Introduction to Icepak in AEDT

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

Release 2020 R1

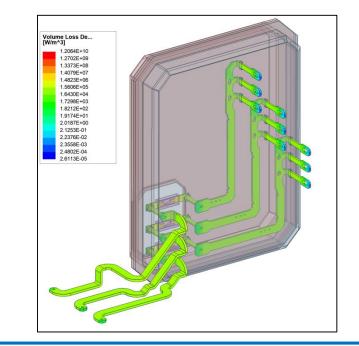


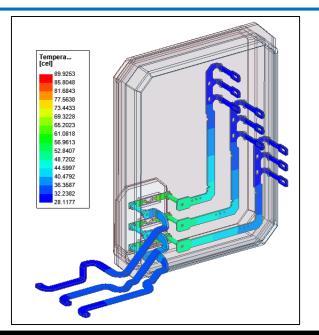
©2020 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

Q3D Extractor and Icepak

ANSYS Q3D Extractor

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





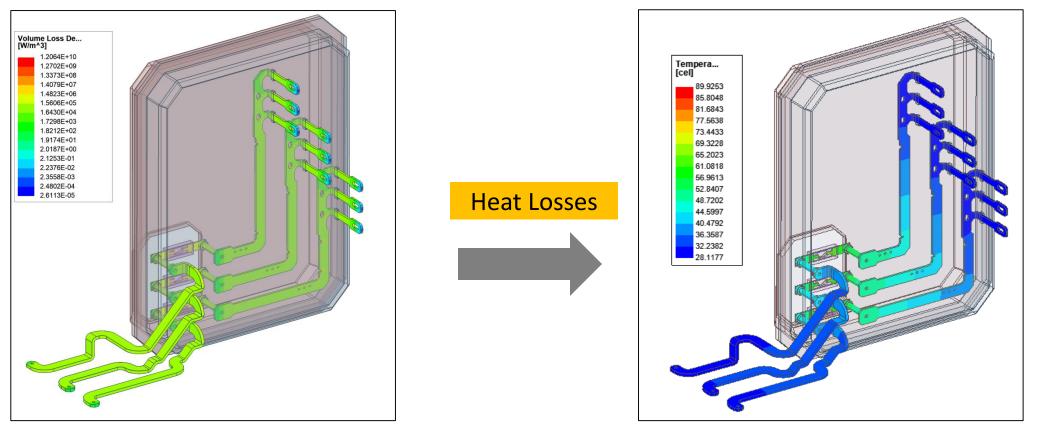
ANSYS Icepak

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



Q3D Icepak Coupling

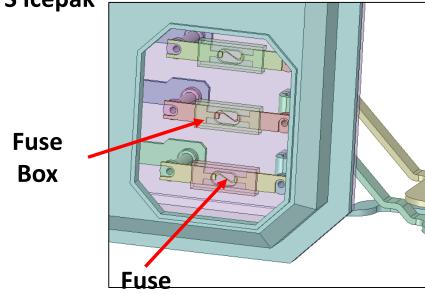
- Q3D calculates both DC and AC Losses.
- The losses are mapped to Icepak. Both surface and volumetric mapping are supported.

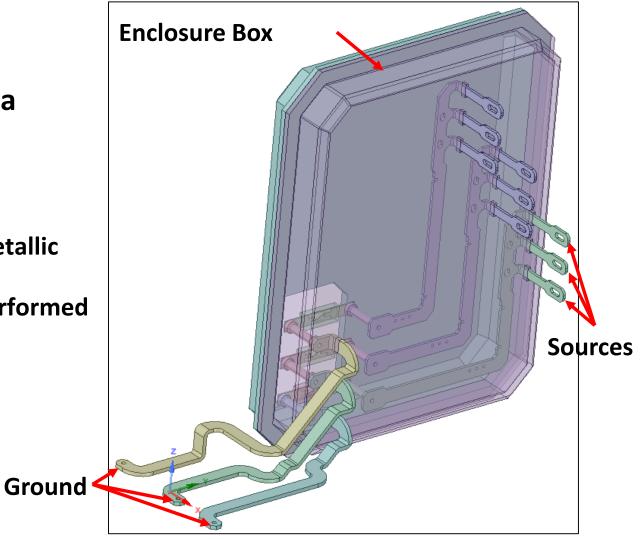






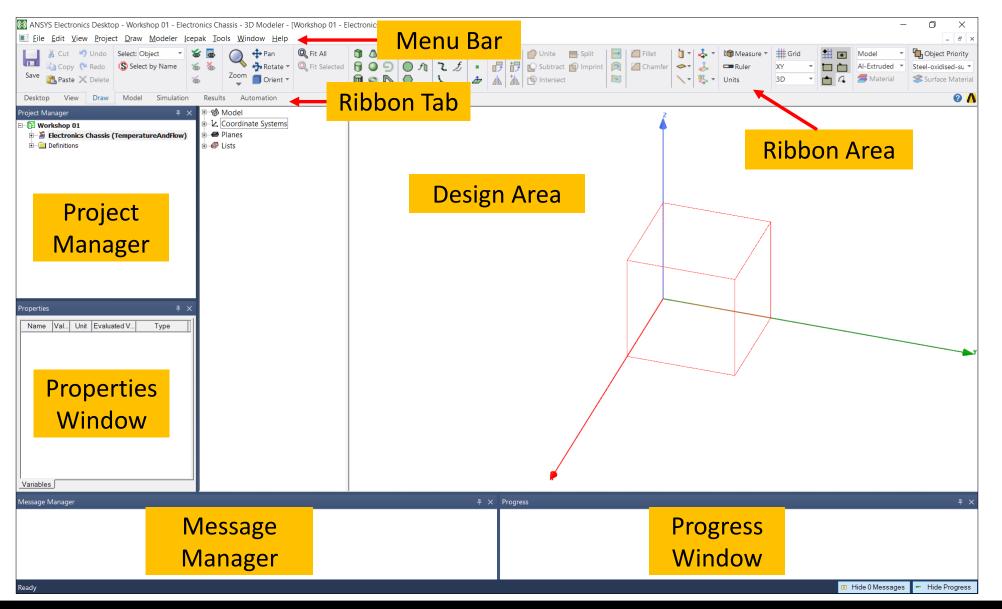
- **Objective:** Calculate the temperatures of a busbar and its enclosure box
- Analysis:
 - A DC RL Analysis is performed in Q3D
 - Q3D is used to calculate the Ohmic losses in metallic parts
 - Thermal analysis with Natural Convection is performed in ANSYS Icepak





//nsys







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

	0 Open				🕸 Project File F	estore Location			×
	Look in: Q3D-Icepa	k _	- 🗕 🖆 📰 -		Save in:	Q3D-Icepak	x 💌	← 🗈 💣 📰▼	
ANSYS Electronics Desktop 2020 R1 File Window Help New Ctrl+N Cut 7 Undo Control Control C	Quick access Desktop Libraries This PC Vetwork C File name: Files of type:	Q3D_Busbar.aedtz Q3D_Busbar.aedtz	ANSYS Electronics	Size 53,847 KB S Open Cancel	Quick access Desktop Libraries This PC	Name Kara Save as type: Overwrite ex Open project	No items match your so Q3D_Busbar.aedt ANSYS Electronics Desktop Project disting files	_	Type Type Save Cancel



Visualize the Nets

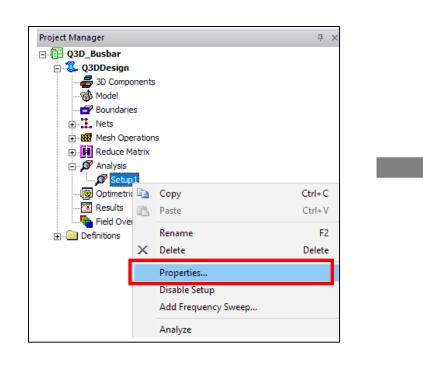
- In the project Manager, under Q3DDesign, Right click Nets and select Visualization ...
- Select "All" in the Group Visualization Options to view the Current Paths for the 4 nets in the model.

			Vis	ualization Options		×			
Project Manager									
🖃 📴 Q3D_Busbar				elect item(s) for visualization					
□ SDDesig B 3D Com B Model				Select 🛨		select All			
- 🗗 Bounda	ries			All	View Geometry	View Name 🔺			
 ■ II Ne ■ II Ne ■ II Ne ■ II Ne II Ne II		> > >		All Signal Nets All Floating Nets All Ground Nets All Sources All Sinks Infinite Ground Plane Thin Conductor		C			
	Delete All			By Name	· ·	·			
	Visualization Set Material Thresholds			Source2_1	∨				
				Ck	ose		0	200	400 (mm)



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



Solve Setup)
General DC RL Exp	pression Cache Defaults	
Name:	Setup1 Finabled	
Solution Frequency:	1 kHz 💌	
Solution Selection -]
	Conductance	
DC 河	Resistance/Inductance	
	C Resistance Only	
AC Resistanc	e/Inductance	
☑ Save fields		
	Use Defaults	1
	HPC and Analysis Options	
-	OK Can	cel

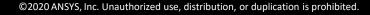


Check Sources

- 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

Project Manager	т >	CG	DC RL AC RL			
🚊 选 Q3DDesign			Name	Value		Unit
		1	BB1:Source1_1		15 A	
Boundaries		2	BB1:Source1_2		15 A	
🕀 🔂 Nets		3	BB1:Source1_3		15 A	
⊞ Mesh Operations		4	BB2:Source2_1		15 A	
		5	BB2:Source2_2		15 A	
Setup1		6	BB2:Source2_3		15 A	
Optimetrics		7	BB3:Source3_1		15 A	
		8	BB3:Source3_2		15 A	
Field Ove →	G Fields >	9	BB3:Source3_3		15 A	
	C R/L Fields >					
D	C R/L PEC Fields >					
D	C R/L Thin Conductor Fields					
A	C R/L Fields >					
PI	ot Mesh					
Ec	lit Sources		○ Voltage	Current		
Ta M	odify Plots					
반 』 M	odify Plot Attributes					1
Se	t Context To Active Window				OK	Ca

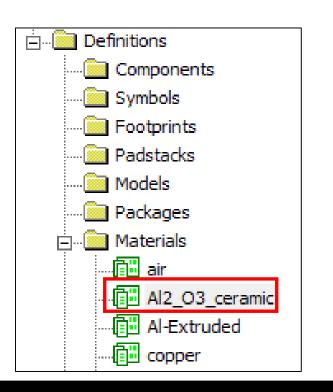
Edit Sources

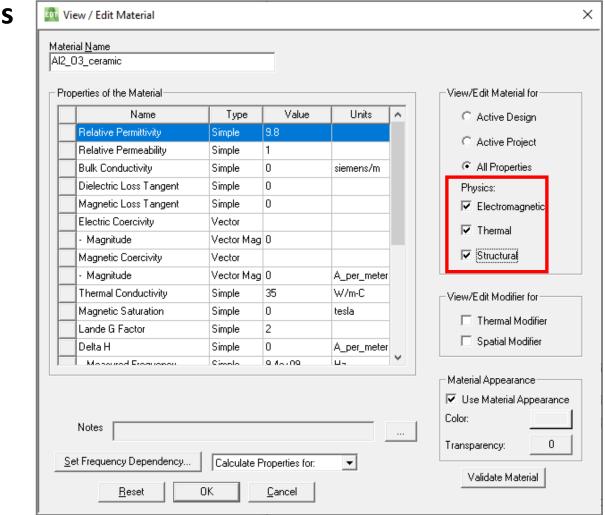


 \times

Material Properties

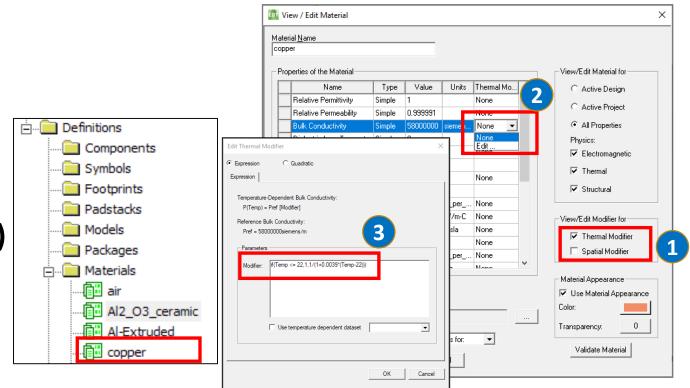
- Explore the material properties by double-clicking on the appropriate entry in the Project Tree under Definitions > Materials
 - Enable All Properties to view the material properties used in electromagnetic, thermal, and structural simulations

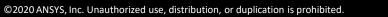




Temperature Dependent Material Settings for a 2-Way Coupling

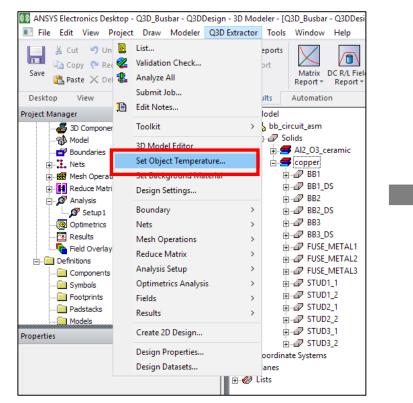
- In order to see the impact of temperature on results in Q3D after the Icepak simulation is completed later, material properties must be temperature dependent.
- Copper bulk conductivity (the reciprocal of electrical resistivity -found under Electromagnetic category) varies with temperature.
- After 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)))</pre>





Enabling Feedback

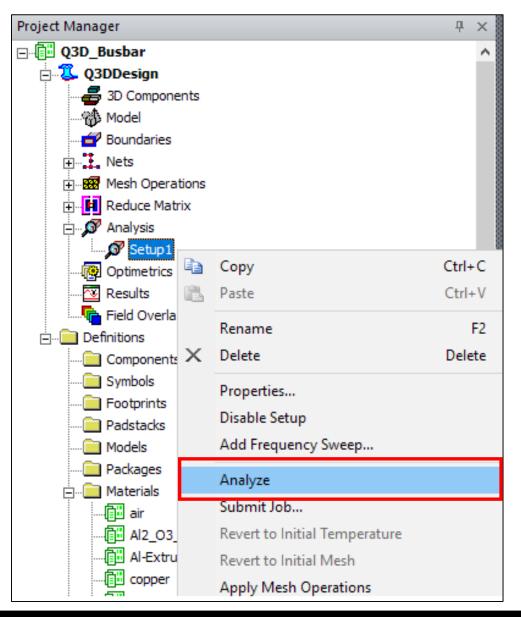
- One more step is needed to prepare the Q3D model for a bidirectional coupling to ANSYS Icepak, enabling feedback.
- From the main menu Q3D Extractor, click on "Set Object Temperature"
 - Check both "Include Temperature Dependence" and "Enable Feedback"



🔿 🔹 Object Name	Material	Temperature Dependent	Temperature	Uni	^
BB1	copper	×	22	cel	
BB1_DS	copper	×	22	cel	
BB2	copper	V	22	cel	
BB2_DS	copper	V	22	cel	
BB3	copper	×	22	cel	
BB3_DS	copper	V	22	cel	
COVER_FUSE_BOX	Al2_03_ceramic				Y
<				>	
elect By Name:		Select	1		
emperature:	22 cel	▼ Set	Set	Defaul	ŀ



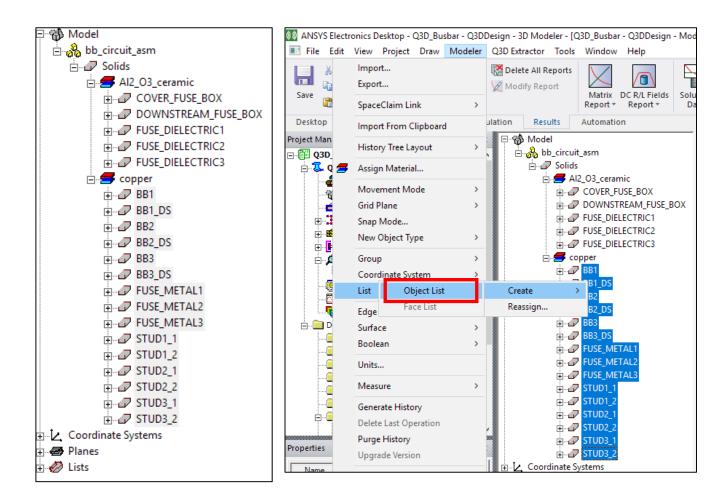
- To Run Q3D Analysis:
 - Right click Setup1 and select Analyze





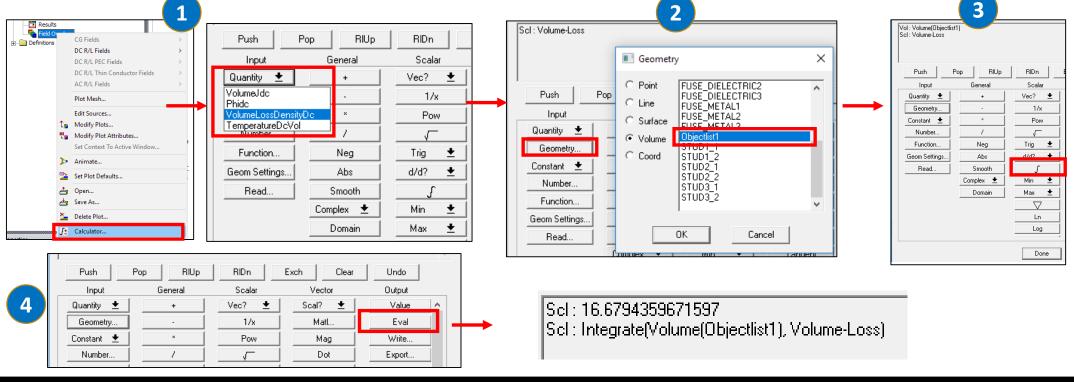
Create an Object List

- 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



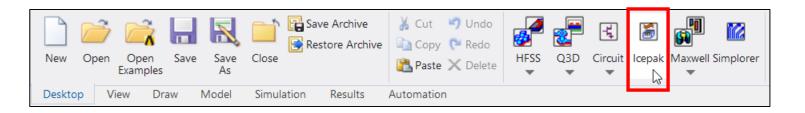
Report Volumetric Losses

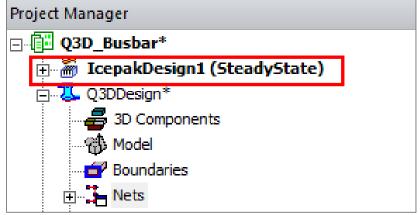
- Use the Field Calculator in Project Manager/Q3D Design/Field Overlays to report the Volume Losses on ObjectList1.
 - From Input/Quantity add VolumeLossDensityDC.
 - From Input/Geometry Add Objectlist1. From Scalar add Integral.
 - From Output click Eval. The Volume integral loss value is displayed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



Insert Icepak Design

• From Ribbon Menu/Desktop/Insert Icepak Design.

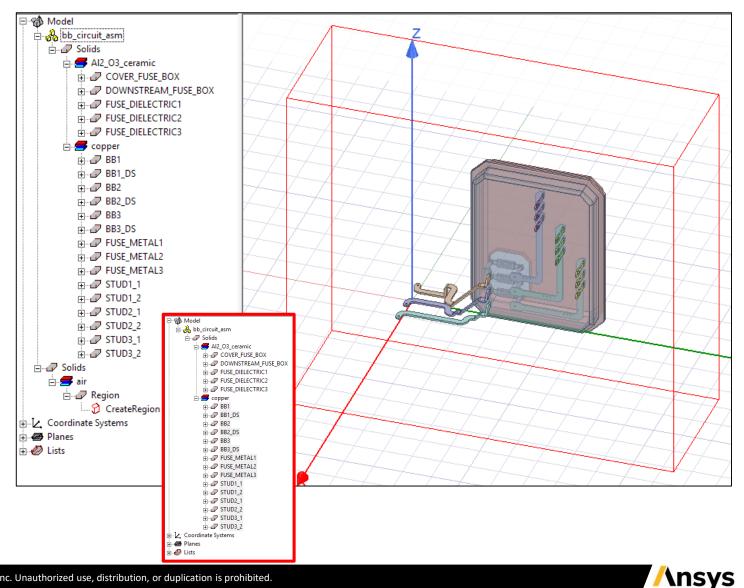






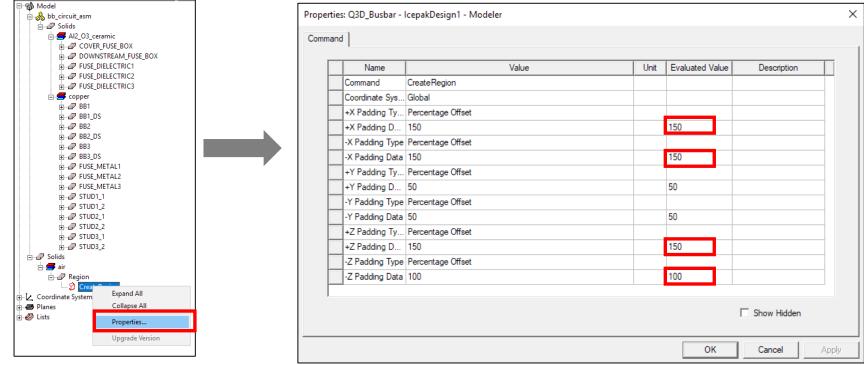
Bring Q3D Geometry and Material Setup into Icepak Design

- 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





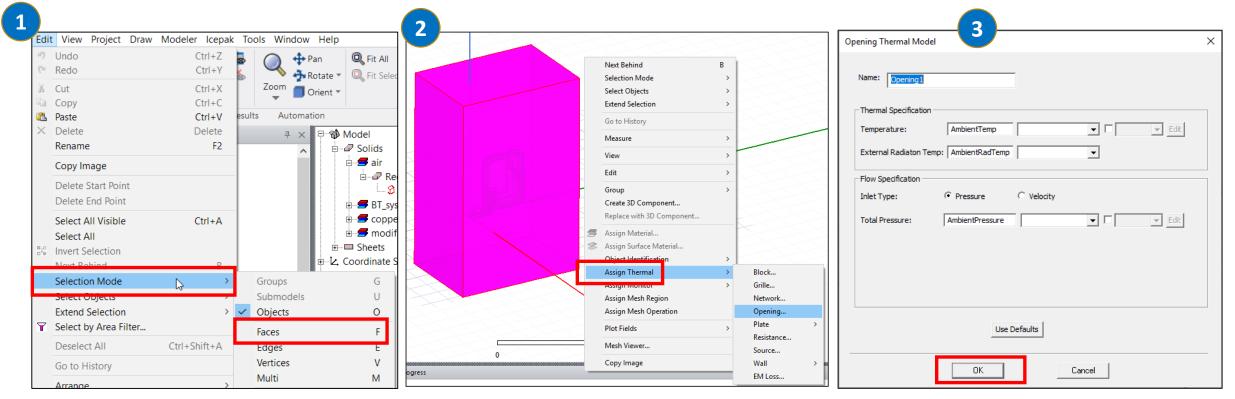
- In the Model tree, Double Click on the Region > Create Region.
- Increase the padding in +Z and –Z direction to 150 and 100 Percentage, respectively.
- Increase the padding in +X and –X direction to 150 and 150 Percentage, respectively.
 - It is recommended to have larger padding in the direction opposite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





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.

T B→ C FUSE_DIELECTRIC2 B→ C FUSE_DIELECTRIC3 D→ BB1 BB1 B→ BB1_DS		2 Setup Link X EM Loss	×
B	Expand All Collapse All Select >	General Variable Mapping Intrinsics Product: ElectronicsDesktop Name: EMLoss1 Source Project: Use This Project Setup Link Save source path relative to: Sync Loss Type From Source © The project directory of selected product Image: Construct of the project directory of selected product Image: Construct of the project directory of selected product Source of Field Sourc	
BUD2_2 BUD3_1 BUD3_1 BUD3_2 STUD3_2 STUD3_2 STUD3_2 STUD3_2 STUD3_2 STUD3_2 STUD3_2 STUD3_2	Edit > Group > Create 3D Component Assign Material View > Properties Object Identification >	Image: Contract of this project Image: This project Imag	^
Block Grille Network Opening Plate > Resistance Source Wall >	Assign Thermal > Assign Monitor > Assign Mesh Region Assign Mesh Level Plot Fields > Mesh Viewer	Simulate source design as needed Preserve source design solution Note: In extractor mode, source project will be saved upon exit. OK Cancel OK Cancel	*



Icepak Design Settings

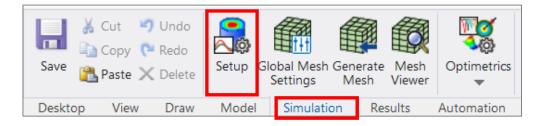
- Right Click the Icepak Project and Select Design Settings...
- In the Ambient Conditions tab, keep the default values at 20°C
- In Gravity tab, select negative Z direction for gravity vector. (Gravity must also be enabled in Setup which will be showed in following slide.)

Image: Comparison of the second se			Ctrl+C	2 Icepak Design Settings	
- 👘 Model	Ē.		Ctrl+V	Icepak Design Settings	Х
EMLoss 1		Rename	F2	Ambient Conditions Gravity Validations Ambient Conditions Gravity Validations	
Monitor Monitor Mesh Mesh Mesh Mesh Mesh Mesh Mesh Mesh		Convert to Full Access Convert to Read Only Solution Type List	Delete	Temperature 20 cel Gravity Vector Global::Z Gauge Pressure 0 n_per_meter_sq C Positive Radiation temp 20 cel C Negative	
	_	Validation Check			
Definitions		Analyze All Edit Notes			
000000000000000000000000000000000000000		Toolkit	>		
Properties Name Vali		Import IDF			
		Design Settings			



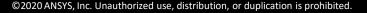
Analysis Setup

• 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"

Icepak Solve Setup Dialog	×
General Convergence Solver Settings Radiation Defaults	_
Name Setup 1 I Enabled	
Maximum Number of Iterations 200	
−Problem Types ✓ Temperature ✓ Flow	
Flow Regime C Turbulent Options	
C Off C Discrete Ordinates C Ray Tracing	
Include Gravity Solve Flow and Energy Equations Sequentially	
Use Defaults HPC and Analysis Options	
Solve Setup Defaults 🛨 OK Cancel Help	





Analysis Setup

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

Icepak Solve Setup Dialog			×
General Convergence Solver	Settings Radiation Default	s	
Flow	0.001		
Energy	1e-07		
Turbulent Kinetic Energy	0.001	•	
Turbulent Dissipation Rate	0.001		
Specific Dissipation Rate	0.001		
Discrete Ordinates	1e-06		
	Use Defaults		
Solve Setup Defaults 👲	ОК	Cancel	Help

- 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	Icepak Solve Setup Dialog		×
-	General Convergence Solver Settings Radiat	tion Defaults	1
– Click "OK"	Iteration Parameters Flow Iterations per Radiation Iteration	5	
	Angular Discretization		
	Theta Divisions:	5	
	Phi Divisions:	2	
	Theta Pixels:	2	
	Phi Pixels:	2	
	Use Defa	ults	
	Solve Setup Defaults	OK Cancel Help	

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.

Icepak Solve Setup Dialog		×
General Convergence Solver Set	tings Radiation Defaults	1
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
	_	
		Advanced Options
	Use Defaults	
Solve Setup Defaults 👲	ОК	Cancel Help

- 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.

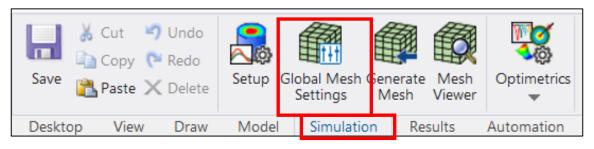
	Under	-relaxa	tion	Discretiza	tion Sche	me		
Pressure	0.7			Standard		•		
Momentum	0.3			First		•		
Temperature	1			First		•	Secondary	Gradi
тагранно кілеас спетду	0.0		-	First		·		_
Turbulent Dissipation Rate	0.8			First		~		
Specific Dissipation Rate	0.8			First		-		
	1			ji nac		_		
Discrete Ordinates near Solver Options	Тур	De		First	Residua Tol	I Reducerance	tion Stabili	zatio
Discrete Ordinates	,	De		First	Residua Tol		tion Stabili	zatio
Discrete Ordinates	, Тур		(First	Tol	I Reduc	Stabil	zatio
Discrete Ordinates	, Тур V		0.1	First	Tol	I Reduc	Stabil	zatio
Discrete Ordinates near Solver Options Pressure Momentum	, Typ V flex	•	0.1	First	Tol	I Reduc	None	zatio
Discrete Ordinates near Solver Options Pressure Momentum Temperature	Typ V flex	•	0.1	First	Tol	I Reduc	None	zatio
Discrete Ordinates near Solver Options Pressure Momentum Temperature Turbulent Kinetic Energy	V flex flex	•	0.1	First	Tol	I Reduc	None	zatio



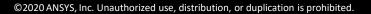
Check Global Mesh Settings

• In the Simulation tab of the ribbon Menu, Click Global Mesh Settings.

• We will use the default Medium Mesh Resolution (Resolution 3).

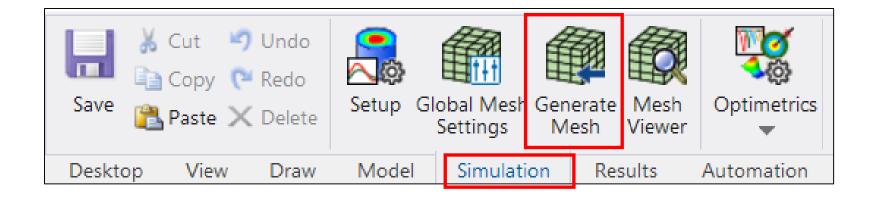


	\square	
General Advanced Defau	ults	
Name Global	✓ Enabled	
Auto Mesh Setting		
Coarse Resolut	tion Fine	
]		
Smal Mesh Si	ize Large	
	Use Defaults	



Mesh Generation

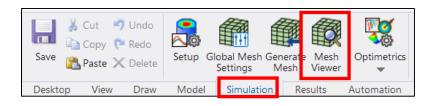
• In the Simulation tab of the ribbon Menu, Click Generate Mesh.





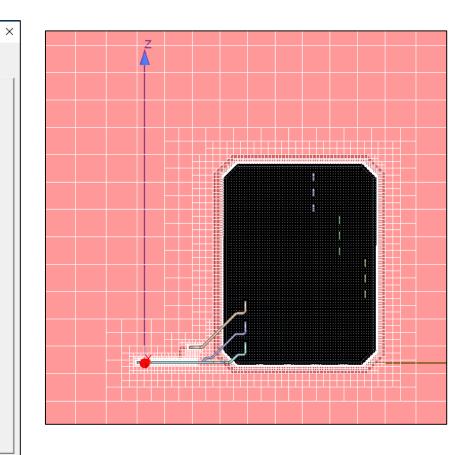
Display Mesh

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



• Display Mesh on X plane through center.

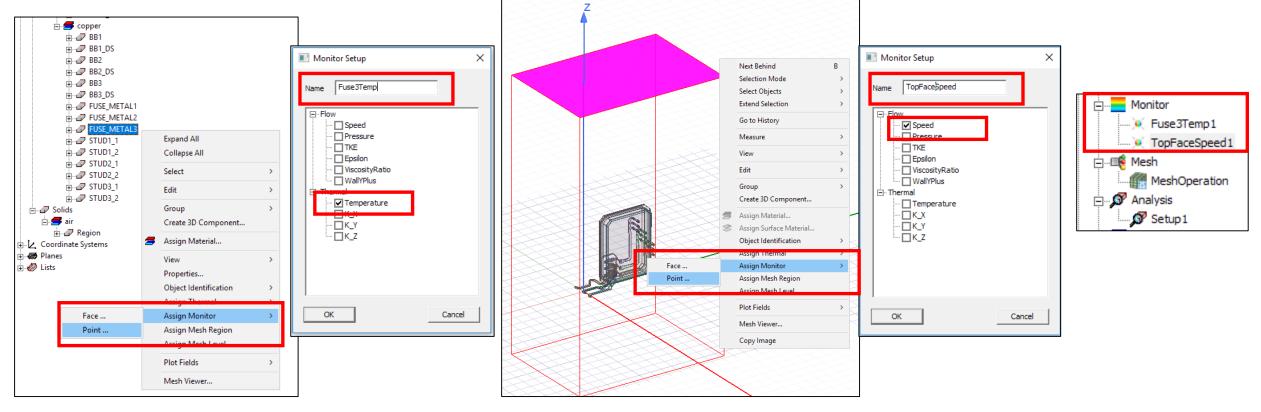
[Q3D_Busbar] IcepakDesign1 Mesh visualization	
Num elements: 3220419 Mesh Display Quality	
Mesh display on Show C Cut plane C Geometry/Boundary selection	
- Plane Location	
Define plane X plane through center _ Update	
PX 1.59999847 PY 0 PZ 0	
NX 1 NY 0 NZ 0	
A 1 B 0 C 0 D 1.59999843	
Offset: 0.500	
Display attributes	
Gectioned C Raw elements	
C Wire 🏵 Shaded 🔽 Grid Color 🖾 Color by object	
Plane transparency 0 1	
Geometry Filters:	
Region FUSE_DIELECTRIC3 FUSE_METAL3 FUSE_DIELECTRIC2 FUSE_METAL2 FUSE_METAL2 FUSE_DIELECTRIC1	
FUSE_METAL1 BB2 BB1	
BB3 ISTUD3 1	
Load mesh Unload mesh	Close



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

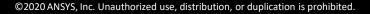






- To run the Icepak analysis (one-way simulation with EM losses from Q3D without a temperature feedback to Q3D):
 - Right click Setup1 and select Analyze

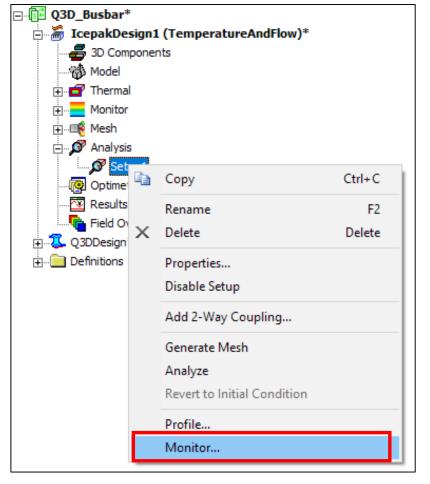
⊡ 🔊 Analysis				
Optimetu Results Field Ov Q3DDesign* ⊡ Definitions		Сору	Ctrl+C	
		Rename Delete	F2 Delete	
		Properties Disable Setup		
		Add 2-Way Coupling		
		Generate Mesh		
		Analyze		
		Revert to Initial Condition		
		Profile		
		Monitor		

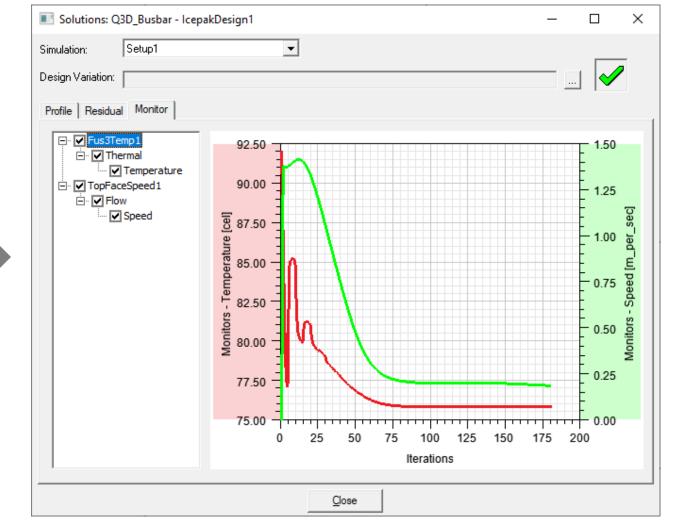




Review Solution Monitors (For Convergence)

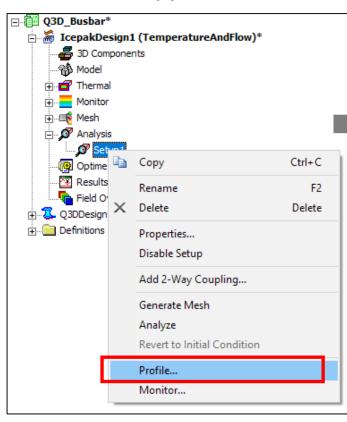
- To review Solution Monitors and residual plots:
 - Right click Setup1 and select Monitor





Review Solution Profile

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

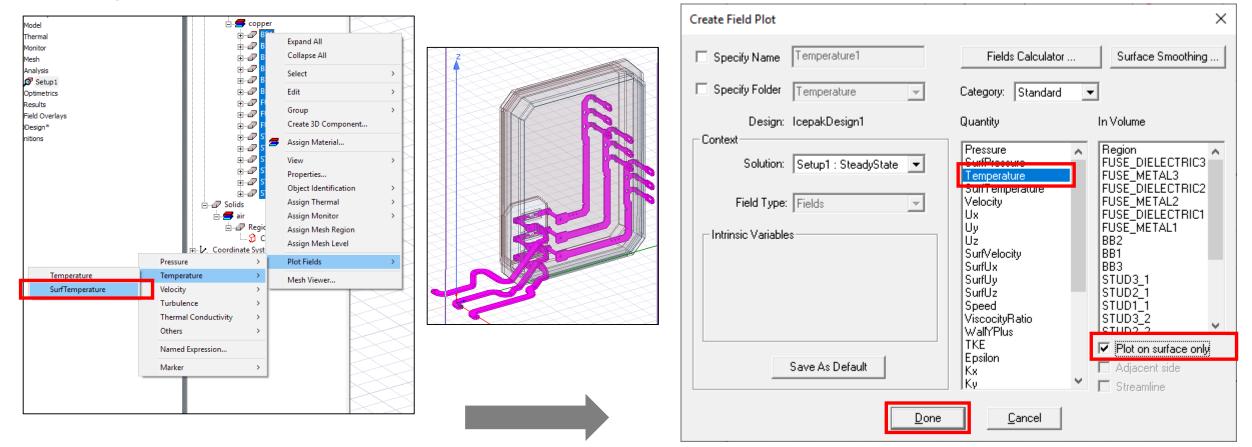


Solutions: Q3D_Bu	sbar - IcepakDes	ign1		- D X				
imulation: Setu	51	•	[
esign Variation:								
Profile Ionitor								
Task	Real Time	CPU Time	Memory	Information				
Populate Solver In	out 00:01:52	00:04:17	5.79 G					
Get External EM Lo	s 00:00:15	00:00:00	0 K	Source: This Project*, Q3DDesign - Setup1 : LastAdaptive				
				Total Loss on Source Object 'FUSE_METAL3': 4.81488W, Scaling Factor: 1.00772.				
				Total Loss on Source Object 'FUSE_METAL2': 4.81587W, Scaling Factor: 1.00798.				
				Total Loss on Source Object 'FUSE_METAL1': 4.81371W, Scaling Factor: 1.00728.				
				Total Loss on Source Object 'BB2': 0.262772W, Scaling Factor: 1.00119.				
				Total Loss on Source Object 'BB1': 0.325167W, Scaling Factor: 1.0003.				
				Total Loss on Source Object 'BB3': 0.24896W, Scaling Factor: 0.9997.				
				Total Loss on Source Object 'STUD3_1': 0.0483642W, Scaling Factor: 1.00569.				
				Total Loss on Source Object 'STUD2_1': 0.0483984W, Scaling Factor: 1.00632.				
				Total Loss on Source Object 'STUD1_1': 0.0483507W, Scaling Factor: 1.00377.				
				Total Loss on Source Object 'STUD3_2': 0.0474632W, Scaling Factor: 1.00309.				
				Total Loss on Source Object 'STUD2_2': 0.0475111W, Scaling Factor: 1.00226.				
				Total Loss on Source Object 'STUD1_2': 0.0475027W, Scaling Factor: 1.00617.				
				Total Loss on Source Object 'BB2_DS': 0.368427W, Scaling Factor: 1.00127.				
				Total Loss on Source Object 'BB1_DS': 0.359824W, Scaling Factor: 1.00209.				
				Total Loss on Source Object 'BB3_DS': 0.377226W, Scaling Factor: 1.00142.				
Solver Initialization	00:01:29	00:01:28	10.4 G	Mapped EM Loss: 16.653 (Watt)				
				15.65 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FUSE_METAL2, FUSE_METAL3, STUL				
Solve	01:02:40	02:50:15	14.6 G					
Solution Process				Elapsed time : 01:07:36 , Icepak ComEngine Memory : 6.73 G				
<				>				
				Export				
				Close				

Ansys

Post-Processing – Contours of Temperature

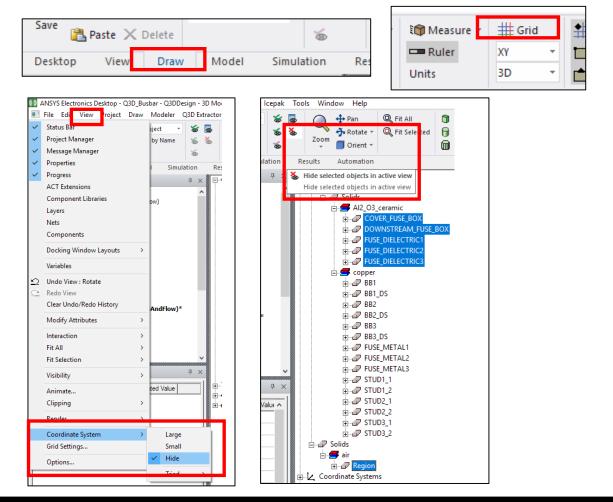
 Select any desired Solids for post-processing: RMB click > Plot Fields > Temperature > Temperature

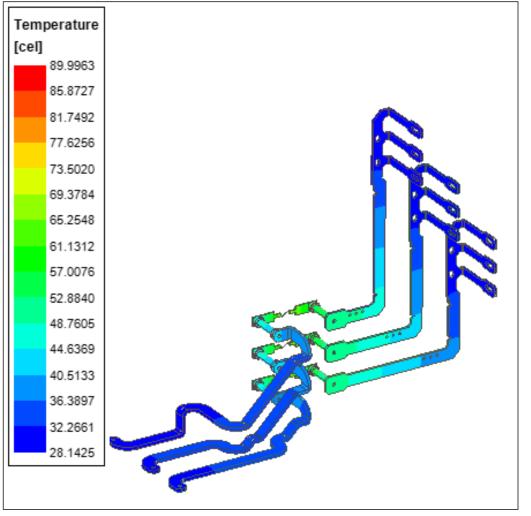




Postprocessing Temperature Contours

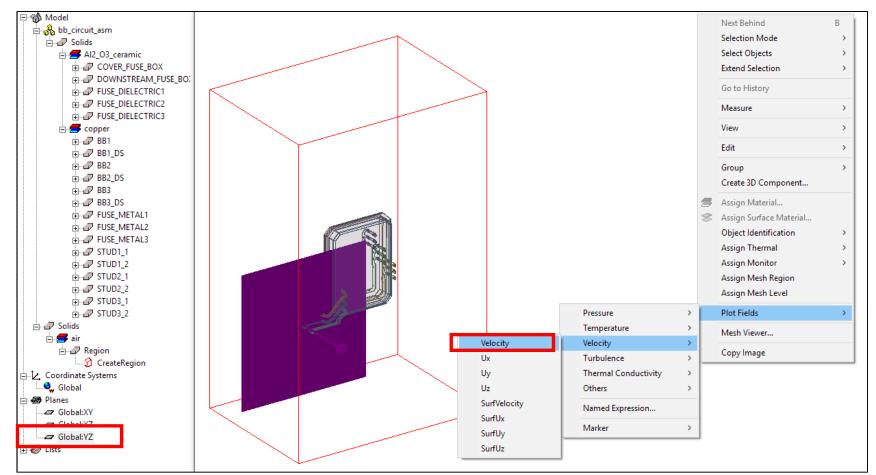
 Optional: user can hide the coordinate system, grids, region, as well as any unused objects for a particular post-processing





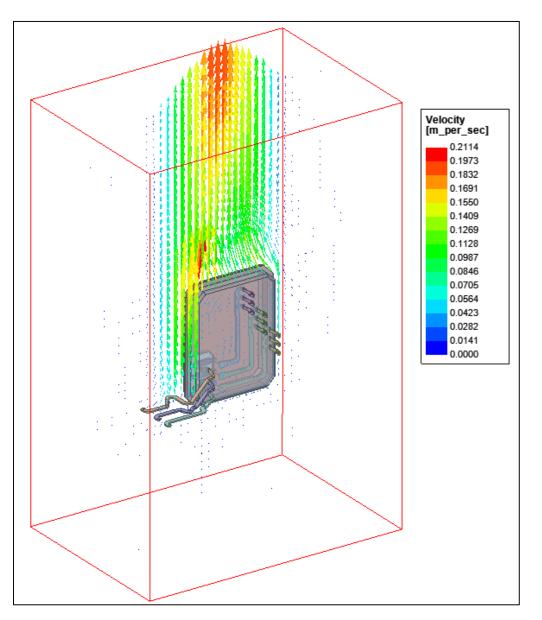
Post-Processing – Velocity Vectors

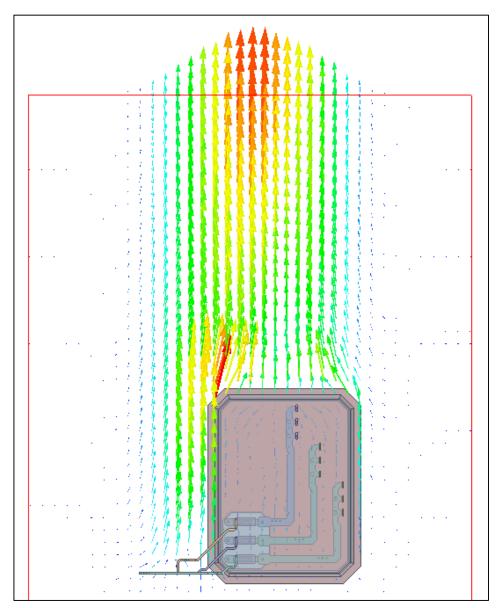
- Select the "Global:YZ" Plane to plot the Velocity Vectors
 - RMB Click Field Overlays > Plot Fields > Velocity > Velocity

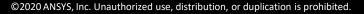




Postprocessing Velocity Vector









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.

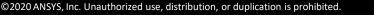
Analysis Setup 1 	Сору	Ctrl+C	2-Way Coupling	×
Results Field Ove Constraints	RenameDeletePropertiesDisable SetupAdd 2-Way CouplingGenerate MeshAnalyzeRevert to Initial ConditionProfileMonitor	F2 Delete	Number of Coupling Iterations: 3 Continue Icepak Iterations During Coupling • Max. Icepak Iterations per Coupling: 20 OK Cancel	

Ansys

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

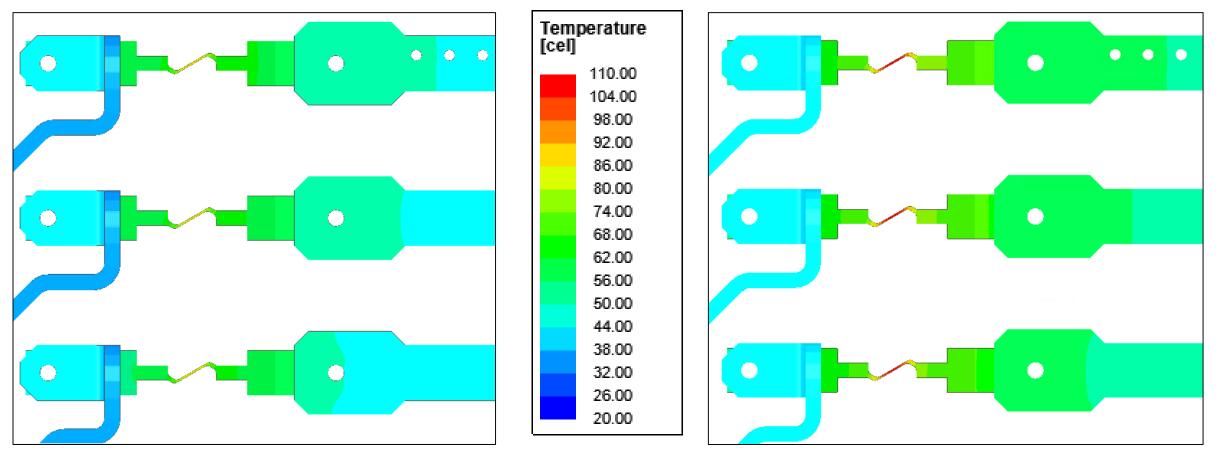
- 2-Way coupling clearly show a higher EM Loss generation and higher temperature values.
- For most accurate solution, a 2-way electro-thermal analyses until full convergence must be conducted.
 - In Q3D simulation at the ambient temperature, the total power was 16.65 W
 - With the two-way coupling, 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%).

	Solver Initialization	00:01:03	00:01:0)2 11.1 (G Mapped EM Loss: 16.653 (Watt)
Coupling Iteration 1					16.65 (Watt) on Volume (BB1, BB1_DS, BB2, BB2_DS, BB3, BB3_DS, FUSE_METAL1, FL
	Solve	00:17:06	02:15:2	28 15 G	
		I	I	I	
Counting Iteration 2	Solver Initialization	00:01:05	00:01:05	11.1 G	Mapped EM Loss: 20.115 (Watt)
Coupling Iteration 2					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	
	Solver Initialization	00:01:03	00:01:03	11.1 G	Mapped EM Loss: 20.88 (Watt)
Coupling Iteration 3					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	
					Total Loss of Source object (bb3_b3, 0.40501179, Scaling Factor, 1.00141.
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
			i ···	İ	



One-Way versus Two-Way Coupling - Temperature

• 20°C difference in the fuse maximum temperature (89.9°C vs. 109.9°C)



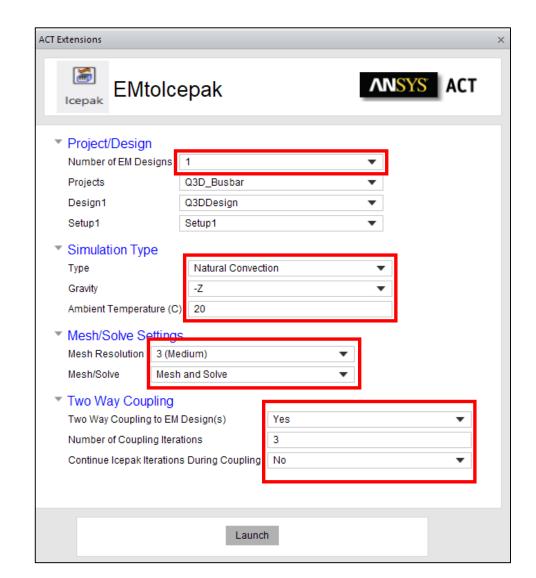
Coupling Iteration 1

Coupling Iteration 3

//nsys

Running the Electro-Thermal ACT (Automated Workflow)

- All the aforementioned steps to create a thermal solution can be automated using the electrothermal 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"





- 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

