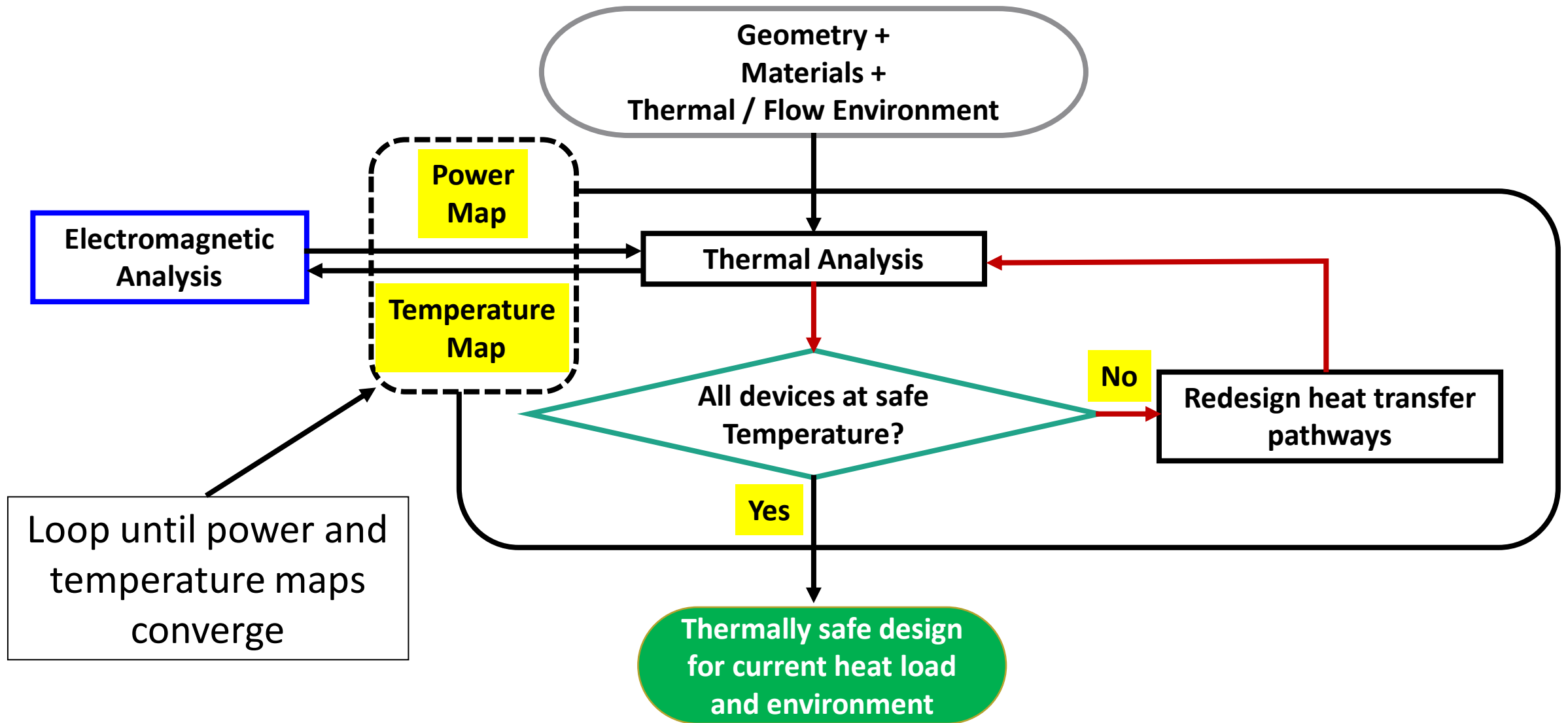


Module 4 – Lecture 1: Electro-Thermal Analysis

Release 2020 R1

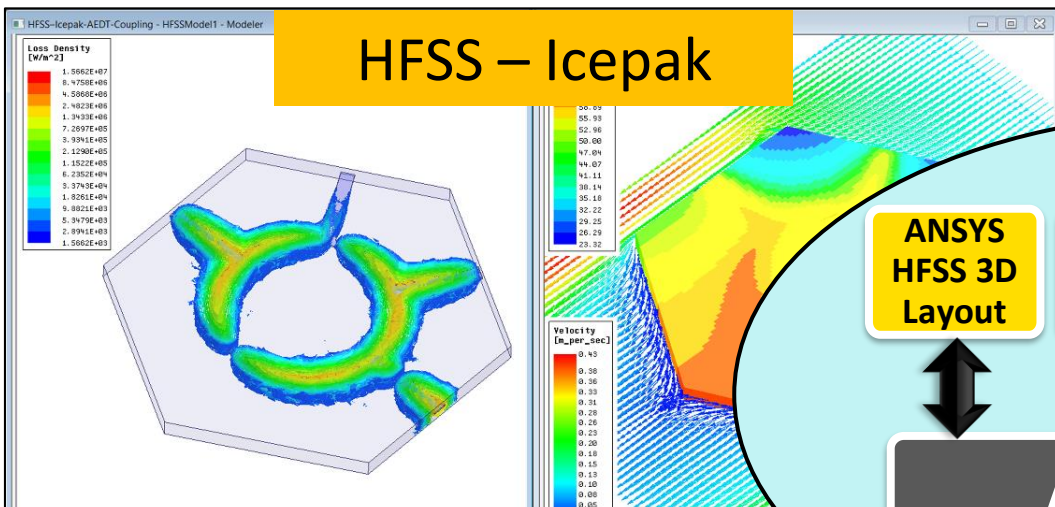
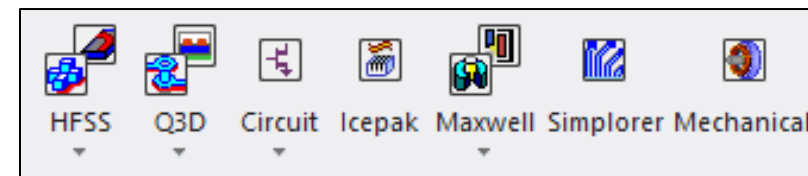


/ Introduction – Need for Electro-Thermal Analysis

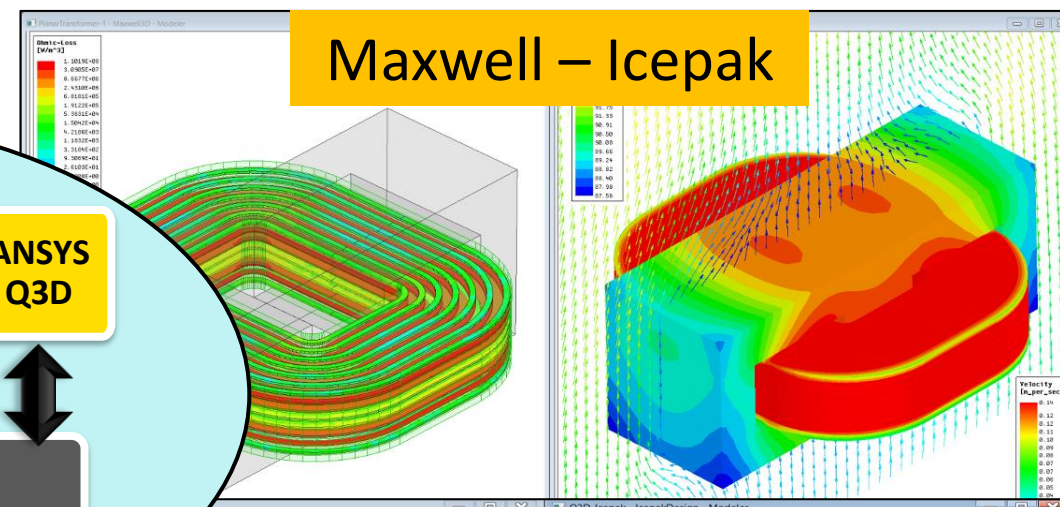


Introduction

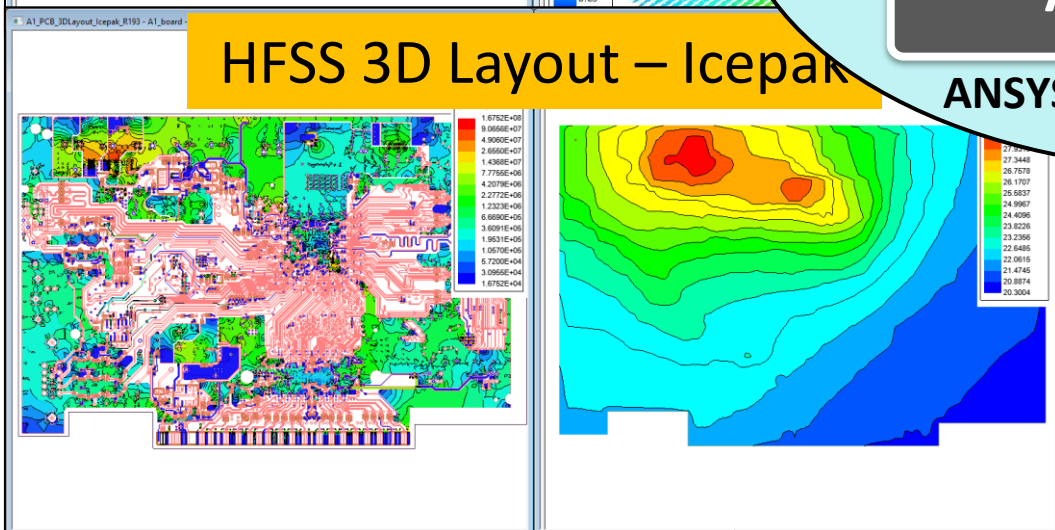
- Electro-Thermal workflows in ANSYS Electronics Desktop



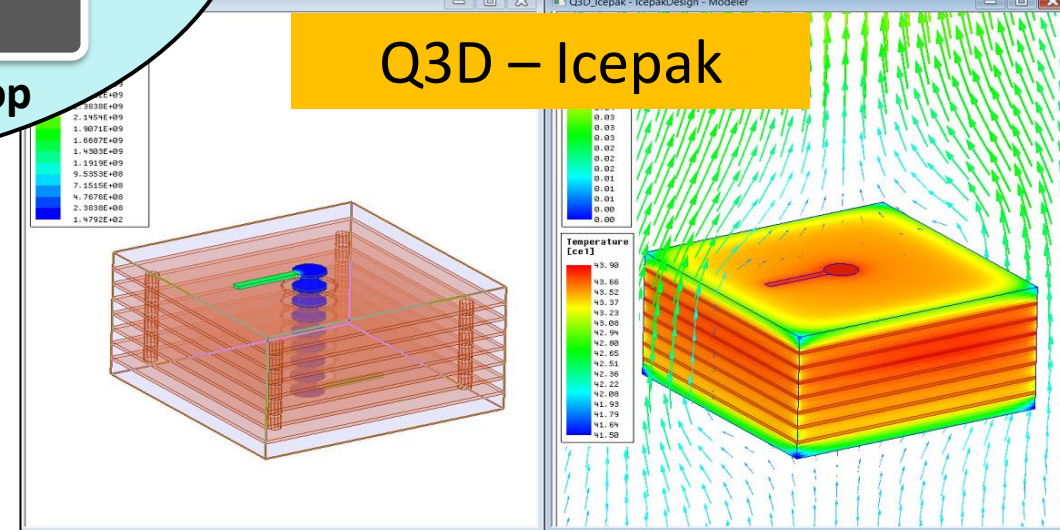
HFSS – Icepak



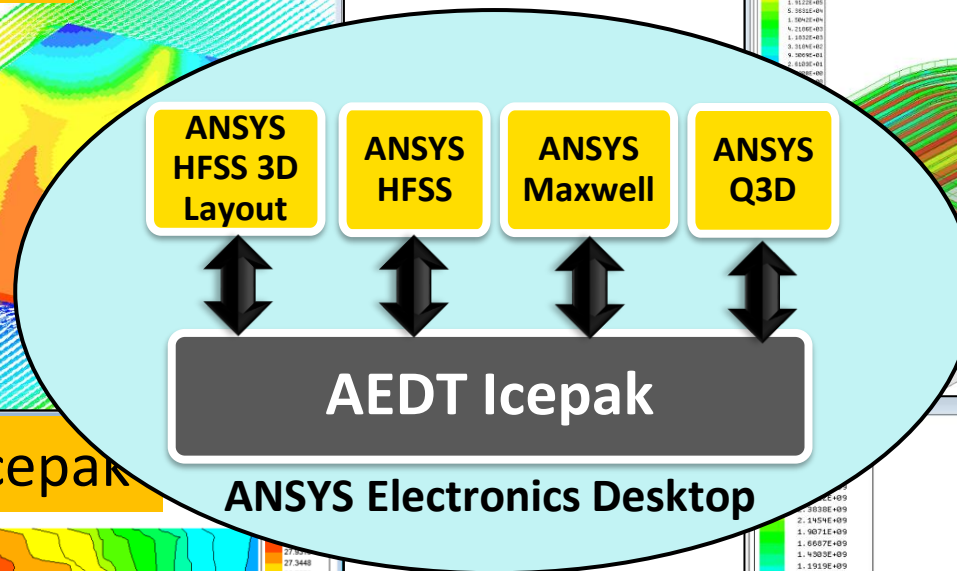
Maxwell – Icepak



HFSS 3D Layout – Icepak



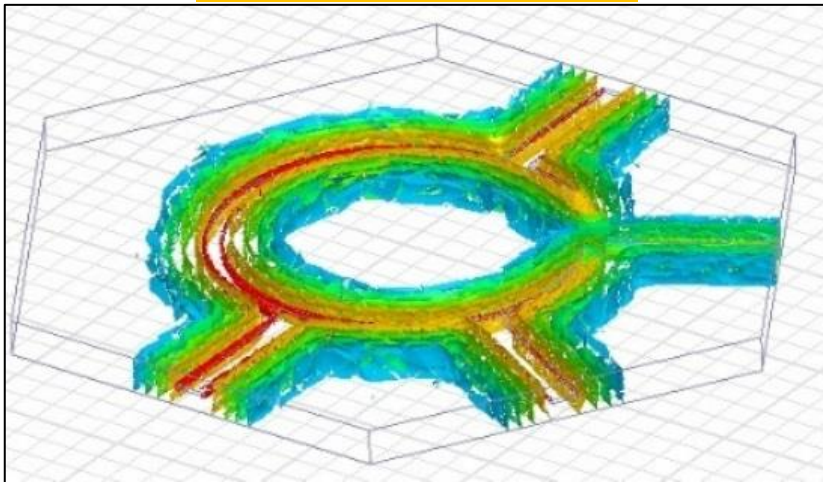
Q3D – Icepak



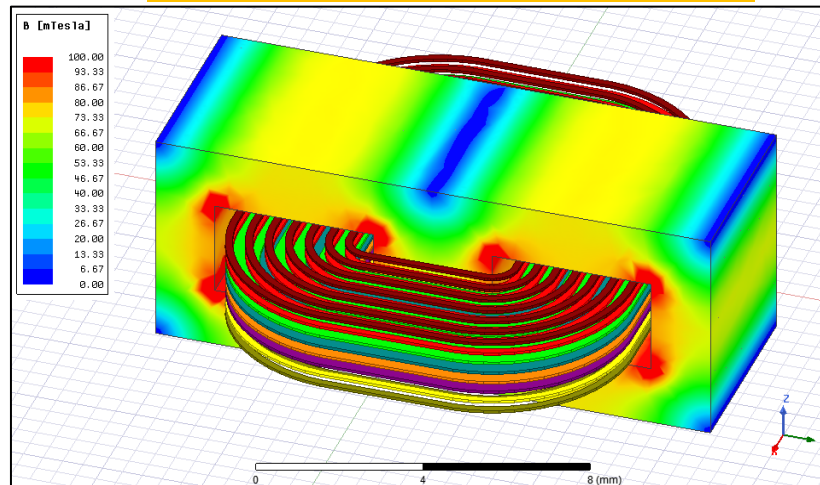
/ Electro-Thermal Workflow with HFSS, Maxwell and Q3D

- **Surface** and **Volume** losses from an EM design (HFSS/Maxwell/Q3D) can be imported into Icepak
- Temperature results from Icepak can be fed back to the EM design to account for temperature dependency of material properties
- EM loss from **multiple frequencies** (for HFSS and Maxwell designs) can be transferred to Icepak. The losses from multiple frequencies are applied as cumulative loss in the solver
- EM losses can be visualized prior to solving the model in Icepak

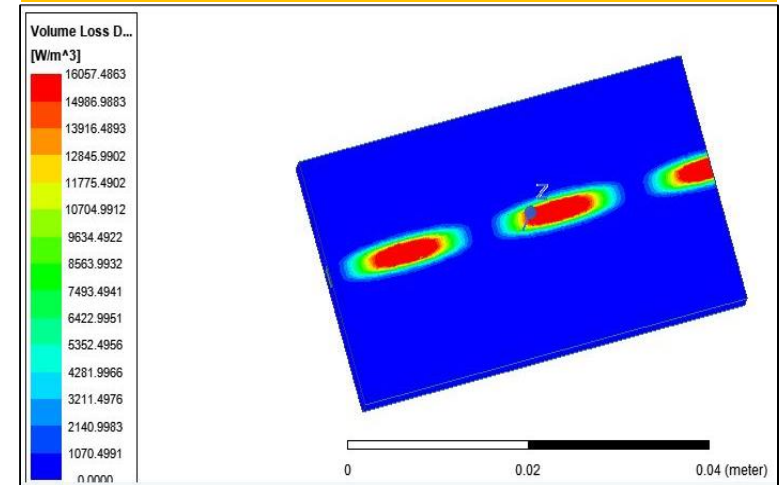
HFSS Model



Maxwell Model

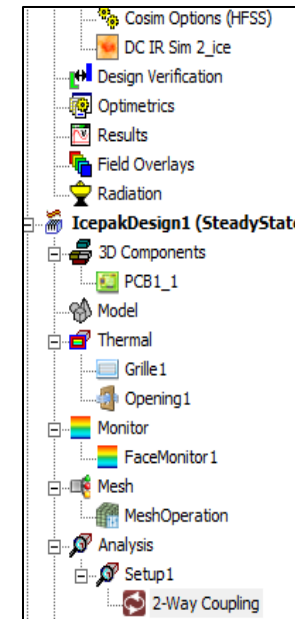


EM Loss Visualization



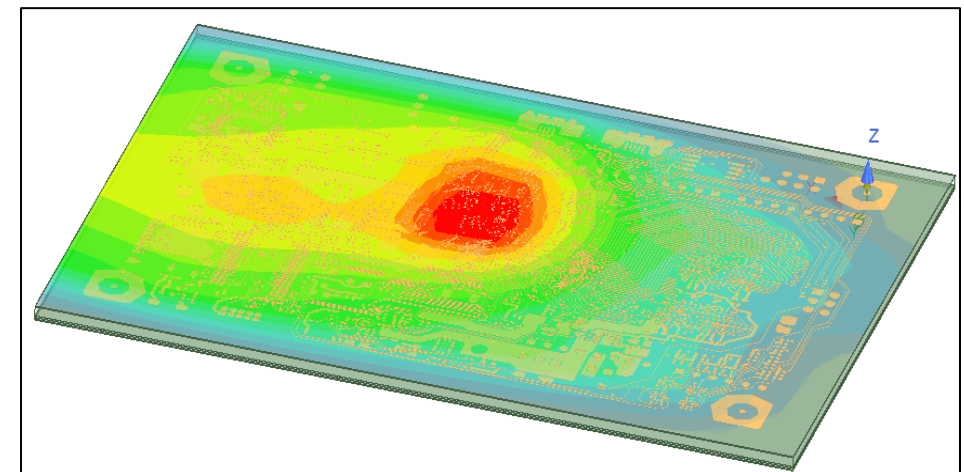
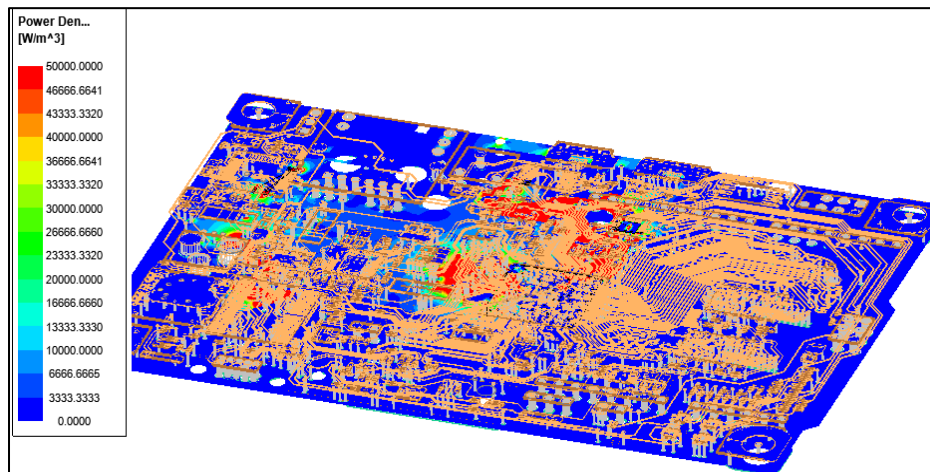
Electro-Thermal Workflow with HFSS 3D Layout

- **DC IR drop** analysis is performed in 3D Layout. The layer-by-layer Joule heating power maps are transferred to Icepak
- Temperature map from Icepak is transferred to 3D Layout to update the material properties (temperature dependent) spatially. DC IR analysis is run again with the updated material properties.
- Both **DC** and **AC** losses can be included for thermal analysis



Name	Value
Number of Coupling Iterations	2
Continue Icepak Iterations	<input checked="" type="checkbox"/>
Icepak Iterations Per Coupled Iter...	20

Task	Real Time	CPU Time	Memory	Information
Coupling Iteration 1				Starting at Icepak solver iteration 1
Get Thermal Loss : P...	00:00:40	00:00:00	0 K	Source: This Project: DCIR_test - DC IR Sim 2_ice - DC IR Sim 2_ice
Solver Initialization	00:00:12	00:00:11	662 M	Mapped PCB EM Loss: 0.016886 (watt)
				0.016886 (watt) on Surface (PCB1_1)
Solve	00:00:02	00:00:06	662 M	
Coupling Iteration 2				Starting at Icepak solver iteration 11
Get Thermal Loss : P...	00:00:37	00:00:00	0 K	Source: This Project: DCIR_test - DC IR Sim 2_ice - DC IR Sim 2_ice
Solver Initialization	00:00:12	00:00:11	672 M	Mapped PCB EM Loss: 0.016754 (watt)
				0.016754 (watt) on Surface (PCB1_1)
Solve	00:00:03	00:00:11	672 M	
Solution Process				Elapsed time : 00:02:58 , Icepak ComEngine Memory : 229 M
Total	00:02:19	00:01:07		Time : 02/26/2019 11:23:37, Status: Normal Completion



Electro-Thermal Workflow in AEDT

Electro-Thermal Workflow

Electro-Thermal ACT

- Ideal for **HFSS/HFSS 3D Layout/Maxwell/Q3D users** and **Mechanical engineers** to analyze critical components independently
- Icepak model includes only the components analyzed in the EM design
- Generic Natural Convection and Forced Convection scenarios are setup by the ACT automatically

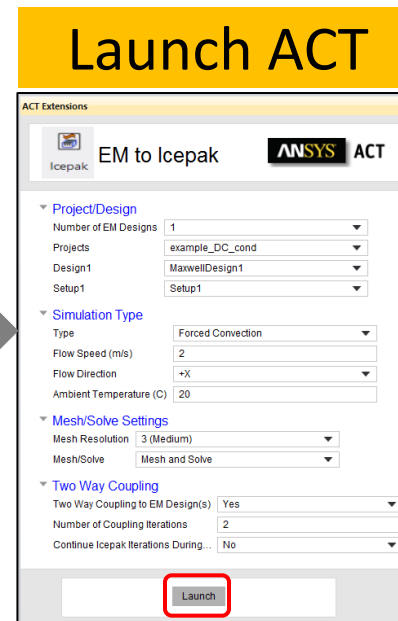
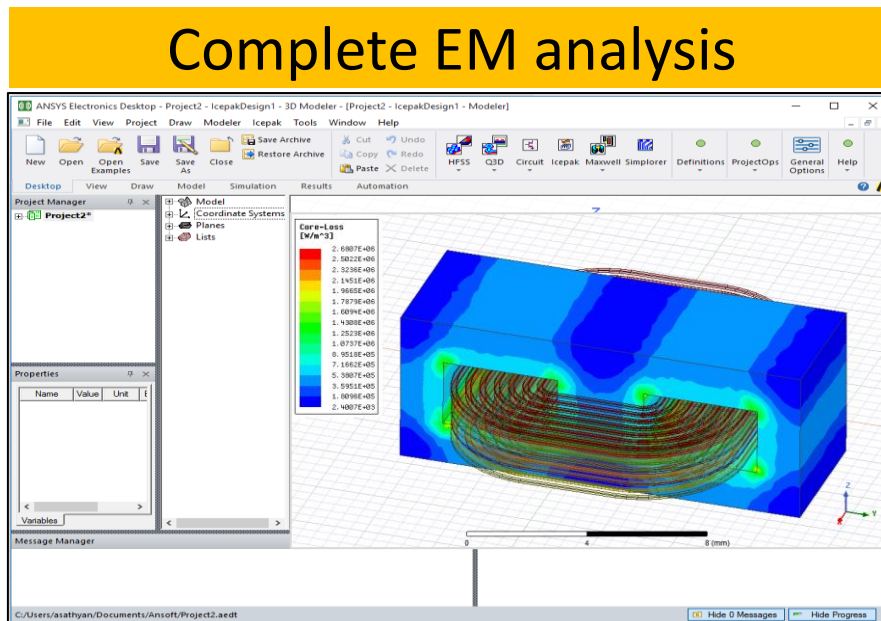
Manual Process

- Ideal for **Mechanical engineers** analyzing detailed system level models
- Icepak model can include several other components (chassis, fans, heatsinks etc.) in addition to the objects analyzed in the EM design
- Any custom scenario can be analyzed

Electro-Thermal ACT Workflow



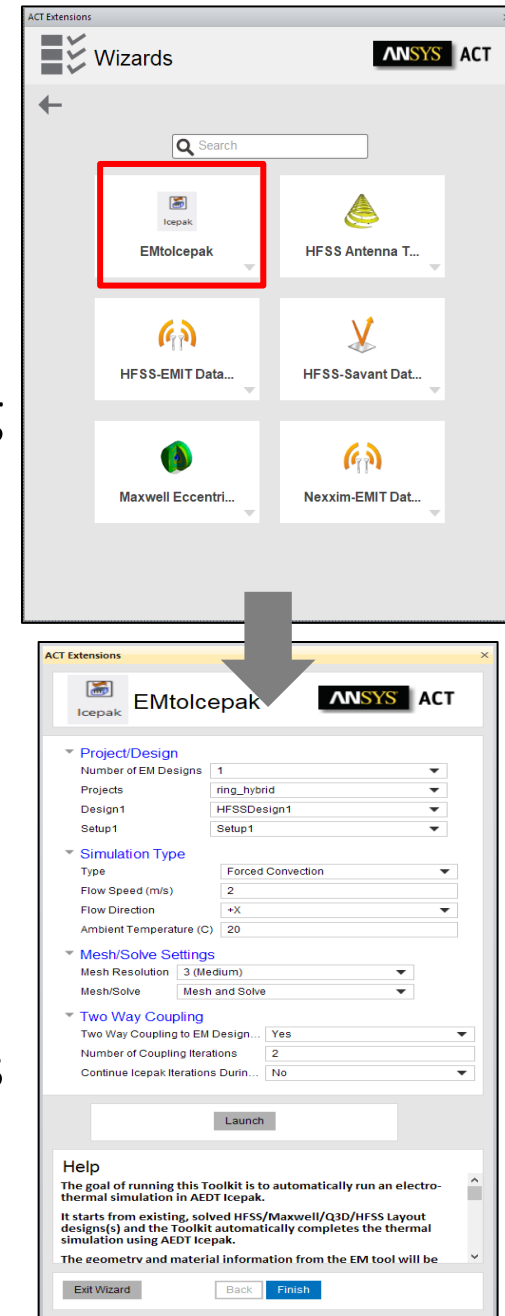
- 1 Complete the EM analysis in HFSS/HFSS 3D Layout/Maxwell/Q3D
- 2 Launch **Electro-Thermal ACT**
- 3 Specify Simulation Type, Mesh/Solve Settings and Two-way Coupling Criteria
- 4 Click **Launch** to run the ACT



Electro-Thermal ACT – Key Features

ACT Objective: Automatically run an electrothermal simulation using AEDT Icepak

- It starts from an existing **HFSS/HFSS 3D Layout/Maxwell/Q3D** design and the toolkit automatically completes the thermal simulation using AEDT Icepak
- **Geometry** and **material information** will be copied over from the EM tool(s) to Icepak
- **EM losses** will be mapped automatically from EM design to Icepak
- **Boundary conditions** for the thermal setup are automatically created depending on the choice of Natural or Forced convection
- **Mesh** and **Solver settings** are set for the Icepak design if those options are enabled
- **Two-way coupling criteria** can be defined for the automatic workflow



Scenario 1: Forced Convection (Active Cooling)

Inputs:

- Flow speed (m/s)
- Flow direction
- Ambient temperature (C)

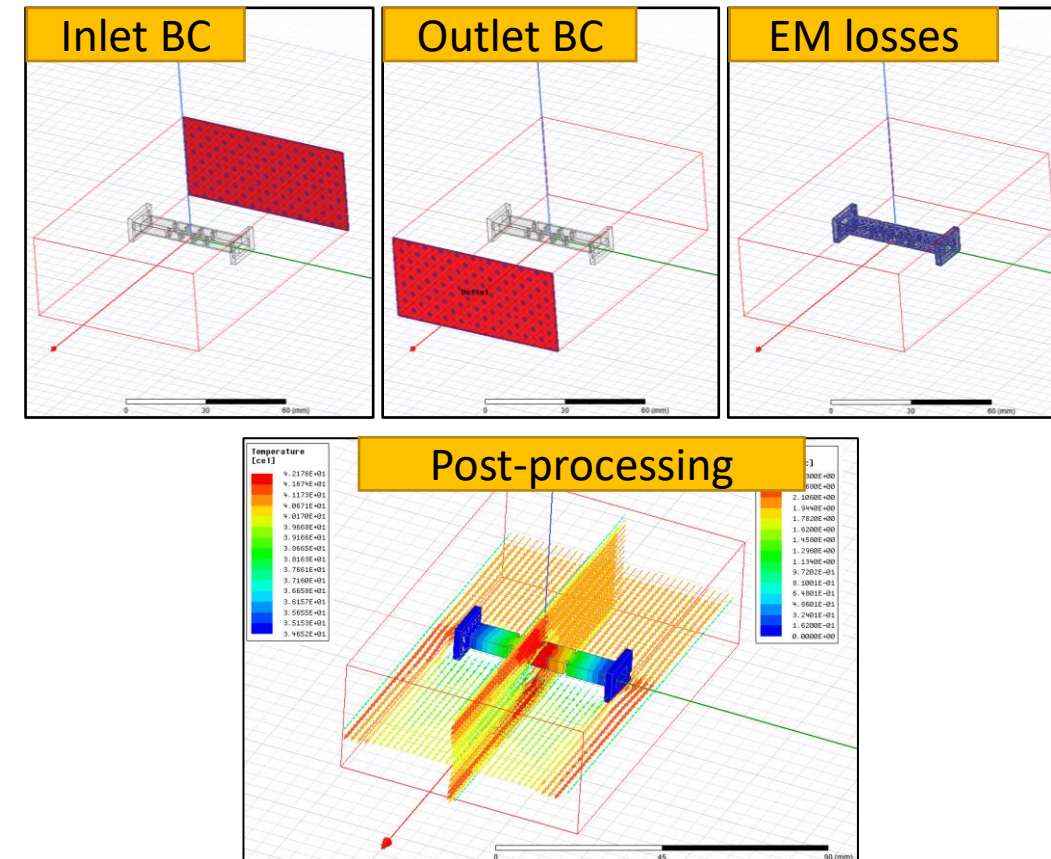
Simulation Type	
Type	Forced Convection
Flow Speed (m/s)	2
Flow Direction	+X
Ambient Temperature (C)	20

Computational Domain:

- Padding is based on the largest length in the geometry
- 100% padding added upstream and downstream of the model in the flow direction
- 25% padding added perpendicular to the flow direction

Solver Settings:

- Sequential solver is used, and Radiation is turned **OFF**



Scenario 2: Natural Convection (Passive Cooling)

Inputs:

- Gravity direction
- Ambient temperature (C)

Computational Domain:

- Padding is based on the largest length in the geometry
- 200% padding added in the direction opposite to gravity
- 100% padding added in the direction of gravity
- 50% padding added perpendicular to the direction of gravity

Solver Settings:

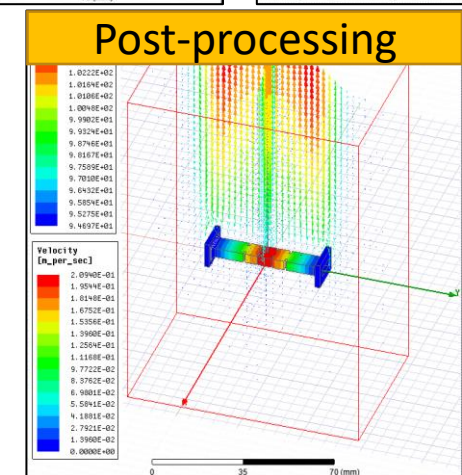
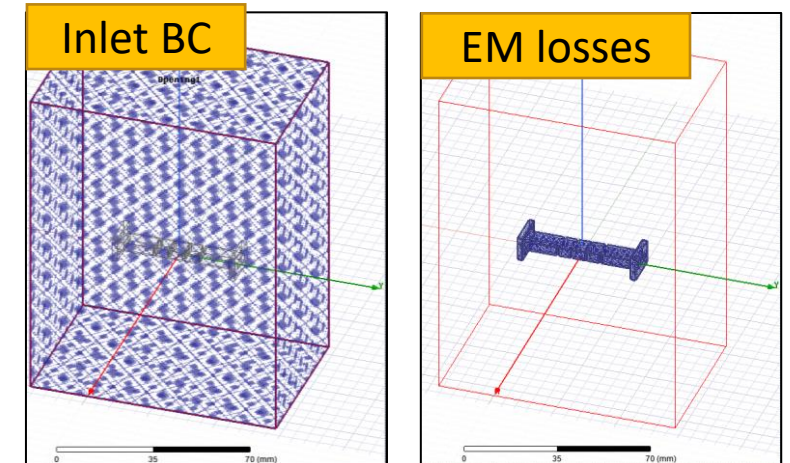
- Radiation is turned **ON**

Simulation Type

Type Natural Convection

Gravity -Z

Ambient Temperature (C) 20



Electro-Thermal ACT – GUI and Options

Project/Design:

- Launching the ACT will populate list of projects, designs and setups for selection
- Up to 5 designs can be selected for coupling to Icepak

Simulation Type:

- Forced or Natural Convection options

Mesh/Solve Settings:

- **Mesh and Solve:** Geometry is copied to a new Icepak design, EM losses are imported, Icepak model is setup based on user inputs, meshed and solved. Basic postprocessing is also displayed
- **Mesh Only:** Automation ends after the mesh is generated. Solving and postprocessing have to be completed manually
- **No:** Automation ends after the thermal analysis has been set in Icepak. Meshing, solving and postprocessing will be completed manually. Useful when advanced meshing is needed.

ACT Extensions

Icepak EMtolcepak ANSYS ACT

Project/Design

Number of EM Designs: 1

Projects: ring_hybrid

Design1: HFSSDesign1

Setup1: Setup1

Simulation Type

Type: Forced Convection

Flow Speed (m/s): 2

Flow Direction: +X

Ambient Temperature (C): 20

Mesh/Solve Settings

Mesh Resolution: 3 (Medium)

Mesh/Solve: Mesh and Solve

Two Way Coupling

Two Way Coupling to EM Design...: Yes

Number of Coupling Iterations: 2

Continue Icepak Iterations Durin...: No

Launch

Help

The goal of running this Toolkit is to automatically run an electro-thermal simulation in AEDT Icepak.

It starts from existing, solved HFSS/Maxwell/Q3D/HFSS Layout designs(s) and the Toolkit automatically completes the thermal simulation using AEDT Icepak.

The geometry and material information from the EM tool will be

Exit Wizard Back Finish

Electro-Thermal ACT – GUI and Options

Two-way Coupling:

- **Two way Coupling to EM Design:** Selecting **Yes** enables bidirectional coupling of EM design and Icepak
- **Number of Coupling Iterations:** Defines the number of times EM design and Icepak simulations must be launched and updated with new results
- **Continue Icepak Iterations During Coupling:** If **Yes** is selected, Icepak will use the previous solved solution as the initial condition for each subsequent loop. This results in faster convergence.

Launch:

- Execute the Electro-Thermal simulation

Finish:

- Quit the ACT GUI and return to the Extensions Wizard

ACT Extensions

Icepak EMtolcepak ANSYS ACT

Project/Design

Number of EM Designs: 1

Projects: ring_hybrid

Design1: HFSSDesign1

Setup1: Setup1

Simulation Type

Type: Forced Convection

Flow Speed (m/s): 2

Flow Direction: +X

Ambient Temperature (C): 20

Mesh/Solve Settings

Mesh Resolution: 3 (Medium)

Mesh/Solve: Mesh and Solve

Two Way Coupling

Two Way Coupling to EM Design...: Yes

Number of Coupling Iterations: 2

Continue Icepak Iterations Durin...: No

Launch

Help

The goal of running this Toolkit is to automatically run an electro-thermal simulation in AEDT Icepak.

It starts from existing, solved HFSS/Maxwell/Q3D/HFSS Layout designs(s) and the Toolkit automatically completes the thermal simulation using AEDT Icepak.

The geometry and material information from the EM tool will be

Exit Wizard Back Finish

/ Electro-Thermal ACT – Important Points

- Toolkit does not support **3D components** from the EM design
- **Material assignments** from the EM design **must** include the thermal properties
- Toolkit uses Icepak **slider bar meshing**. For complex geometries, **No** option should be used for **Mesh/Solve** to manually mesh the model
- For 2-way coupling, make sure the EM design is setup appropriately (**temperature dependent material properties** are set, and **Enable Feedback** is checked in EM design)
- For **HFSS 3D Layout**, two-way coupling is available. The toolkit will automatically create mesh regions and assign appropriate mesh settings

The screenshot shows the 'ACT Extensions' dialog box for 'EMtolcepak'. The dialog is organized into several sections:

- Project/Design:** Includes dropdowns for 'Number of EM Designs' (set to 1), 'Projects' (set to 'ring_hybrid'), 'Design1' (set to 'HFSSDesign1'), and 'Setup1' (set to 'Setup1').
- Simulation Type:** Includes a 'Type' dropdown (set to 'Forced Convection'), 'Flow Speed (m/s)' (set to 2), 'Flow Direction' (set to '+X'), and 'Ambient Temperature (C)' (set to 20).
- Mesh/Solve Settings:** Includes a 'Mesh Resolution' dropdown (set to '3 (Medium)') and a 'Mesh/Solve' dropdown (set to 'No'). The 'Mesh/Solve' dropdown is highlighted with a red rectangle.
- Two Way Coupling:** Includes a 'Two Way Coupling to EM Design...' dropdown (set to 'Yes'), 'Number of Coupling Iterations' (set to 2), and 'Continue Icepak Iterations Durin...' (set to 'No').

At the bottom of the dialog, there is a 'Launch' button. Below the main configuration area is a 'Help' section with text explaining the goal of the toolkit and the workflow. At the very bottom, there are 'Exit Wizard', 'Back', and 'Finish' buttons.

/ Electro-Thermal ACT



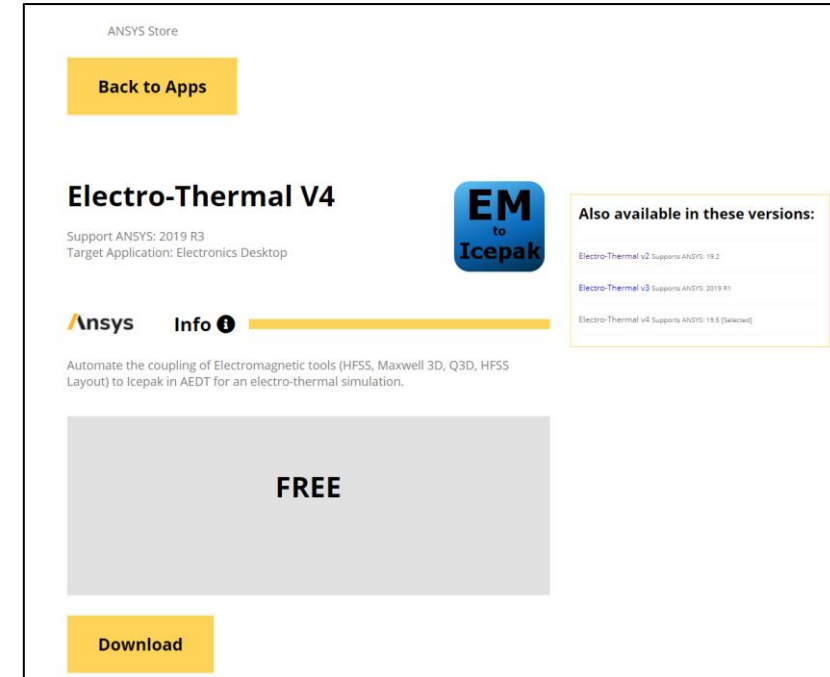
- The Electro-Thermal ACT is available for download from the ANSYS App Store

<https://catalog.ansys.com>

- **ANSYS App Store is a great place to get started**

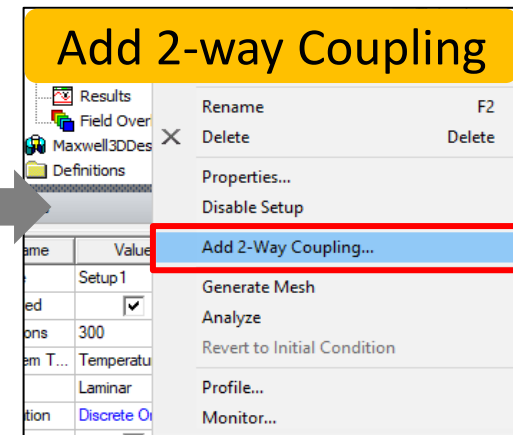
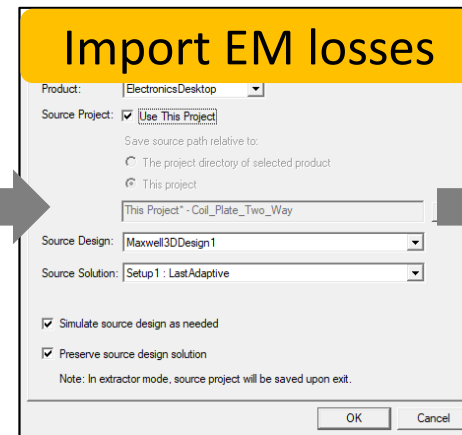
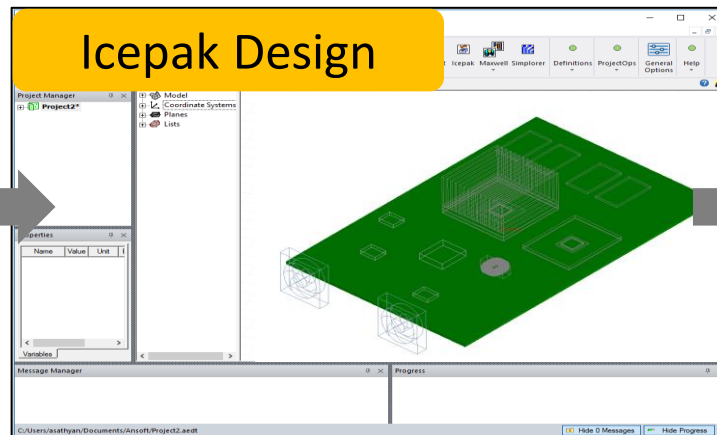
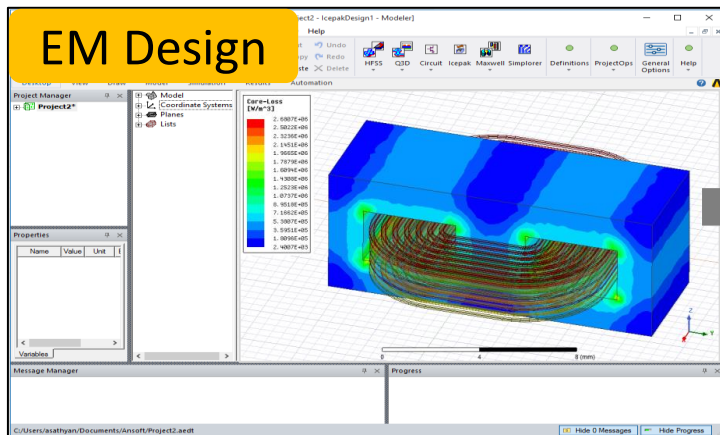
- A library of helpful applications available to any ANSYS customer
- New apps are added regularly
- Applications made available in either binary format (.wbex file) or scripted format (Python and XML files)
- Scripted extensions are great examples
- Documentation and training materials available on the ANSYS Customer **Portal**:

https://support.ansys.com/AnsysCustomerPortal/en_us/Downloads/ACT+Resources



Manual 2-way Electro-Thermal Workflow

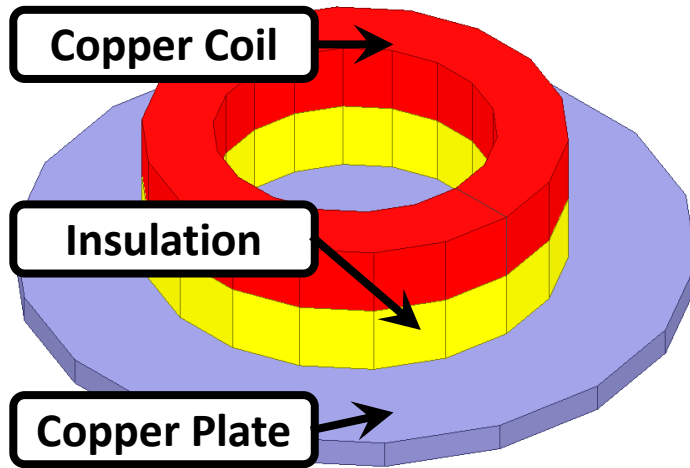
- 1 Complete the EM analysis in HFSS/HFSS 3D Layout/Maxwell/Q3D
- 2 Insert an Icepak design and copy-paste the geometry from EM design. **Add additional components**
- 3 Specify the inputs, boundary conditions, mesh and solver settings in Icepak
- 4 Select EM design objects and import EM losses using **Assign Thermal > EM losses**
- 5 Add a solution setup and select **Add 2-way Coupling**. Right-click and select **Analyze**



Appendix: Electro-Thermal Workflow Example

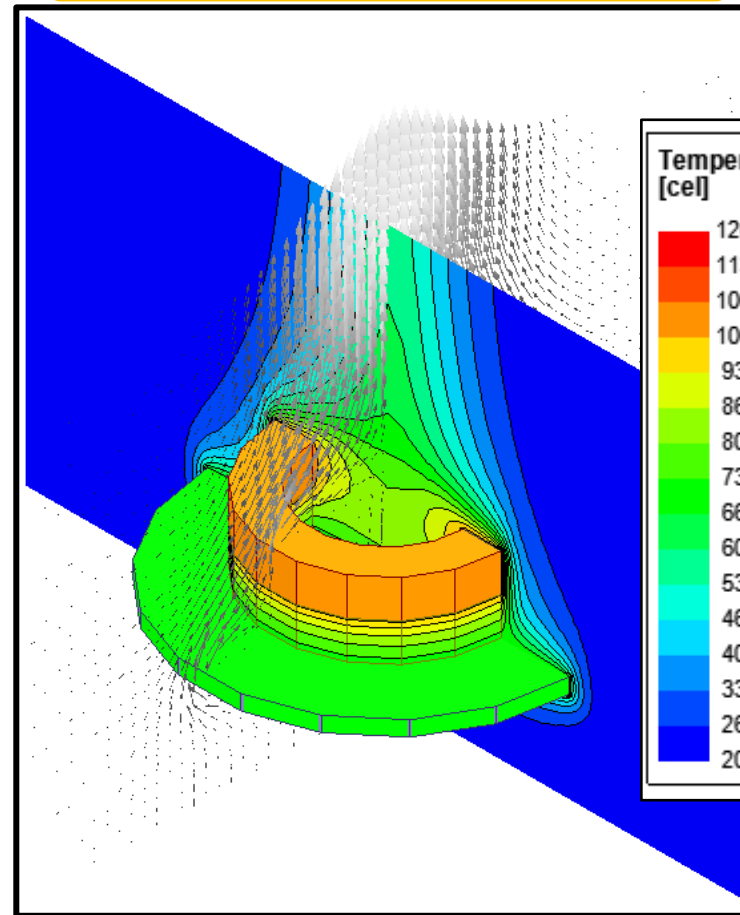


Example



- Induction heating application at 50 kHz
- Stranded coil above copper plate has induced eddy current losses
- Insulation material between coil and plate.

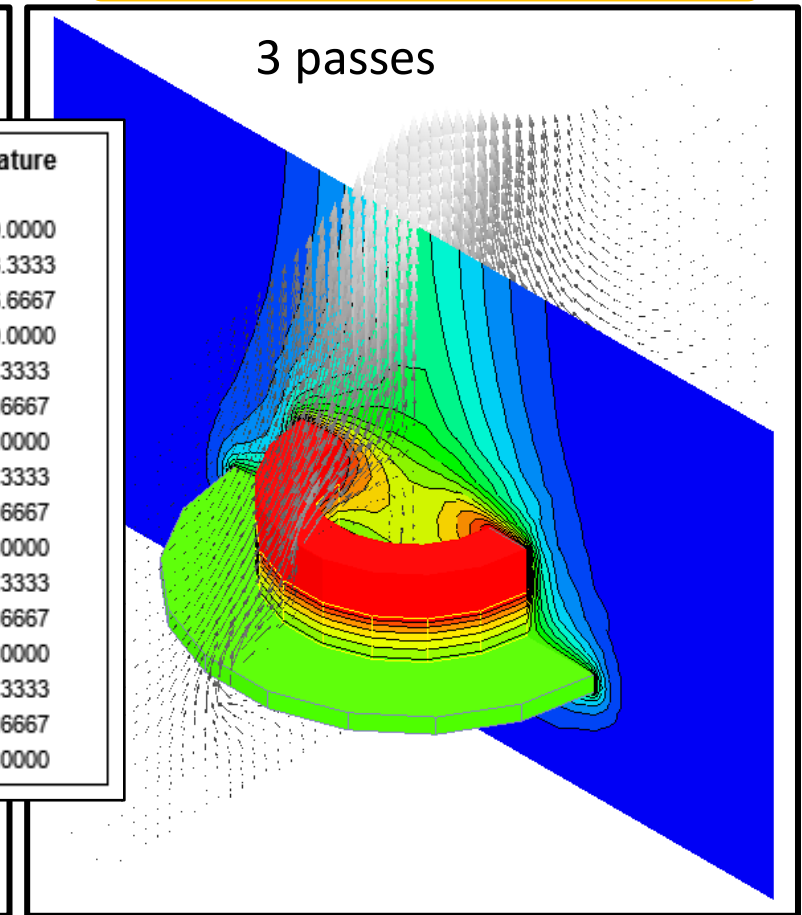
First Coupling Iteration



$$T_{max} = 103.2 \text{ }^{\circ}\text{C}$$

$$Q_{total} = 3.89 \text{ Watts}$$

Final Coupling Iteration

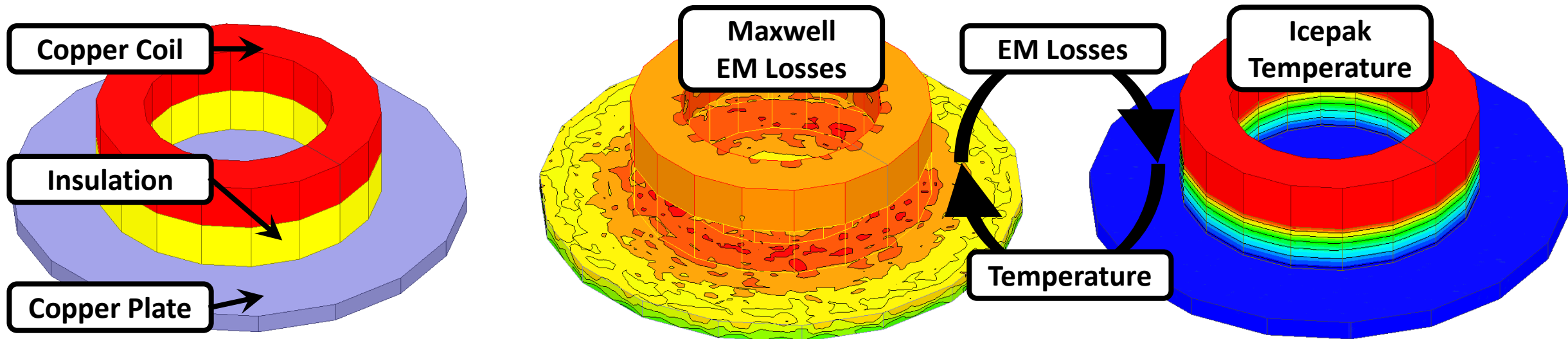


$$T_{max} = 122.0 \text{ }^{\circ}\text{C}$$

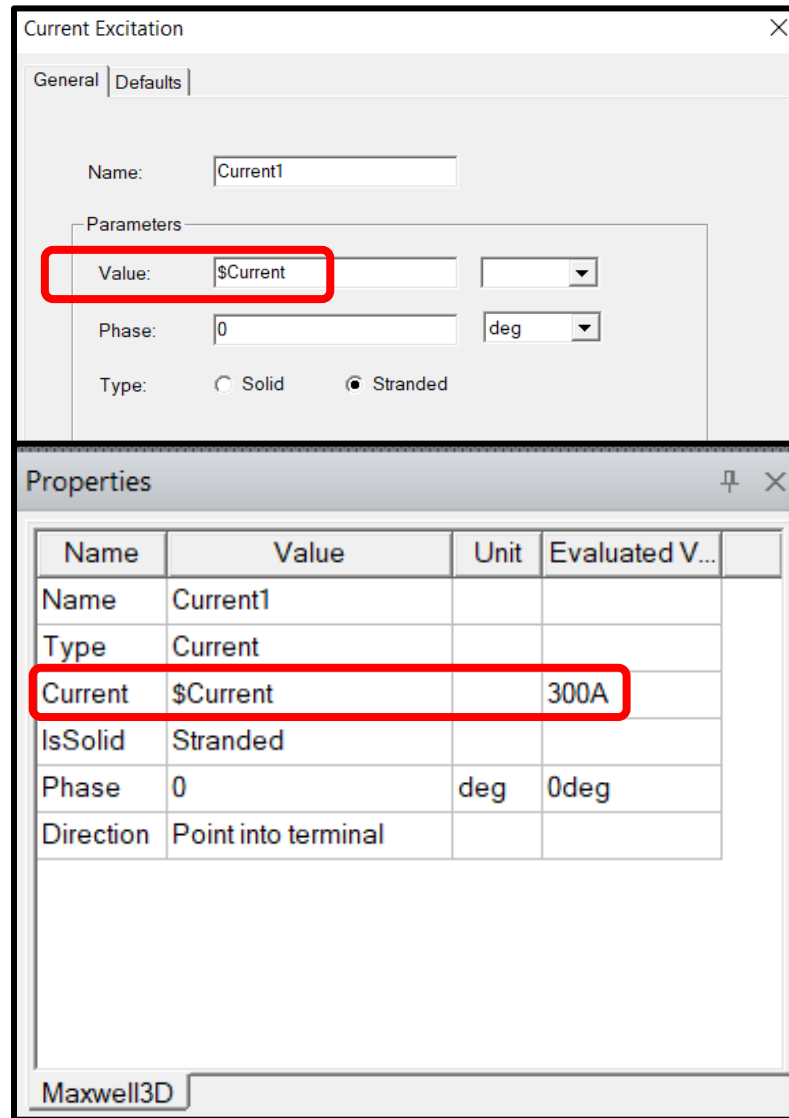
$$Q_{total} = 5.0 \text{ Watts}$$

Sample Problem Definition

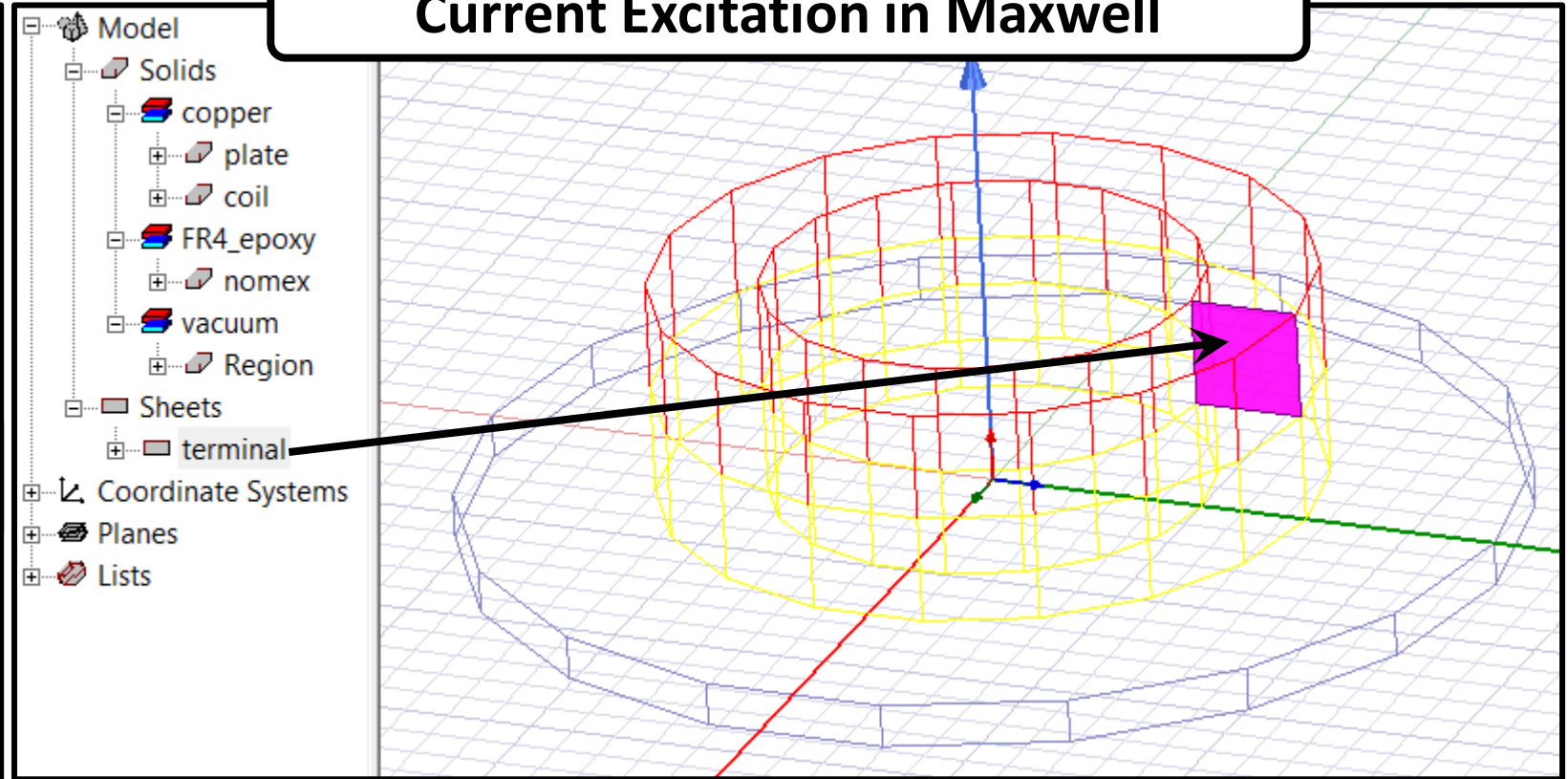
- Induction heating application at 50 kHz.
- Stranded coil above copper plate has induced eddy current losses.
- Insulation material between coil and plate.
- Loss density transfer from Maxwell to Icepak.
- Icepak solves temperature field in a natural convection environment.
- Temperature field transfer from Icepak to Maxwell.
- Use built-in two way coupling to loop Icepak and Maxwell solvers until losses and temperature fields converge.



Maxwell – Eddy Current Setup



Current Excitation in Maxwell



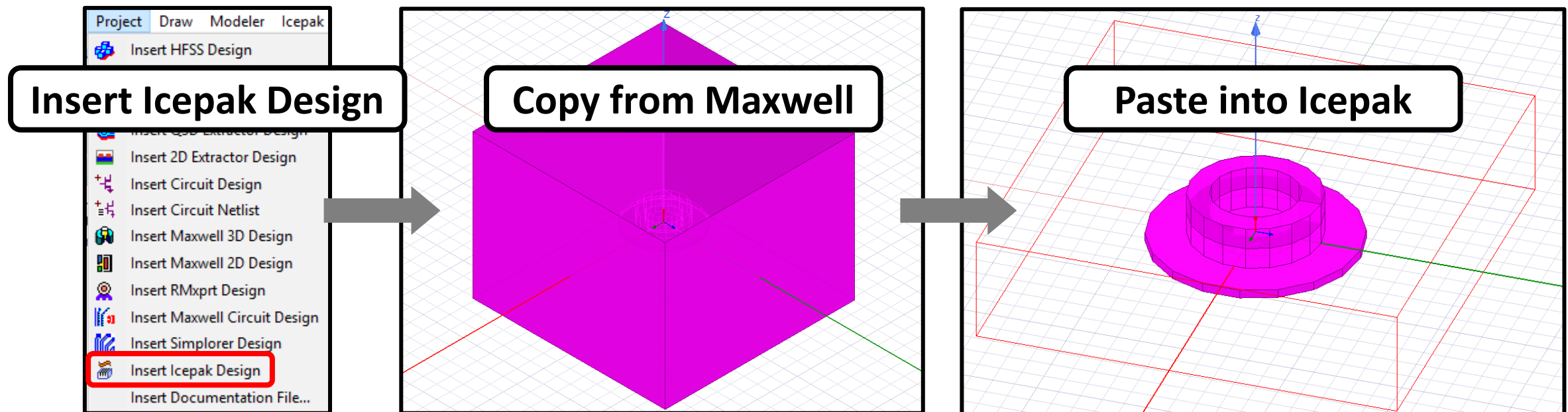
From Maxwell3D > Fields > Calculator

Sc1 : 3.89294171117504

Sc1 : Integrate(Volume(AllObjects), EM-Loss)

/ Icepak – Eddy Current Setup – Geometry Setup

- After completing the Maxwell setup, insert an Icepak Design into the AEDT project.
- In the Maxwell Graphics window, use Ctrl + A, Ctrl + C to copy the entire Maxwell Geometry and Material information.
- In the Icepak Graphics window, use Ctrl + V to paste the Geometry and Material information from Maxwell.
 - Note the automatic creation of the “air” region around the copied geometry.



/ Icepak – Eddy Current Setup – Solution Type and Design Settings

- Set up the Icepak Design for Natural Convection CFD analysis.

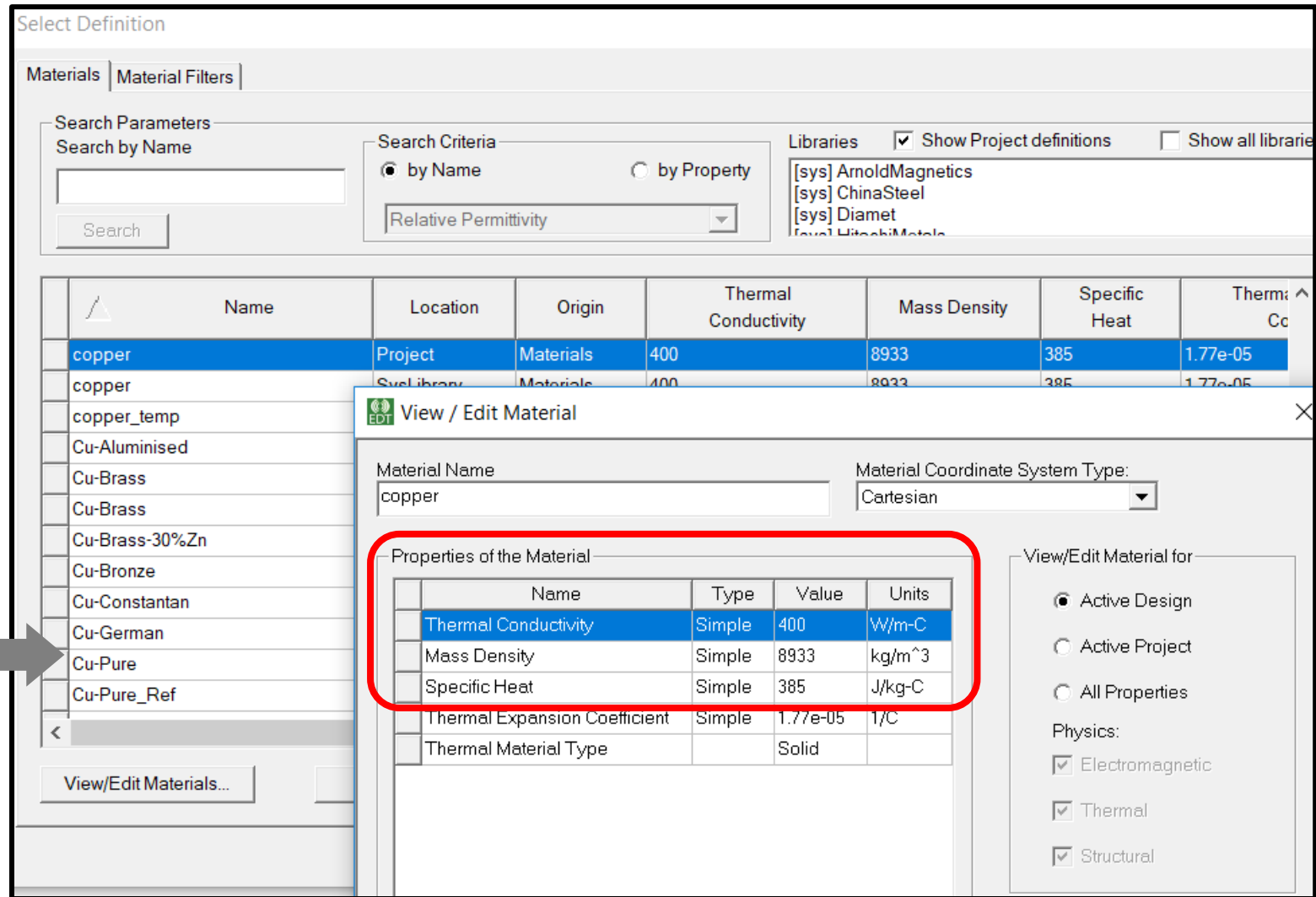
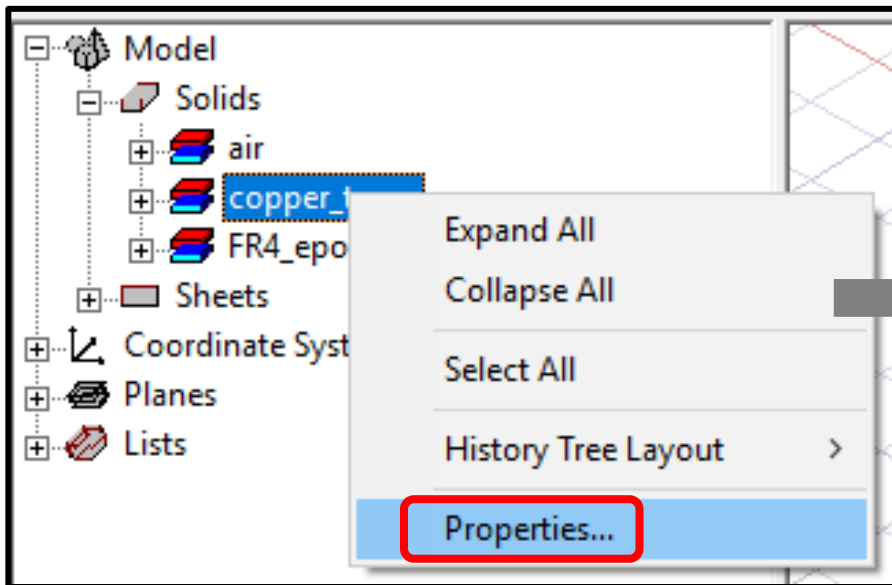
The image shows the Icepak software interface with three main panels highlighted by callouts:

- Solution Type:** A dialog box with three radio buttons: ☒ Temperature and Flow, ☐ Temperature Only, and ☐ Flow Only. A callout box points to this panel with the text: "For Natural Convection models we need to solve temperature and flow equations."
- Design Settings:** A panel with tabs for Ambient Conditions, Gravity, and Validations. The Gravity tab is selected. It contains three rows of settings: Temperature (20, cel), Gauge Pressure (0, n_per_meter_sq), and Radiation temp (20, cel). A callout box points to this panel.
- Gravity Vector:** A panel with tabs for Ambient Conditions, Gravity, and Validations. The Gravity tab is selected. It contains a checked checkbox for Gravity Vector, a dropdown menu set to Global::Z, and two radio buttons: ☐ Positive and ☒ Negative. A callout box points to this panel.

On the left, the Icepak menu is visible with "Solution Type..." and "Design Settings..." highlighted in red boxes. Arrows point from these menu items to their respective panels.

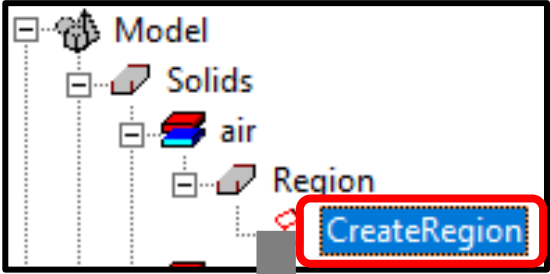
/ Icepak – Eddy Current Setup – Material Setup

- Review the Material setup
 - For steady state models, only thermal conductivity is used.
 - For transient models, thermal conductivity, density and specific heat are all used.



/ Icepak – Eddy Current Setup – Thermal Setup

- Expand the air region size for Natural Convection CFD Analysis.
- **Percentage offset:** computes the offset based on the min and max coordinate of that axis.
- **Transverse percentage offset:** will add a bit more padding with respect to percentage offset.
- **Absolute offset:** distance between closest object and edge of air region.
- **Absolute position:** location is based on the air region's coordinate system.



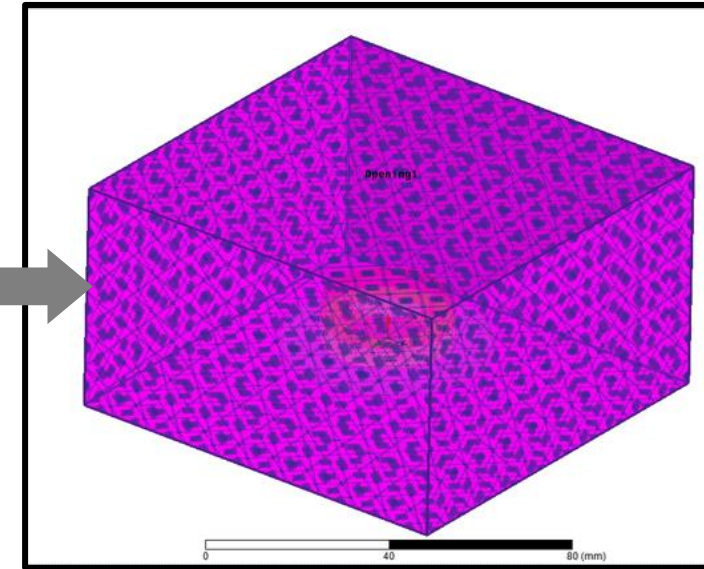
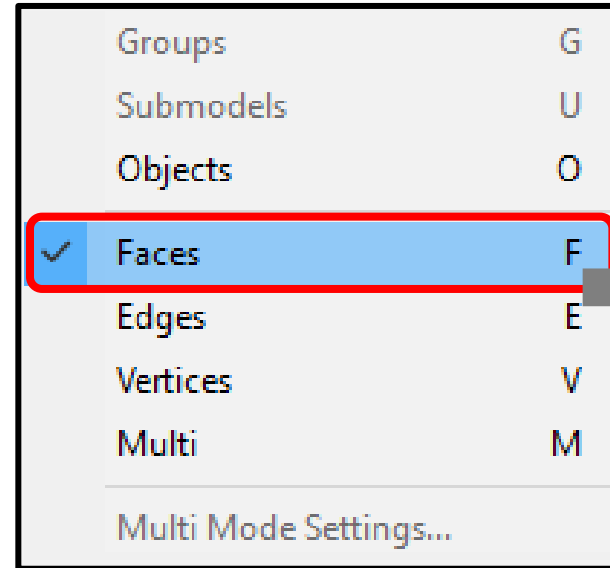
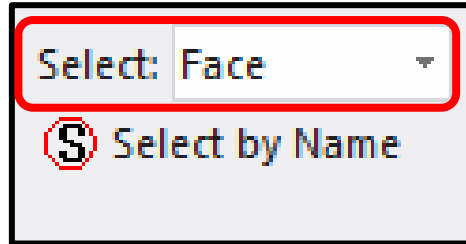
	Name	Value	Unit	Evaluated
Command	CreateRegion			
Coordinate ...	Global			
+X Padding...	Percentage Offset			
+X Padding...	50		50	
-X Padding ...	Percentage Offset			
-X Padding ...	50		50	
+Y Padding...	Percentage Offset			
+Y Padding...	50		50	
-Y Padding ...	Percentage Offset			
-Y Padding ...	50		50	
+Z Padding...	Percentage Offset			
+Z Padding...	250		250	
-Z Padding ...	Percentage Offset			
-Z Padding ...	100		100	

Gravity is in the minus Z direction, so the air plume will rise towards the plus Z side.

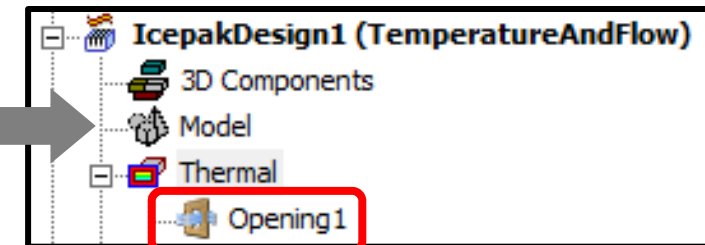
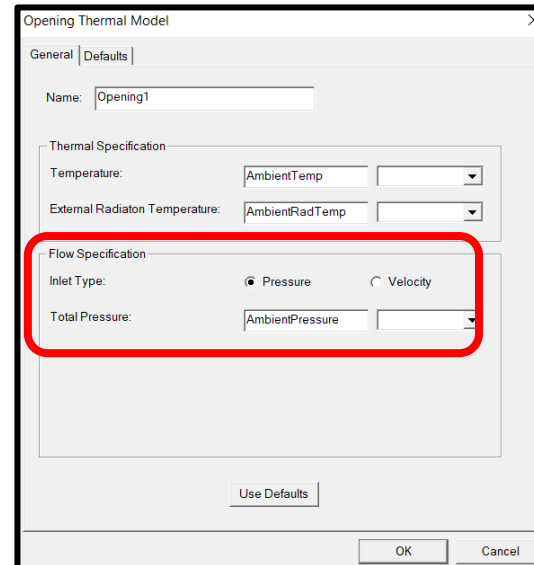
/ Icepak – Eddy Current Setup – Thermal Setup

- **Selection Modes:**

- Right-click in GUI > Selection Mode.
- From menu Edit > Selection Mode.
- Or from Ribbon Drop Down.
- Object (O)
- Face (F)
- Edge (E)
- Vertex (V)
- Multi (M)

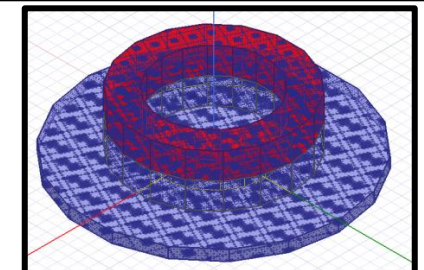
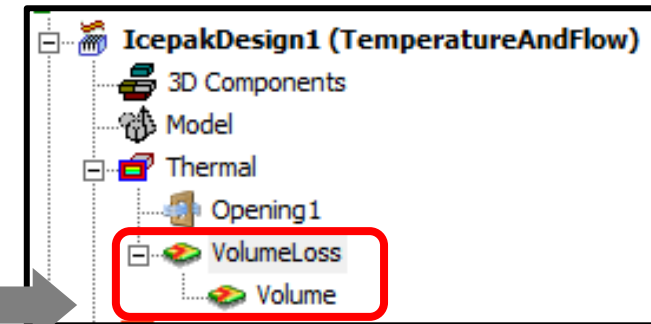
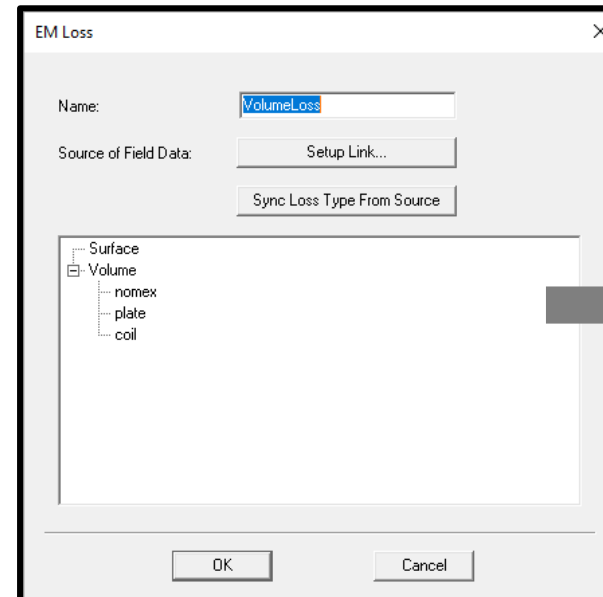
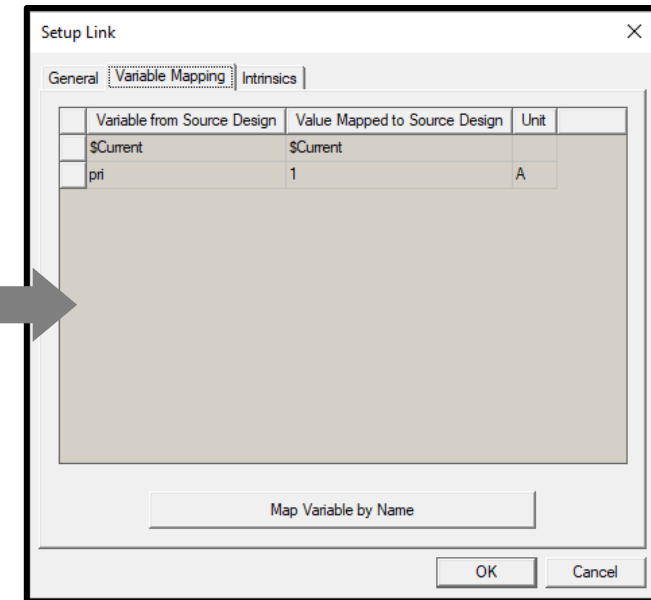
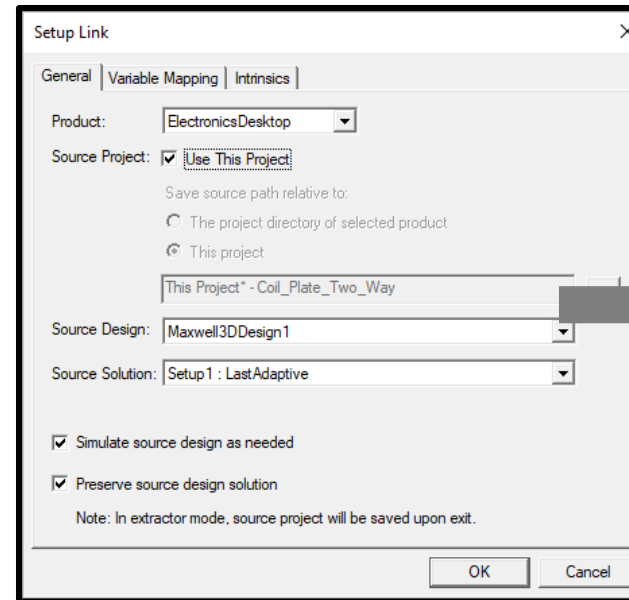


- **Enter Face Selection Mode.**
- **Select all air region faces.**
- **Right-click > Assign Thermal > Opening.**
- **Use default Pressure Inlet settings.**



/ Icepak – Eddy Current Setup – Thermal Setup

- Set up the Electro-Thermal link to map EM losses.
- Enter Object selection mode (O).
- Select the coil, plate and insulation objects.
- Right-click > Assign Thermal > EM Loss.
- “Use this Project” will automatically map losses from existing Maxwell design.
- “Map Variable by Name” will use the original variable from Maxwell.



/ Icepak – Eddy Current Setup – Solution Setup

- Add an Analysis Setup using laminar flow, radiation, and gravity as shown below.

Right-click...

The first screenshot shows a right-click context menu on the 'Analysis' node in the project tree. The 'Add Solution Setup...' option is highlighted. The second screenshot shows the 'Icepak Solve Setup Dialog' with the following settings: Name: Setup1, Maximum Number of Iterations: 300, Problem Types: Temperature and Flow, Flow Regime: Laminar, Radiation Model: Discrete Ordinates, and Include Gravity checked. The third screenshot shows the 'Icepak Solve Setup Dialog' with the following settings: Name: Setup1, Maximum Number of Iterations: 300, Problem Types: Temperature and Flow, Flow Regime: Laminar, Radiation Model: Discrete Ordinates, Include Gravity checked, Flow Iterations per Radiation Iteration: 5, and Angular Discretization: Theta Divisions: 2, Phi Divisions: 2, Theta Pixels: 2, and Phi Pixels: 2.

Icepak Solve Setup Dialog

General | Convergence | Solver Settings | Radiation | Defaults

Name: Setup1 ☒ Enabled

Maximum Number of Iterations: 300

Problem Types

☒ Temperature

☒ Flow

Flow Regime

☒ Laminar

☐ Turbulent Options

Radiation Model

☐ Off

☒ Discrete Ordinates

☐ Ray Tracing

☒ Include Gravity

☐ Solve Flow and Energy Equations Sequentially

Use Defaults

HPC and Analysis Options...

OK Cancel

Icepak Solve Setup Dialog

General | Convergence | Solver Settings | Radiation | Defaults

Iteration Parameters

Flow Iterations per Radiation Iteration: 5

Angular Discretization

Theta Divisions: 2

Phi Divisions: 2

Theta Pixels: 2

Phi Pixels: 2

Use Defaults

OK Cancel

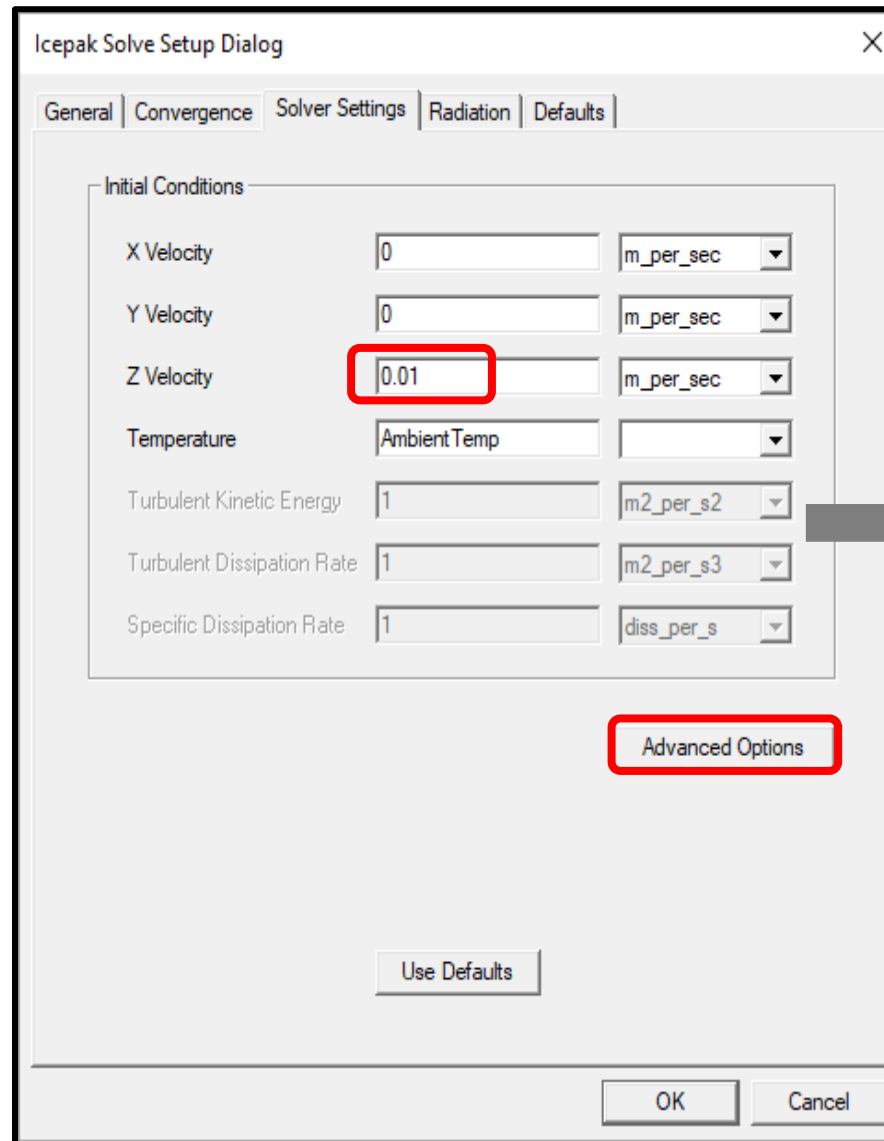
Properties

Name	Value	Unit	Evaluated Value
Name	Setup1		
Enabled	<input checked="" type="checkbox"/>		
Iterations	300		
Problem Type	TemperatureA...		
Flow	Laminar		
Radiation	Discrete Ordin...		
Include Gravity	<input checked="" type="checkbox"/>		

General | Convergence | Initialization | Radiation

/ Icepak – Eddy Current Setup – Solution Setup

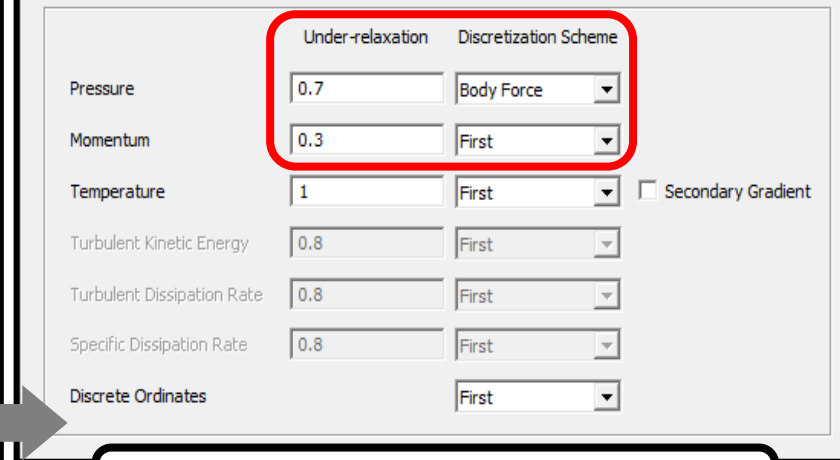
- Initialize the flow field with a small velocity opposite to the gravity vector.
- Best practices for under-relaxation factors discretization and convergence for natural convection models are shown.
- These settings can help improve convergence and accuracy.



The 'Icepak Solve Setup Dialog' is shown with the 'Initial Conditions' tab selected. The 'Z Velocity' field is highlighted with a red box and contains the value '0.01'. The 'Advanced Options' button at the bottom right is also highlighted with a red box. A grey arrow points from this button to the 'Advanced SolverOptions' dialog.

Initial Conditions	Value	Units
X Velocity	0	m_per_sec
Y Velocity	0	m_per_sec
Z Velocity	0.01	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

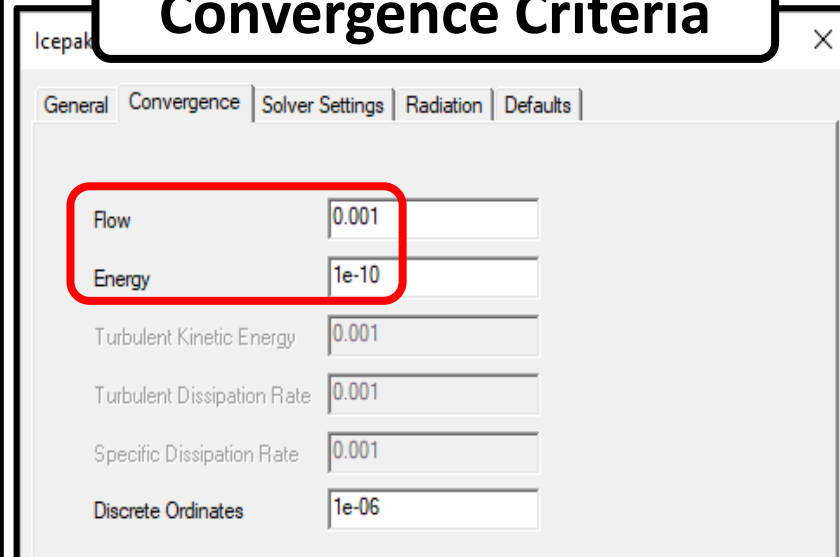
Advanced SolverOptions



The 'Advanced SolverOptions' dialog is shown with the 'Under-relaxation' and 'Discretization Scheme' sections highlighted by a red box. The 'Under-relaxation' section shows values of 0.7 for Pressure and 0.3 for Momentum. The 'Discretization Scheme' section shows 'Body Force' for Pressure and 'First' for Momentum.

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

Convergence Criteria



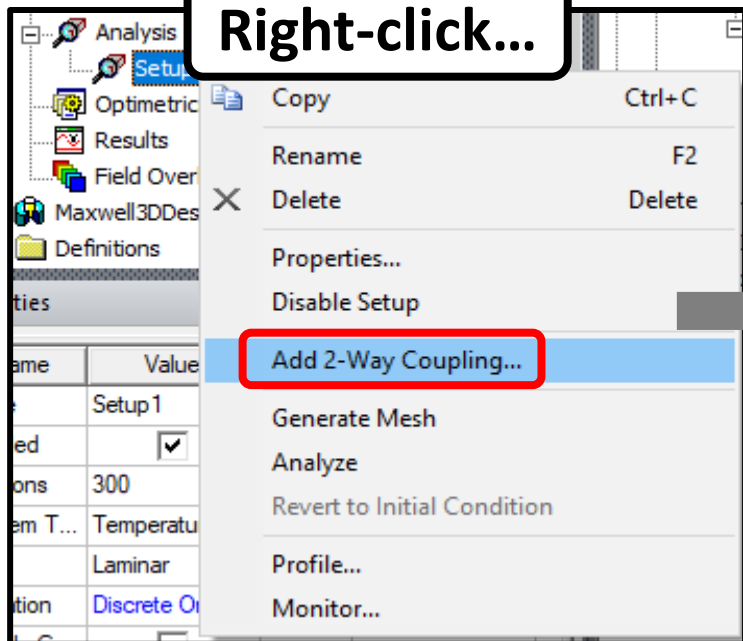
The 'Icepak Convergence Criteria' dialog is shown with the 'Flow' and 'Energy' fields highlighted by a red box. The 'Flow' field has a value of 0.001 and the 'Energy' field has a value of 1e-10.

Parameter	Value
Flow	0.001
Energy	1e-10
Turbulent Kinetic Energy	0.001
Turbulent Dissipation Rate	0.001
Specific Dissipation Rate	0.001
Discrete Ordinates	1e-06

/ Icepak – Eddy Current Setup – Solution Setup

- Two-way coupling will improve the accuracy of the EM losses.
- This in turn will improve the thermal accuracy.
- Typically two or three coupling iterations are sufficient to get good convergence.
- “Continue Icepak Iterations...” will use restart data files for shorter solve times.
- “Max Icepak Iterations...” is the iteration limit after the first solution.

Right-click...



The screenshot shows the 'Analysis' tree on the left with 'Setup1' selected. A right-click context menu is open, and the 'Add 2-Way Coupling...' option is highlighted with a red box. An arrow points from this option to the '2-Way Coupling' dialog box.

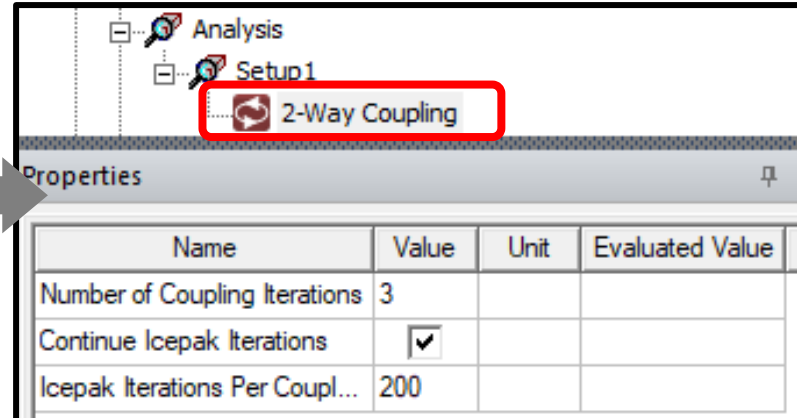
2-Way Coupling

Number of Coupling Iterations: 3

Continue Icepak Iterations During Coupling ☒

Max. Icepak Iterations per Coupling: 200

OK Cancel



The screenshot shows the 'Analysis' tree with 'Setup1' and a '2-Way Coupling' icon highlighted with a red box. An arrow points from the '2-Way Coupling' dialog box to this icon. Below the tree is a 'Properties' table.

Name	Value	Unit	Evaluated Value
Number of Coupling Iterations	3		
Continue Icepak Iterations	<input checked="" type="checkbox"/>		
Icepak Iterations Per Coupl...	200		

Maxwell – Eddy Current Setup – Solution Setup

- Two-way coupling needs to have temperature dependence in the EM solver.
- In the Maxwell design, Maxwell 3D > Set Object Temperature.
- “Include Temperature Dependence” and “Enable Feedback” must be checked ON.
- The temperature dependent materials must have the “Thermal Modifier” active.

Temperature of Objects

☒ Include Temperature Dependence ☒ Enable Feedback

Object Na...	Material	Temperature Dependent	Temperature	Unit
Region	vacuum	<input type="checkbox"/>		
coil	copper_temp	<input checked="" type="checkbox"/>	22	cel
nomex	FR4_epoxy	<input type="checkbox"/>		
plate	copper_temp	<input checked="" type="checkbox"/>	22	cel

Select By Name:

Temperature:

View / Edit Material

Material Name: copper_temp Material Coordinate System Type: Cartesian

Name	Type	Value	Units	Thermal Modifier
Relative Permittivity	Simple	1		None
Relative Permeability	Simple	0.999991		None
Bulk Conductivity	Simple	58000000	siemens/m	$1 / (1 + 0.0039 * (Temp - 22))$
Dielectric Loss Tangent	Simple	0		None
Magnetic Loss Tangent	Simple	0		None
Core Loss Model		None	w/m^3	
Mass Density	Simple	8933	kg/m^3	None
Composition		Solid		

View/Edit Material for:

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

Physics:

- ☒ Electromagnetic
- ☒ Thermal
- ☒ Structural

View/Edit Modifier for:

- ☒ Thermal Modifier

Material Appearance:

- ☐ Use Material Appearance

Color:

Transparency:

Notes:

Calculate Properties for:

Edit Thermal Modifier

☒ Expression ☐ Quadratic

Expression:

Temperature-Dependent Bulk Conductivity:
 $P(Temp) = Pref [Modifier]$

Reference Bulk Conductivity:
Pref = 58000000siemens/m

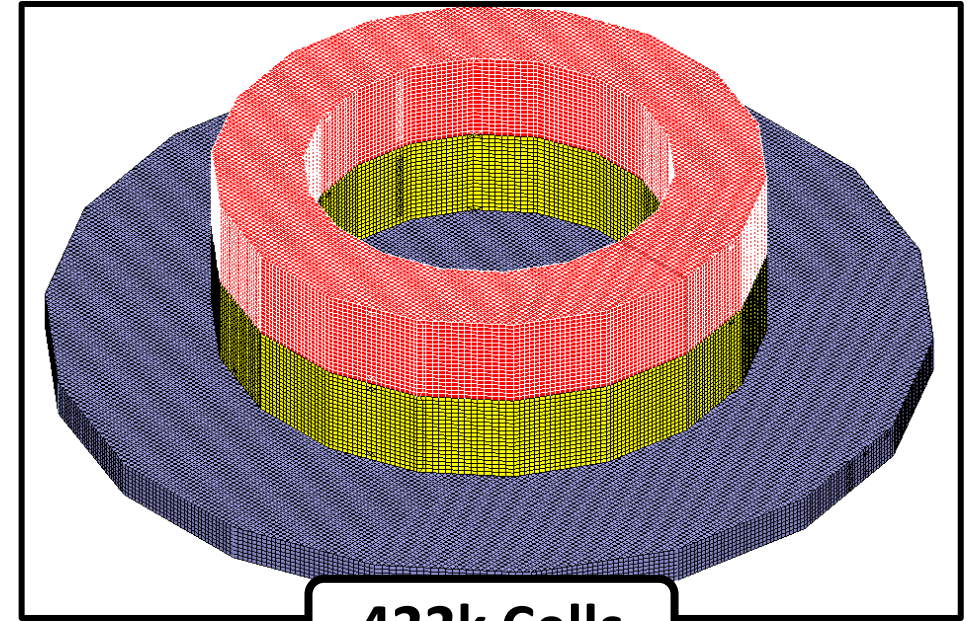
Parameters:

Modifier: $1 / (1 + 0.0039 * (Temp - 22))$

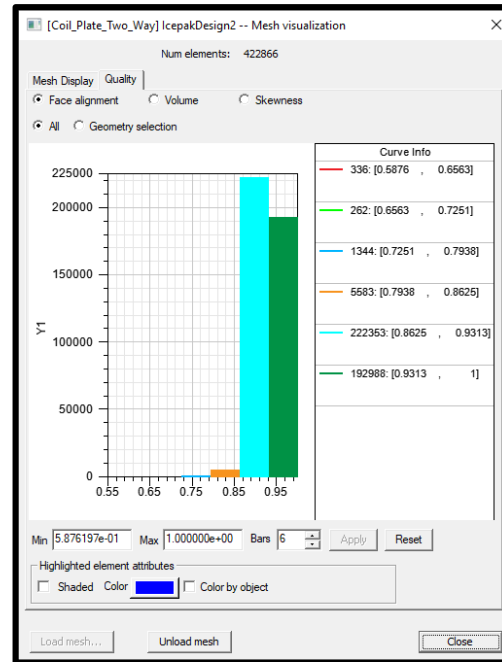
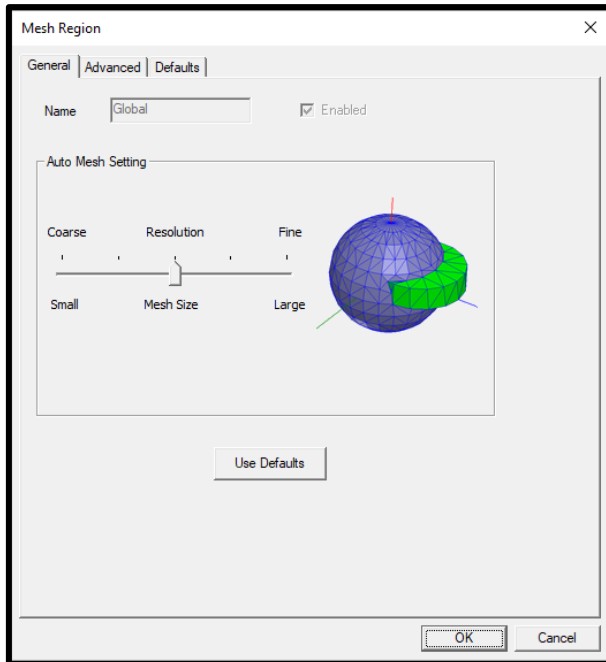
☐ Use temperature dependent dataset

/ Icepak – Eddy Current Setup – Mesh Setup

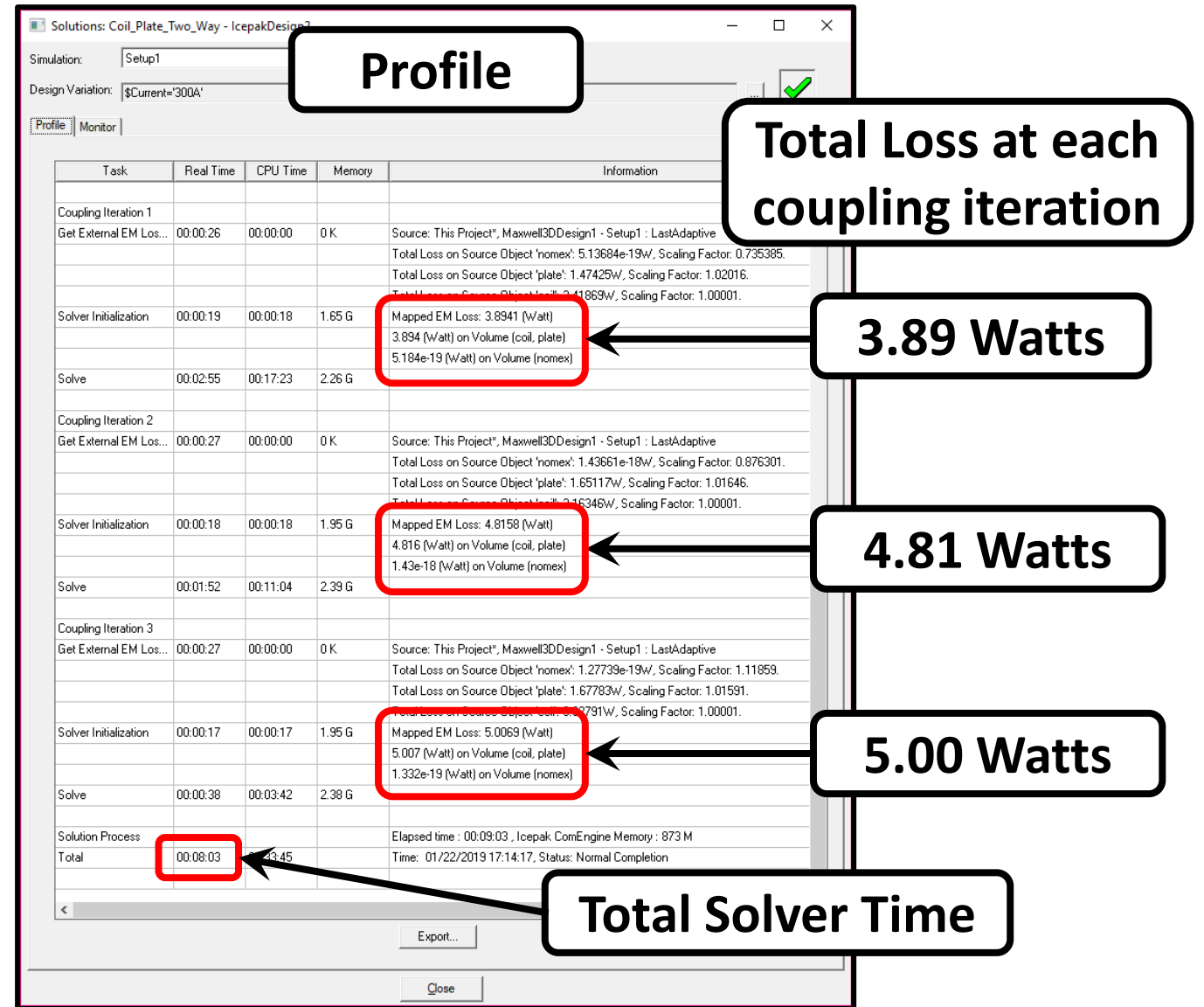
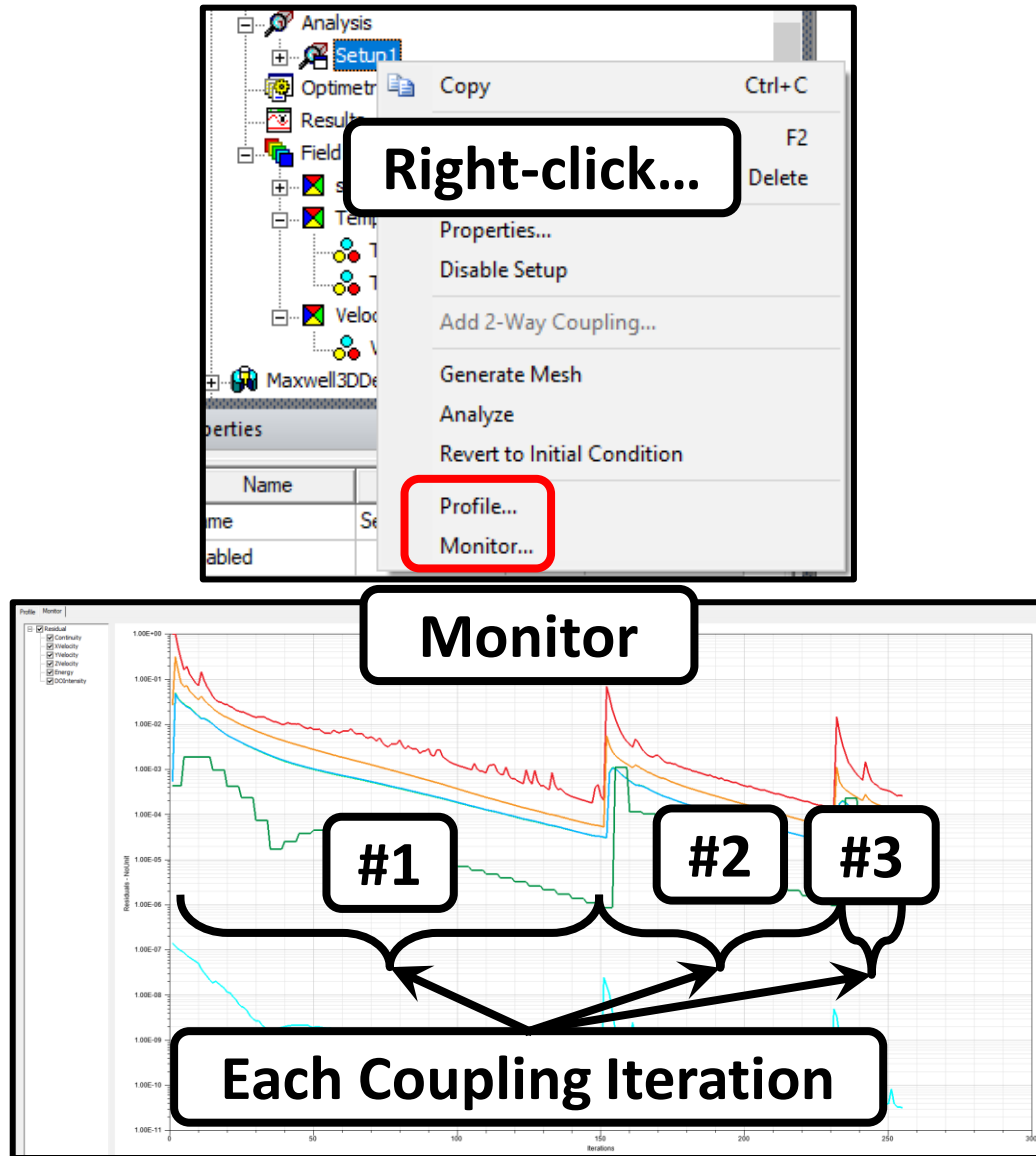
- The default middle slider bar setting provides a good Icepak mesh.
- Models that need additional mesh refinement should use a combination of mesh levels, and mesh regions.



422k Cells



Icepak – Eddy Current Setup – Solution Profile and Solution Monitors



/ Icepak – Eddy Current Setup – Results

Velocity
[m_per_sec]

0.2000
0.1867
0.1733
0.1600
0.1467
0.1333
0.1200
0.1067
0.0933
0.0800
0.0667
0.0533
0.0400
0.0267
0.0133
0.0000

Temperature
[cel]

120.0000
113.3333
106.6667
100.0000
93.3333
86.6667
80.0000
73.3333
66.6667
60.0000
53.3333
46.6667
40.0000
33.3333
26.6667
20.0000

First Coupling Iteration

$T_{max} = 103.2\text{ }^{\circ}\text{C}$

$Q_{total} = 3.89\text{ Watts}$

Final Coupling Iteration

$T_{max} = 122.0\text{ }^{\circ}\text{C}$

$Q_{total} = 5.0\text{ Watts}$



End of presentation