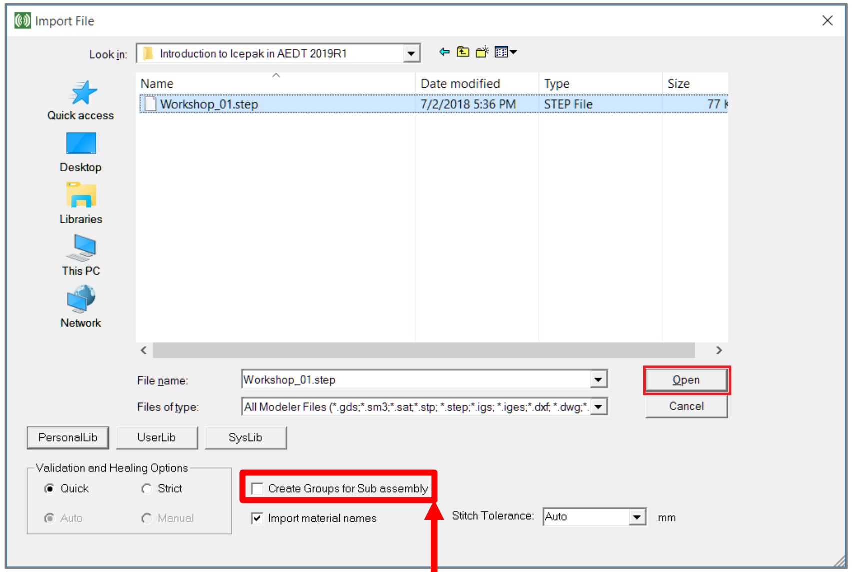
Set the units to mm



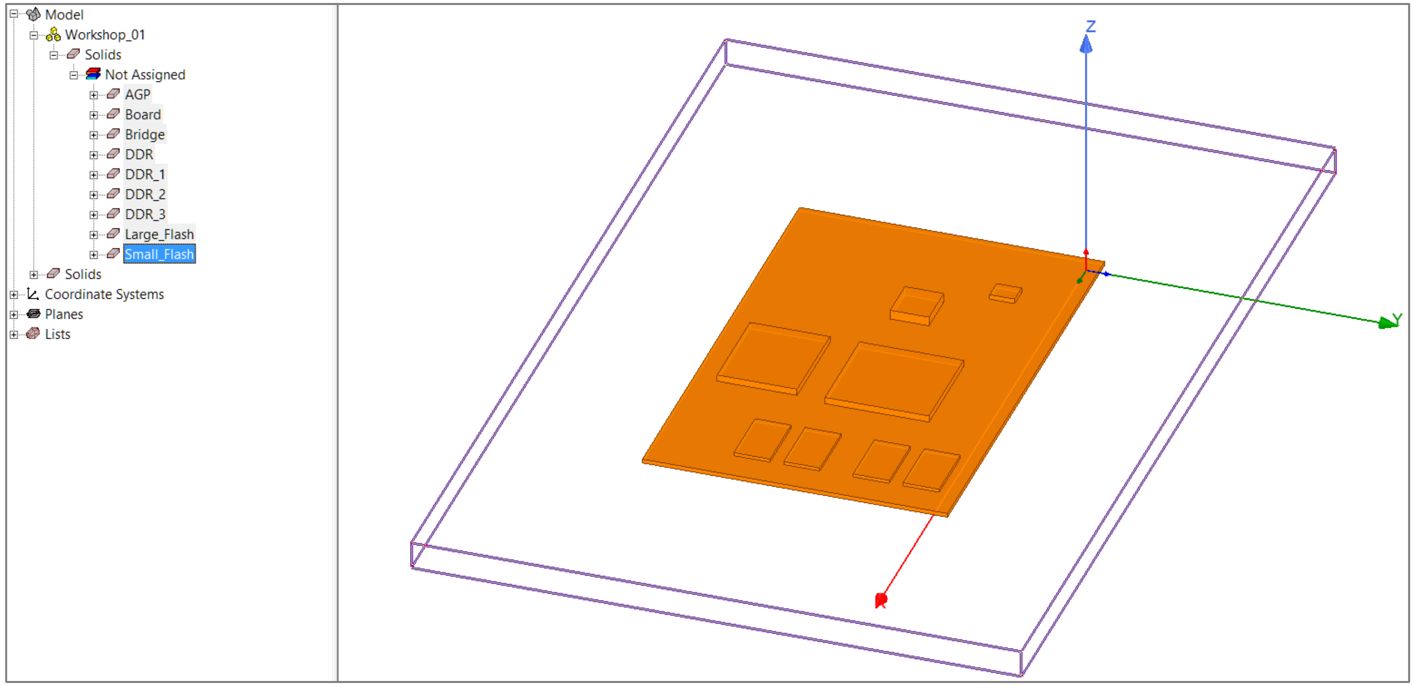
Ansys

Import the Geometry and Setup Domain

- Click on Modeler in the dropdown menu and click Import
- Select the file **Workshop_01.step** from the workshop directory, confirm that 'Create groups for assembly' is unchecked and click on 'Open'
- To switch between wireframe and smooth shade mode, press F6 (wireframe) and F7 (smooth shade)

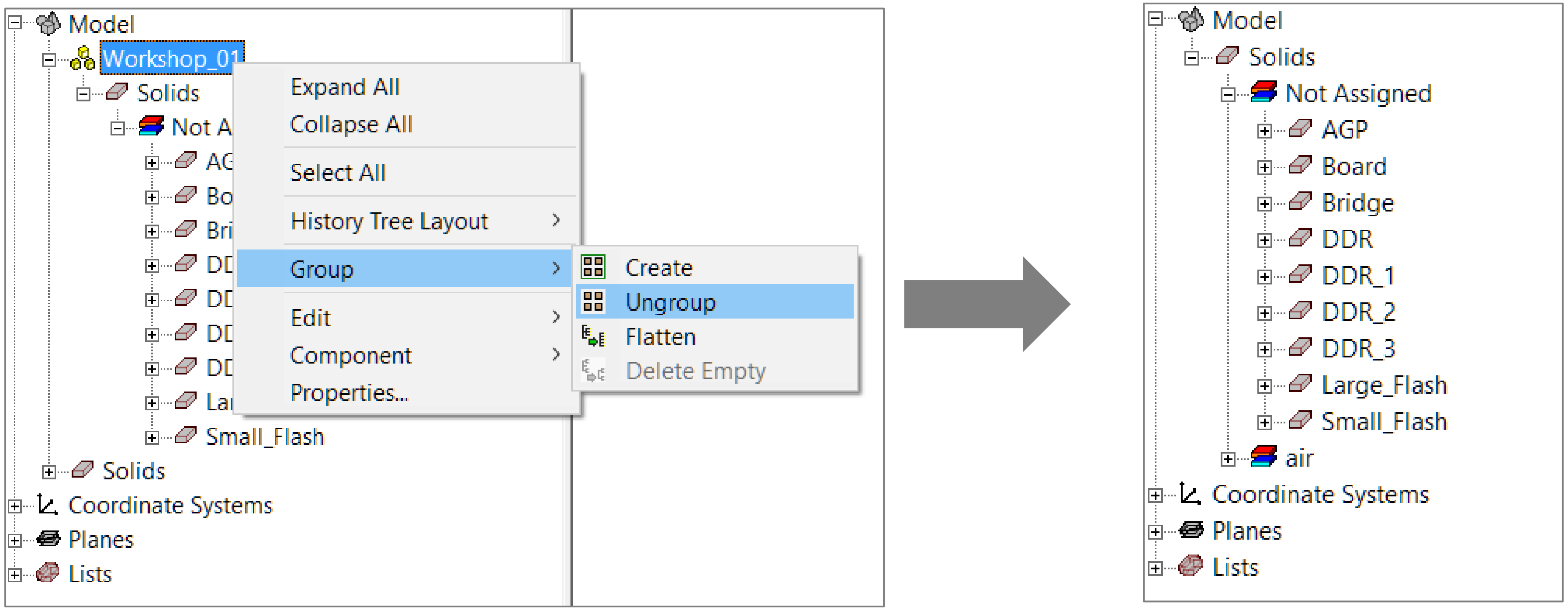


Uncheck Create Groups for Sub Assemblies



Group and Ungroup Objects

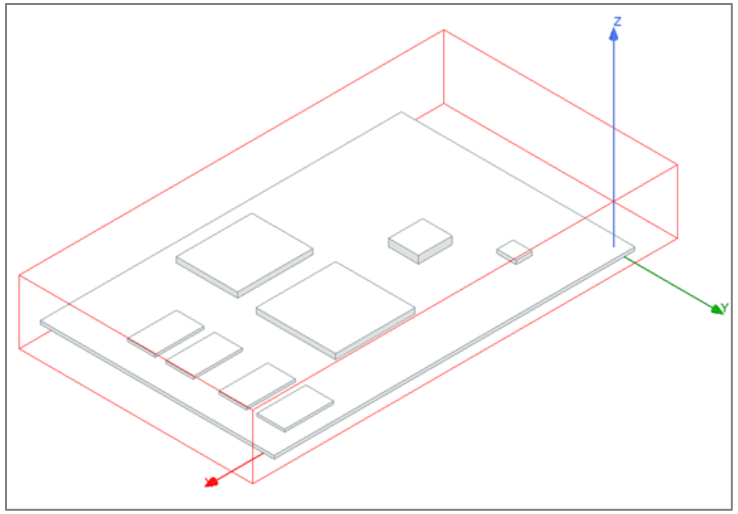
- The 3D Modeler allows you to group objects in the History tree
- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups
- Ungroup the group named Workshop_01 to bring all objects under the Solids node
- To ungroup, right-click on Workshop_01 → Group → Ungroup



Solution Procedure in AEDT Icepak

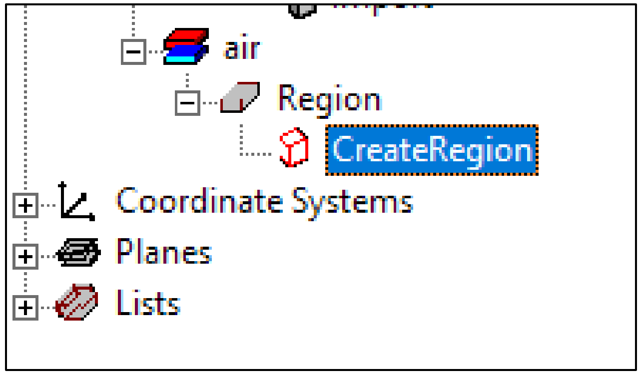
- Domain setup and model preparation includes all steps that are followed to setup the geometry such as creating fans, heat sinks, etc.
- Boundary condition assignment includes flow boundaries such as openings, grilles, etc. and heat sources such as network thermal resistance, power, etc.
- Material properties such as density, specific heat, thermal conductivity, viscosity, etc. tells the solver how much heat is retained and dissipated out of an object
- Meshing is a critical step as all the governing equations are solved in the computational cells created during meshing process
- Solver setup tells the solver how long the problem should run and under what settings
- Once the solution is obtained the post processing step helps the user to make sense of the simulation by analyzing the results

Domain Setup

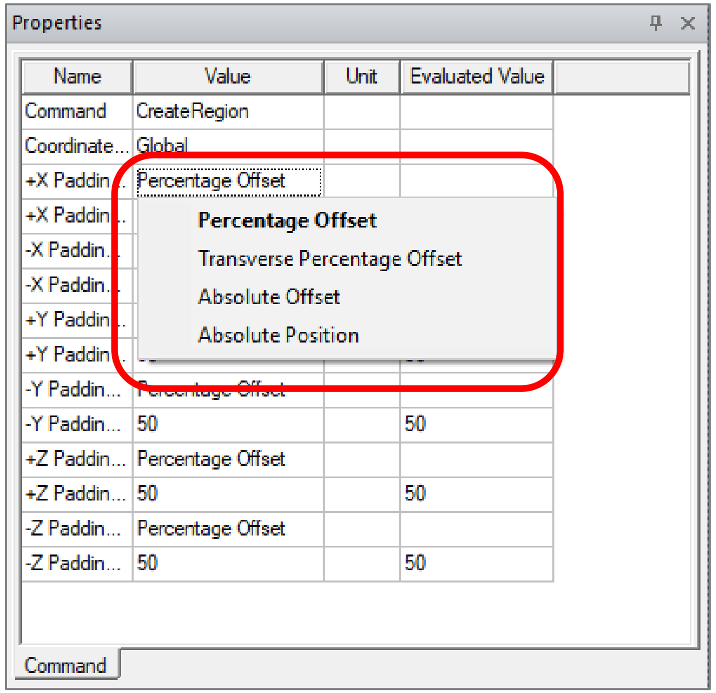


- Right-click on Solids and select Expand All
- Select CreateRegion under air
- In the Properties panel, input the padding values shown below
- Notice that some inputs use values by <u>Absolute Position</u>, and others by <u>Absolute Offset</u>.

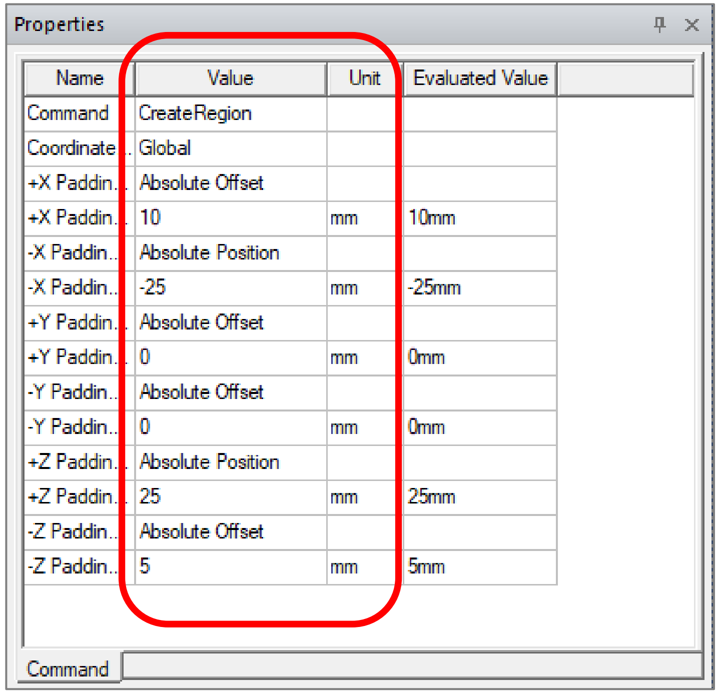
1



2



3

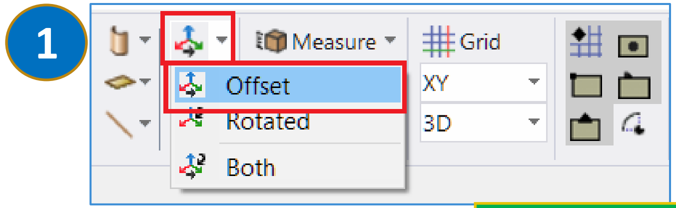


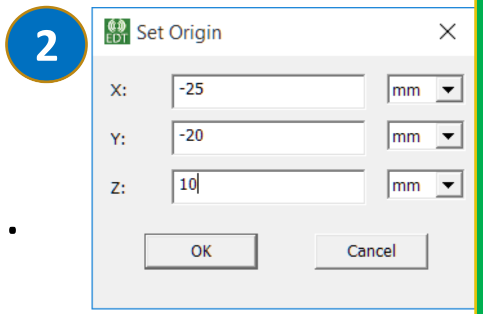
Ansys

Create Intake Fans

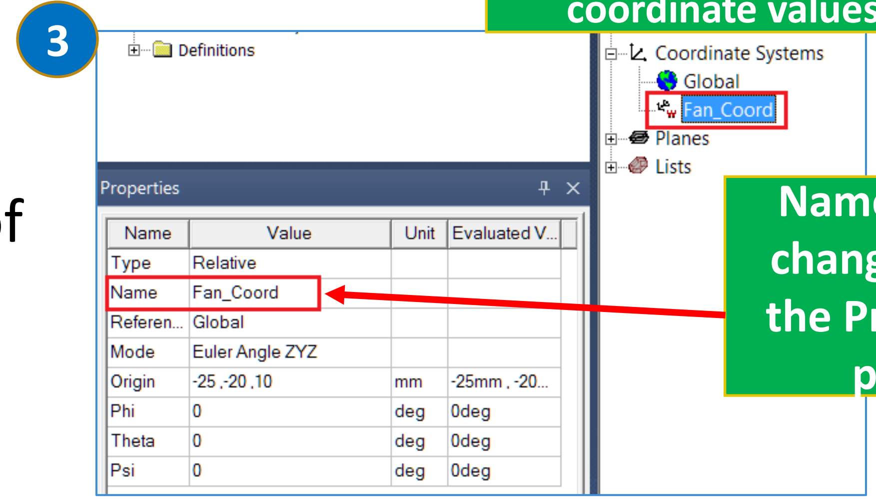
Fans will be added by creating relative coordinate systems (CS) and placing the fan object on the Relative CS

- Choose the Global Coordinate System from the Coordinate Systems node
- Click on Offset Origin Icon in the Draw ribbon.
- Create relative coordinate system by offsetting origin at -25, -20, 10
- (Alternatively use Modeler → Coordinate System → Create → Relative CS → Offset)
- Press F4 and enter the values for the origin of the relative coordinate system as shown
- Rename the relative coordinate as ‘Fan_Coord)





- Toggle between point and dialog entry mode by pressing F3/F4
- Point entry mode allows you to select a point in the Graphics window
- Dialog entry mode (F4, shown on left) allows you to enter the coordinate values



Name can be changed from the Properties panel

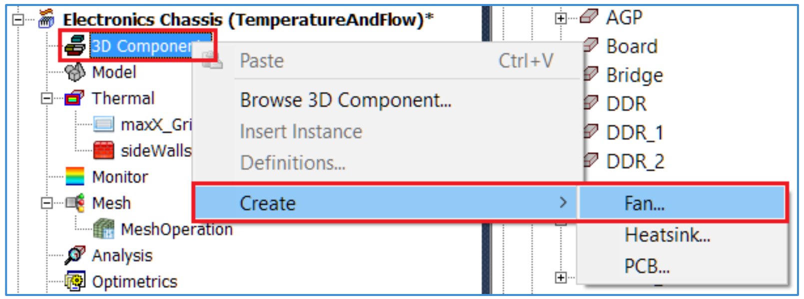
Note: Deleting the relative coordinate system will delete the associated object (in this case, fan)

Ansys

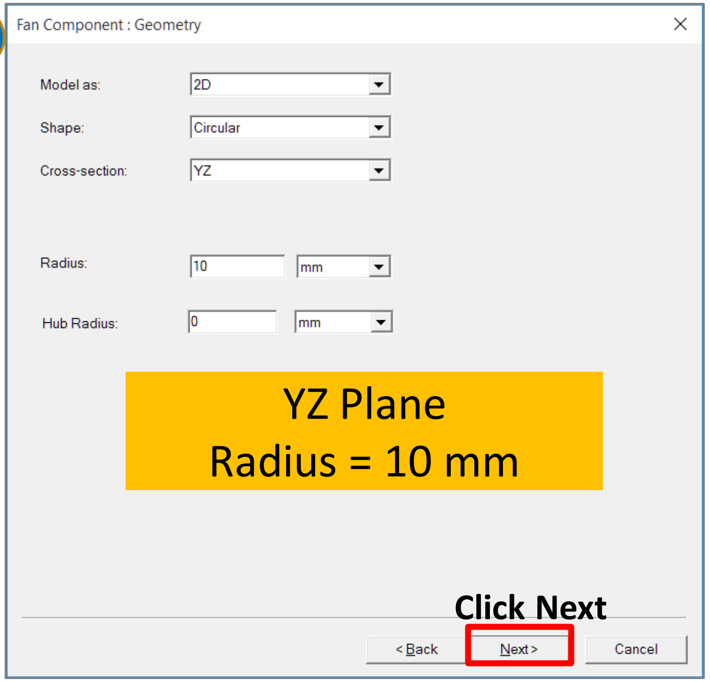
Create Intake Fans

- In the Project Manager, under Electronics Chassis, right click on 3D components and choose Create → Fan…
- Enter 'Fan' for name, click Next and Enter the details as shown and click Finish
- The process will create Fan1 in the Model Tree

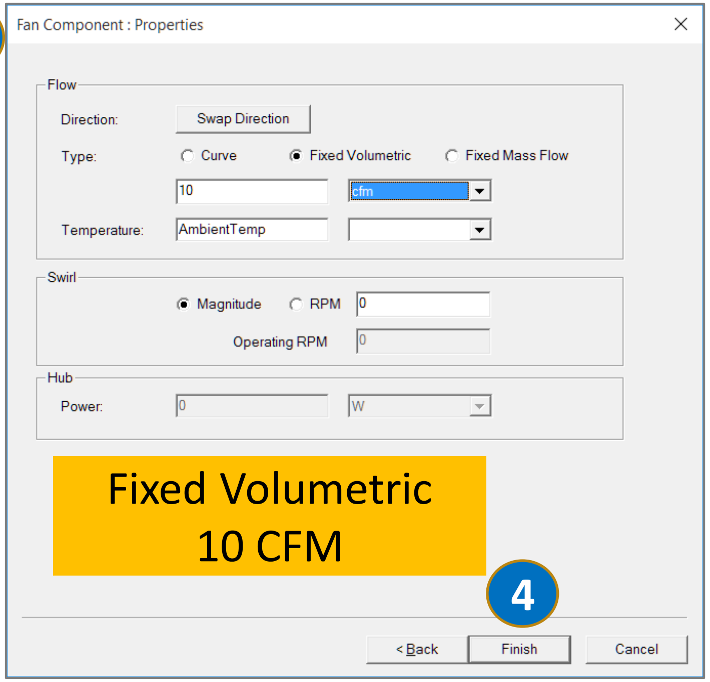
1

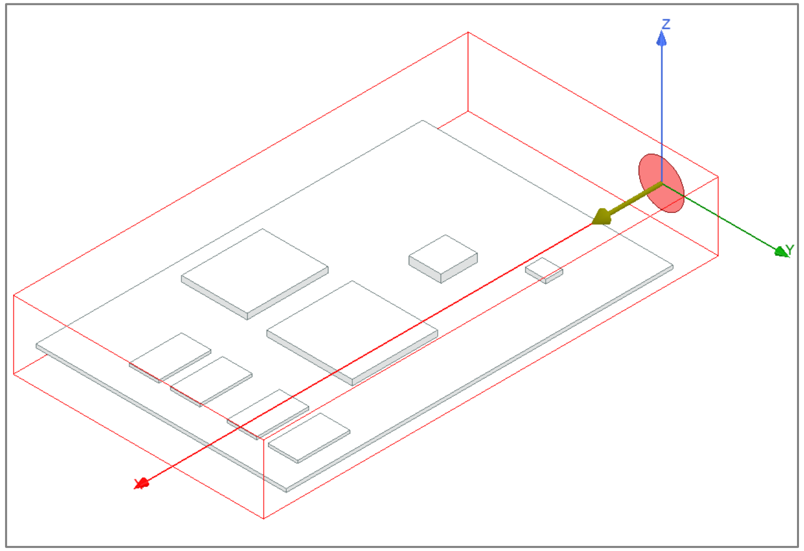


2



3





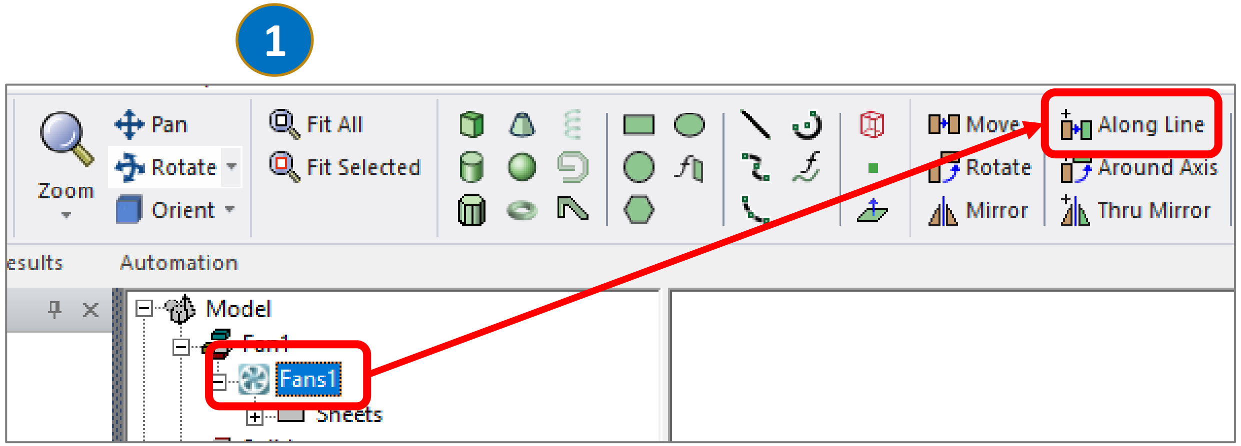
Ansys

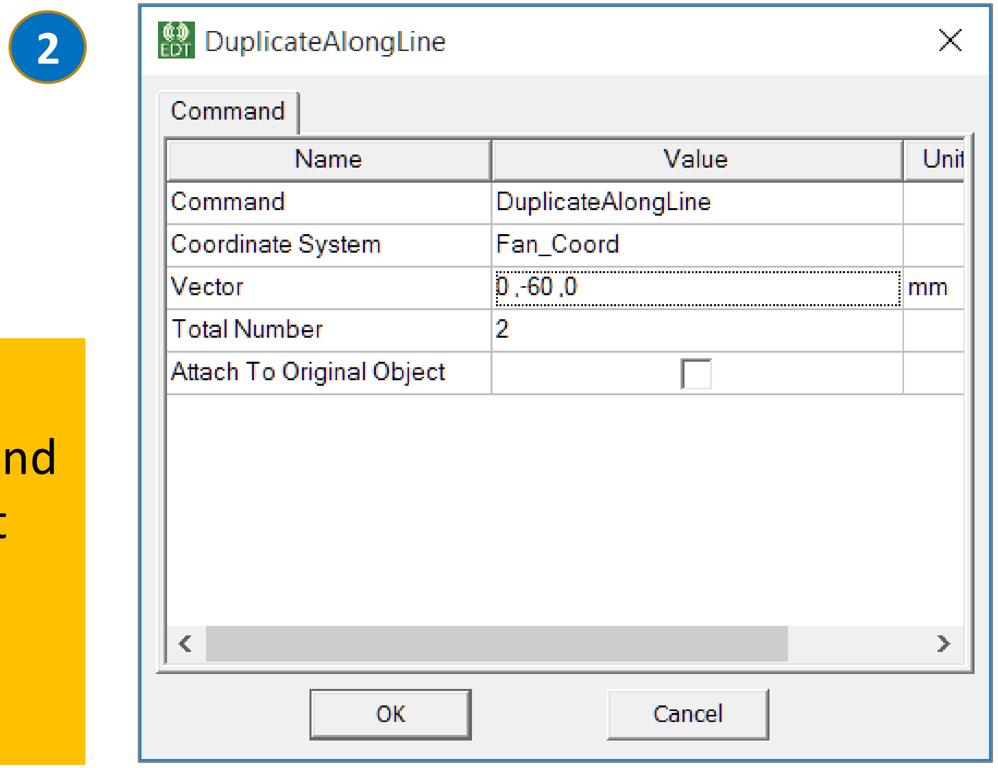
Copy the Intake Fans

- In the Model tree, select Fan1
- In the Draw tab, select Duplicate along line (see image for icon)
- Use the Dialog entry mode (F4) to enter the values
- Translate the fan by -60 mm in the y-direction
- Total Number = 2
- Click OK and it will create Fan2

Note: This method will change the properties of any duplicates/copies of the fan. For example, if you created a second fan by copying the first fan, then changing the flow rate for first fan will change the flow rate for the second fan as well.

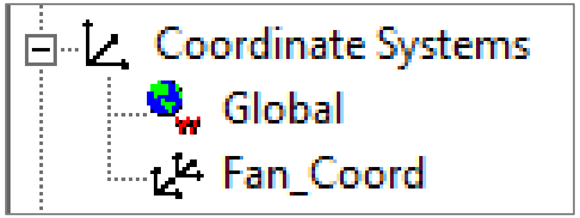
Workaround: Create two distinct fans



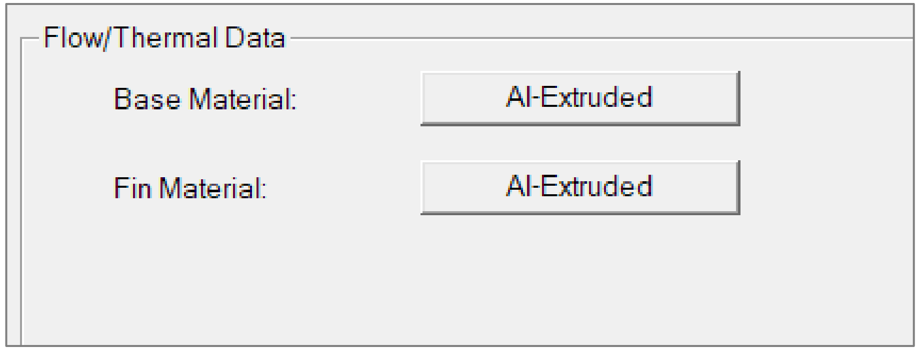


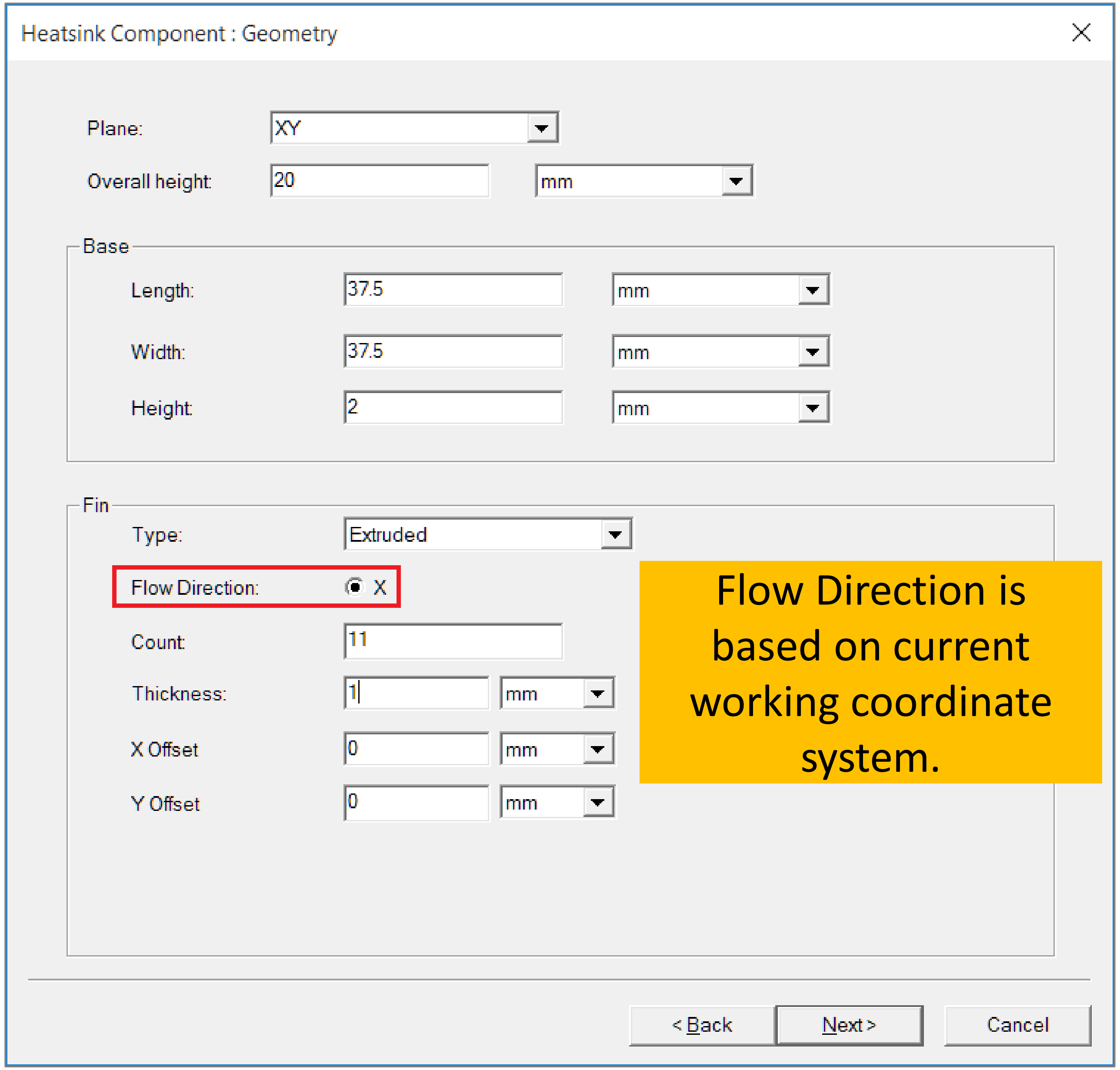
Create Heat Sink

- Select ‘Global’ under coordinate system, right-click on 3D Components and choose Create → Heatsink



- Give the name ‘ Heatsink’ and Enter the values for heatsink dimensions and properties as shown
- Click Next and retain the defaults for materials and click Finish





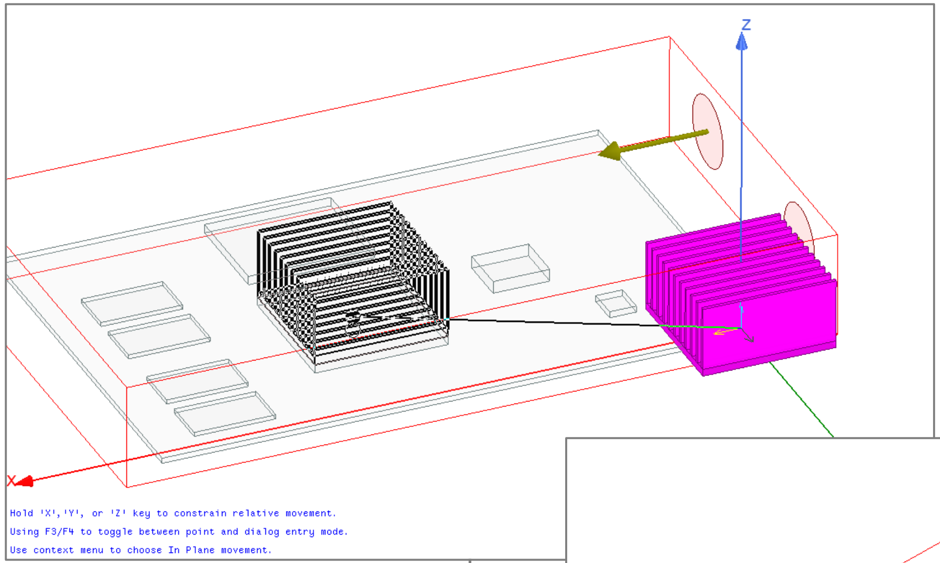
Flow Direction is based on current working coordinate system.

Ansys

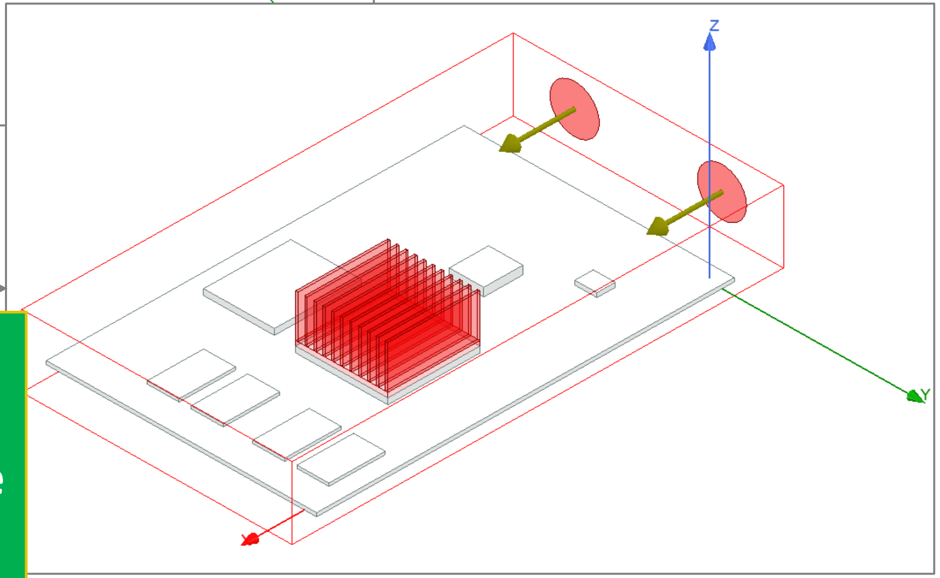
Move the Heatsink

Use F3 / F4 to toggle between point and dialog entry modes.

- Select the Heatsink1 and from the Draw tab, select Move icon
- Hover the mouse pointer at the bottom face of the heat sink until the pointer turns into a circle. Click to select the center of the face
- Next, hover the mouse pointer over the top face of the AGP to search for the face center. When the pointer turns into a circle, click to select the center of the face
- The heat sink will be moved to the new location



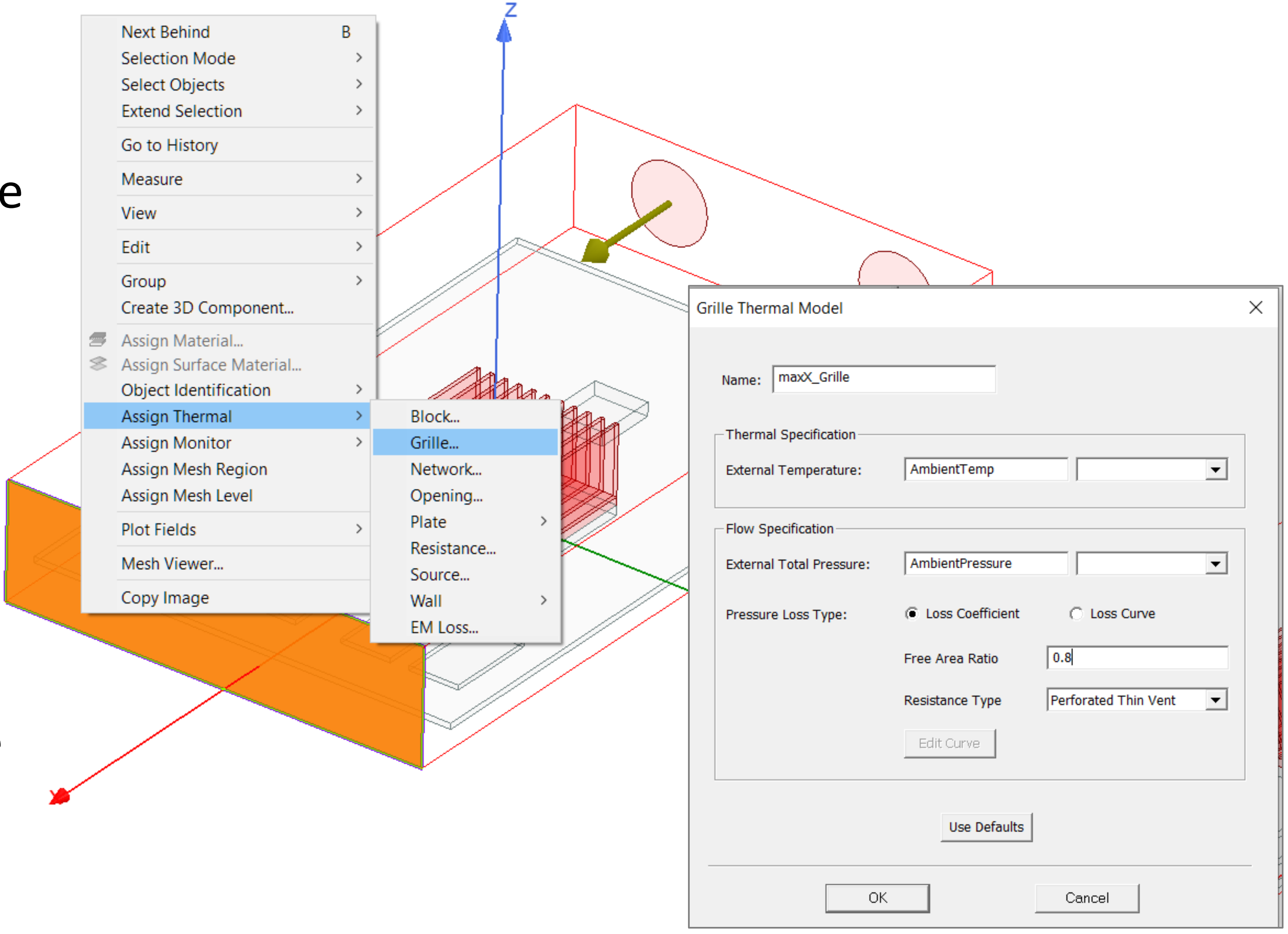
Move objects icon in Draw tab



Note: The heatsink can also be aligned to AGP using any edge at the base of the heatsink as the anchor point

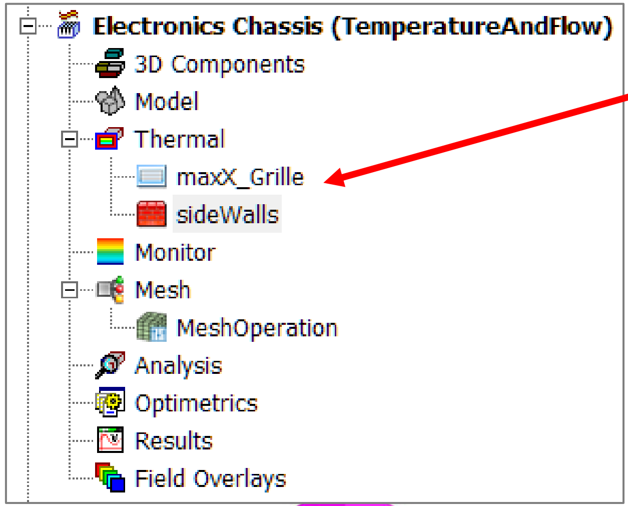
Assign Boundary Conditions – Flow Boundary

- Click on Region under Solids → air
- In the Properties window, change the name to Domain
- Next, press F to change to face selection mode
- Select the max X face of the domain
- Right click and select Assign Thermal → Grille…
- Enter MaxX_Grille for name, use 0.8 for Free Area Ratio and click OK to accept other default settings

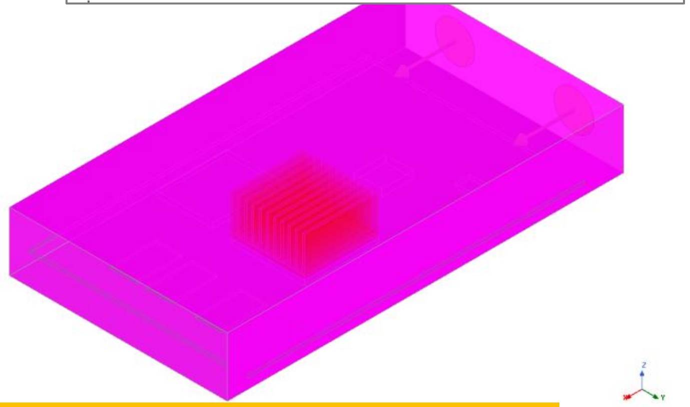


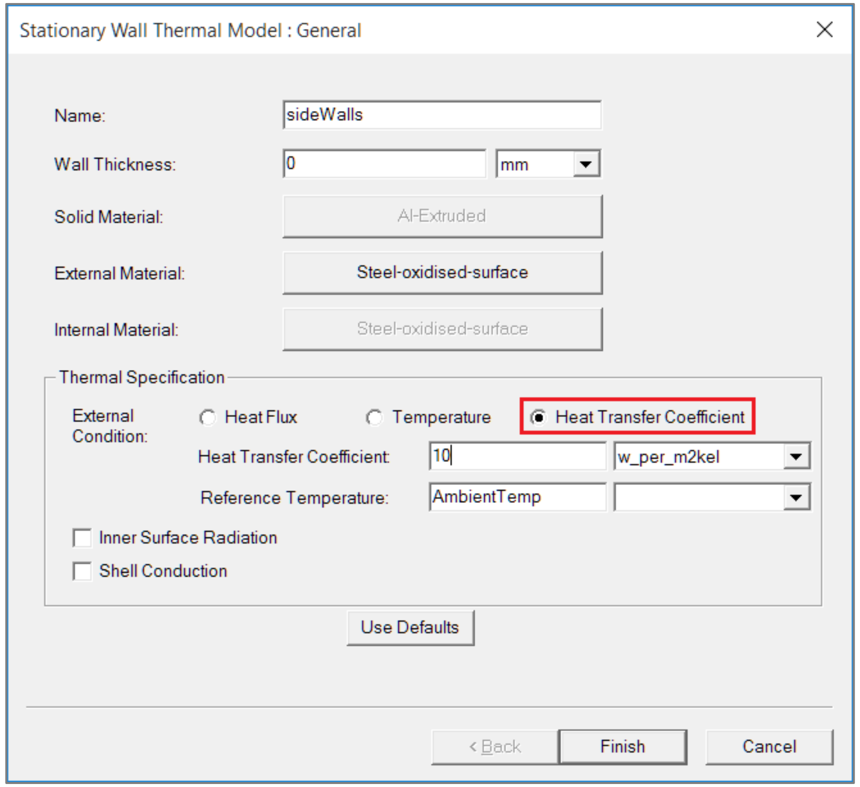
Assign Boundary Conditions – Wall Boundary

- Select the lateral sides of the domain, i.e., minY, maxY, minZ and maxZ
- Right click → Assign Thermal → Wall → Stationary
- Enter SideWalls as name in the panel
- Select Heat Transfer Coefficier under Thermal Specification
- Enter a value of 10 W/(m²K)
- Click Finish



Boundary Conditions appear under the Thermal node in Project Manager

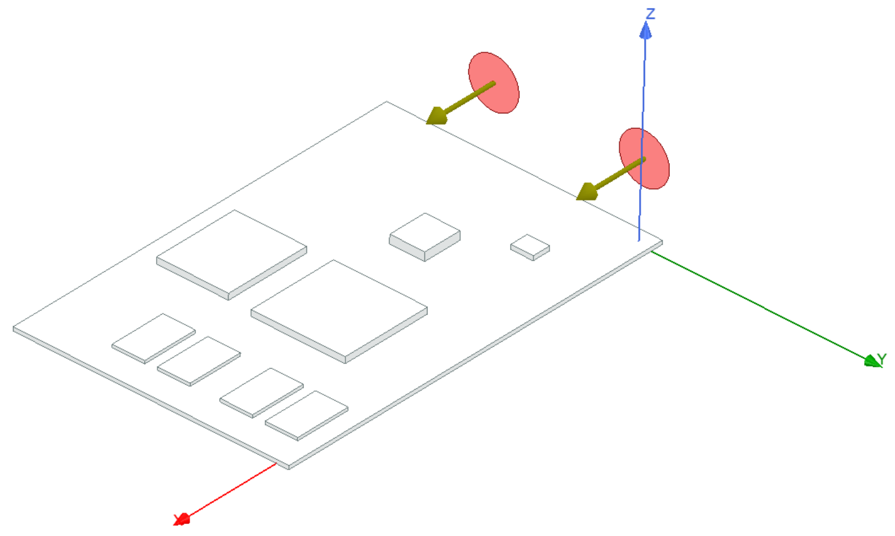


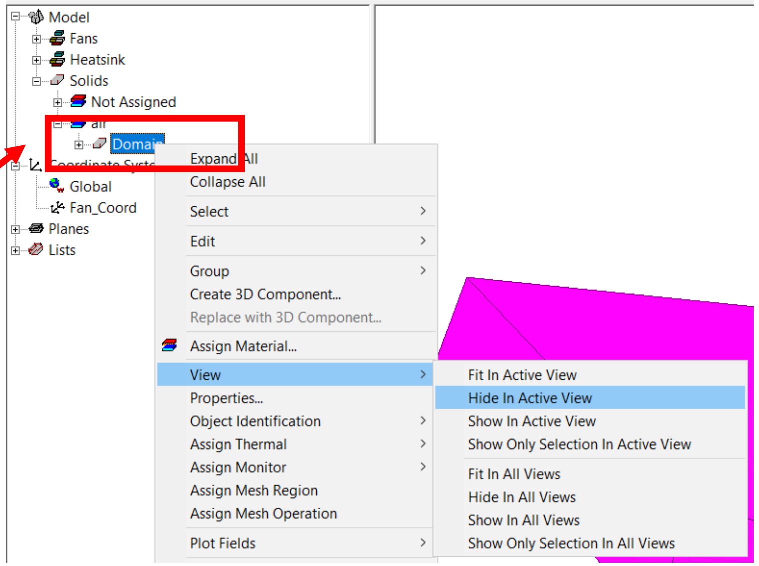


Note: Spaces are not allowed in names

Hiding Objects

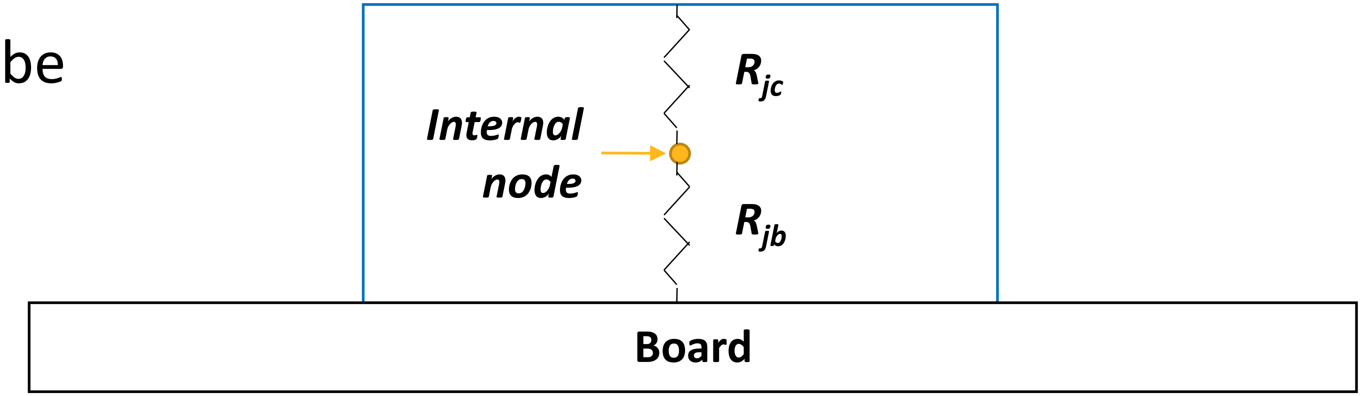
- To hide objects in the graphics area, select the objects in the model tree
- You can select multiple objects at once by pressing CTRL key and clicking on the respective object names in the tree
- Right-click → View → Hide in Active View
- Alternatively, you select a body in the graphics area by pressing O and then clicking on the body
- Next, press CTRL+H to hide the body
- Hide the 'Domain' under 'air' in the model



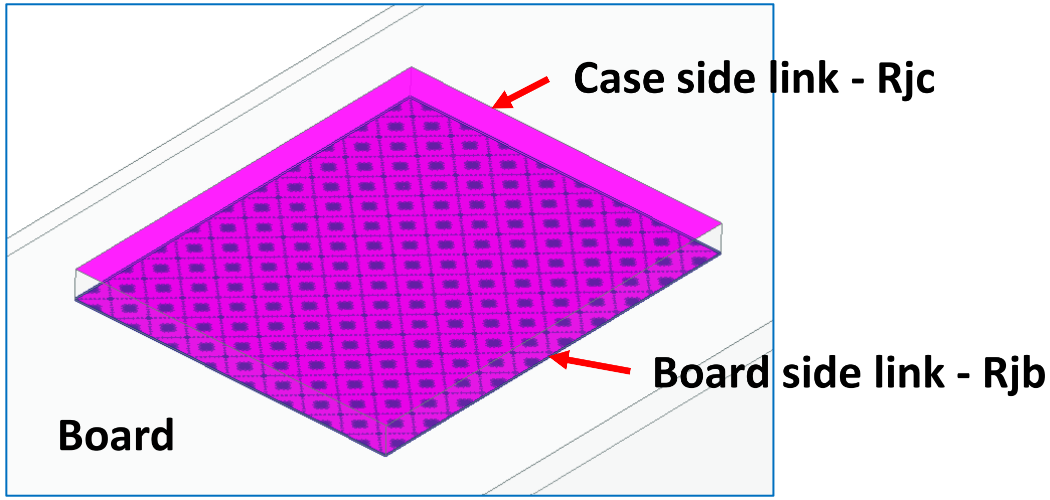


Basics of 2R Thermal Network

- Package objects 'Bridge' and 'AGP' will be modeled as 2R Thermal Network

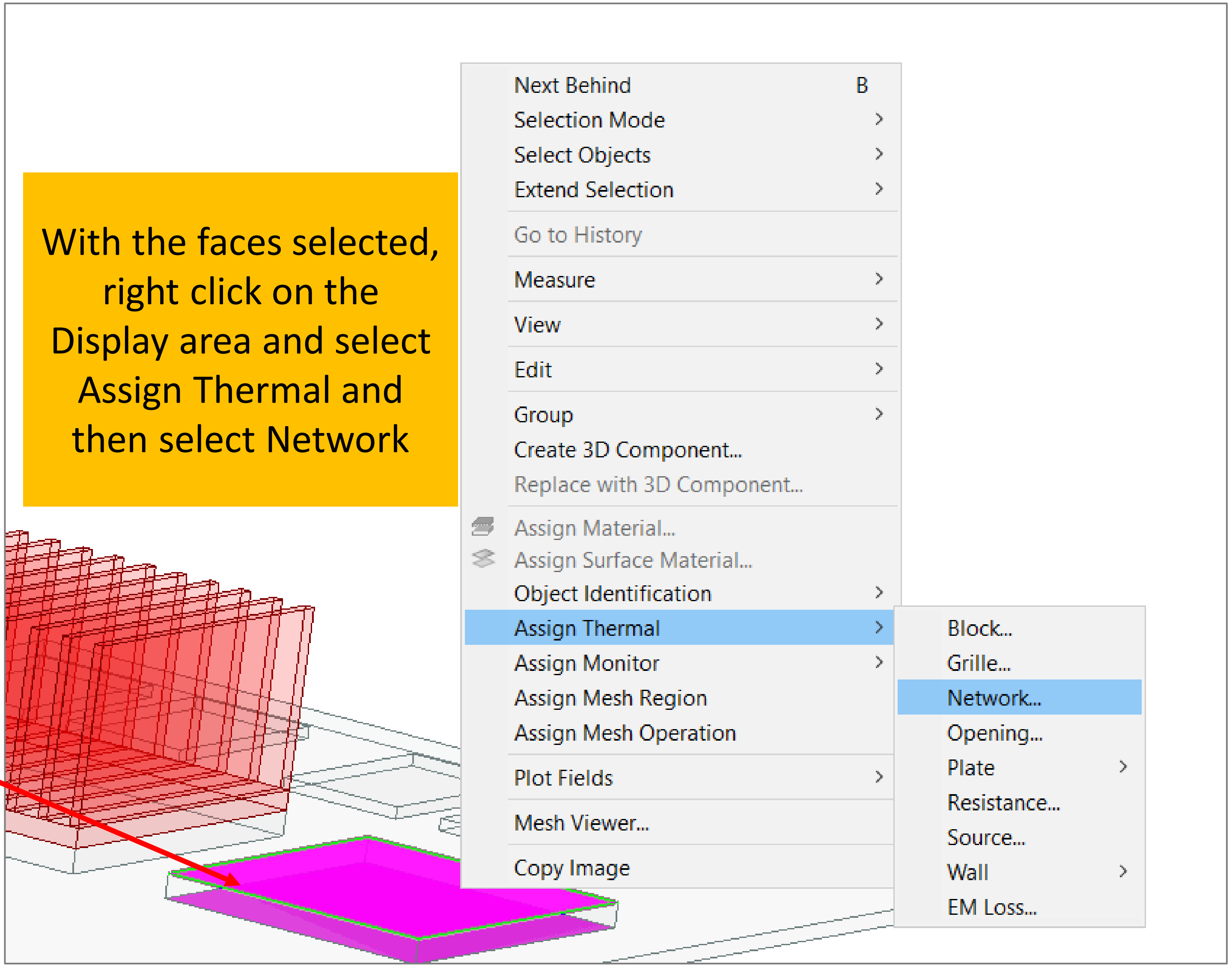


- 2R Network Basics:
  - The bottom face (coincident with the board) of the package is board side and the upper face is case side
  - The power is specified at the Internal node
  - R_{jb} is the thermal resistance from the internal junction to board
  - R_{jc} is the thermal resistance from internal junction to case



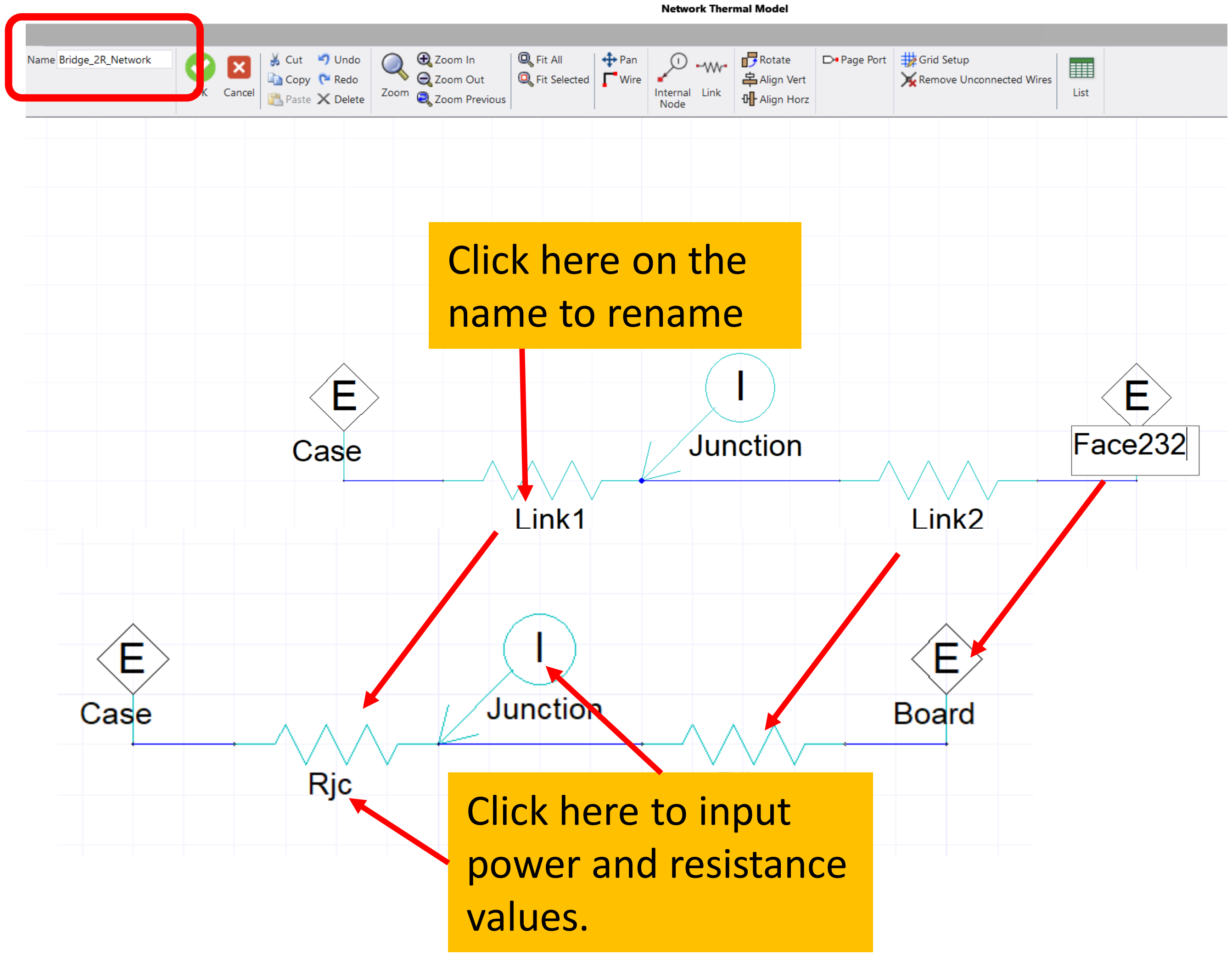
Creating 2 R Thermal Network for Bridge

- Switch to face selection mode by pressing F
- Select the top face of ‘Bridge’
- Press B to switch selection to the back face
- Press CTRL and then select the top face of the Bridge again
- The two faces of Bridge are now selected as shown in the figure



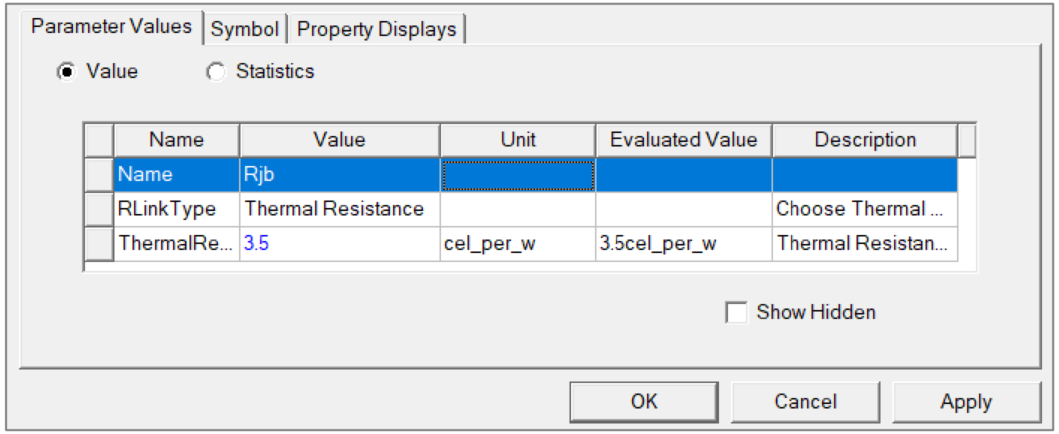
Creating 2 R Thermal Network for Bridge

- Assigning Network will open ' Network Thermal Model' Window.
- Review the Face nodes, Internal node and Links diagram.
- Name the Network as ' Bridge_2R_Network'
- Rename the end face nodes as 'Case' and 'Board' by double-clicking on the respective node names. (Double check the selected face in GUI to confirm board side and case side faces)
- Rename the Internal node as 'Junction', and 'Link1' and 'Link2' as Rjc and Rjb respectively.
- To edit the resistances and power values, double click on the symbols which will pop up the edit panel. (Explained on the next slide.)



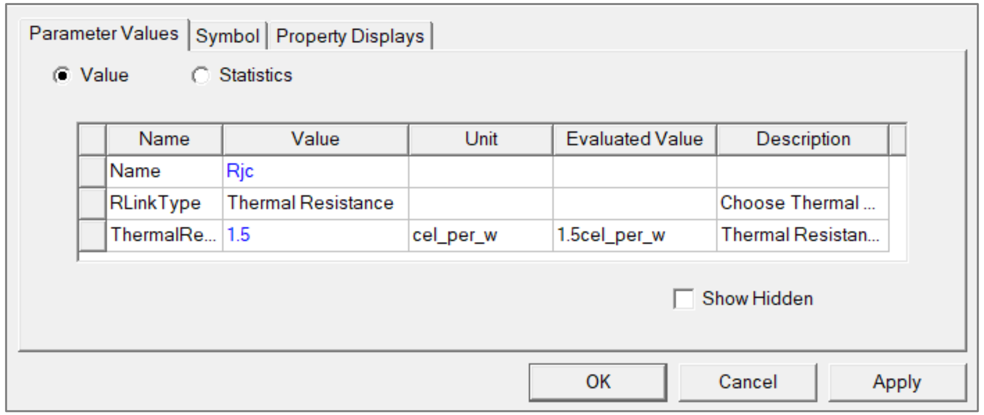
Creating 2 R Thermal Network for Bridge

- Use the following entries for Rjb = 3.5 C/W, Rjc = 1.5 C/W and internal node power as 2.5 W



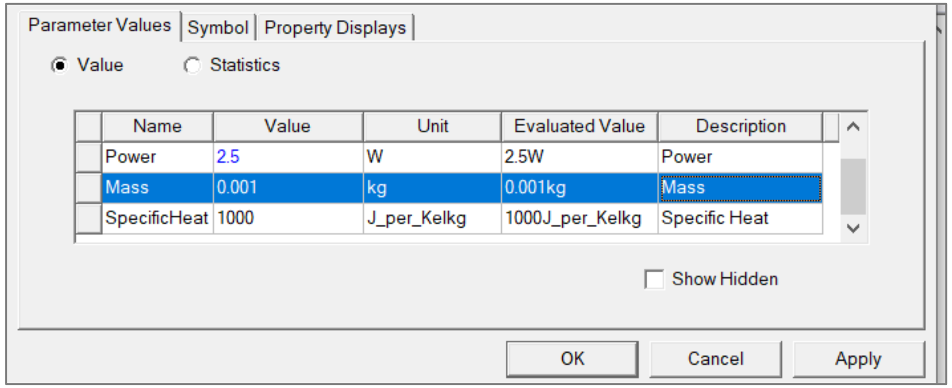
Name	Value	Unit	Evaluated Value	Description
Name	Rjb			
RLinkType	Thermal Resistance			Choose Thermal ...
ThermalRe...	3.5	cel_per_w	3.5cel_per_w	Thermal Resistan...

Junction to board thermal resistance



Name	Value	Unit	Evaluated Value	Description
Name	Rjc			
RLinkType	Thermal Resistance			Choose Thermal ...
ThermalRe...	1.5	cel_per_w	1.5cel_per_w	Thermal Resistan...

Junction to case thermal resistance

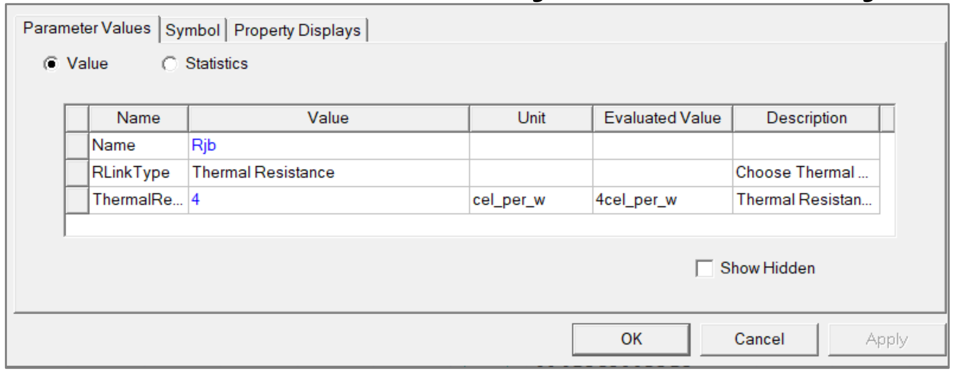


Name	Value	Unit	Evaluated Value	Description
Power	2.5	W	2.5W	Power
Mass	0.001	kg	0.001kg	Mass
SpecificHeat	1000	J_per_Kelkg	1000J_per_Kelkg	Specific Heat

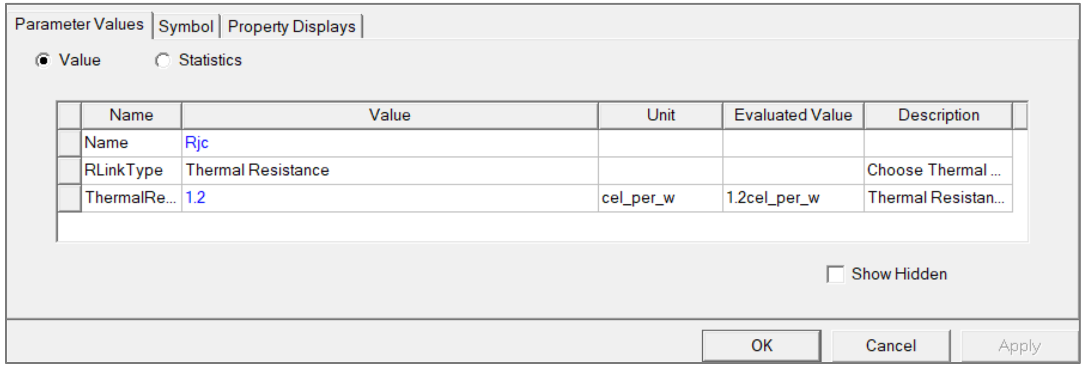
Junction power – Bridge

Creating 2 R Thermal Network for AGP

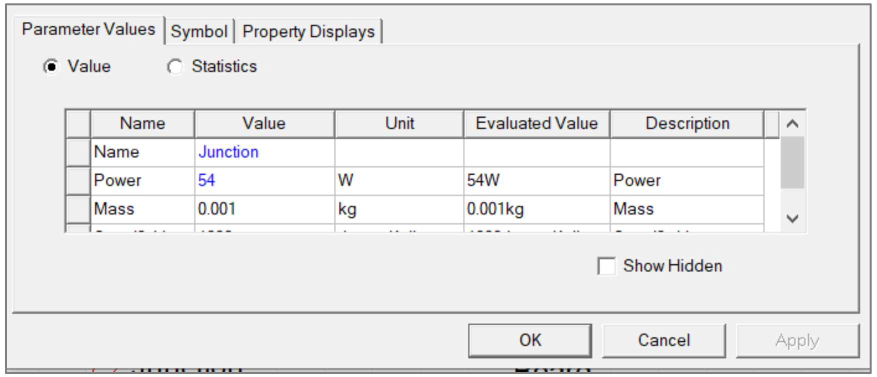
- Repeat the steps discussed for AGP. You can hide the 'Heatsink' object to select the faces.
- Use 'AGP_2R_Network' for the network name.
- Enter the values for Rjb = 4 C/W, Rjc = 1.2 C/W, and junction node Power = 54 W as below.



Junction to board thermal



Junction to case thermal resistance

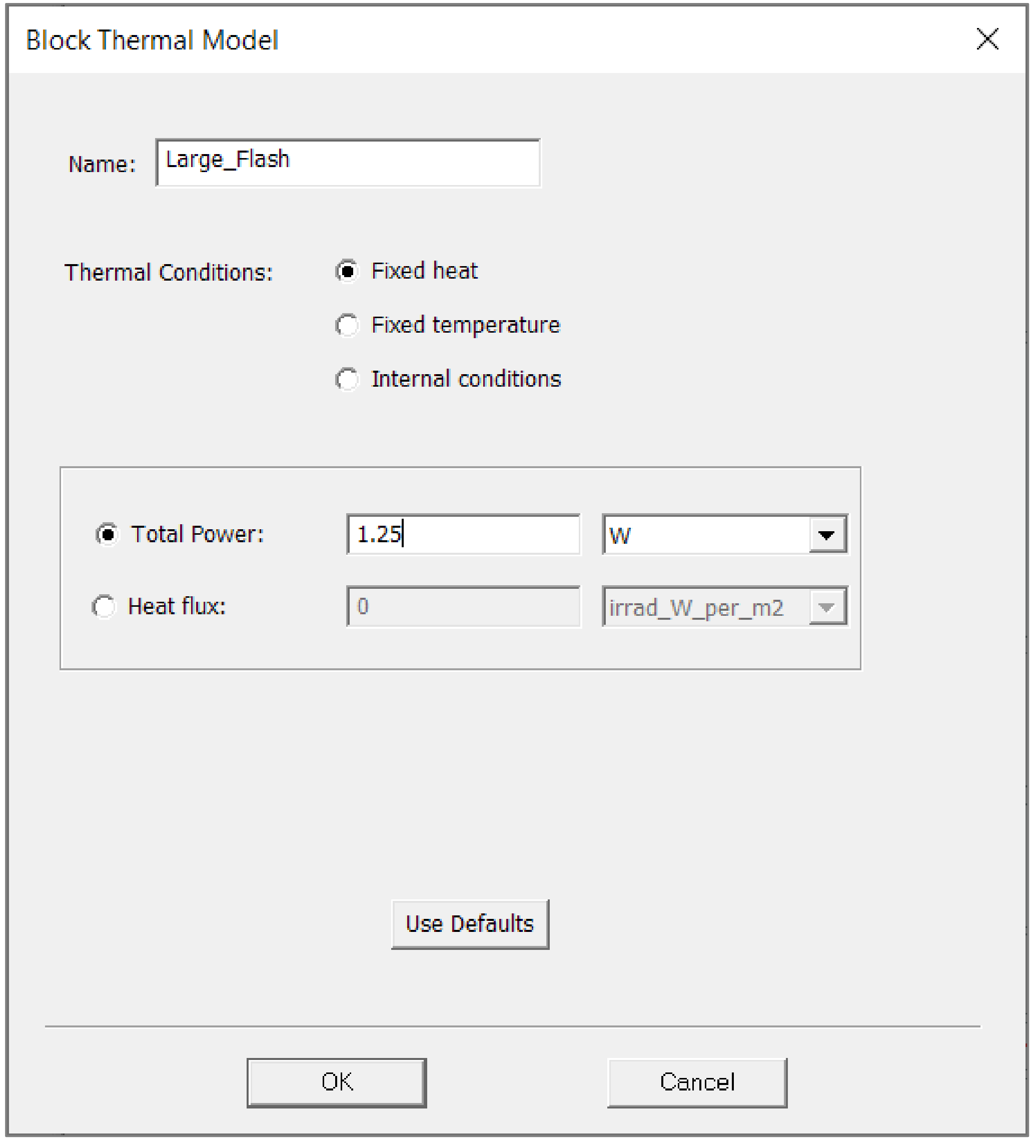


Junction power – Bridge

Add Heat Sources – Power Assignment

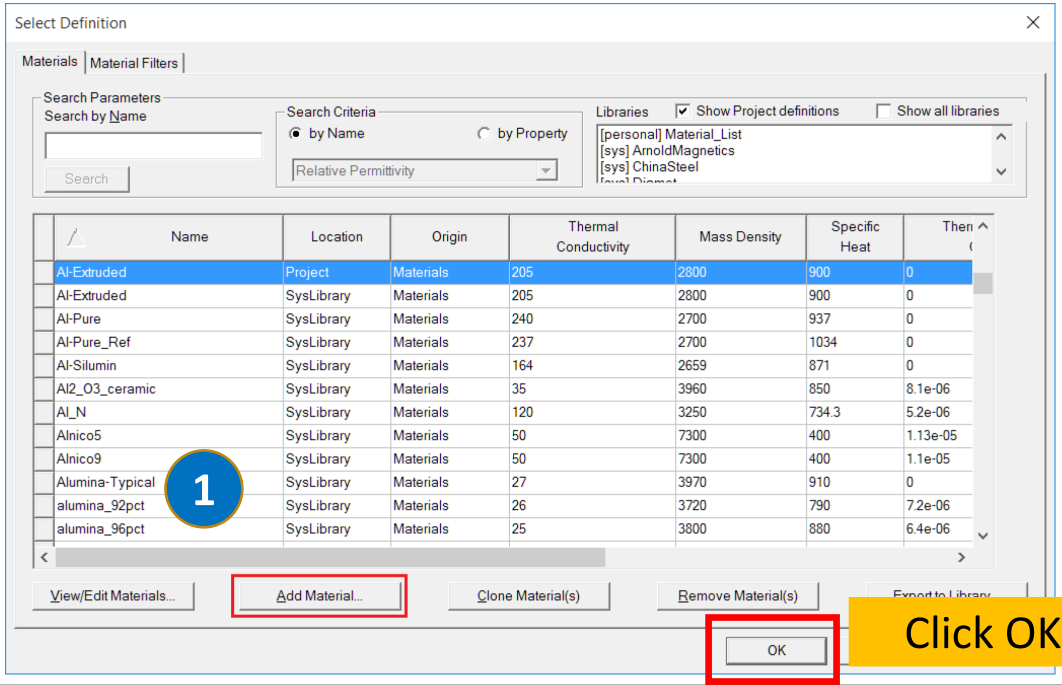
- Select Large_Flash from the model tree
- Right click → Assign Thermal → Block…
- Enter the Name as Large_Flash and use value of 1.25 W for total power
- Repeat the above steps for Small_Flash and DDRs and use the powers listed below
- Tip: Multiple objects can be selected to create a single thermal block object. The same power input in the block panel is applied to all objects.
- Select all four DDRs and input 1.125W as the Total Power. Each DDR will have 1.125W for a total of 4.5W.

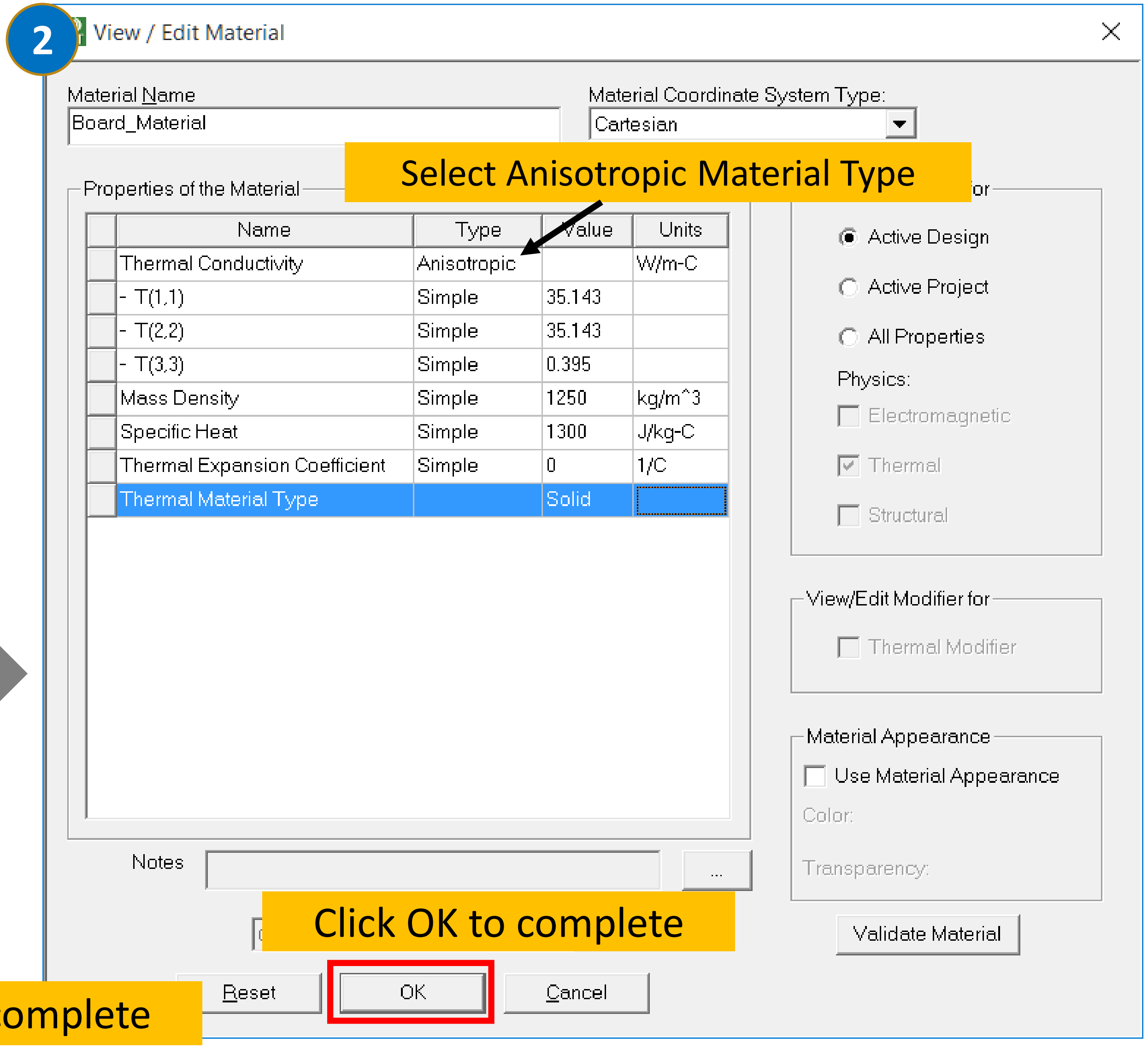
Small Flash – 0.5 W
Large Flash – 1.25 W
DDRs – 1.125 W



Assign Material Properties – Board

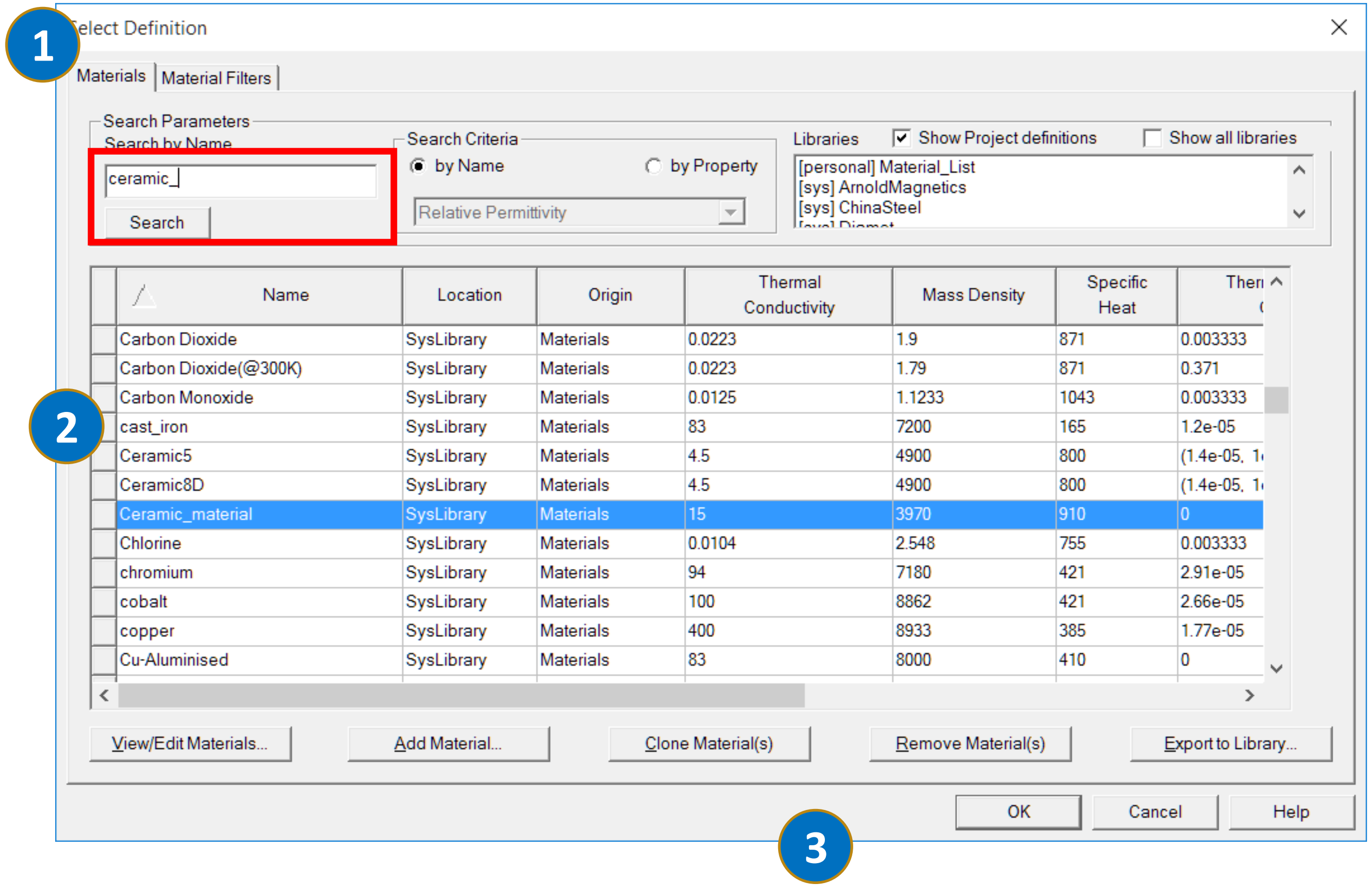
- Right click on Board in the Model Tree and select Assign Material…
- In the Select Definition panel click Add Material to create a new custom material
- Enter the values shown in the figure





Assign Material Properties to Components

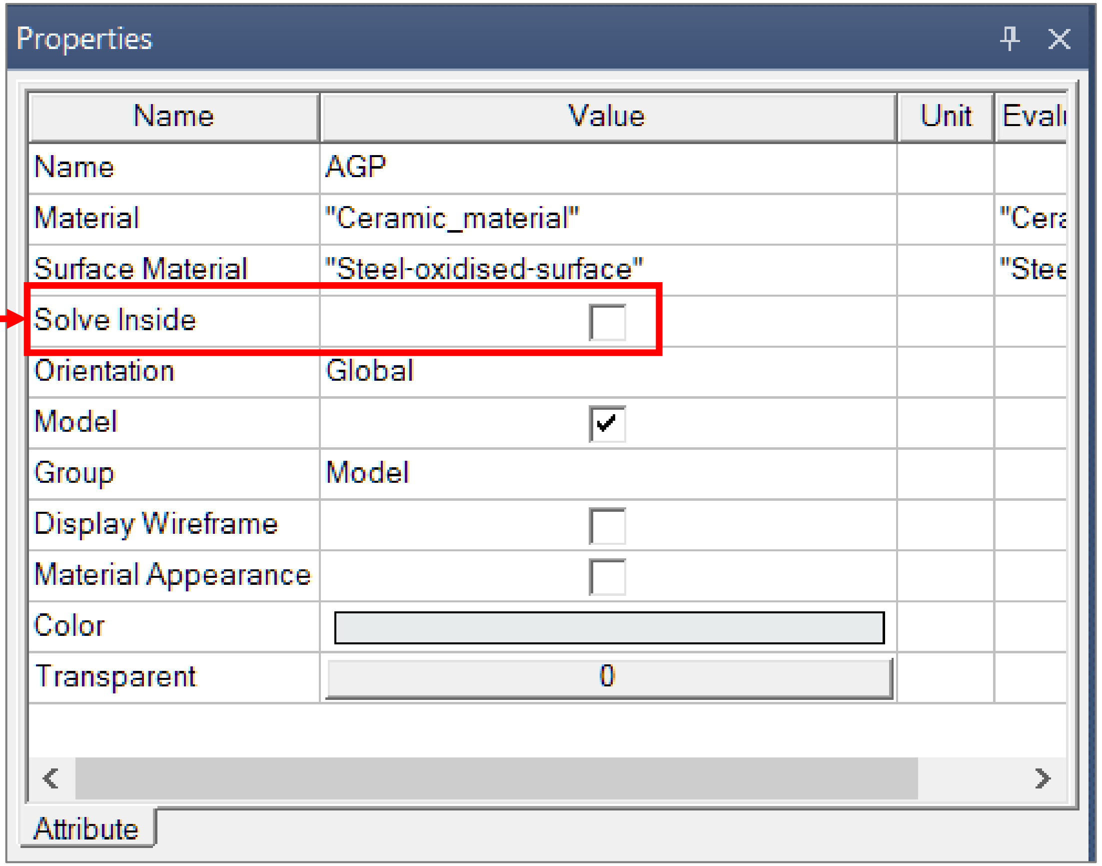
- Select the DDR components, AGP, BRIDGE, Large Flash and Small Flash from the Model tree
- You can select multiple objects at once by pressing the CTRL key and selecting the object names in the model tree
- Right click on the components in Model Tree and select Assign Material…
- You can search for a material by name or by property
- Search for the Ceramic_material and click OK to assign the material and close the Select Definition panel



Ansys

Assign Material Properties – Contd.

- Select all objects in the Model tree
- In the Properties box of the object, select Steel-oxidized-surface for surface material
- Network blocks should have Solve Inside turned off
  - Make sure 'Solve Inside' option is turned off for both AGP and Bridge objects.

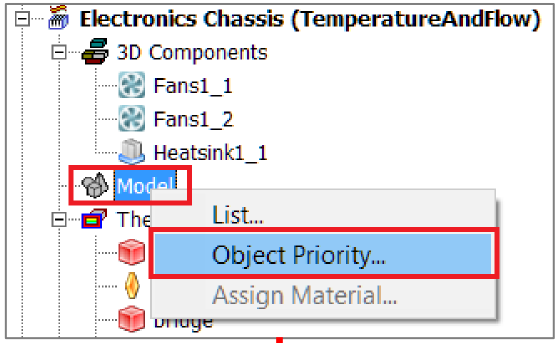


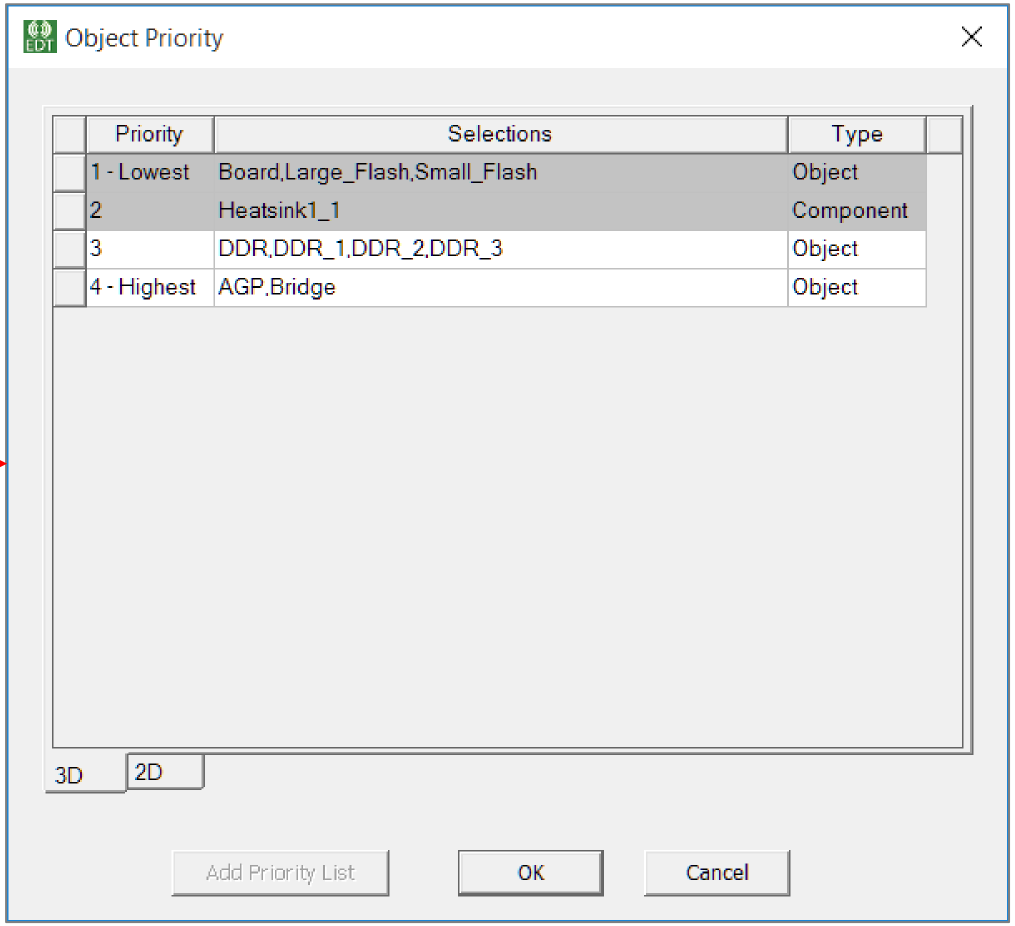
Solve Inside should be turned off for network objects

Ansys

Changing Object Priority

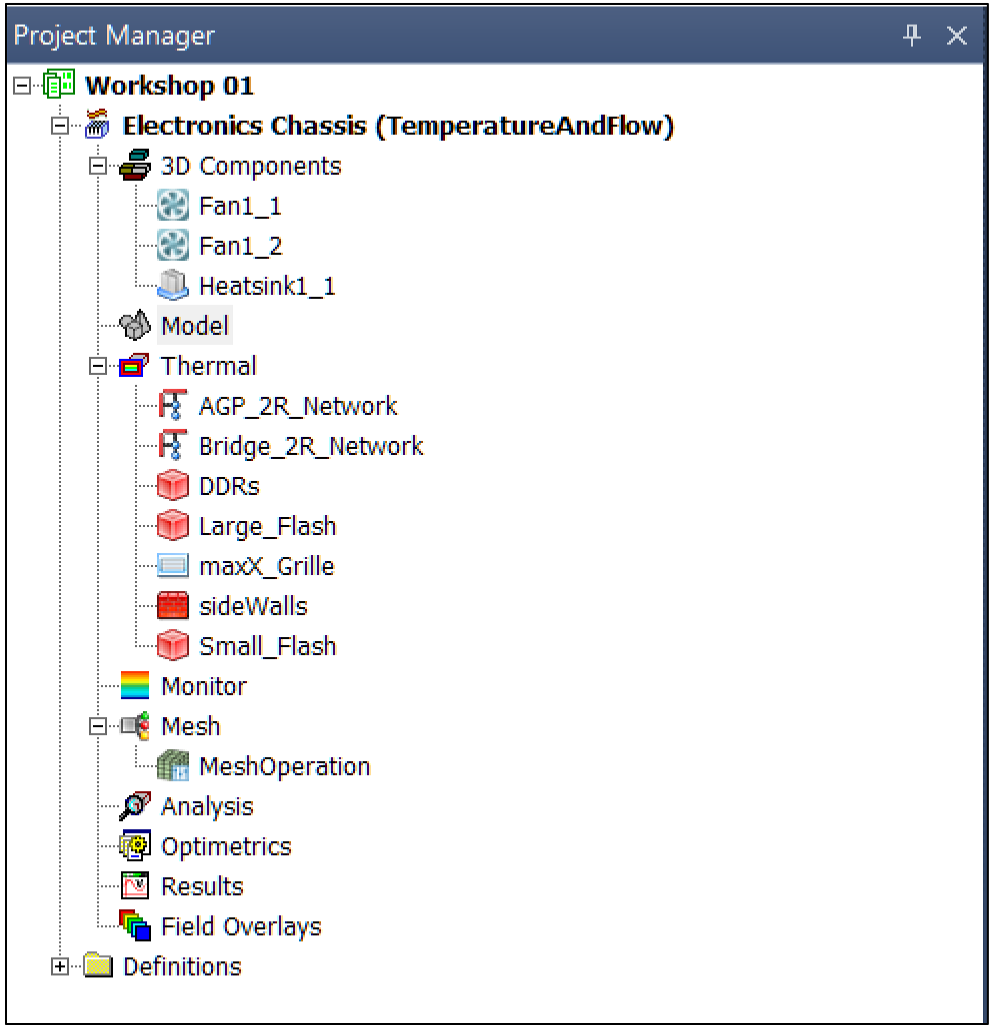
- Right click on Model under the Project Manager and select Object Priority...
- Select the DDR objects from the model tree and then click on Add Priority List in the Object Priority panel
- Next select AGP and Bridge and then click Add Priority List in the Object Priority panel
- Click OK to close the panel

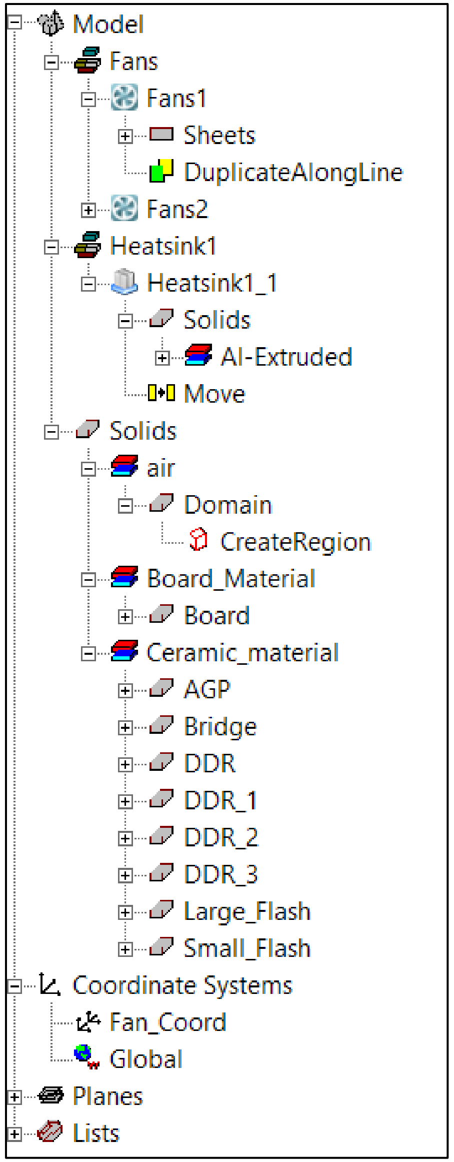




Ansys

Summary of Problem Setup





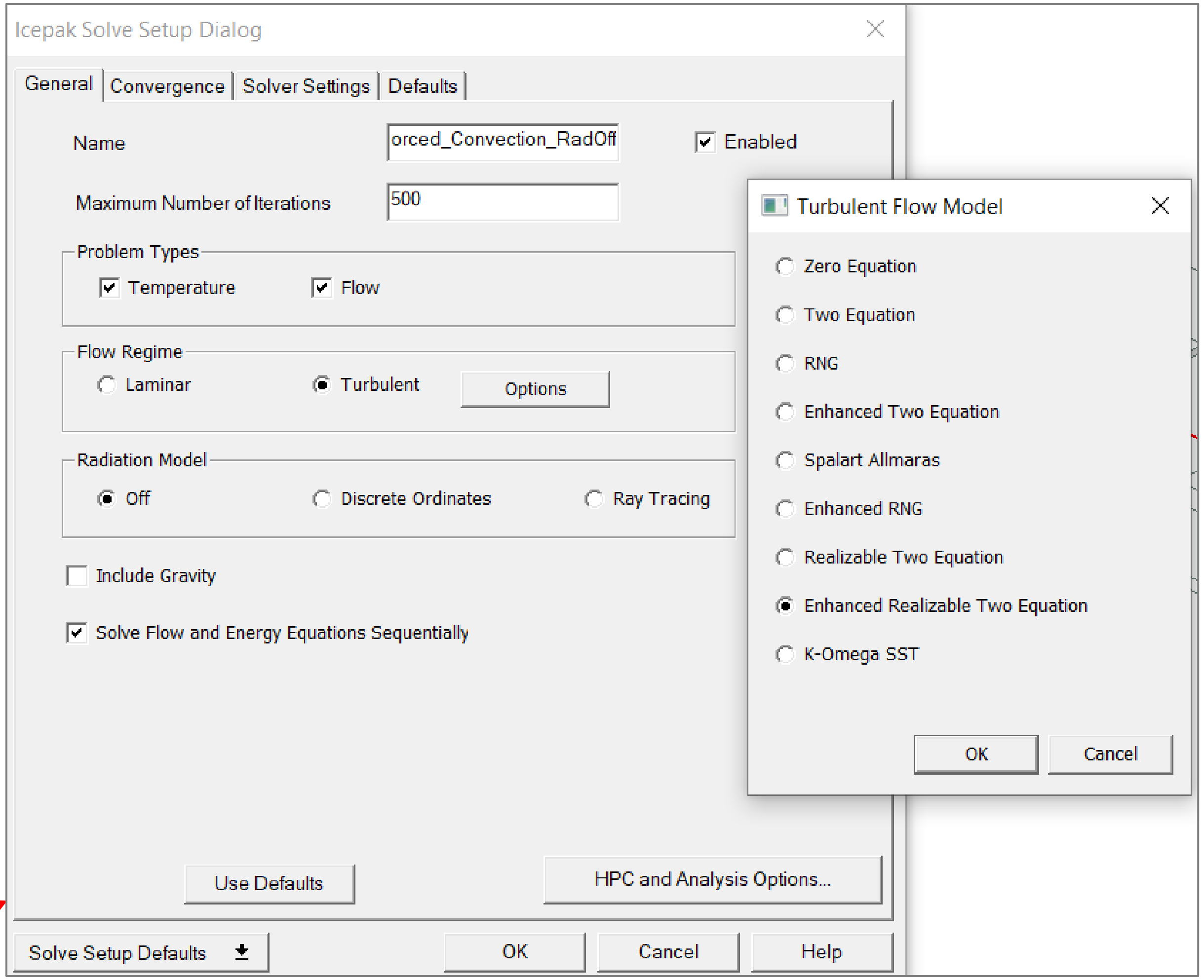
Fan Flow Rate = 10 CFM each

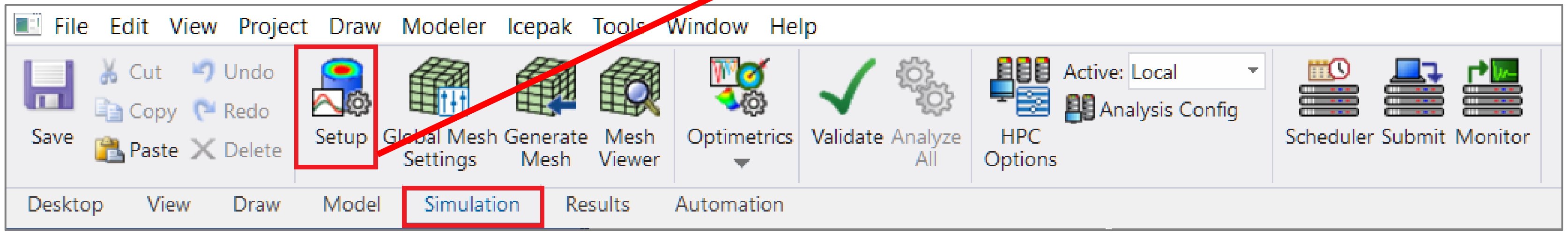
Object	Power (W)	Material	Thermal Conductivity (W/(m.K))
DDR1	1.125	Ceramic Material	15
DDR2	1.125	Ceramic Material	15
DDR3	1.125	Ceramic Material	15
DDR4	1.125	Ceramic Material	15
Large Flash	1.25	Ceramic Material	15
Small Flash	0.5	Ceramic Material	15
Board	0	Custom Board	(35.143, 35.143, 0.395)

Object	Power (W)	Rjb (C/W)	Rjc (C/W)
AGP	54	4	1.2
Bridge	2.5	3.5	1.5

Solver Settings

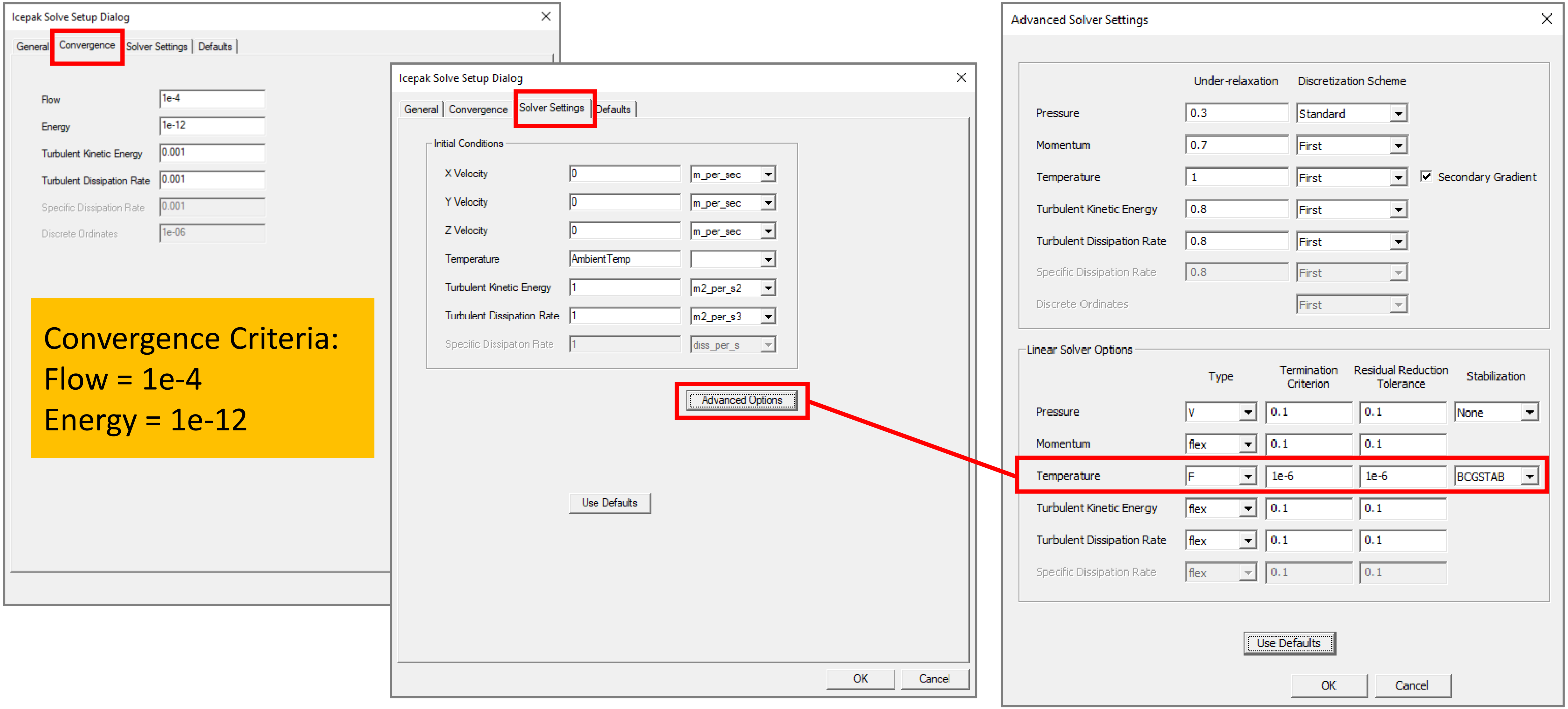
- Click on Setup in the Simulation tab
- Enter ForcedConvection_RadOff as simulation name
- Set the maximum number of iterations to 500
- Use Enhanced Realizable Two Equation as turbulence model
- Click Radiation Off
- Check the box for Flow and Energy Equations Sequentially (With this option 400 of 500 iteration will be used for flow iteration and rest will be used for Energy iteration)





Solver Settings

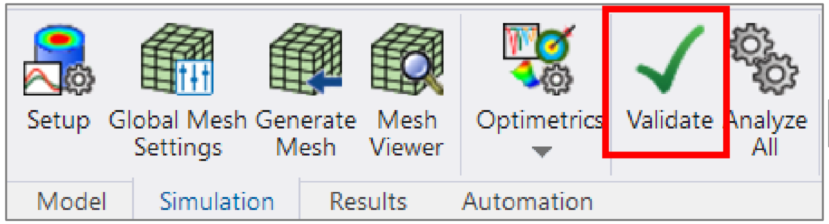
Recommended Settings for Sequential Solution of Flow and Energy

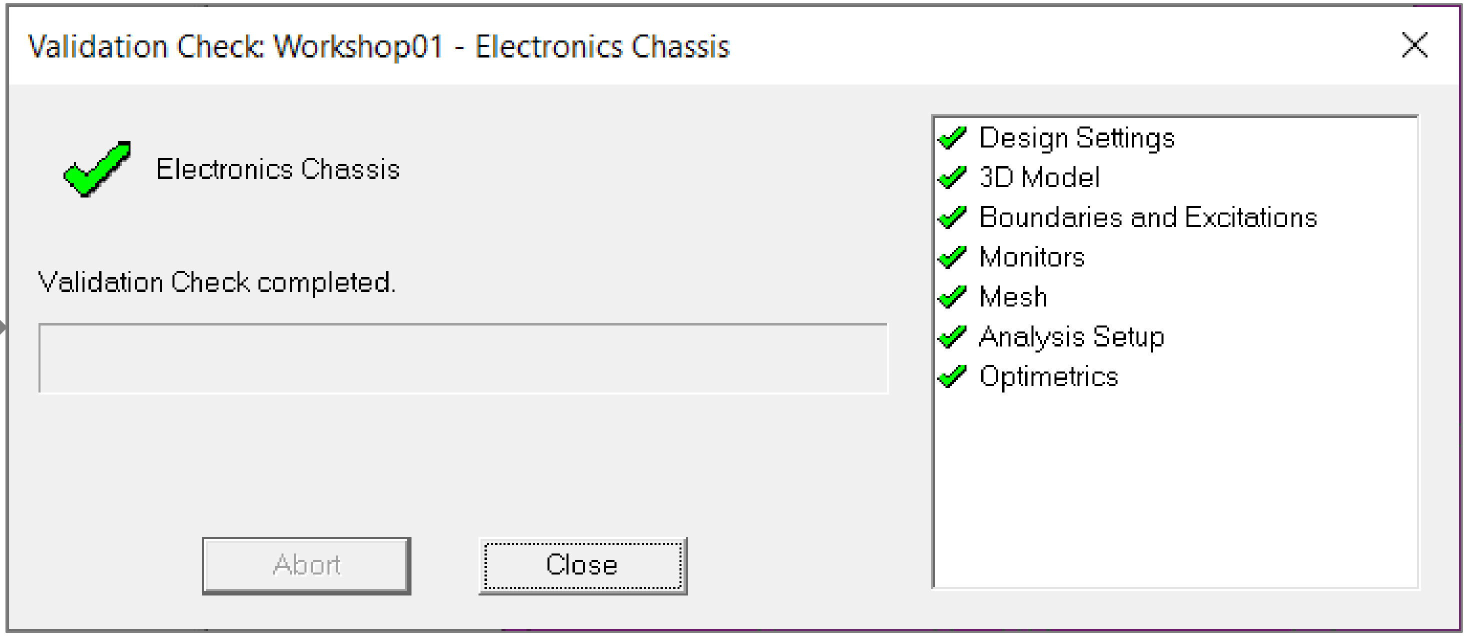


Convergence Criteria:
Flow = 1e-4
Energy = 1e-12

Solver Settings

- Finally check for any errors in the model
- Under the Simulation tab, click on Validate
- Click Close to close the validation check panel





Workflow for Mesh Generation and Viewing

Global Mesh Settings

- **Slider bar (automated)**
- **Advanced (manual)**

→

Generate Mesh

- **Mesh generation step**

→

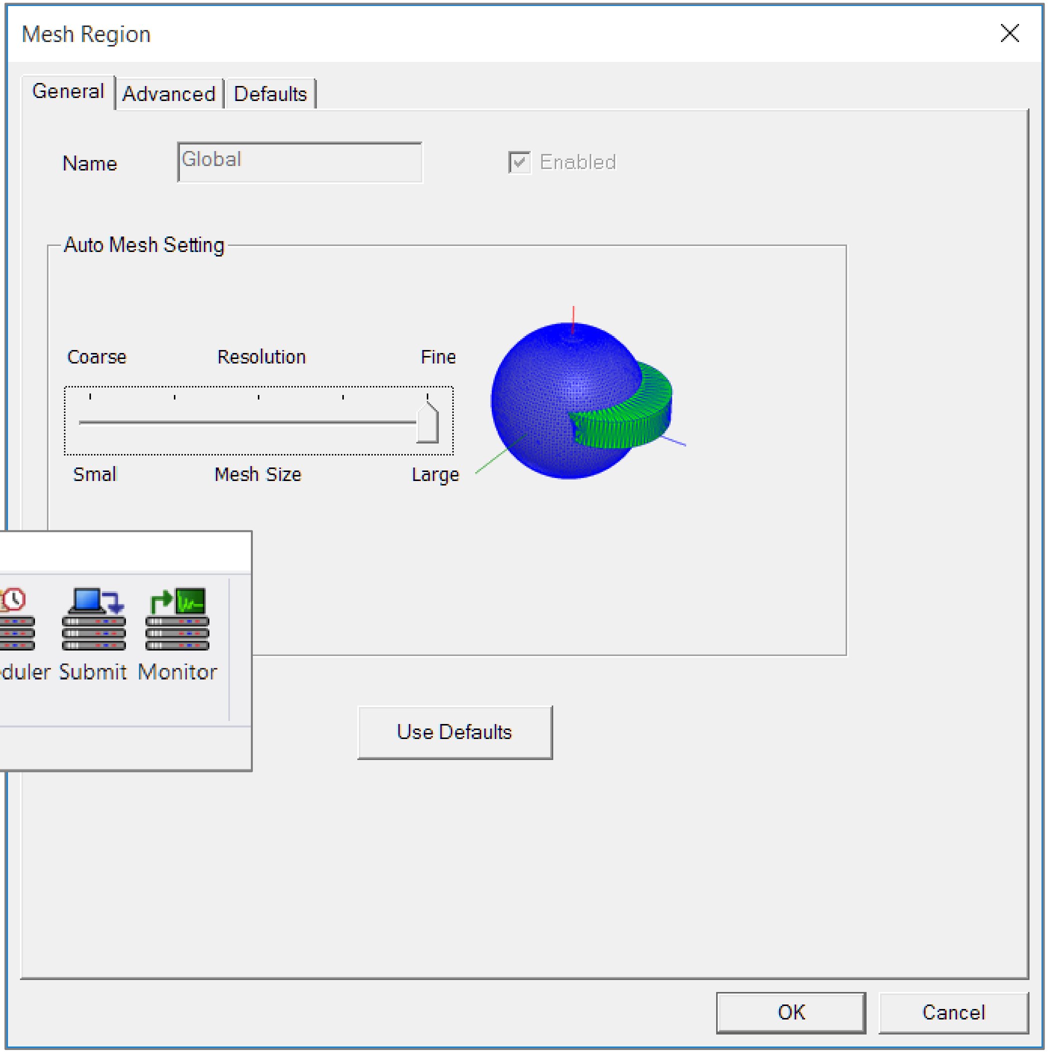
Mesh Viewer

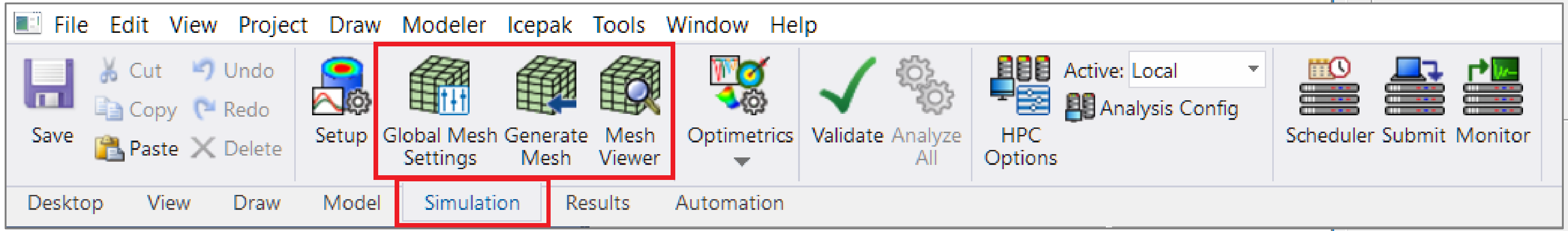
- **View mesh for quality**
- **View mesh over a plane or on a selected object**

<u>Note:</u> Only slider bar mesher will be used in this workshop

Meshing the Model

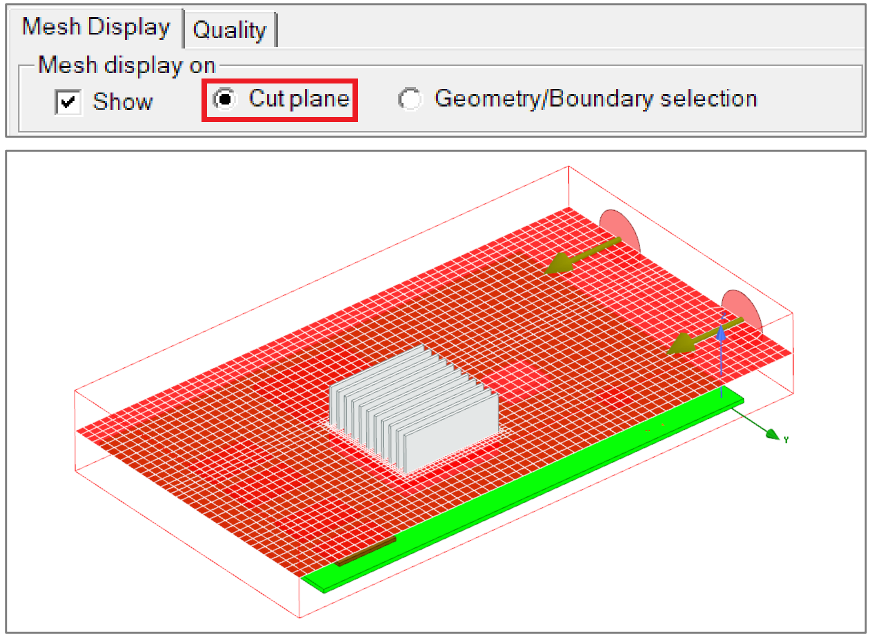
- Click on Global Mesh Settings under Simulation tab
- Use the Fine setting in the slider bar for mesh
- Click OK
- Click Generate Mesh



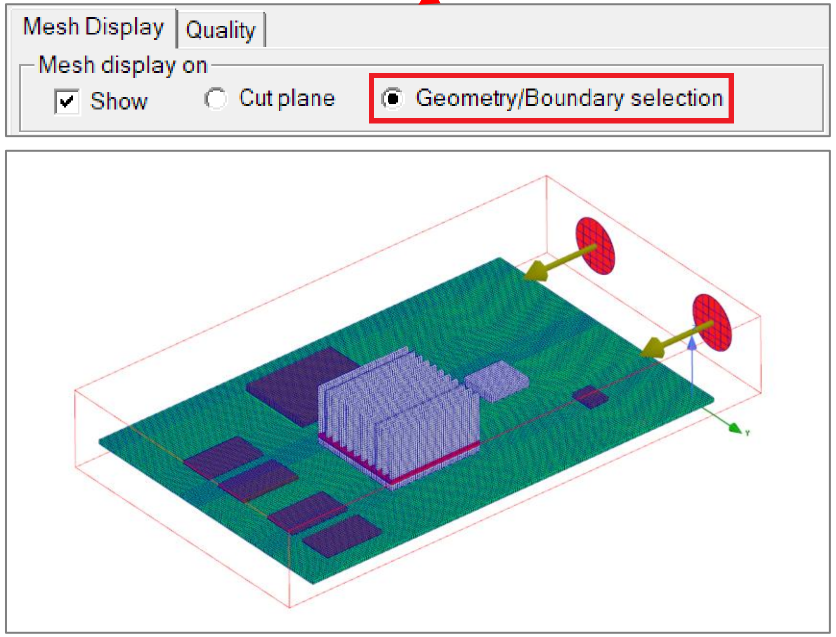


Viewing the Mesh

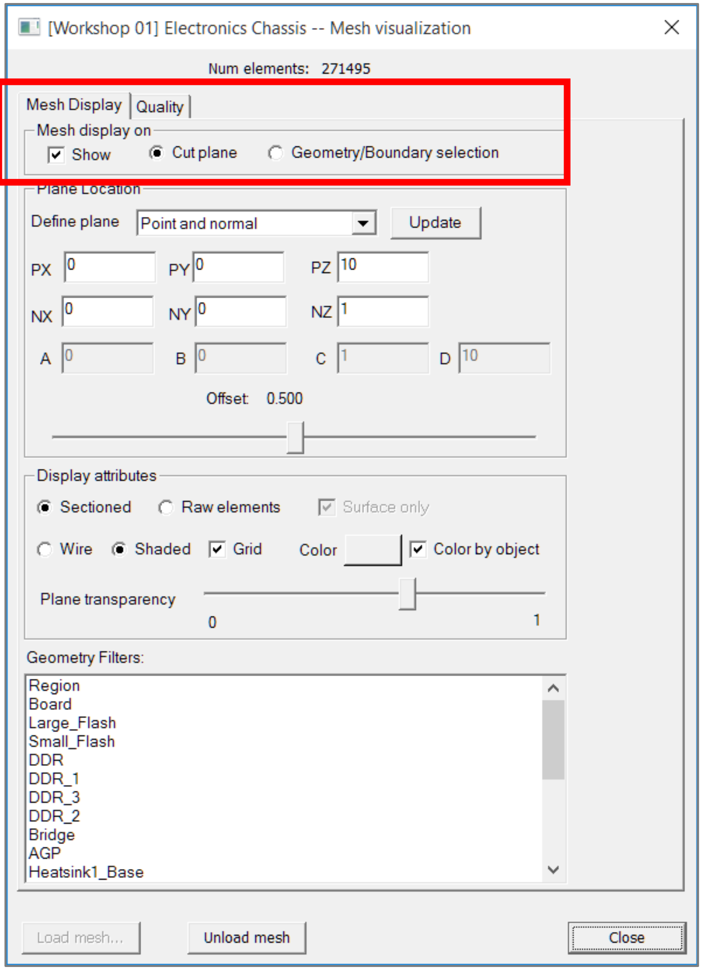
- Click on Mesh Viewer
- Use the cut plane option on a plane or Geometry/Boundary selection option to view the mesh on objects



Cut plane

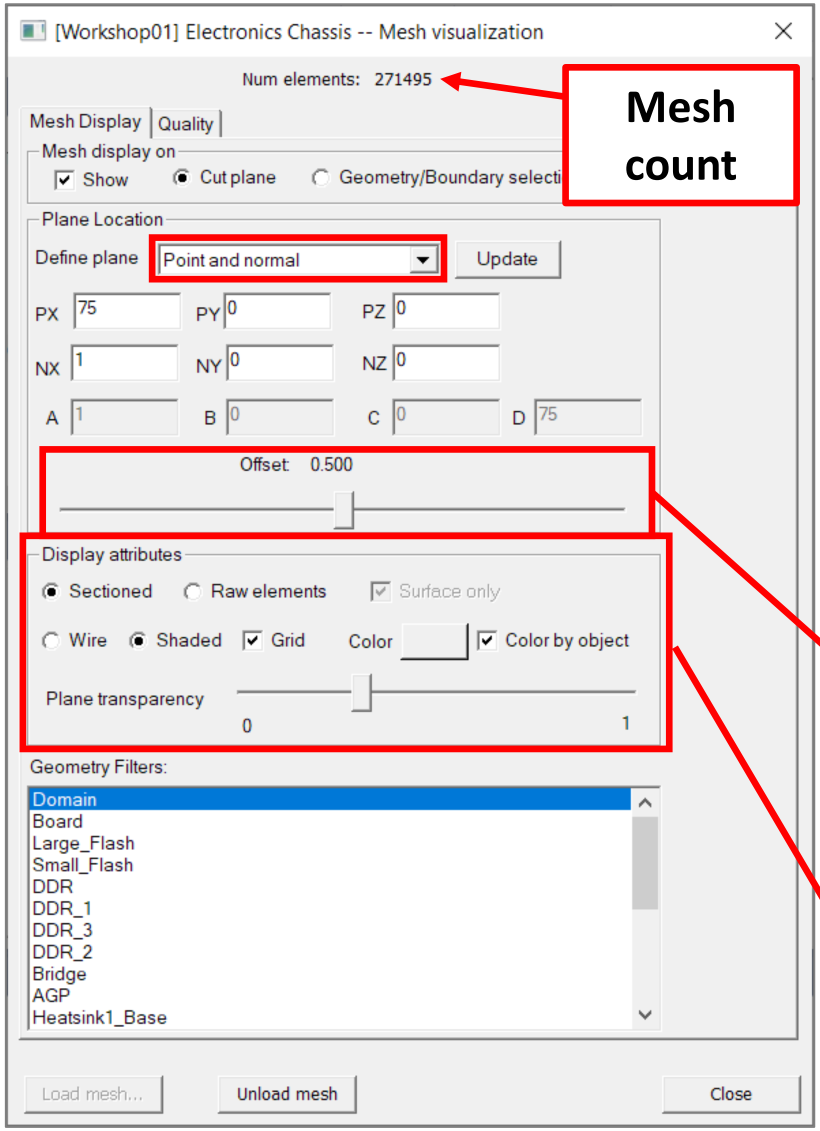


Geometry Selection



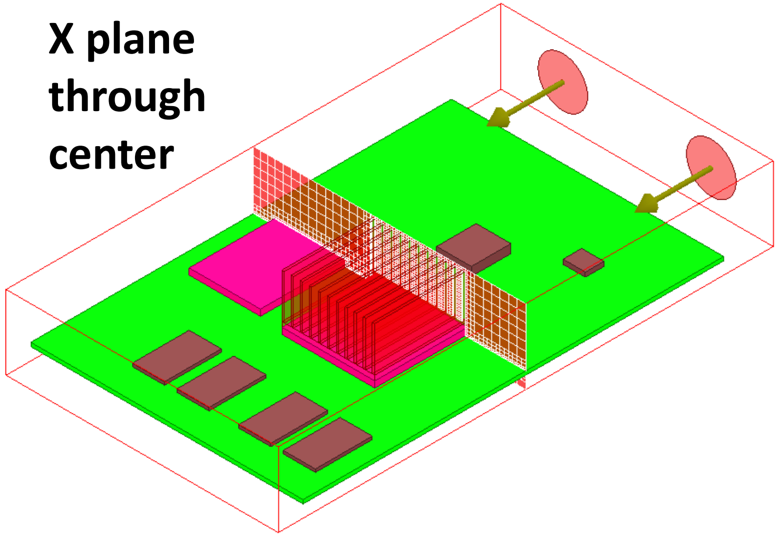
Ansys

Viewing the Mesh

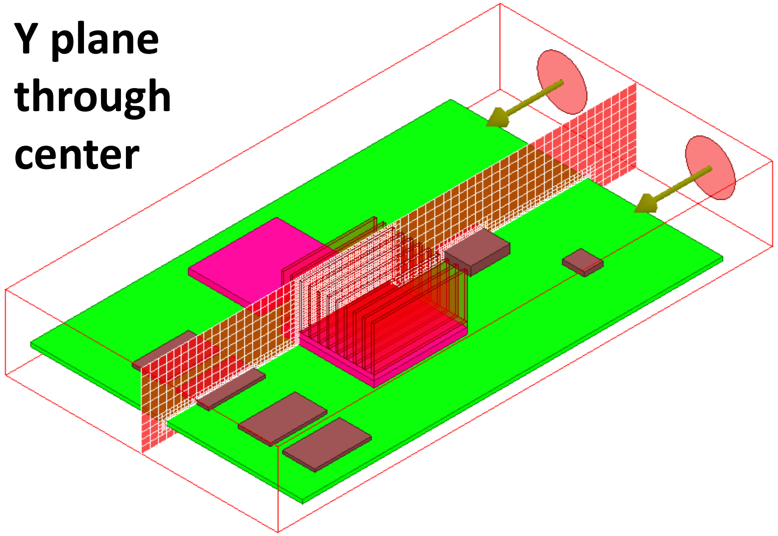


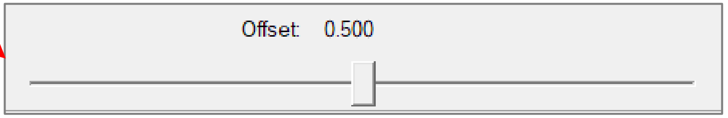
Mesh count

X plane through center



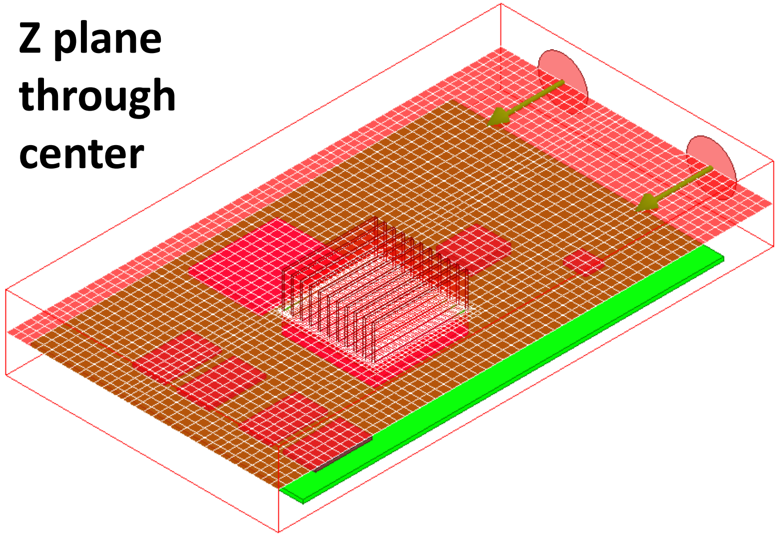
Y plane through center

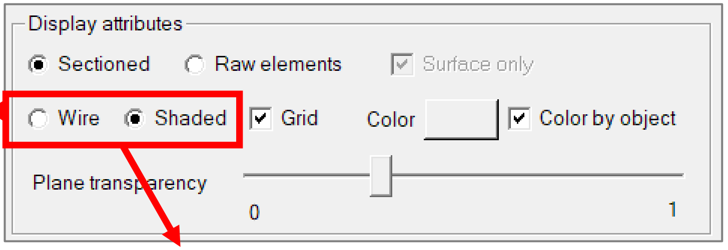




Slider to move cut plane

Z plane through center

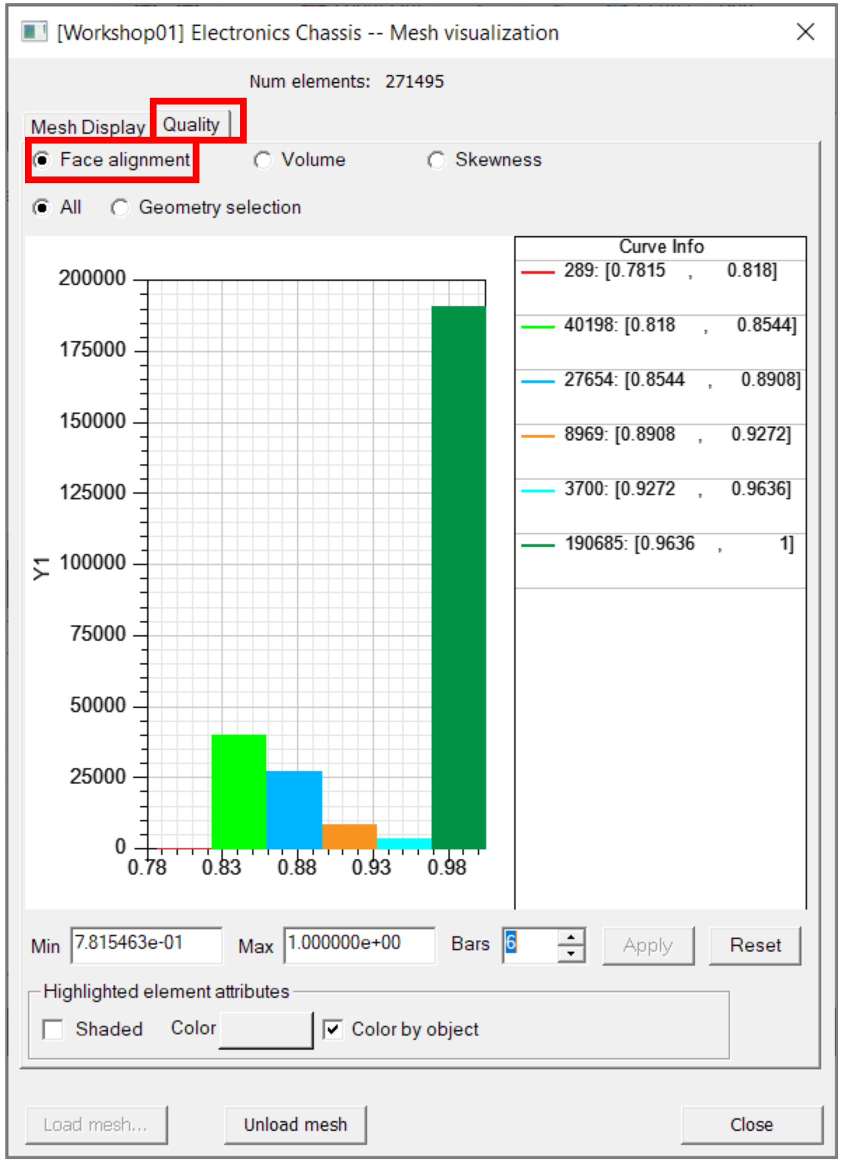


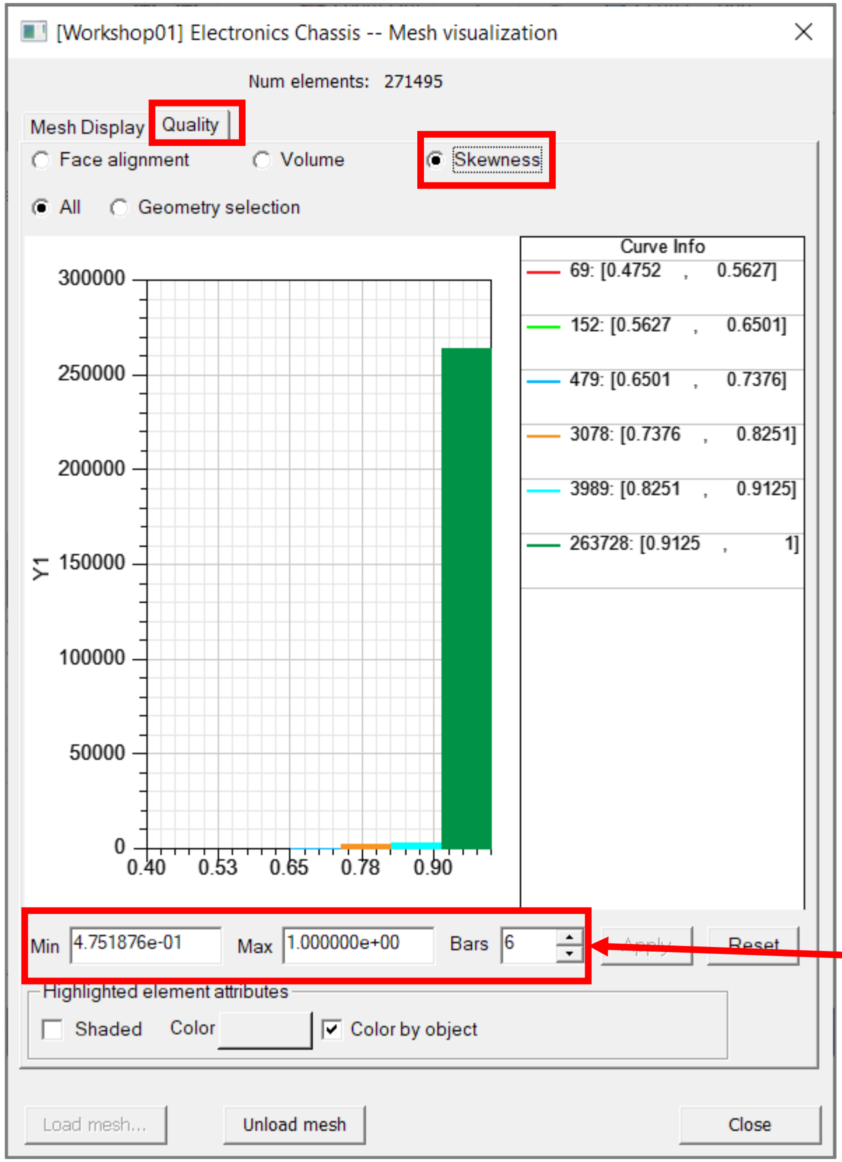


Wire or shaded cut plane

Ansys

Mesh Quality

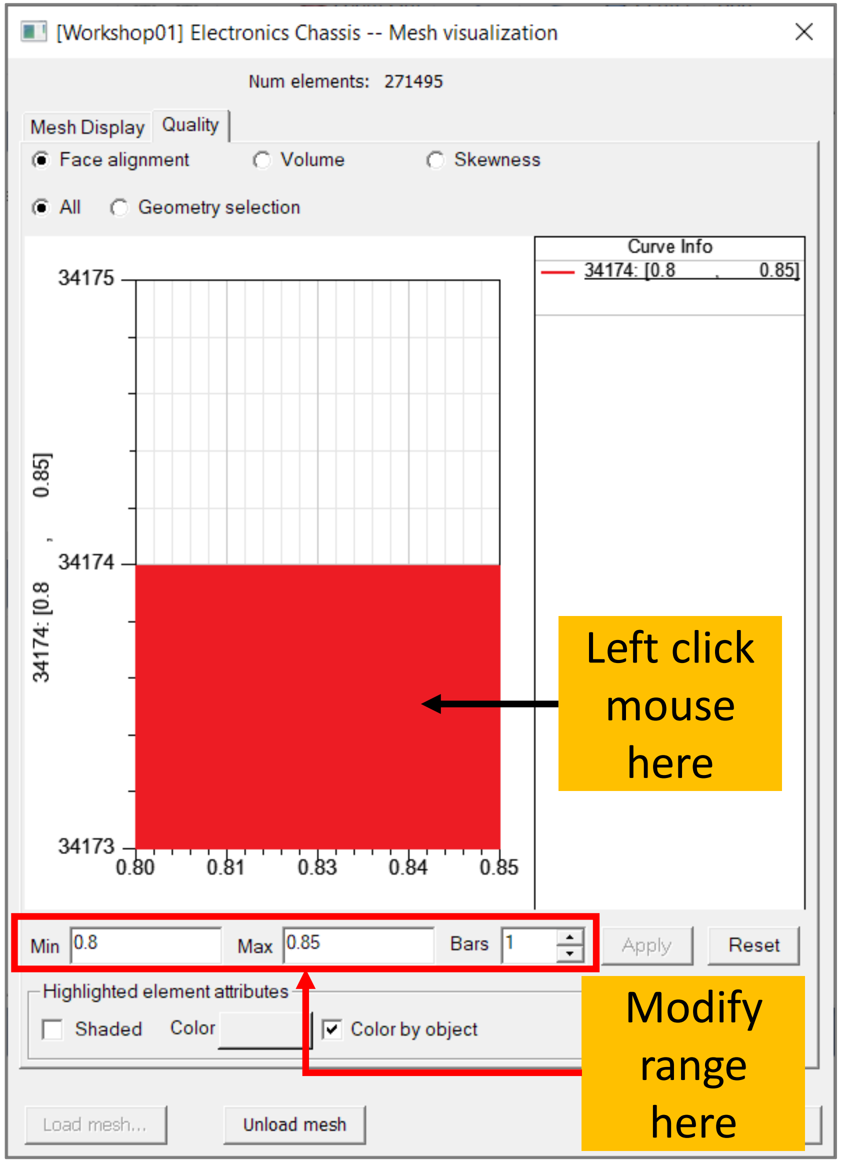




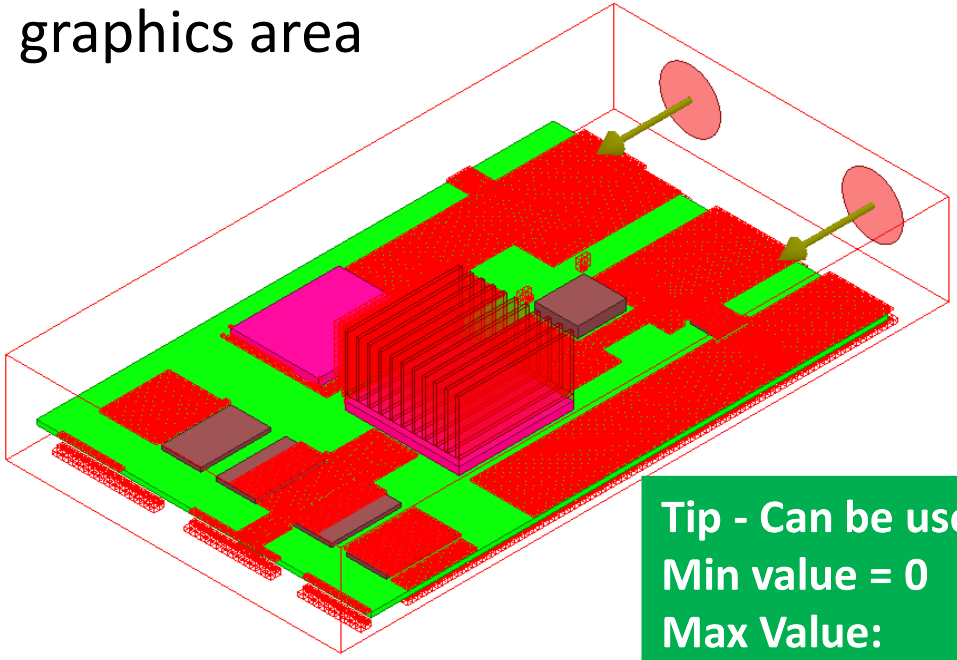
- **For a good quality mesh:**
  - Min. Face Alignment > 0.05
  - Min. Skewness > 0.02
- **It is recommended to check mesh quality before running a simulation**

Min and Max values for face alignment and skewness are shown here

Mesh Quality



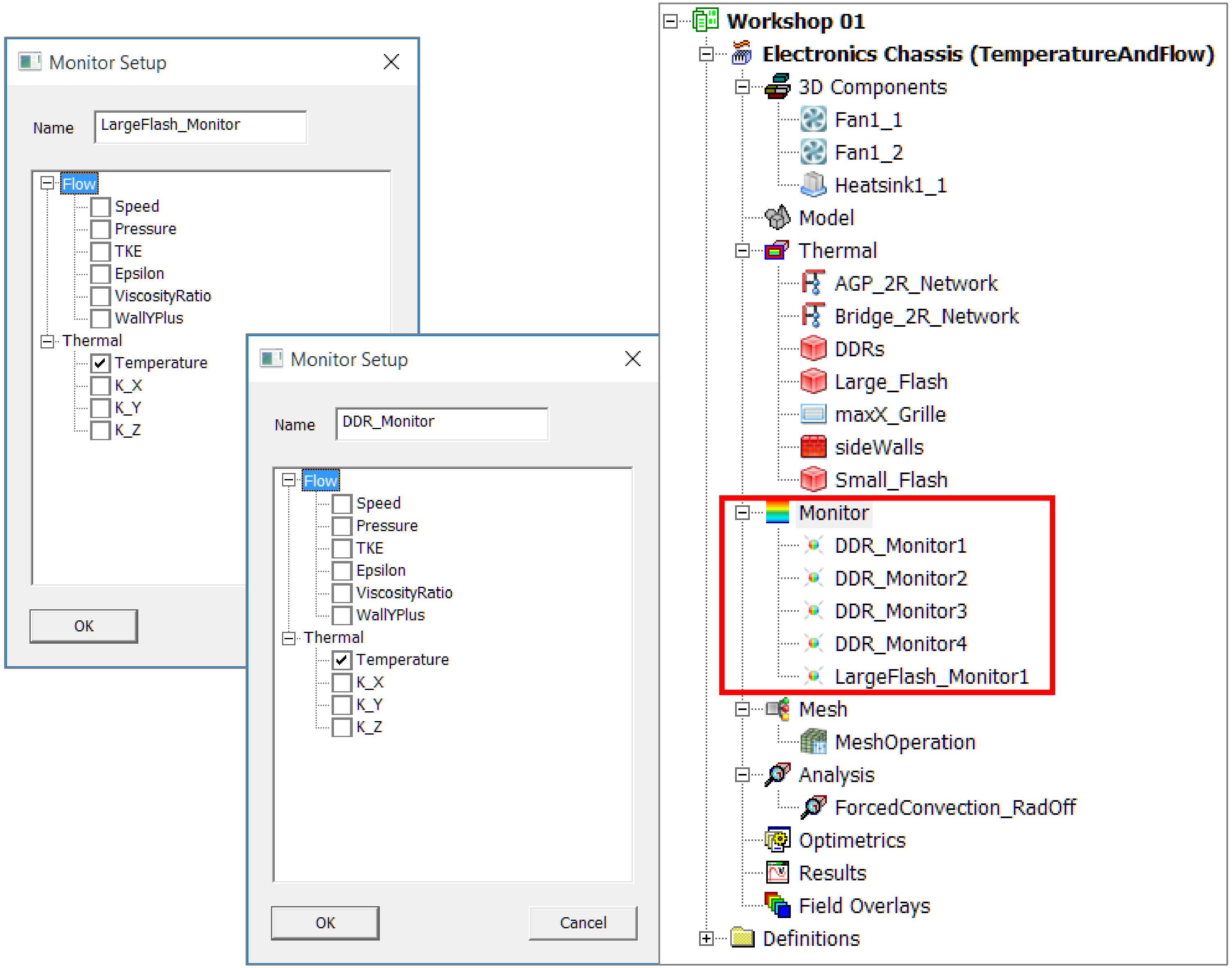
- Modify the histogram range to locate the cells between desired range
- Change the number of bars to 1
- Click on histogram to highlight the cells in the graphics area



Tip - Can be used to locate bad cells:
Min value = 0
Max Value:
0.05 (Face alignment)
0.02 (Skewness)

Create Monitor Points

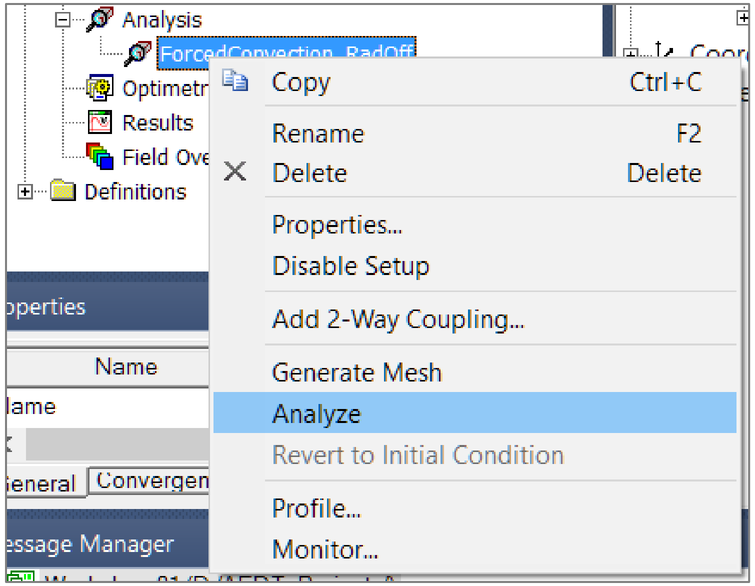
- Right click on Large_Flash in the model tree and click Assign Monitor → Point…
- Enter the name and click on Temperature
- Control select all the DDR object in the model tree and right click Assign Monitor → Point…
- Enter the name and click on Temperature

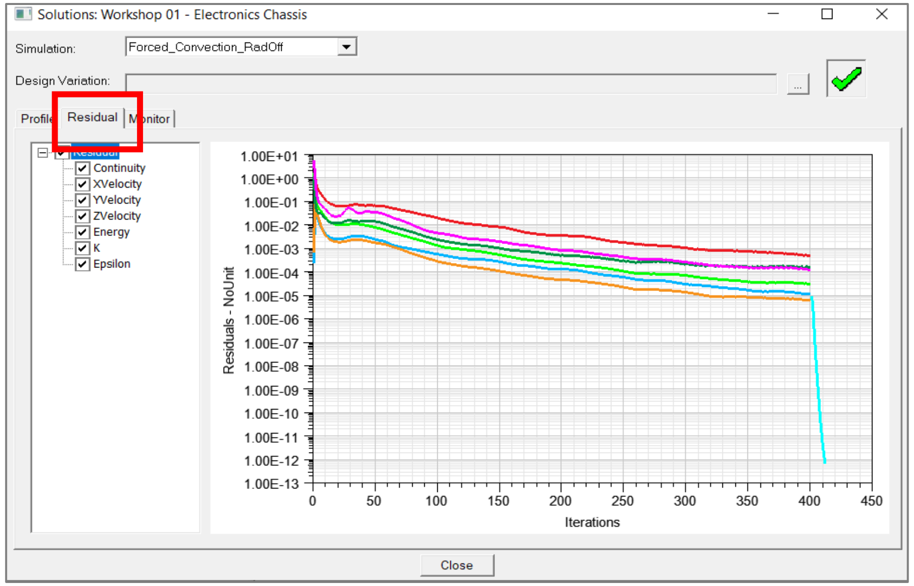


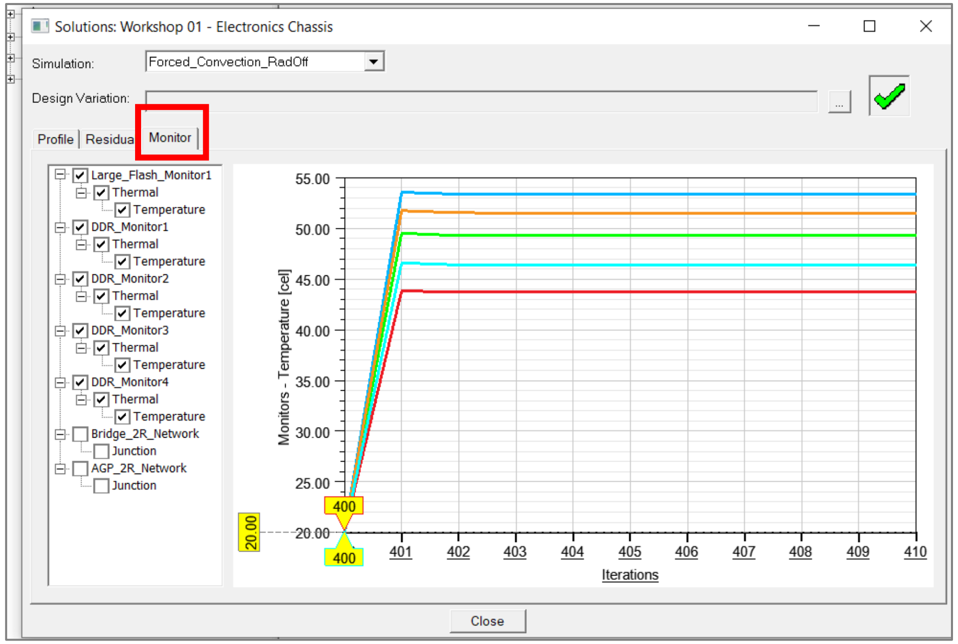
Ansys

Analyze the Setup

- Right click on the setup ForcedConvection_RadOff under the Analysis node in Project Manager
- Click on Analyze
- Right click on the ForcedConvection_RadOff and click Monitor.
- Residual and Monitor Tab should give information on convergence

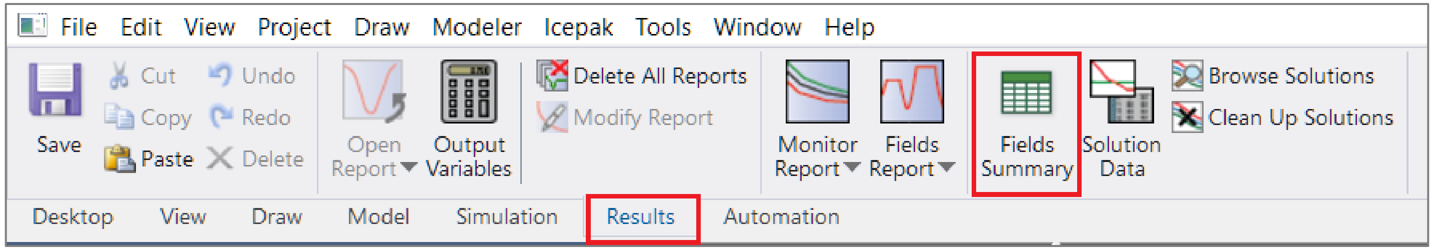


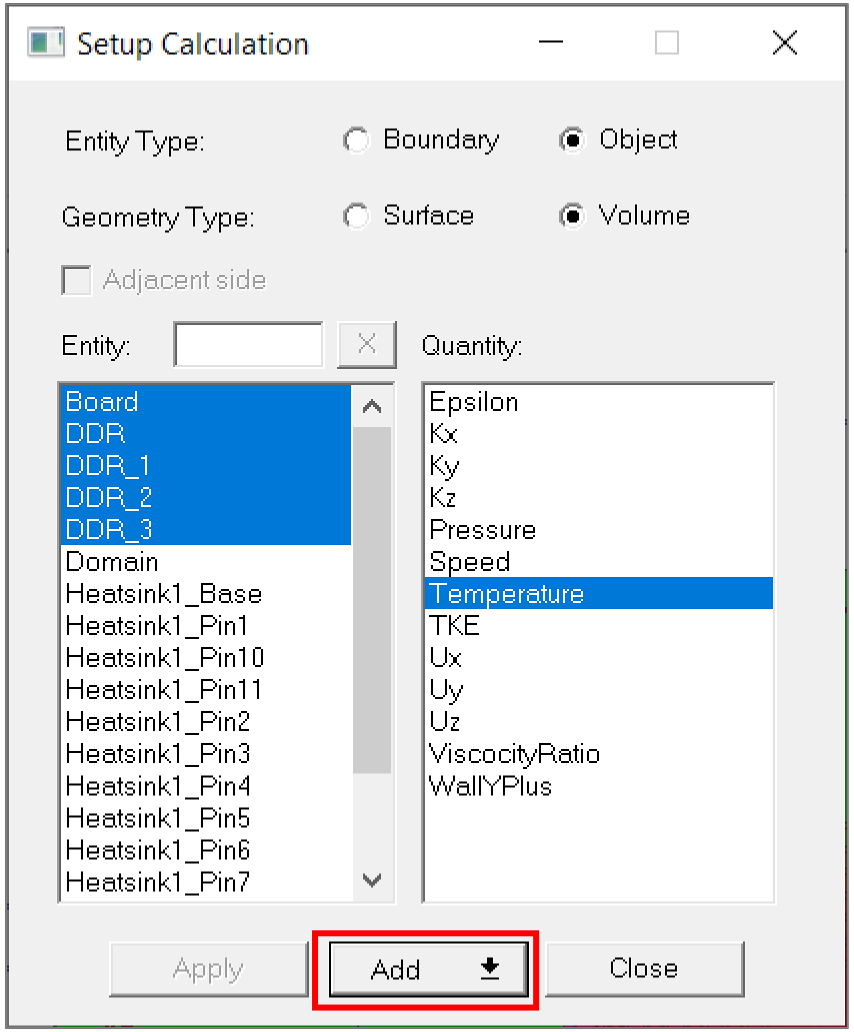




Post Processing

- Click on the Results tab and select Fields Summary
- Select Object for Entity Type and Volume for Geometry Type
- Select Board, DDR objects, Large Flash, and Small Flash from the object list
- Select Temperature under Quantity list
- Click on Add and select Add as Multiple Calculations





Post Processing

Fields Summary: Workshop 01 - Electronics Chassis

Inputs

Solution: Forced_Convection_RadOff : SteadyState

Design Variation: Nominal

Calculations:

Entity Type	Geometry Type	Entity	Quantity	Side	Normal	Min	Max	Mean	Stdev	Area/Volume
Object	Volume	Board	Temperature[C]	Default		23.5361	87.5397	41.361	12.6685	2.8611e-05 m^3
Object	Volume	DDR	Temperature[C]	Default		44.5645	49.269	48.7192	0.437491	3.588e-07 m^3
Object	Volume	DDR_1	Temperature[C]	Default		46.5141	54.8063	52.8855	1.11476	3.588e-07 m^3
Object	Volume	DDR_2	Temperature[C]	Default		44.1497	53.5388	51.1006	1.64162	3.5742e-07 m^3
Object	Volume	DDR_3	Temperature[C]	Default		39.8176	47.5125	45.9191	0.946477	3.588e-07 m^3
Object	Volume	Large_Flash	Temperature[C]	Default		36.446	44.1177	43.4453	0.511525	7.84e-07 m^3
Object	Volume	Small_Flash	Temperature[C]	Default		28.798	34.6158	34.2506	0.479396	1.215e-07 m^3

Setup... Delete

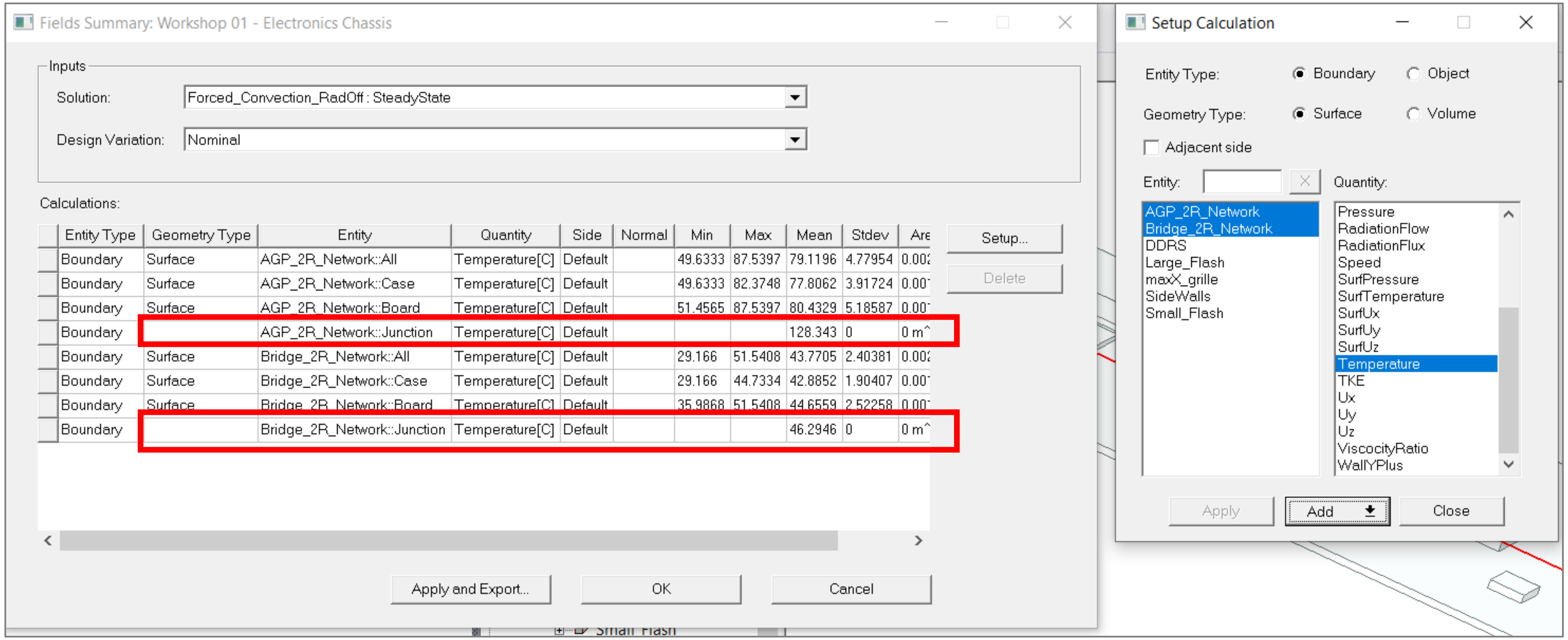
Results can be exported as a CSV file

Apply and Export... OK Cancel

Ansys

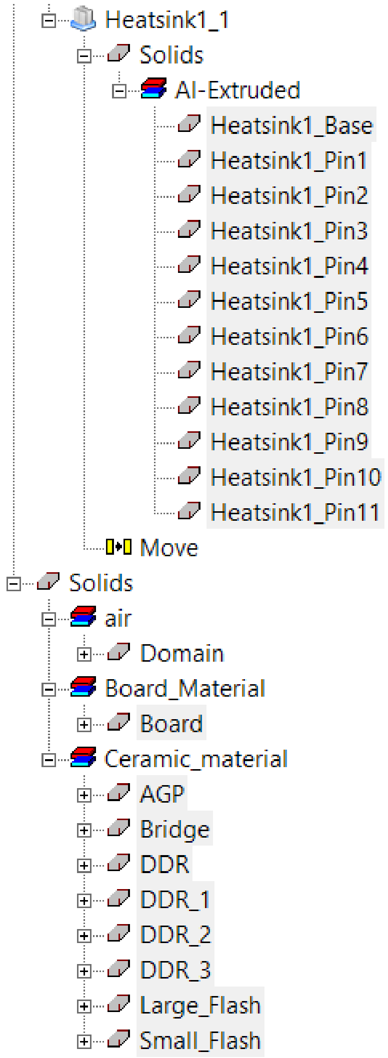
Reporting Network Temperatures

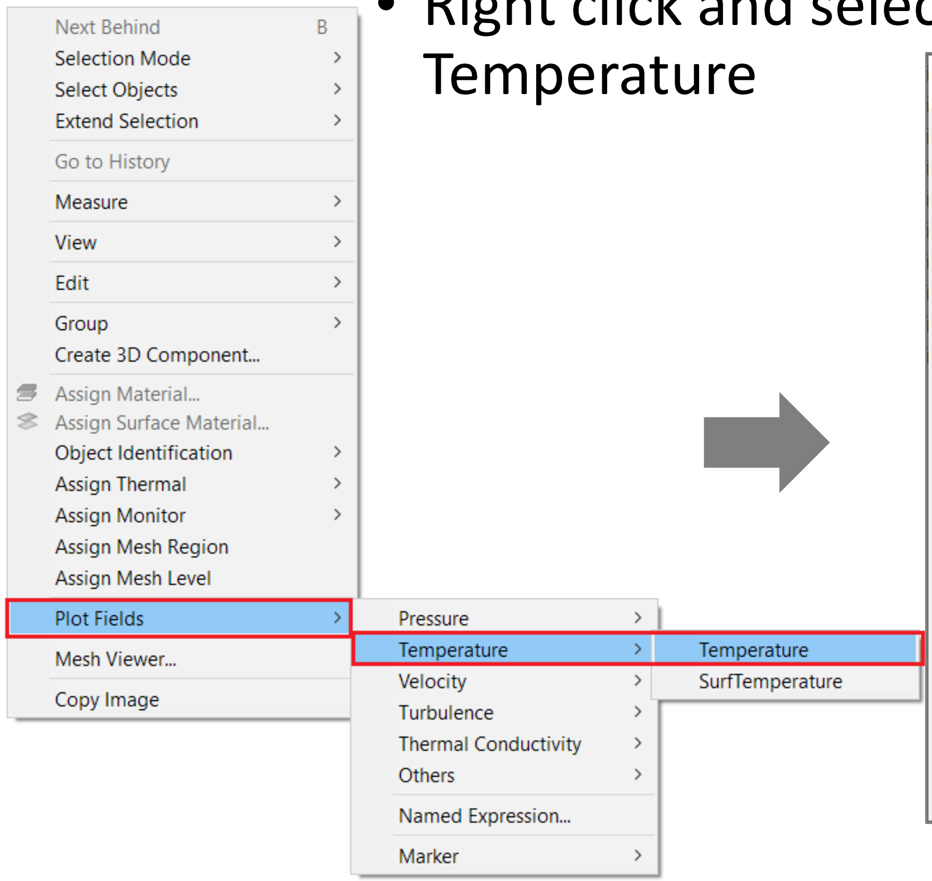
- In the Setup Calculation panel, select Boundary for Entity Type and Surface for Geometry Type
- In the Entity list, select AGP_2R_Network and Bridge_2R_Network
- Select Temperature in Quantity list
- Click on Add and select Add as Multiple Calculations
- Note the Network Junction temperatures for AGP and Bridge

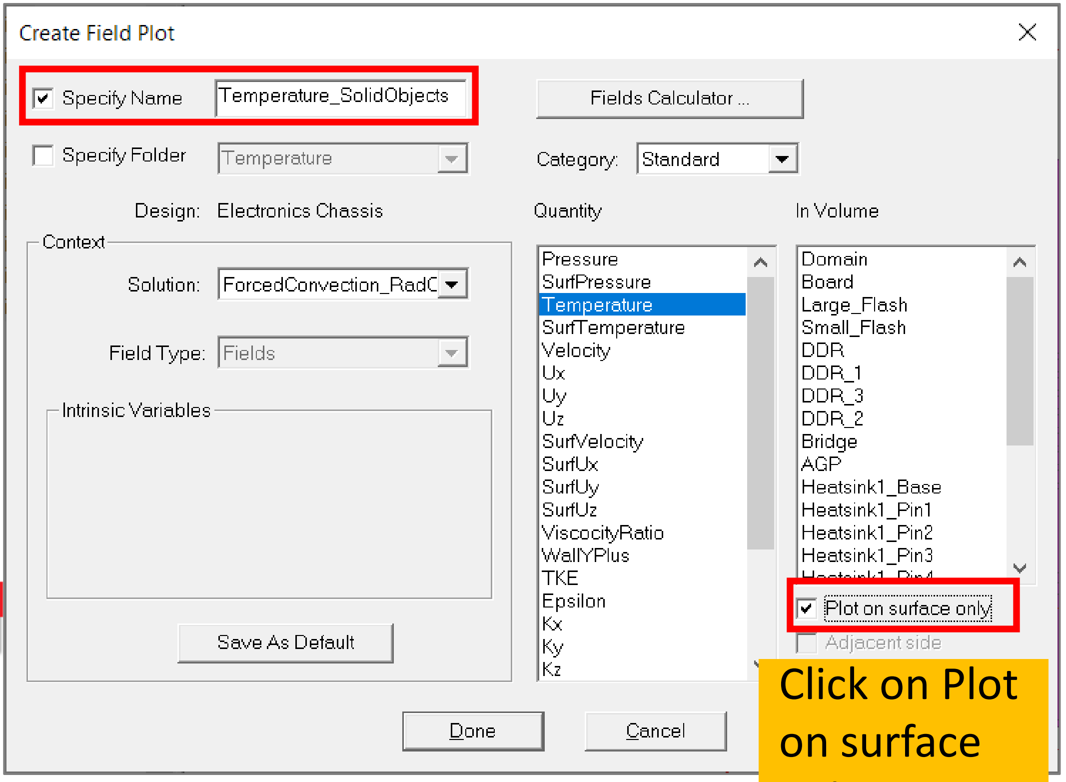


Post Processing

- Select the all the model components from the model tree
- To select the heat sink, expand the Heat sink node and select all the solids
- Right click and select Plot Fields → Temperature → Temperature

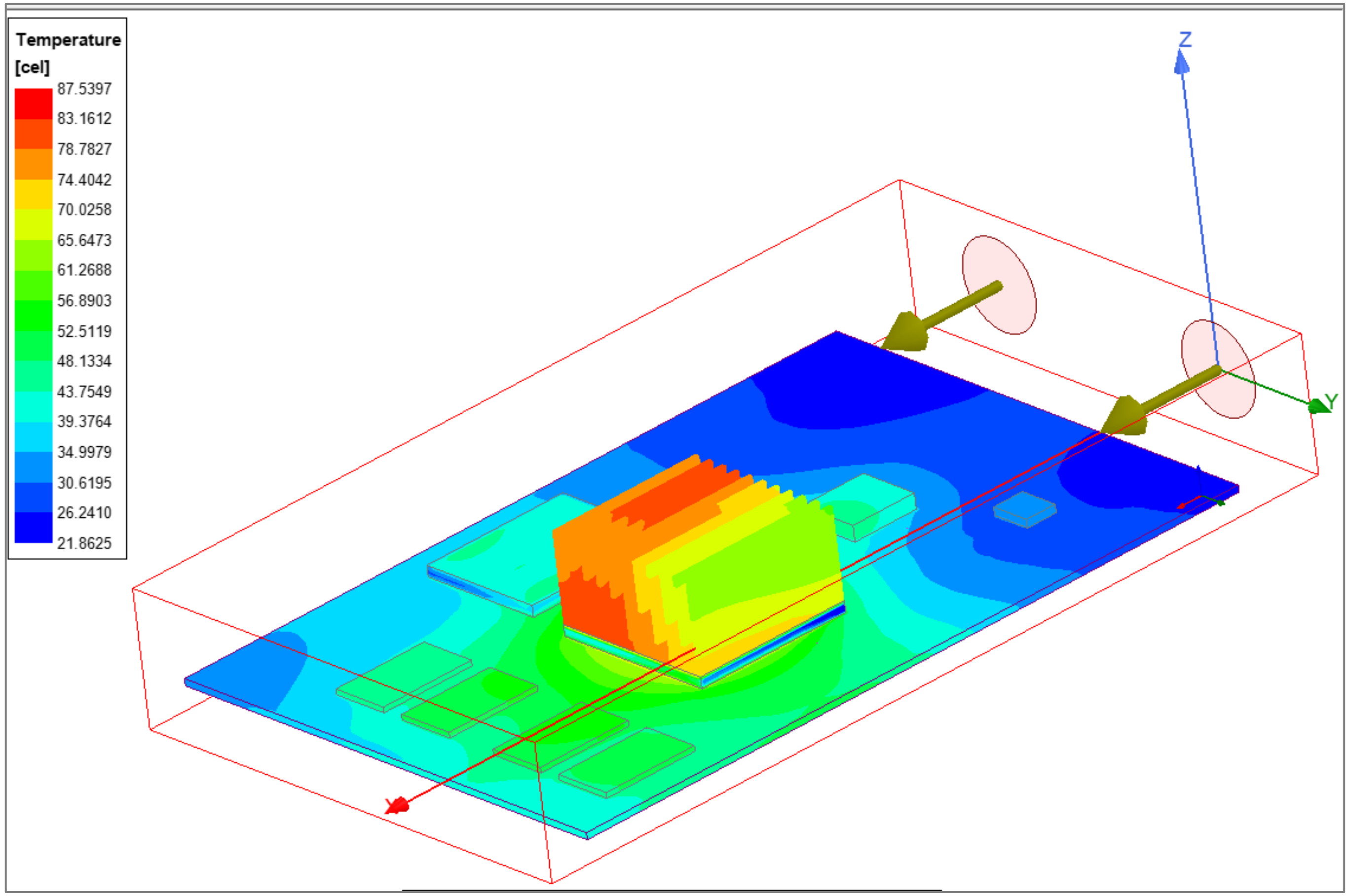




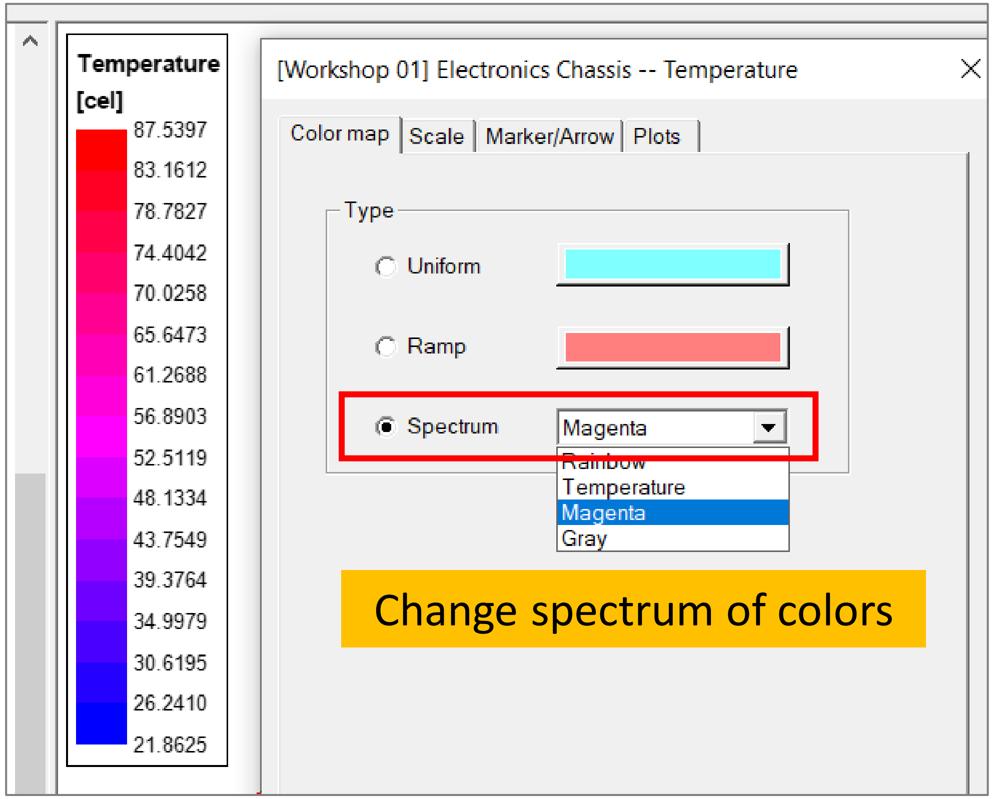


Click on Plot on surface only

Post Processing: Temperature contour on the component surface



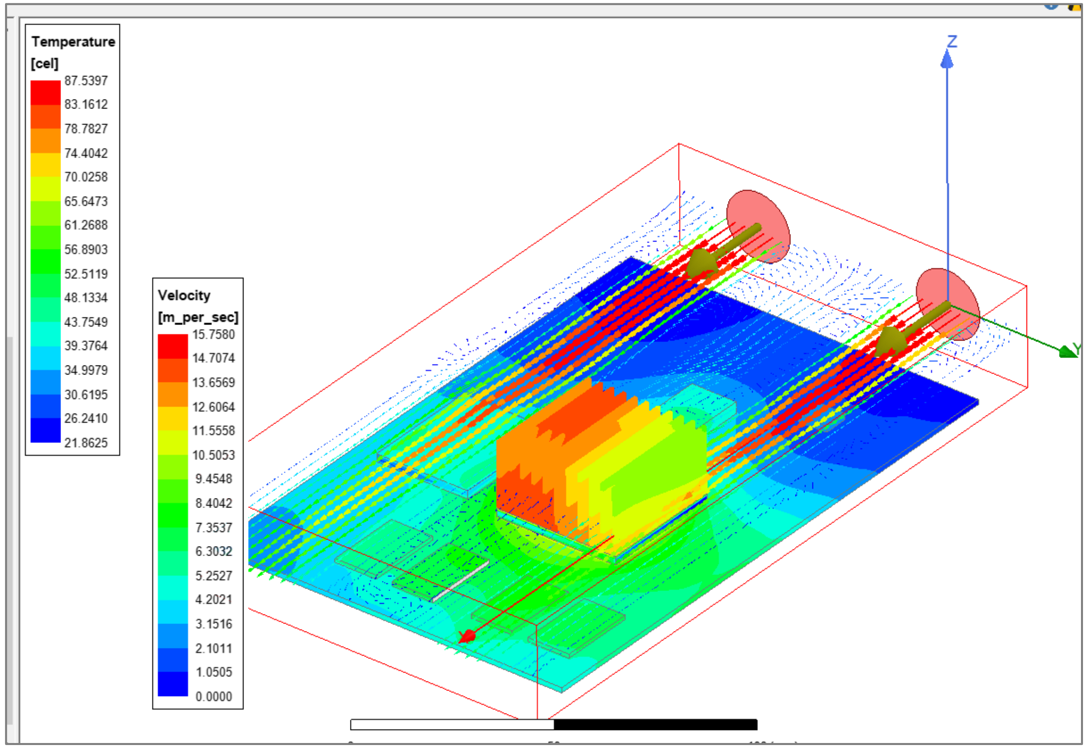
Double click on color legend to get the property window for legend

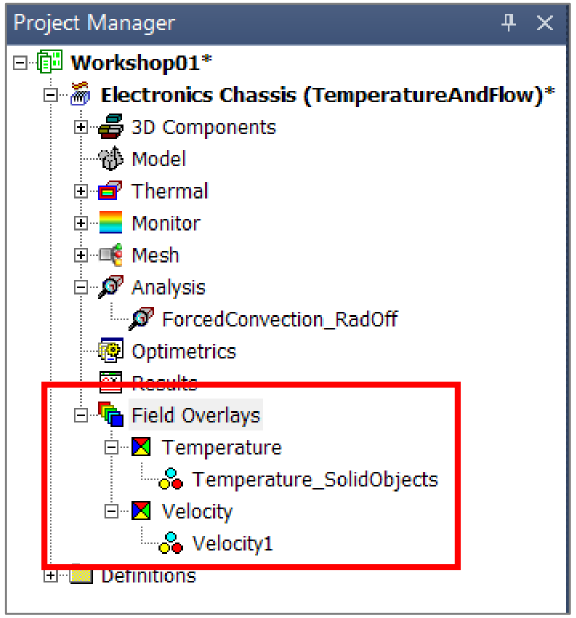


Change spectrum of colors

Post Processing

- Expand the planes node and select Fan_Coord:XY
- Right click in the Graphics area and select Plot Fields → Velocity → Velocity
- Click Done on the Plot Fields panel to accept the defaults
- Drag the legends to show temperature and velocity legends simultaneously

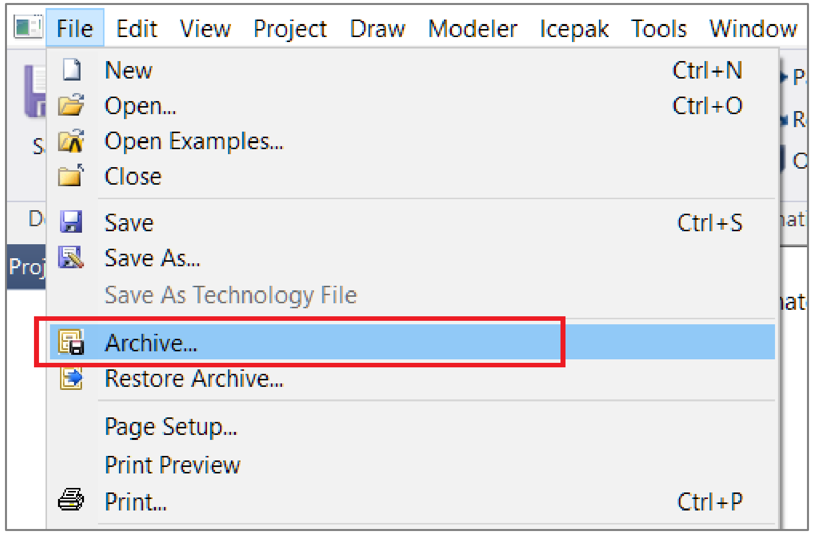


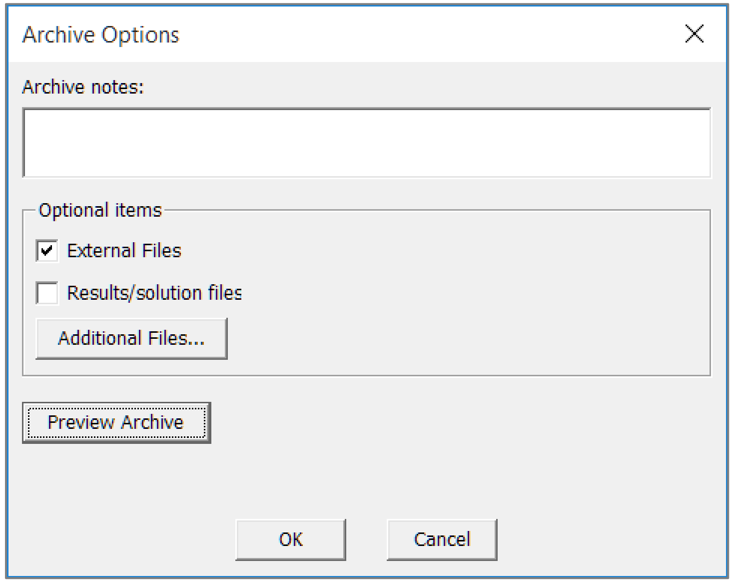


The plots appear under the Field Overlays node in the Project Manager

Save the Model *.aedtz File

- The *.aedtz file is useful for sharing a model with colleagues and Icepak Technical Support
- The *.aedtz file compresses the model and associated boundary conditions, material properties and mesh settings
- Select Results/solution files to include completed solutions
- Additional files can be included in the archive, *.pdf, *.pptx, etc.
- Click OK

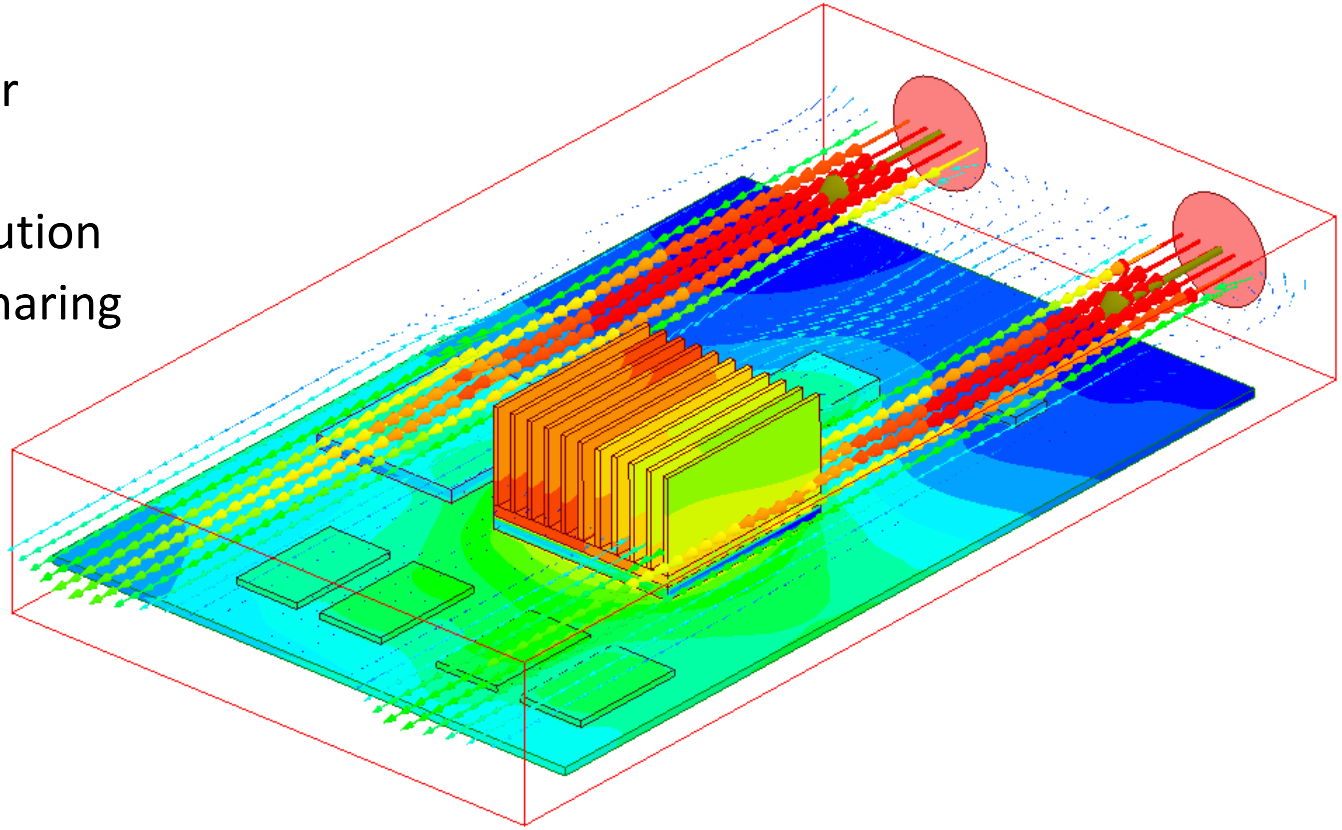




Ansys

Ideal Workshop Outcome

- You should be able to:
  - Build Icepak objects like fans and heatsinks in AEDT Icepak
  - Assign boundary conditions
  - Assign material properties
  - Generate a mesh using the Slider Bar mesher
  - Setup physics and solver settings
  - Run the simulation and post process the solution
  - Save a project and archive it for backup or sharing



Appendix – Additional Exercise

- AEDT is able to import packed Classic Icepak model (*.tzr) except certain features which are not yet available and should provide similar mesh and results
- To import the tzr file (Classic-WS1.tzr available in workshop folder) into AEDT Icepak:
  - simply drag and drop the tzr file into AEDT. You don't have to open an Icepak design type first. AEDT automatically inserts an Icepak design type
- Review the imported model
  - Check for any messages
  - Objects imported successfully should include – all blocks, 2D fans, and boundaries such as walls and grilles
  - Objects not imported –heat sink
- Objects that are not imported can be recreated following the steps in the main workshop.
- Physics settings should also be imported successfully
- Solve and Post process following the steps in the main workshop.

End of presentation