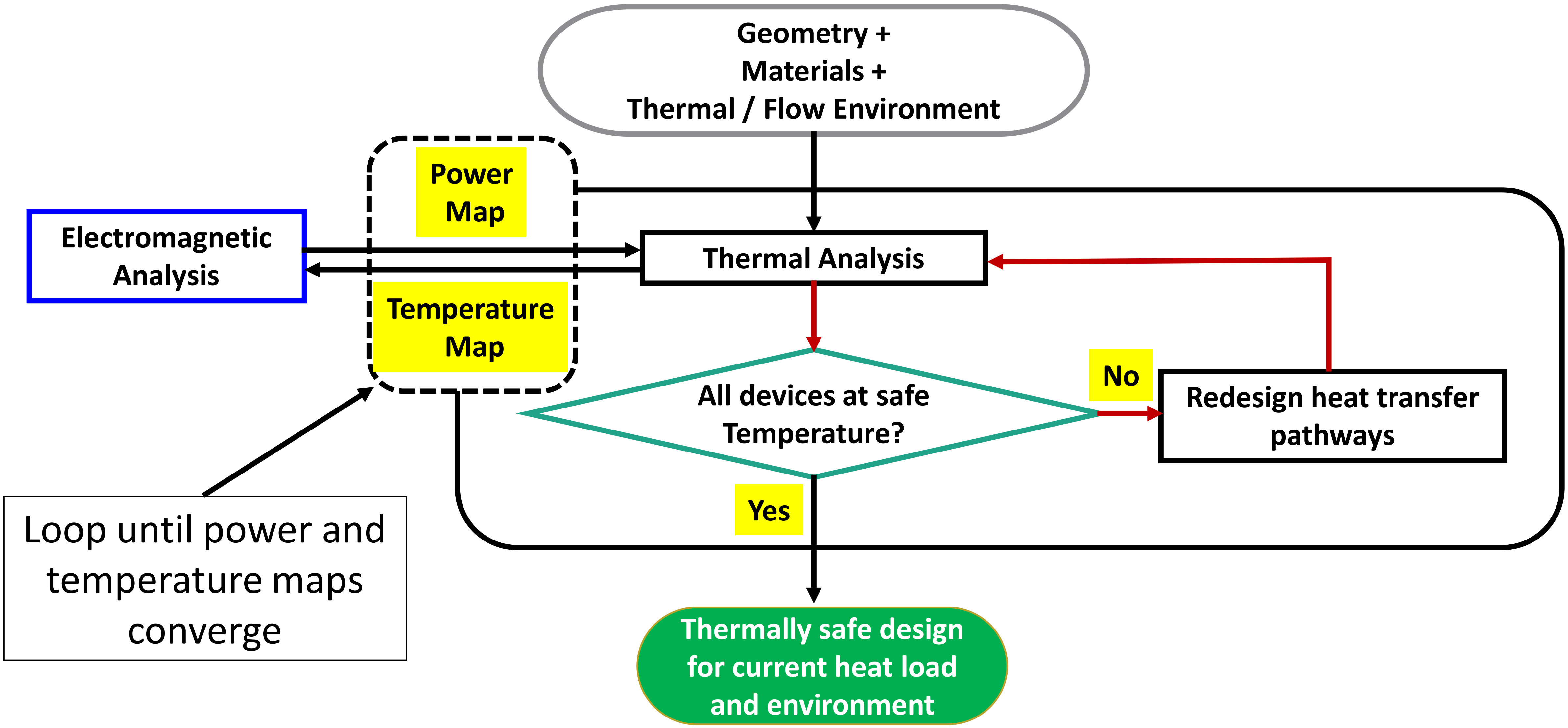
Module 4 – Lecture 1: Electro-Thermal Analysis

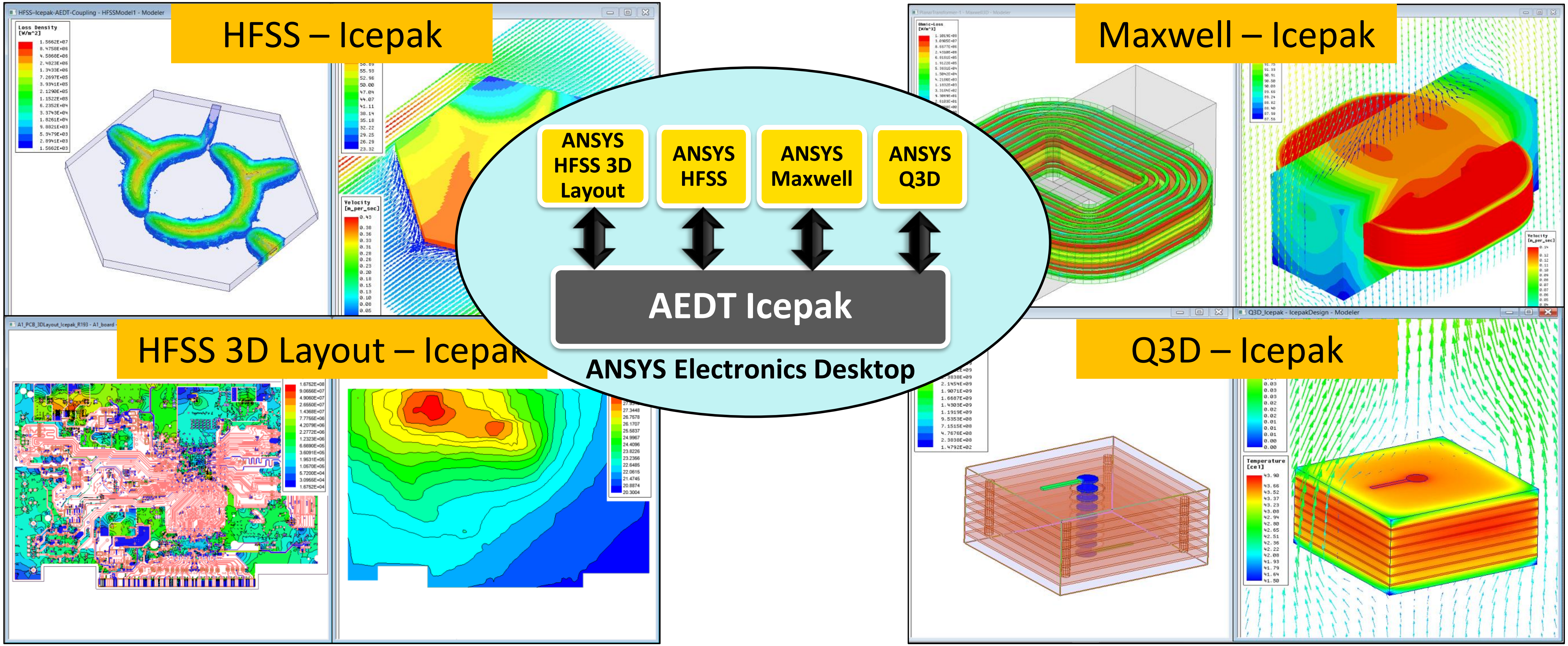
Release 2020 R1

Introduction – Need for Electro-Thermal Analysis



Introduction

- Electro-Thermal workflows in ANSYS Electronics Desktop

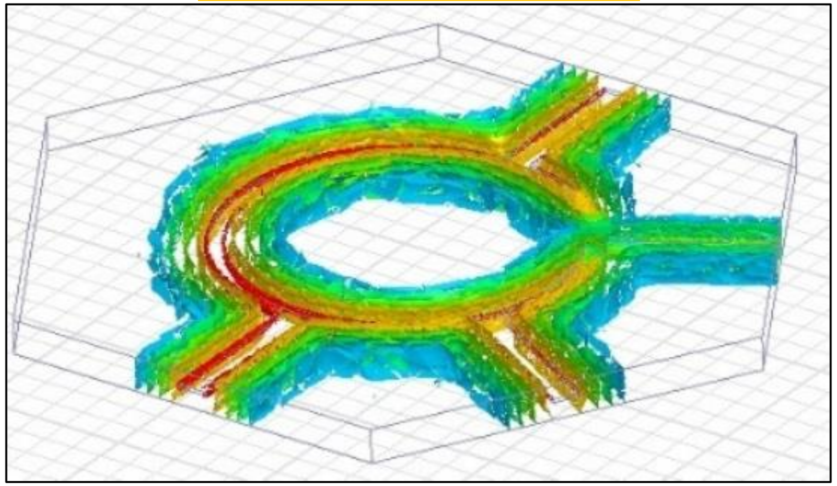


Ansys

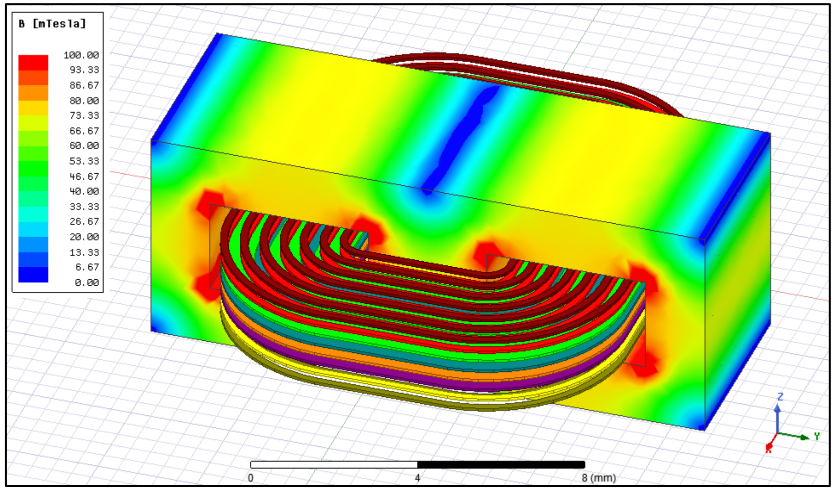
Electro-Thermal Workflow with HFSS, Maxwell and Q3D

- **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

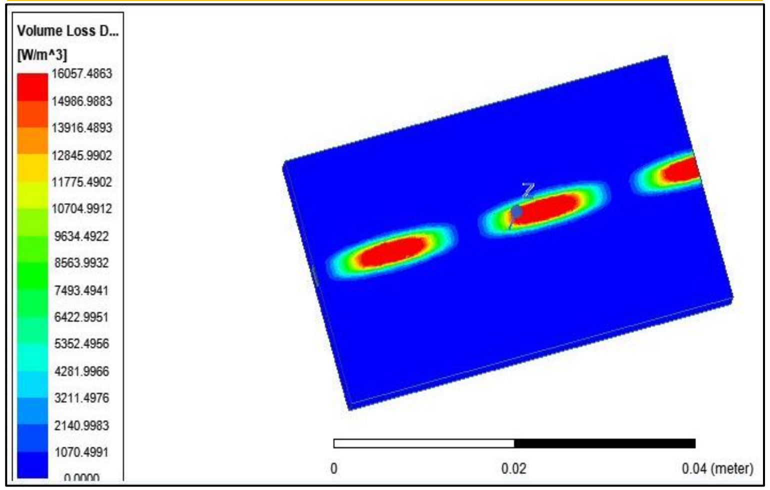
HFSS Model



Maxwell Model



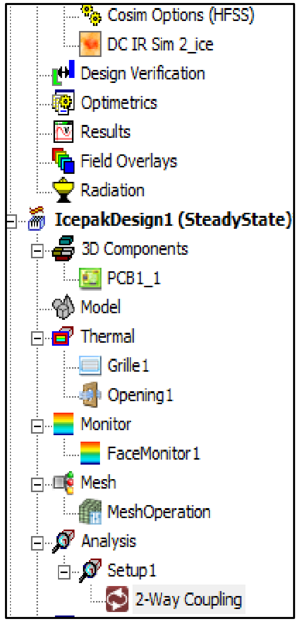
EM Loss Visualization

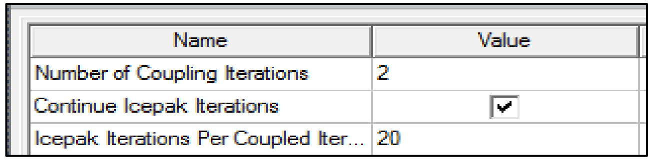


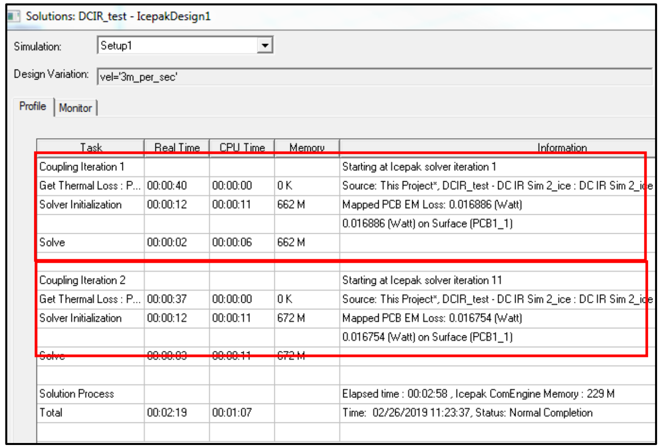
Ansys

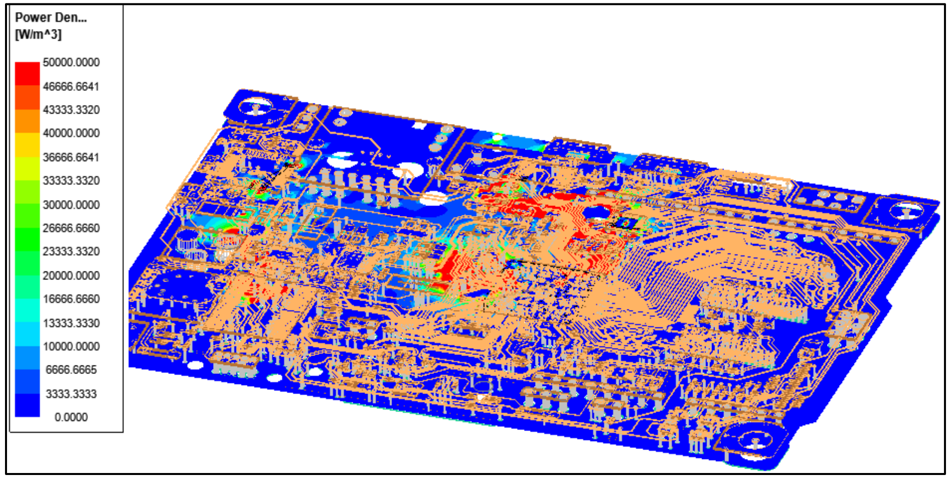
Electro-Thermal Workflow with HFSS 3D Layout

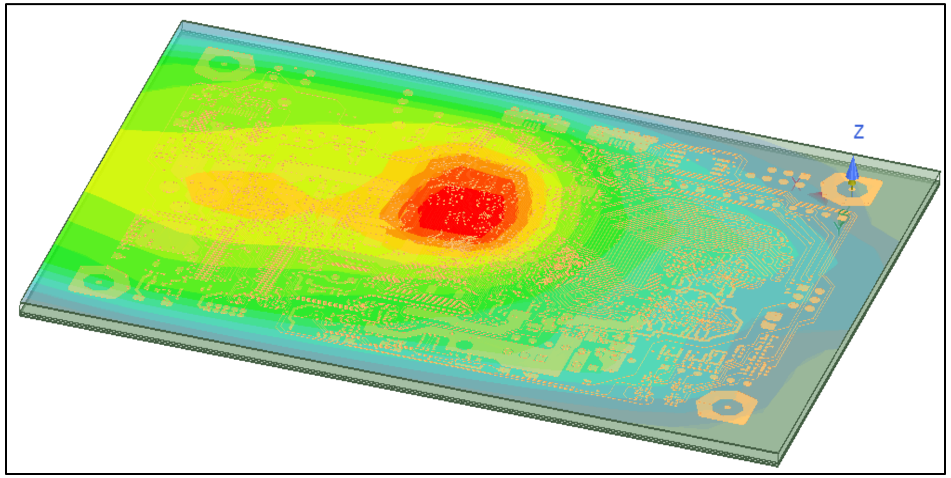
- **DC IR drop** analysis is performed in 3D Layout. The layer-by-layer Joule heating power maps are transferred to Icepak
- Temperature map from Icepak is transferred to 3D Layout to update the material properties (temperature dependent) spatially. DC IR analysis is run again with the updated material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







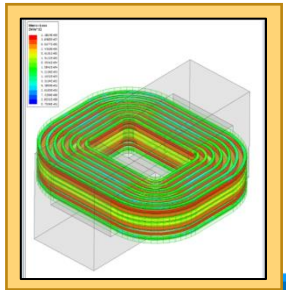




Ansys

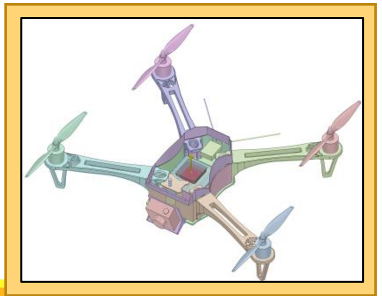
Electro-Thermal Workflow in AEDT

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



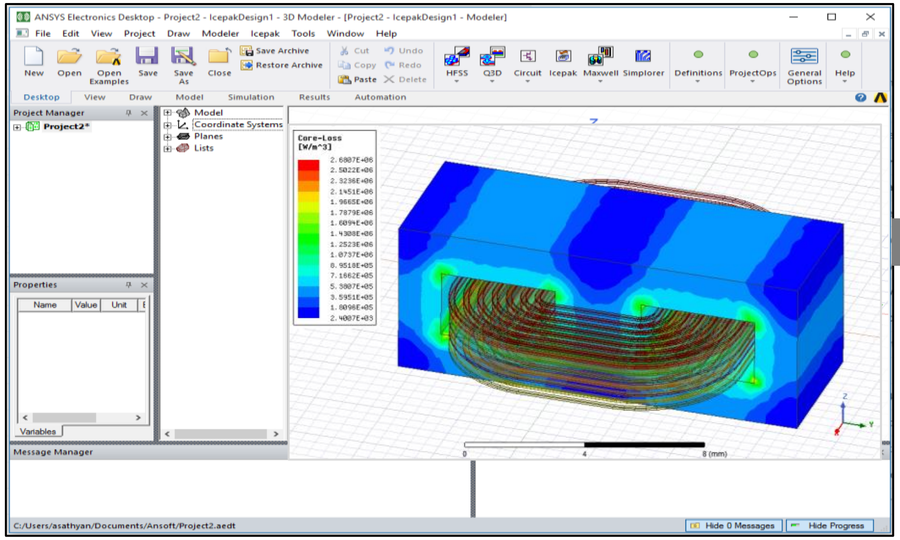
Manual Process

- 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

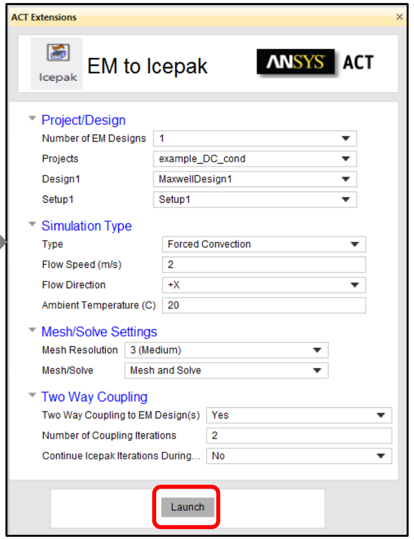
Electro-Thermal ACT Workflow

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

Complete EM analysis



Launch ACT

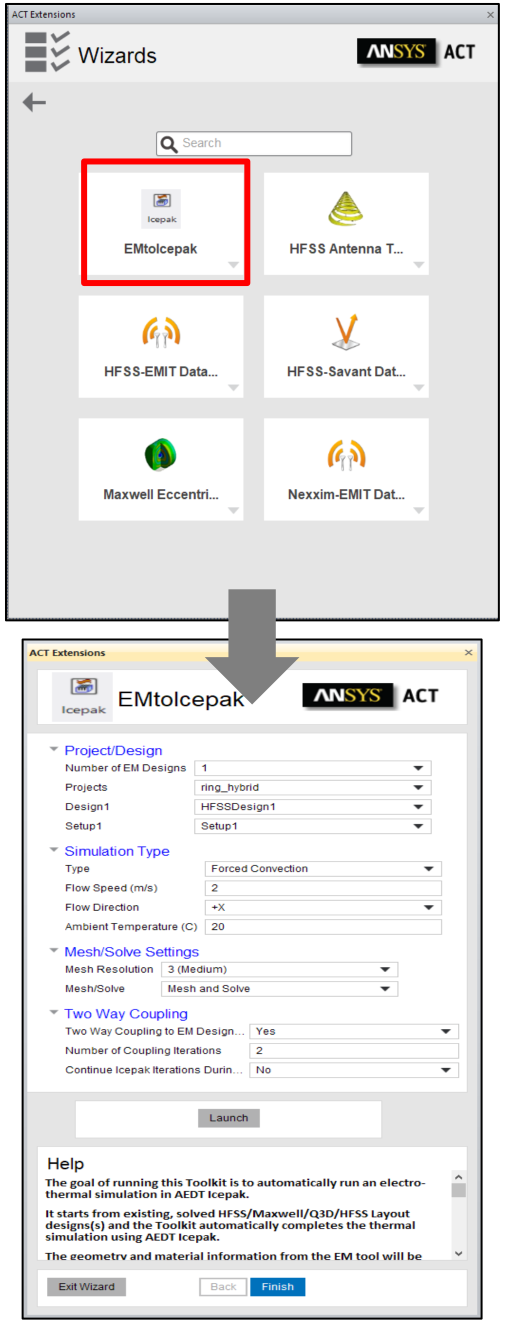


Ansys

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:

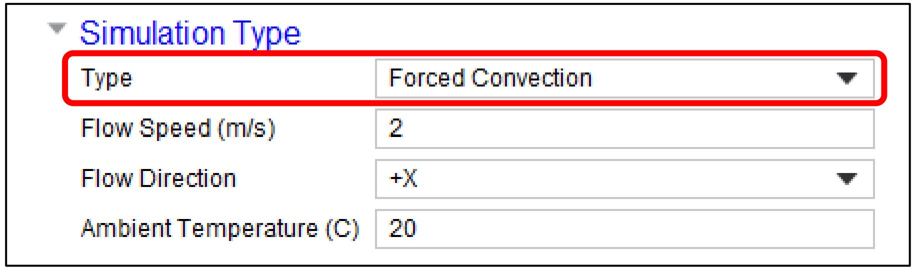
- Flow speed (m/s)
- Flow direction
- Ambient temperature (C)

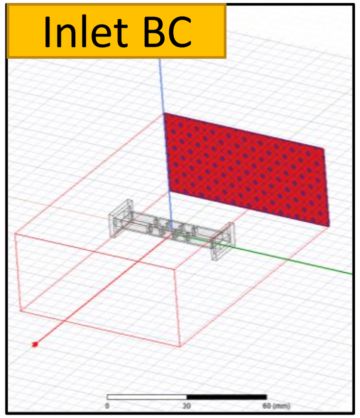
Computational Domain:

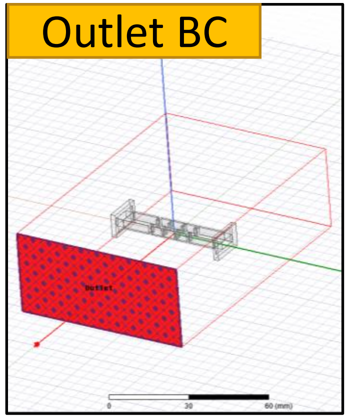
- Padding is based on the largest length in the geometry
- 100% padding added upstream and downstream of the model in the flow direction
- 25% padding added perpendicular to the flow direction

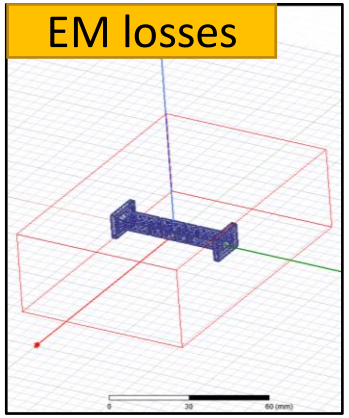
Solver Settings:

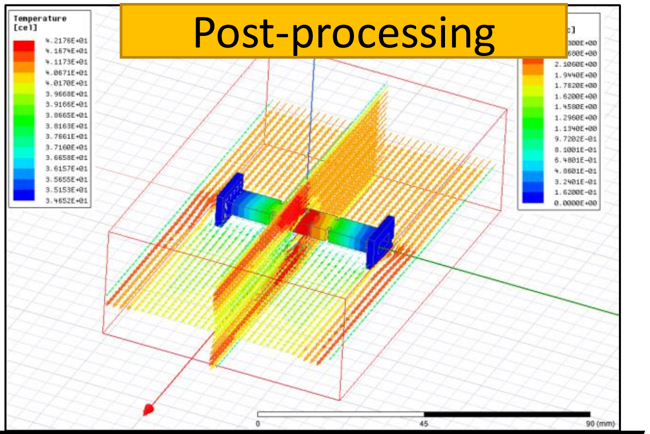
- Sequential solver is used, and Radiation is turned **OFF**



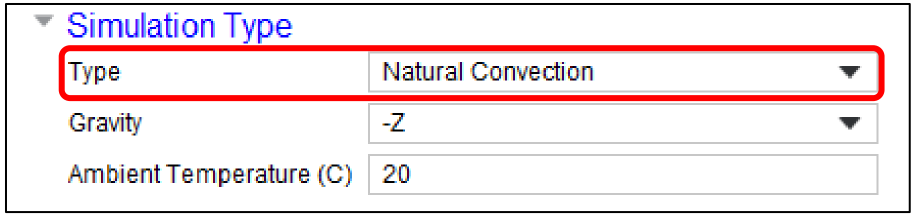








Scenario 2: Natural Convection (Passive Cooling)



Inputs:

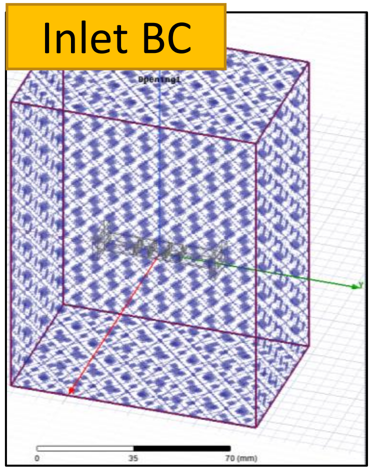
- Gravity direction
- Ambient temperature (C)

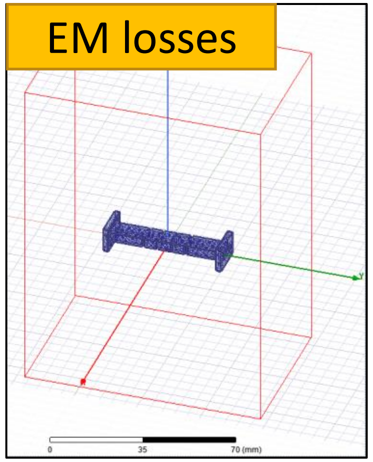
Computational Domain:

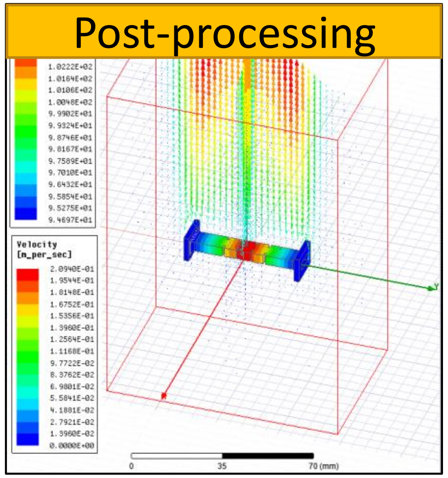
- Padding is based on the largest length in the geometry
- 200% padding added in the direction opposite to gravity
- 100% padding added in the direction of gravity
- 50% padding added perpendicular to the direction of gravity

Solver Settings:

- Radiation is turned **ON**







Ansys

Electro-Thermal ACT – GUI and Options

Project/Design:

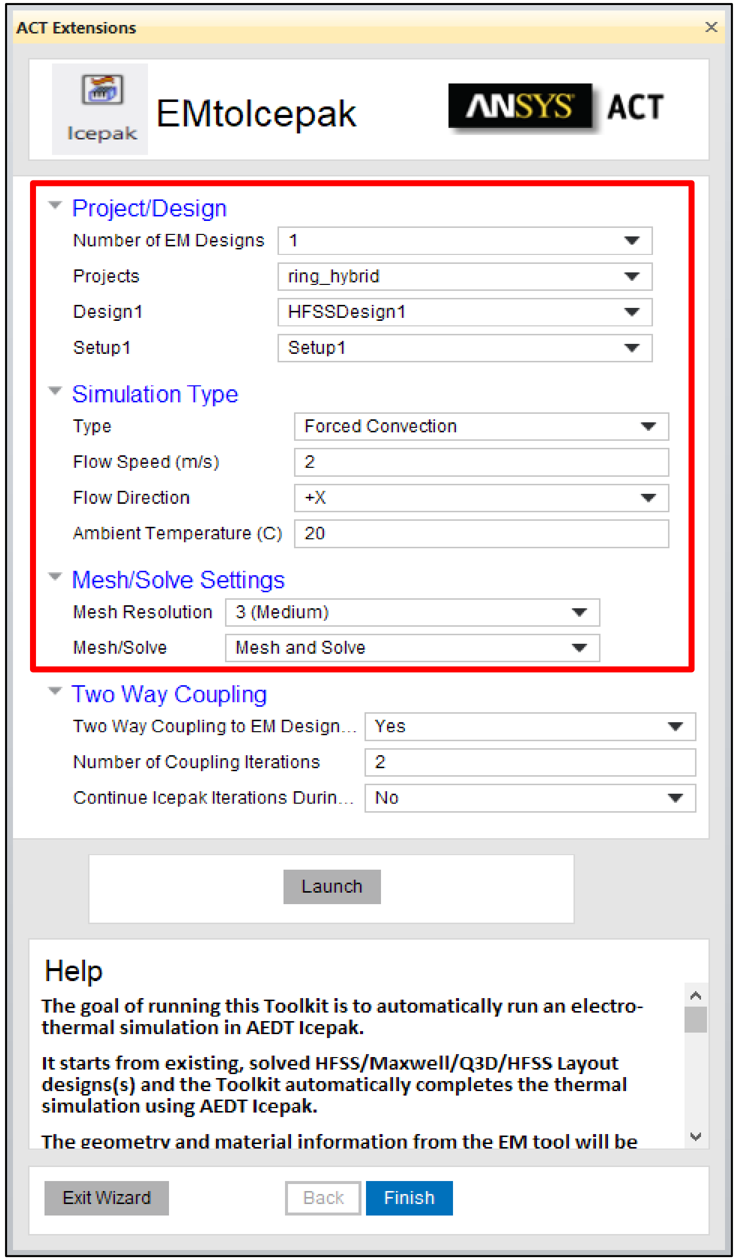
- Launching the ACT will populate list of projects, designs and setups for selection
- Up to 5 designs can be selected for coupling to Icepak

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.



Electro-Thermal ACT – GUI and Options

Two-way Coupling:

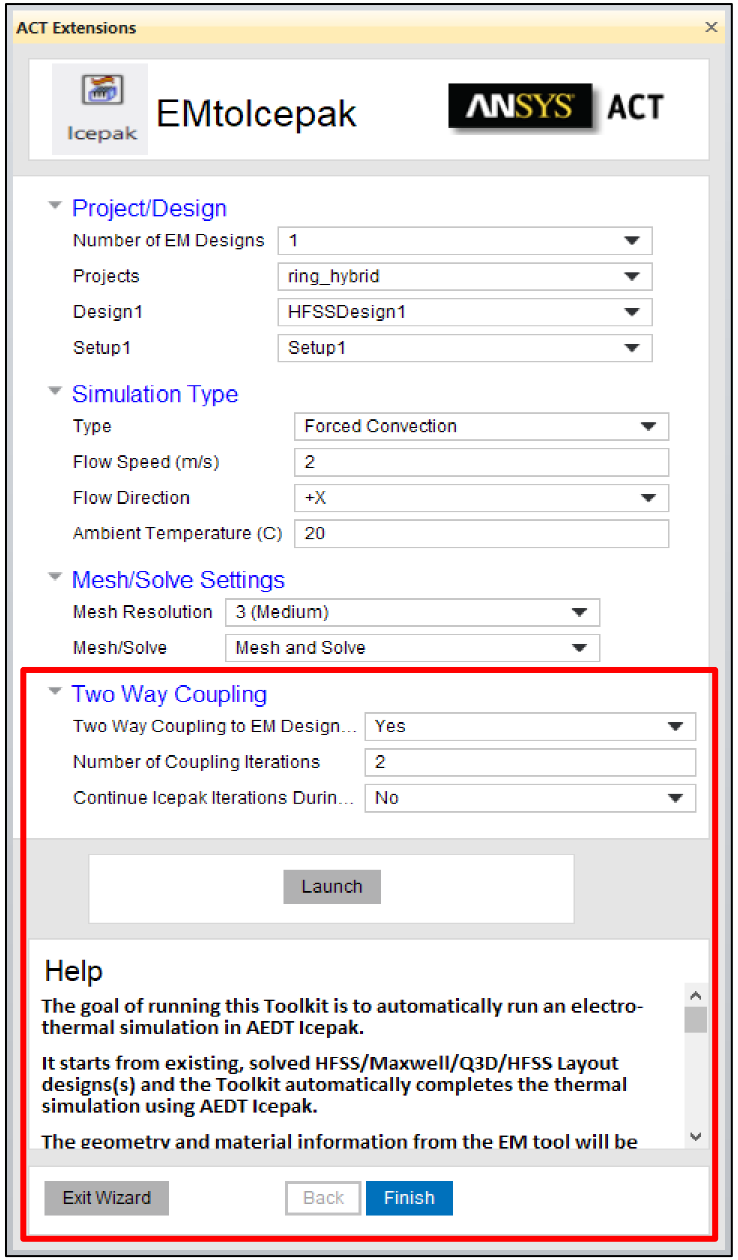
- **Two way Coupling to EM Design**: Selecting **Yes** enables bidirectional coupling of EM design and Icepak
- **Number of Coupling Iterations:** Defines the number of times EM design and Icepak simulations must be launched and updated with new results
- **Continue Icepak Iterations During Coupling:** 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

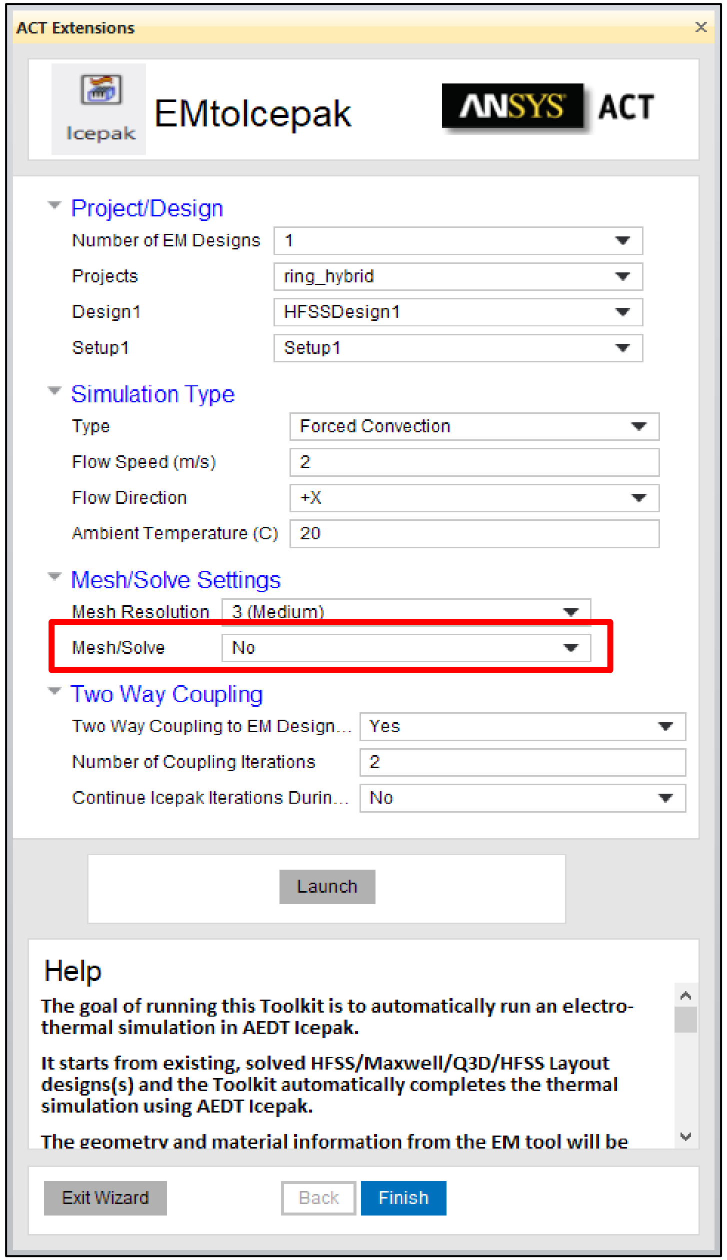
Finish:

- Quit the ACT GUI and return to the Extensions Wizard



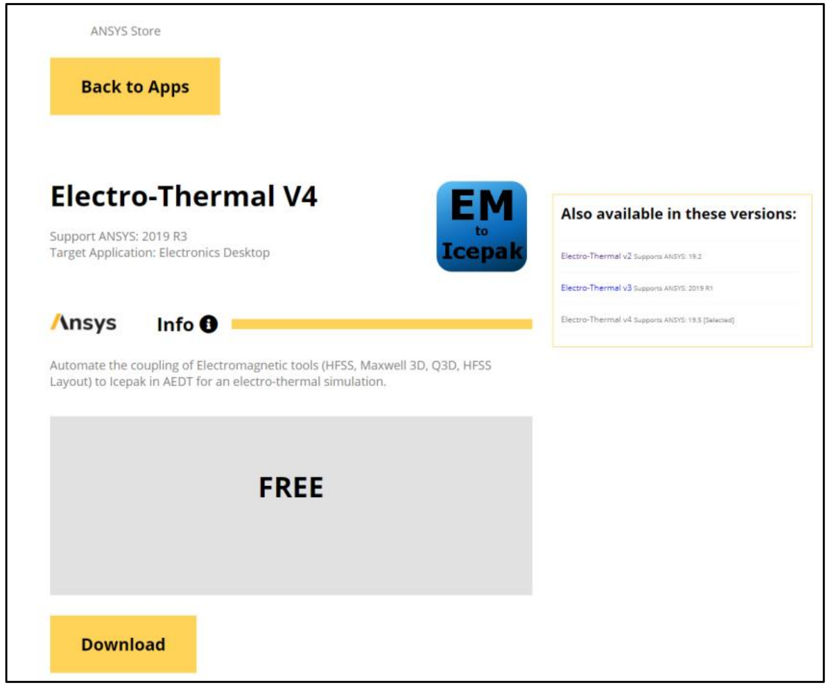
Electro-Thermal ACT – Important Points

- 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



Electro-Thermal ACT

- The Electro-Thermal ACT is available for download from the ANSYS App Store
  https://catalog.ansys.com

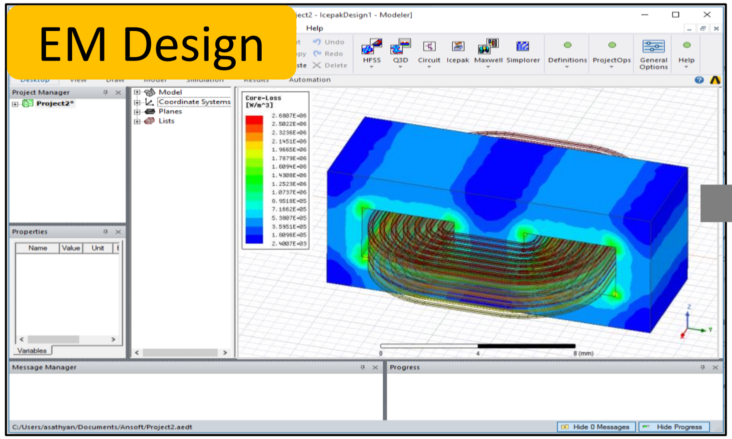


- **ANSYS App Store is a great place to get started**
  - A library of helpful applications available to any ANSYS customer
  - New apps are added regularly
  - Applications made available in either binary format (.wbex file) or scripted format (Python and XML files)
  - Scripted extensions are great examples
  - Documentation and training materials available on the ANSYS Customer **Portal: https://support.ansys.com/AnsysCustomerPortal/en_us/Downloads/ACT+Resources**

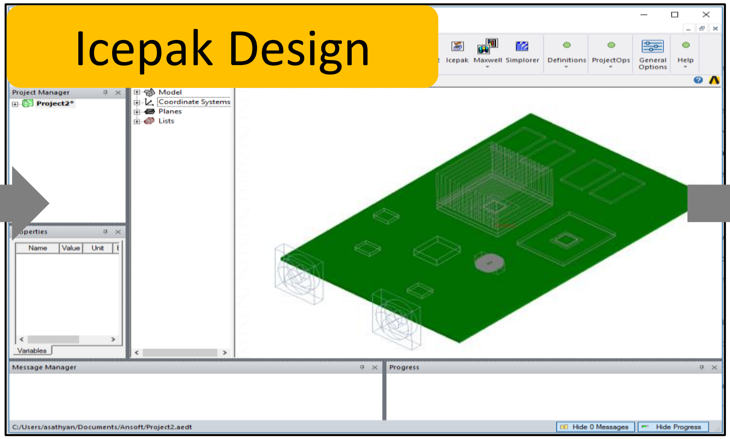
Manual 2-way Electro-Thermal Workflow

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**

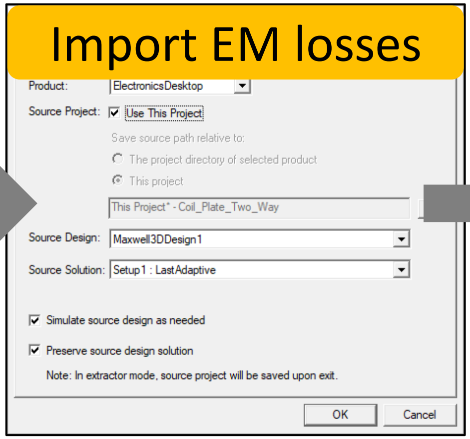
EM Design



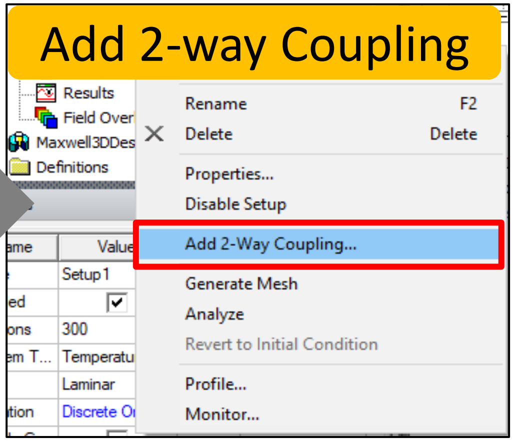
Icepak Design



Import EM losses



Add 2-way Coupling

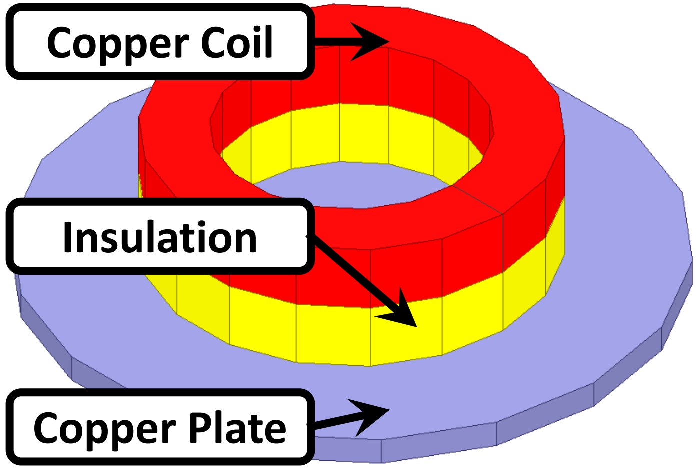


Ansys

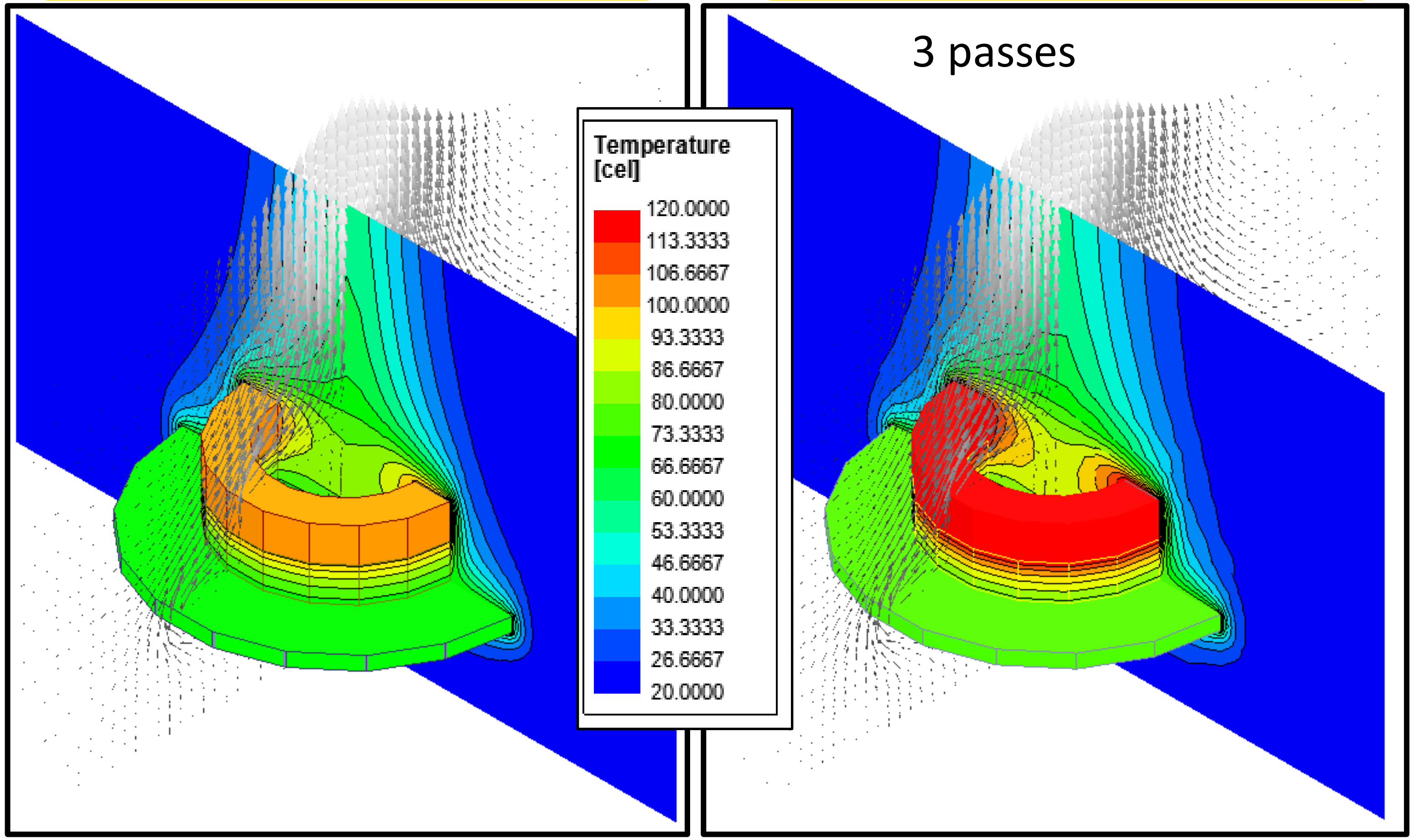
Appendix: Electro-Thermal Workflow Example

Ansys

Example



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



$T_{max} = 103.2$ °C

$Q_{total} = 3.89$ Watts

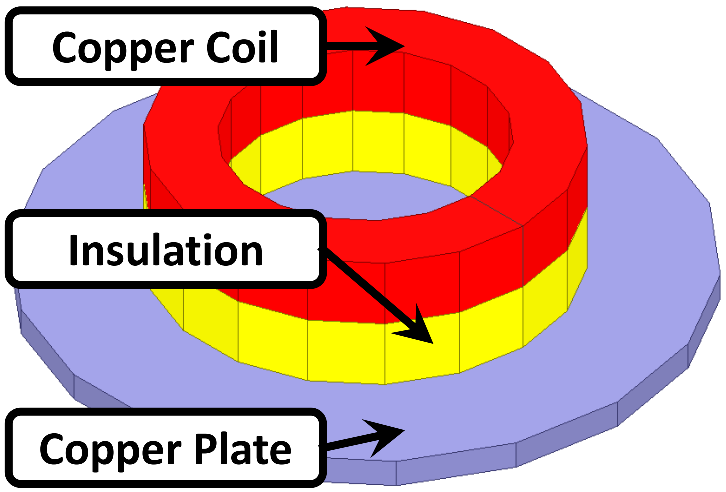
$T_{max} = 122.0$ °C

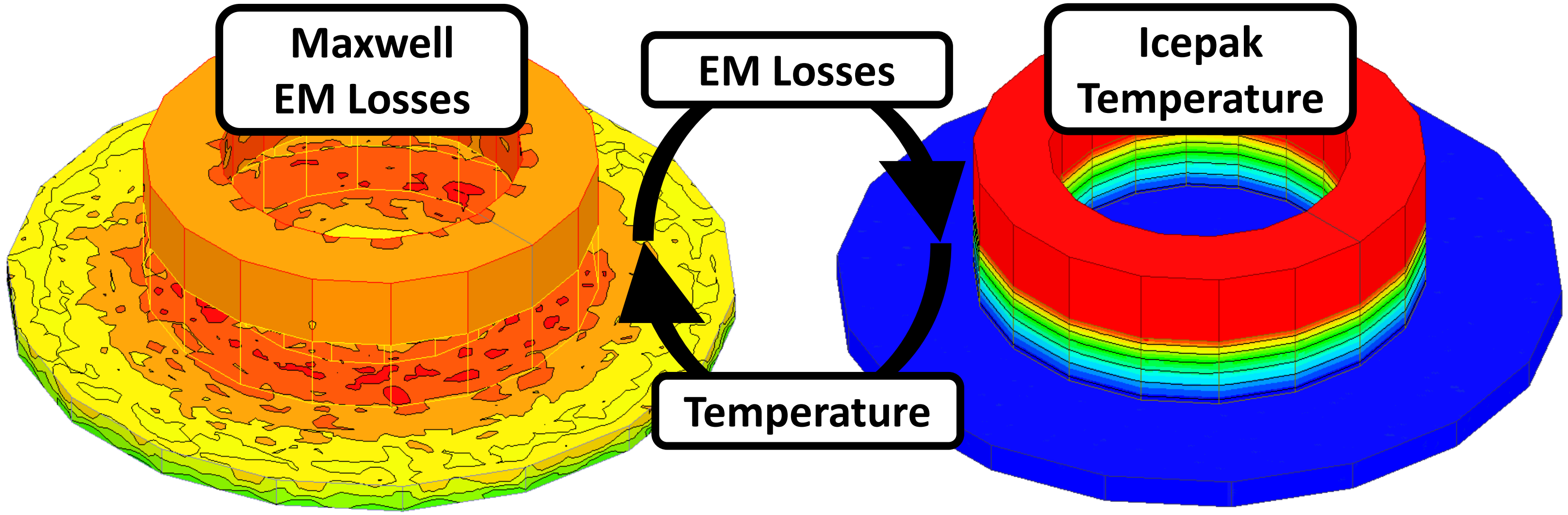
$Q_{total} = 5.0$ Watts

Ansys

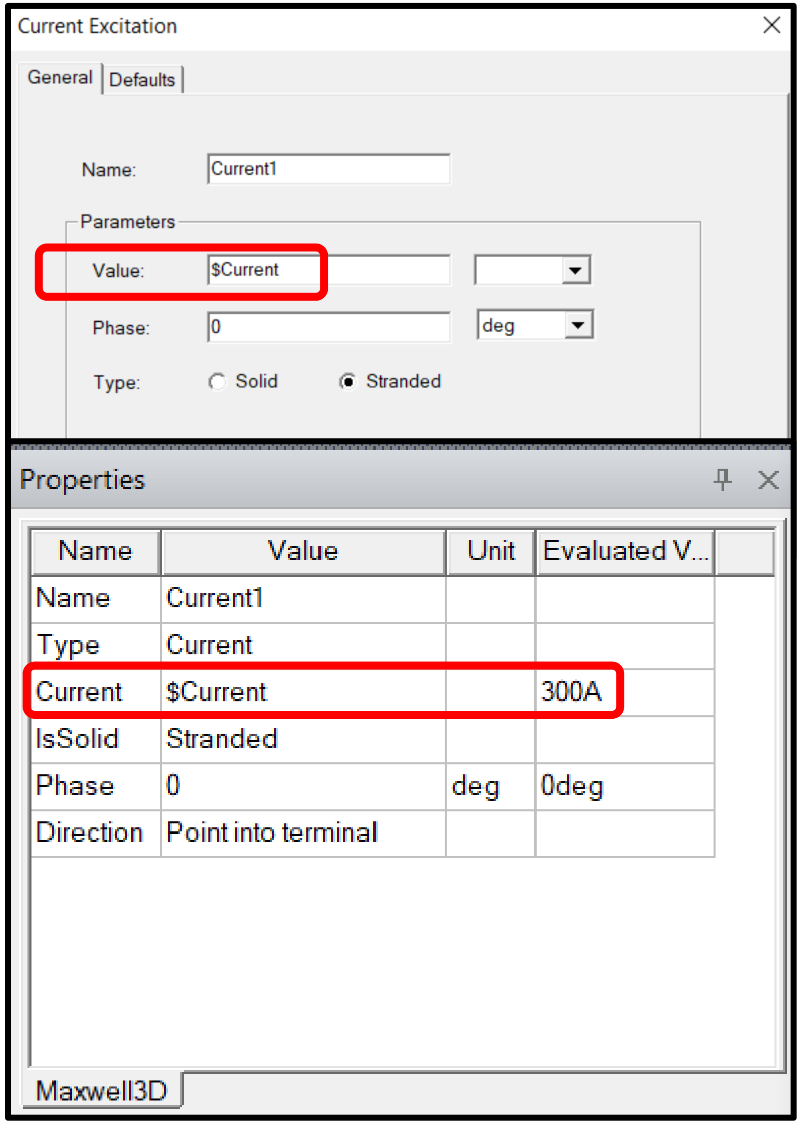
Sample Problem Definition

- **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.**

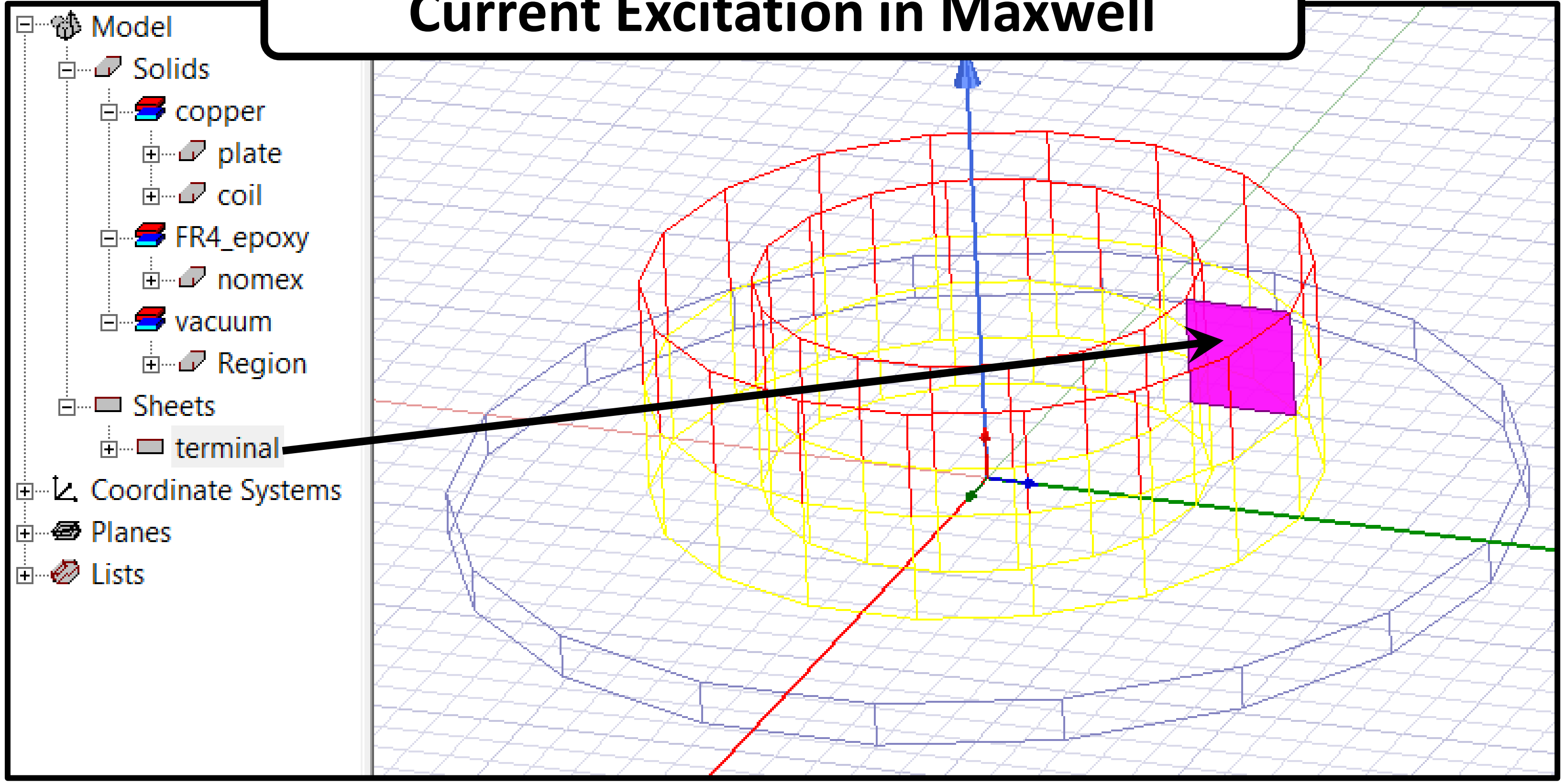




Maxwell – Eddy Current Setup



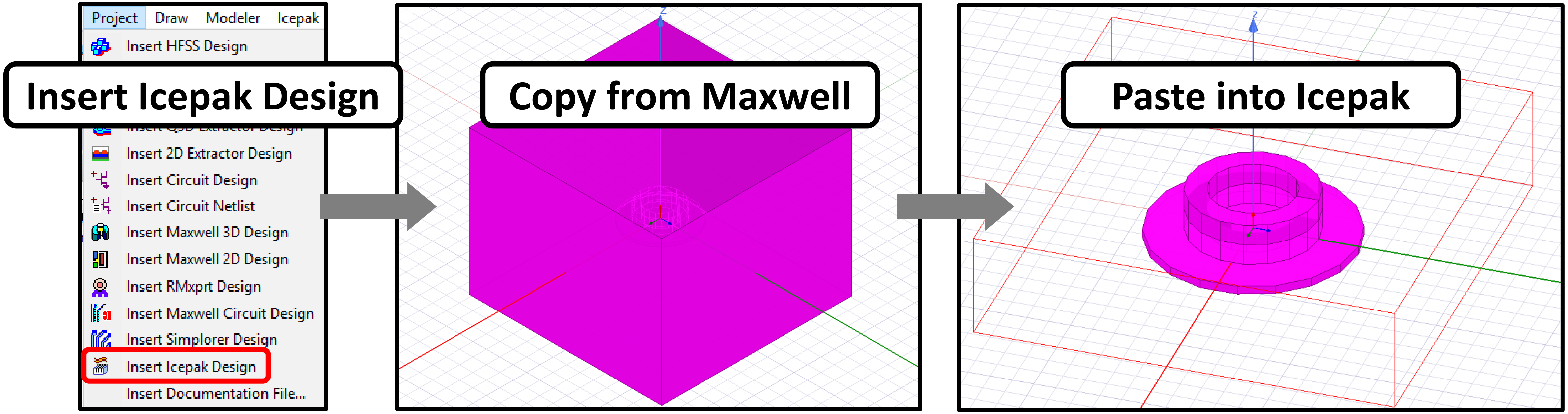
Current Excitation in Maxwell



From Maxwell3D > Fields > Calculator

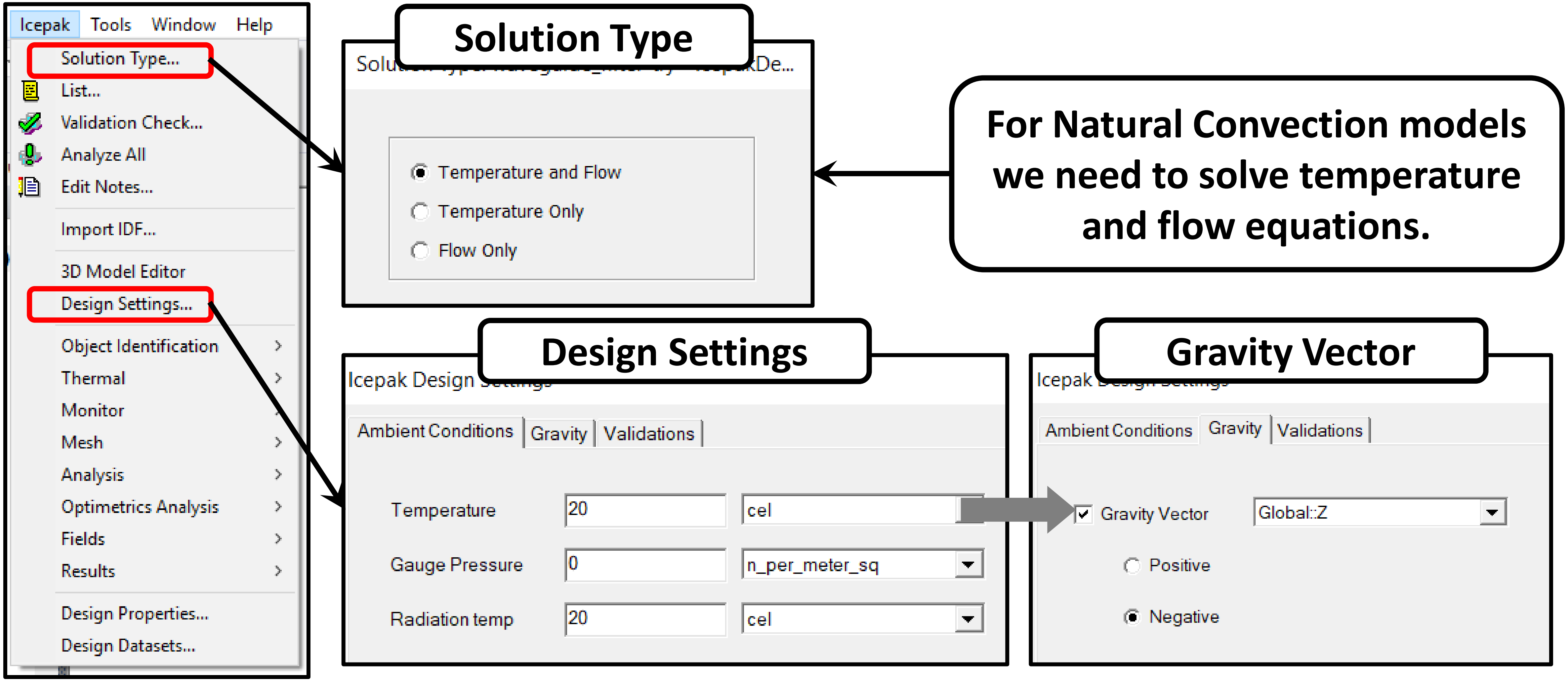
Icepak – Eddy Current Setup – Geometry Setup

- **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.**



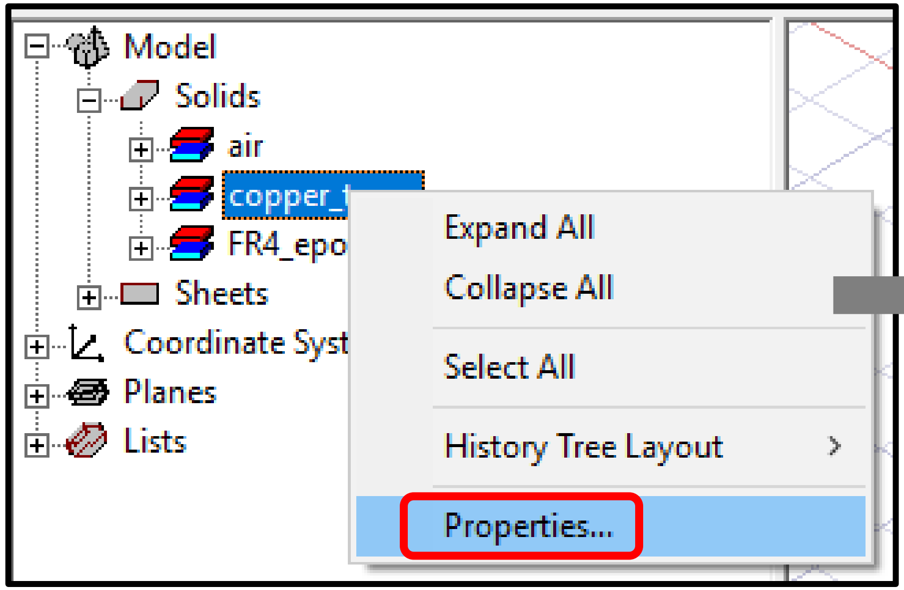
Icepak – Eddy Current Setup – Solution Type and Design Settings

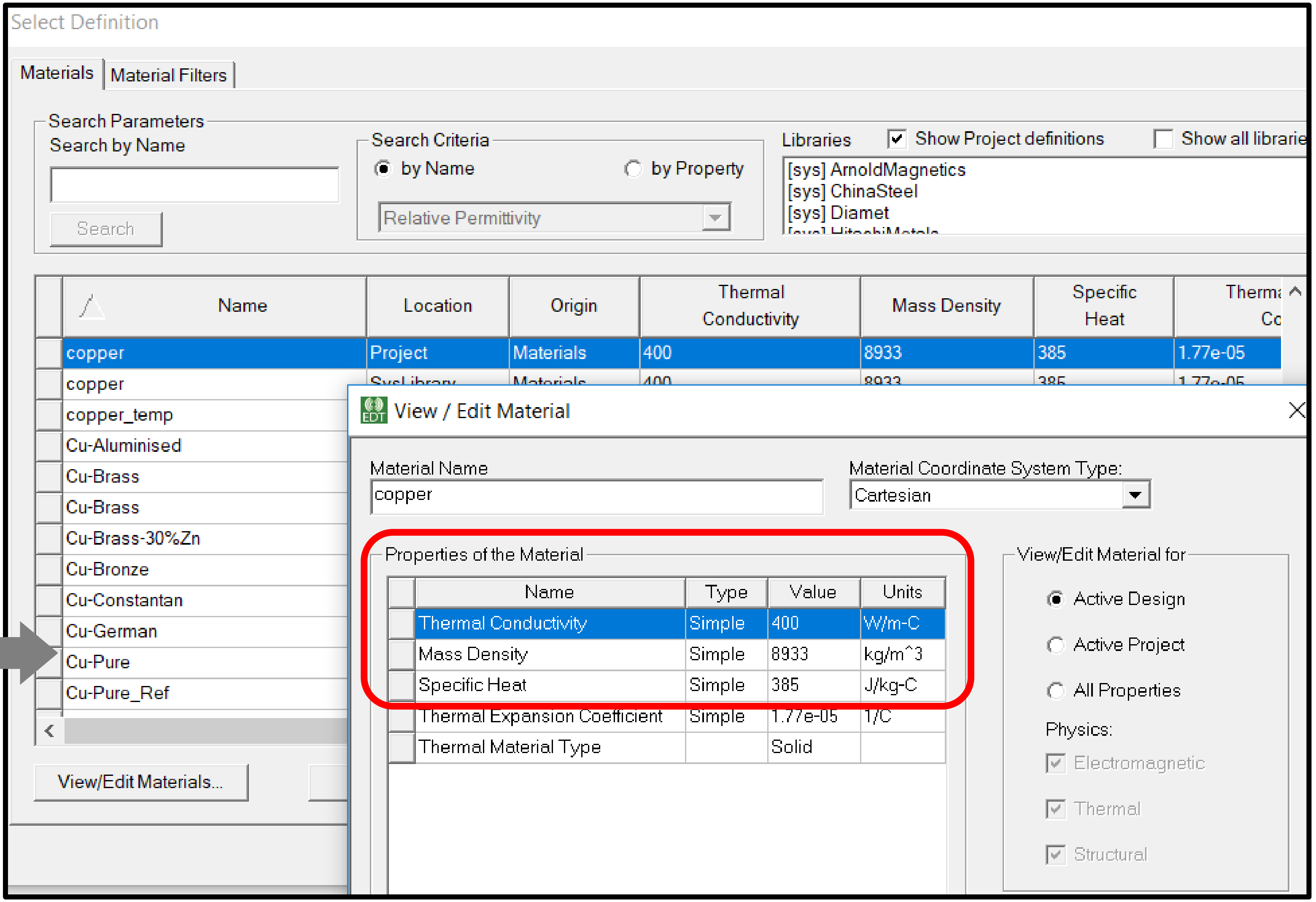
- **Set up the Icepak Design for Natural Convection CFD analysis.**



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.**

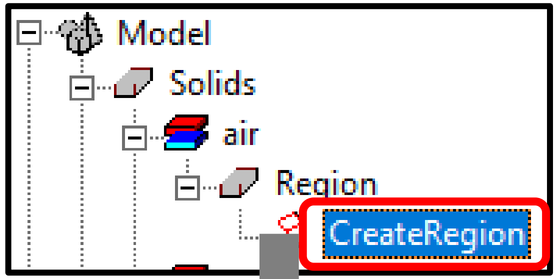




Ansys

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.**



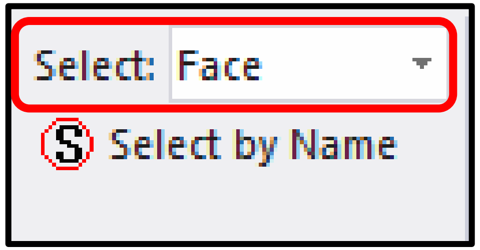
	Name	Value	Unit	Evaluated
	Command	CreateRegion		
	Coordinate ...	Global		
	+X Padding...	Percentage Offset		
	+X Padding...	50		50
	-X Padding ...	Percentage Offset		
	-X Padding ...	50		50
	+Y Padding...	Percentage Offset		
	+Y Padding...	50		50
	-Y Padding ...	Percentage Offset		
	-Y Padding ...	50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

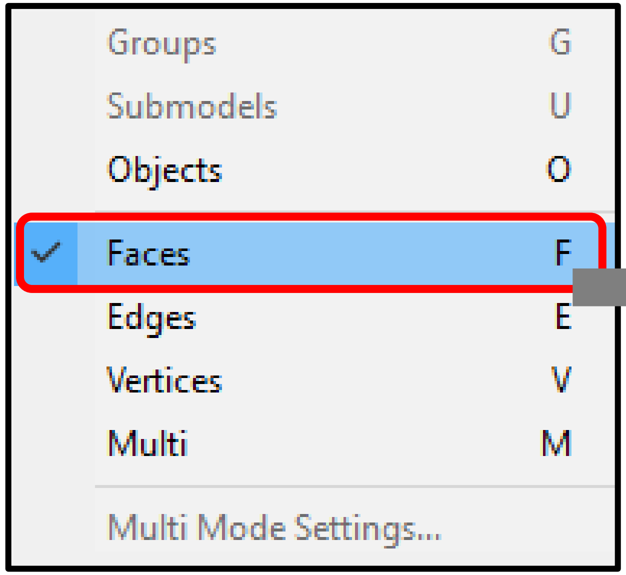
Ansys

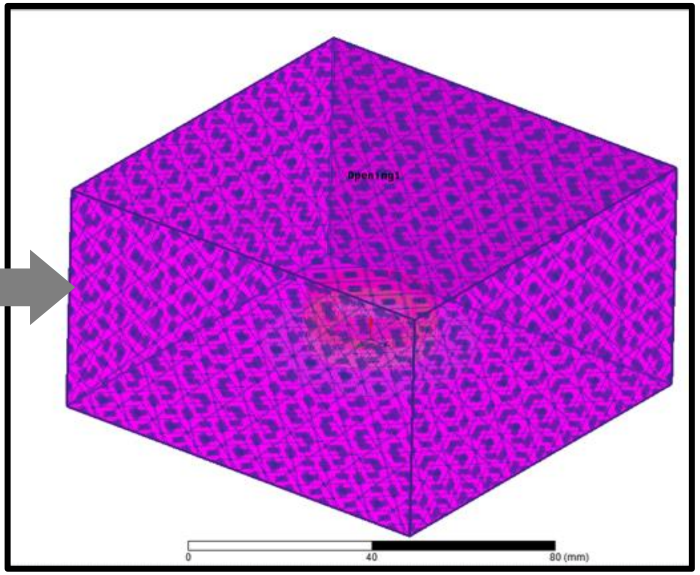
Icepak – Eddy Current Setup – Thermal Setup

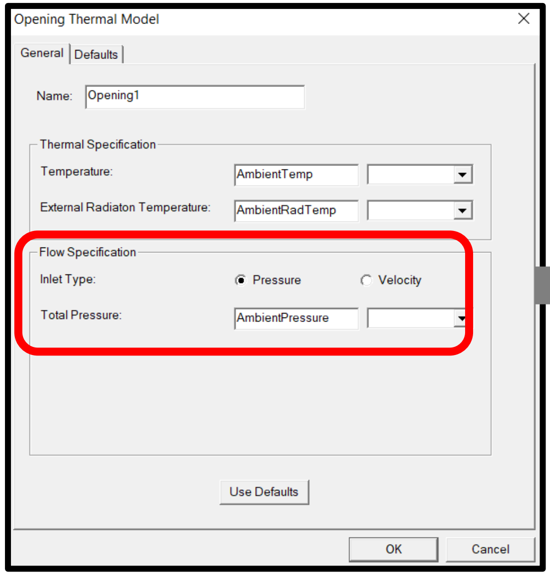
- **Selection Modes:**
  - **Right-click in GUI > Selection Mode.**
  - **From menu Edit > Selection Mode.**
  - **Or from Ribbon Drop Down.**
  - **Object (O)**
  - **Face (F)**
  - **Edge (E)**
  - **Vertex (V)**
  - **Multi (M)**

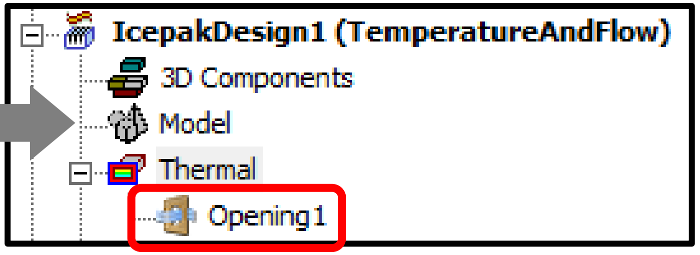


- **Enter Face Selection Mode.**
- **Select all air region faces.**
- **Right-click > Assign Thermal > Opening.**
- **Use default Pressure Inlet settings.**





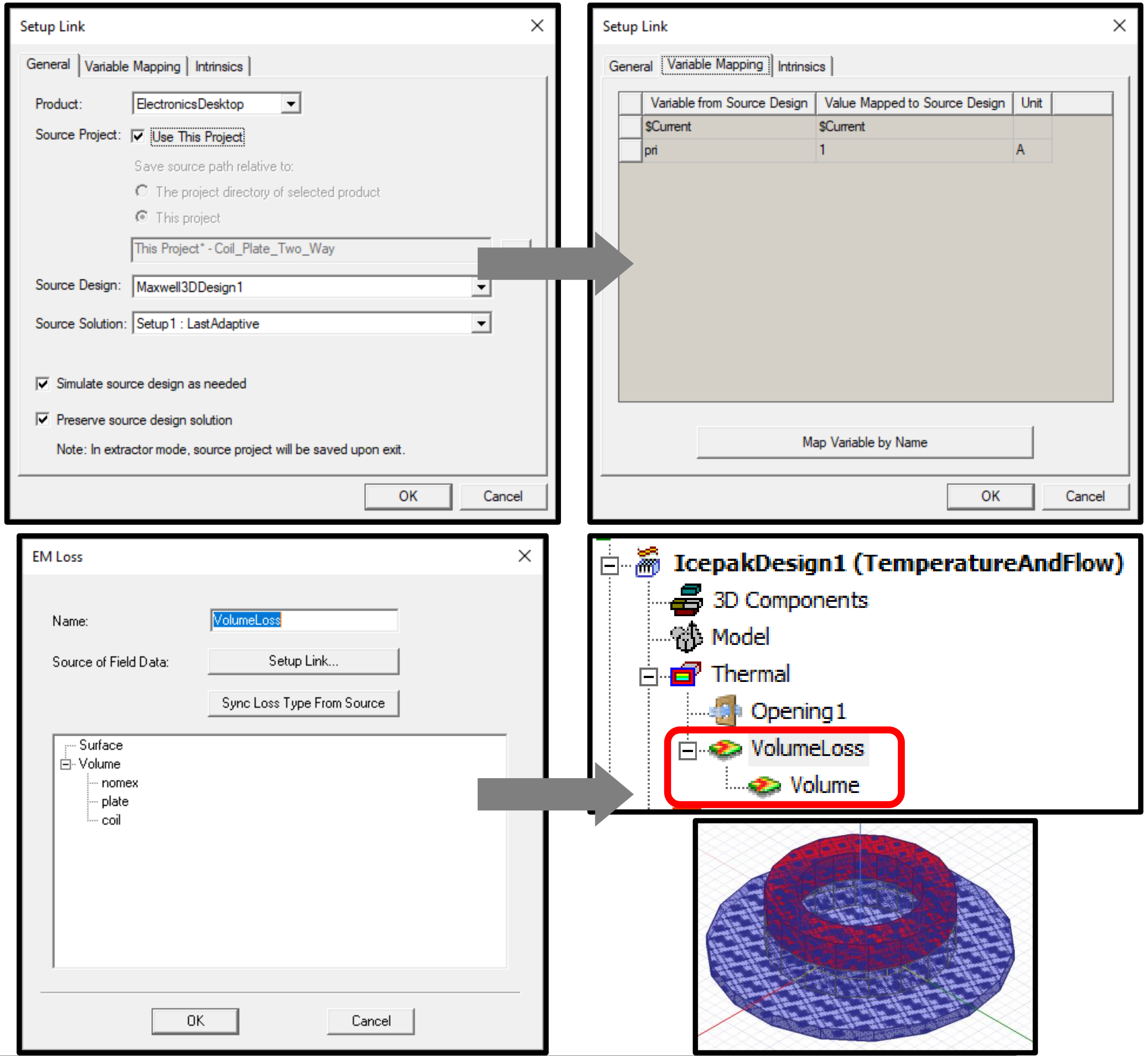




Ansys

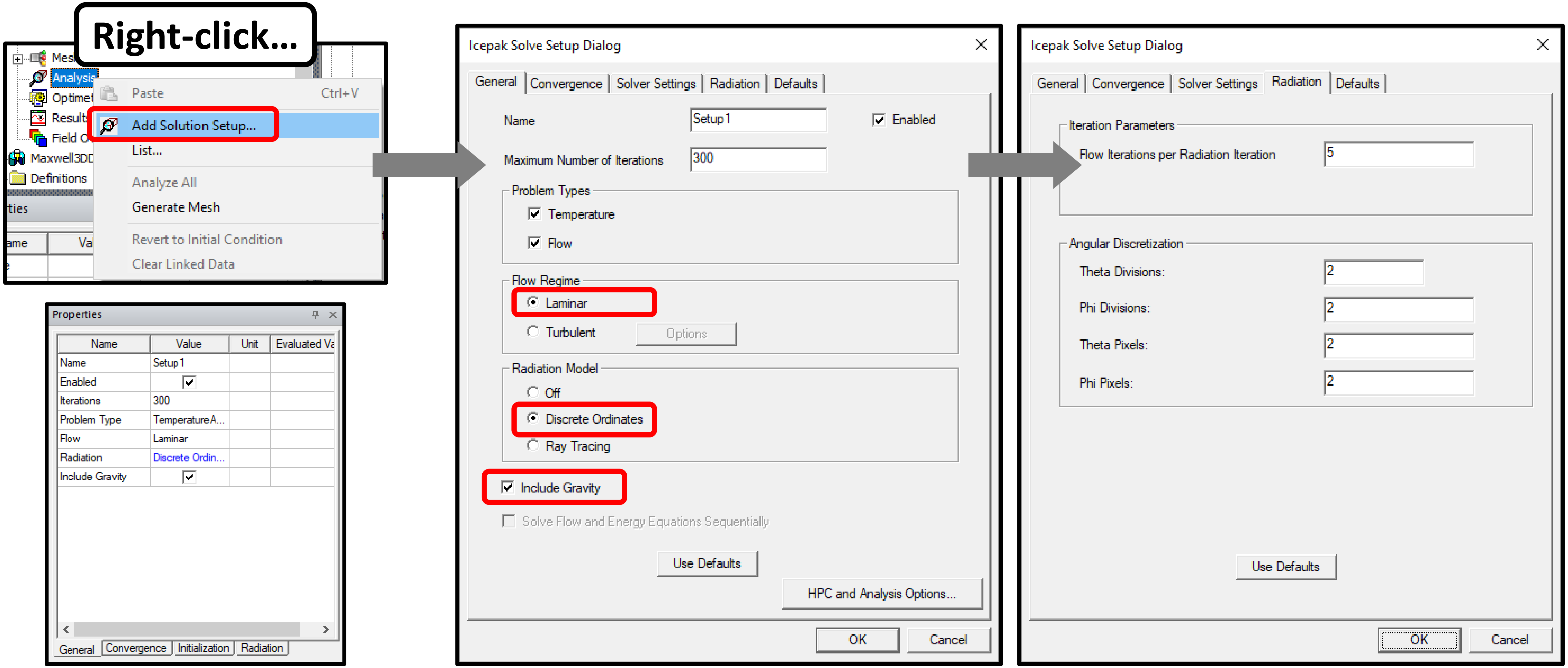
Icepak – Eddy Current Setup – Thermal Setup

- **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.**



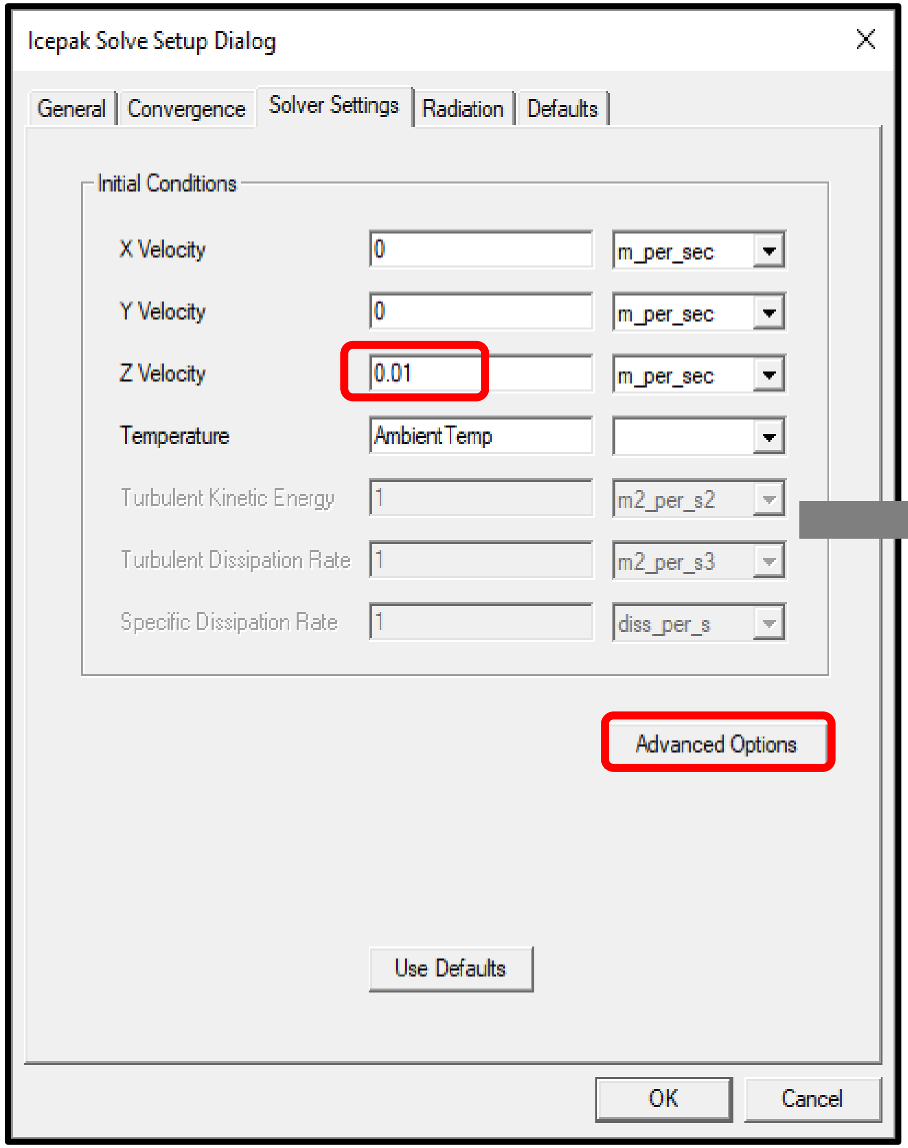
Icepak – Eddy Current Setup – Solution Setup

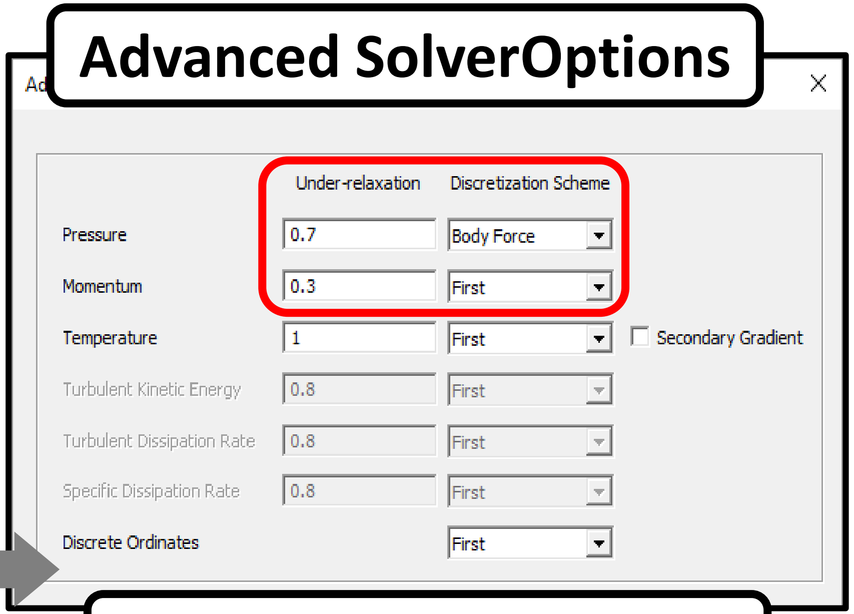
- **Add an Analysis Setup using laminar flow, radiation, and gravity as shown below.**

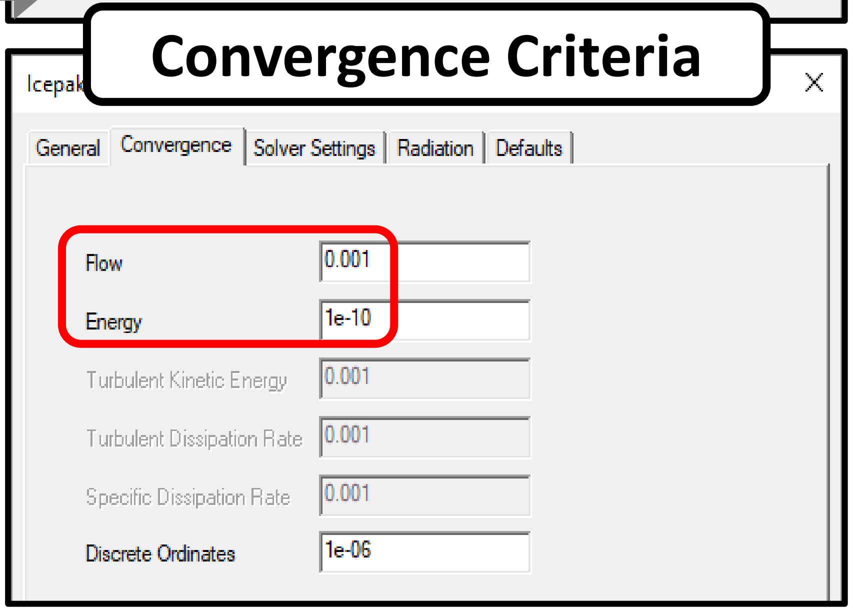


Icepak – Eddy Current Setup – Solution Setup

- **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.**

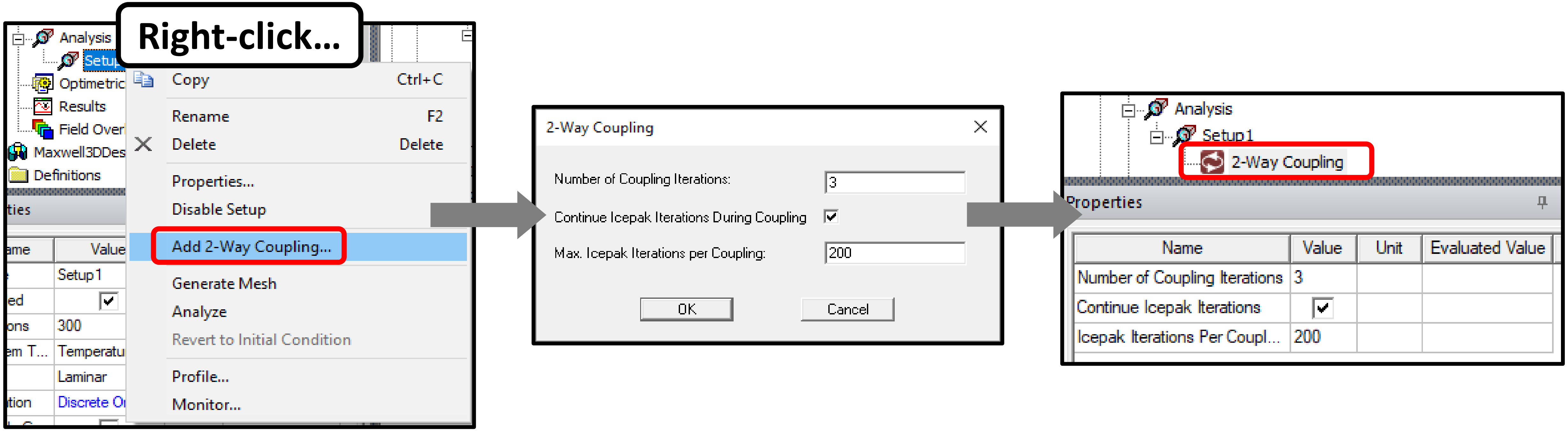






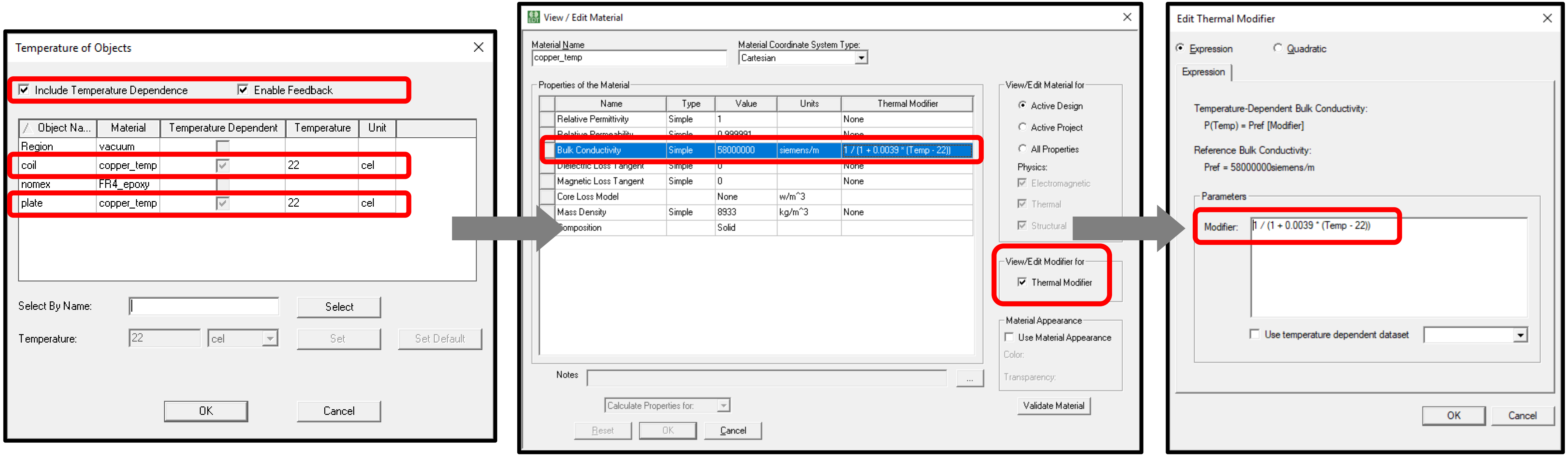
Icepak – Eddy Current Setup – Solution Setup

- **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.**



Maxwell – Eddy Current Setup – Solution Setup

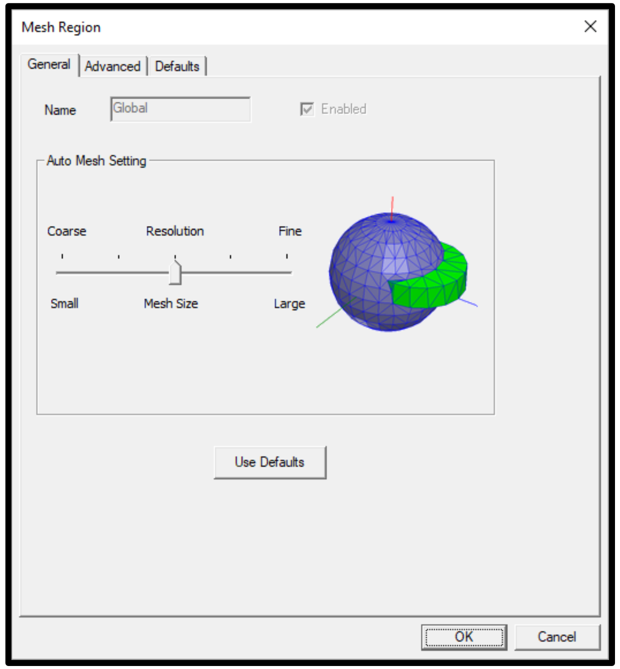
- **Two-way coupling needs to have temperature dependence in the EM solver.**
- **In the Maxwell design, Maxwell 3D > Set Object Temperature.**
- **“Include Temperature Dependence” and “Enable Feedback” must be checked ON.**
- **The temperature dependent materials must have the “Thermal Modifier” active.**

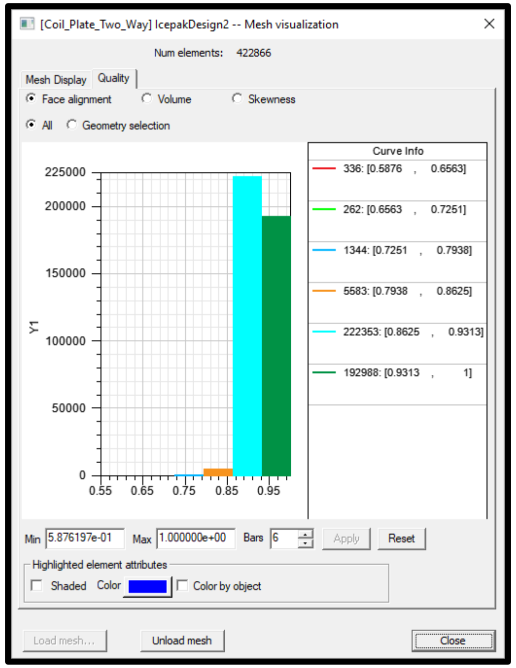


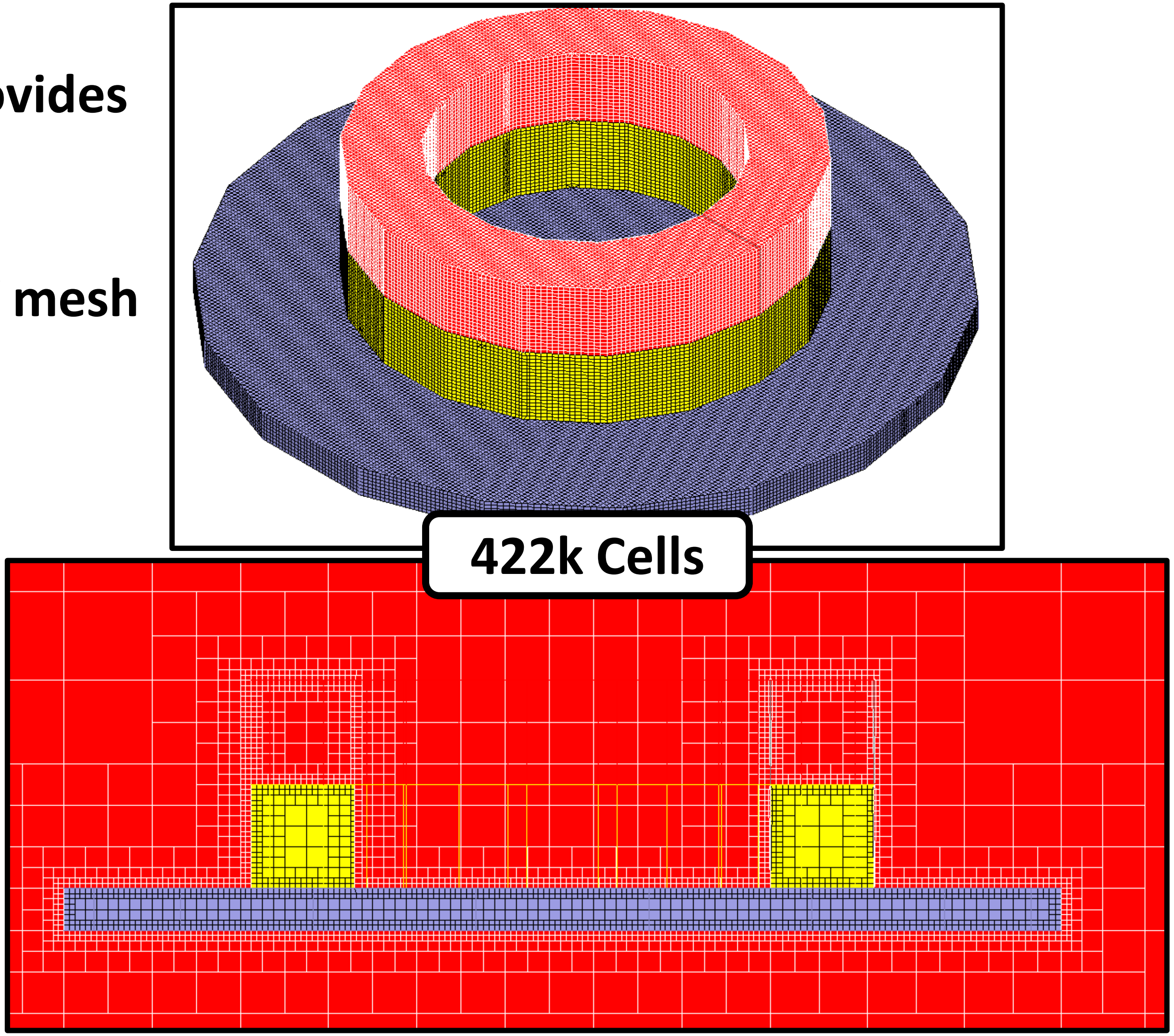
Ansys

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.**

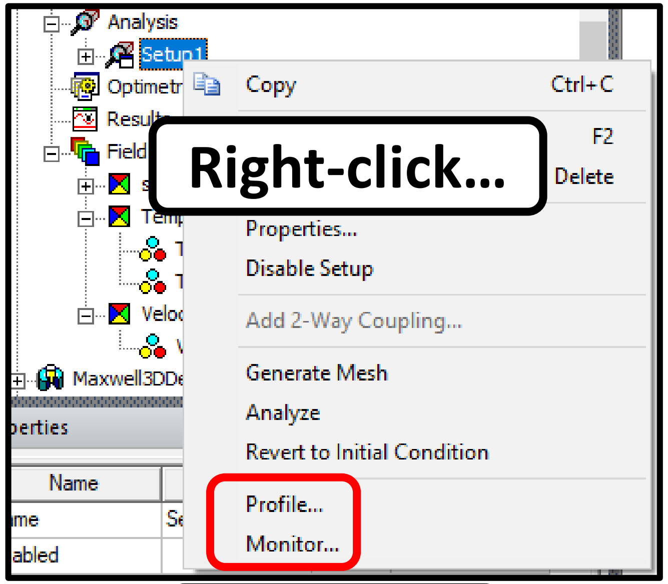


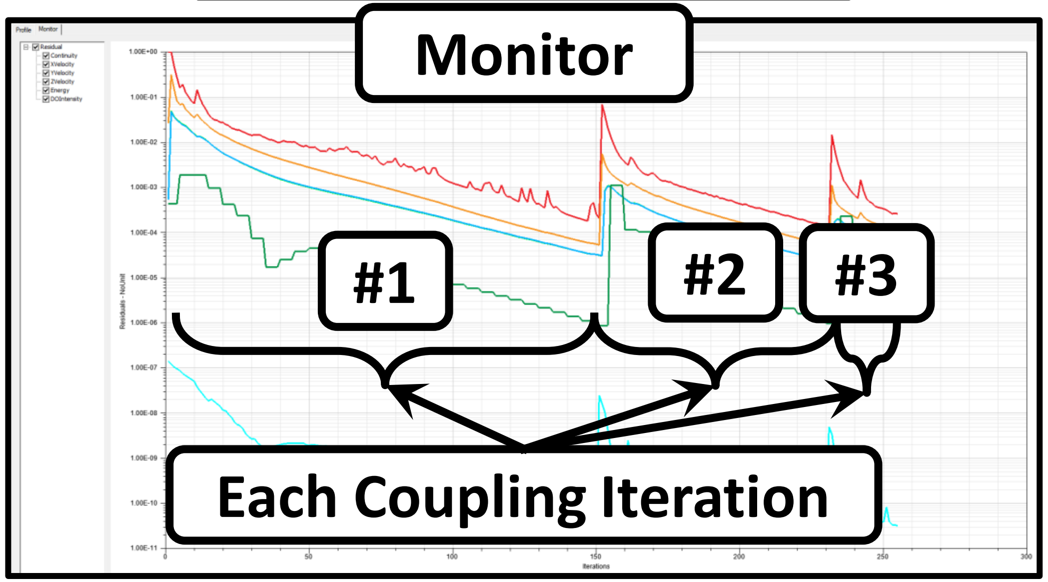


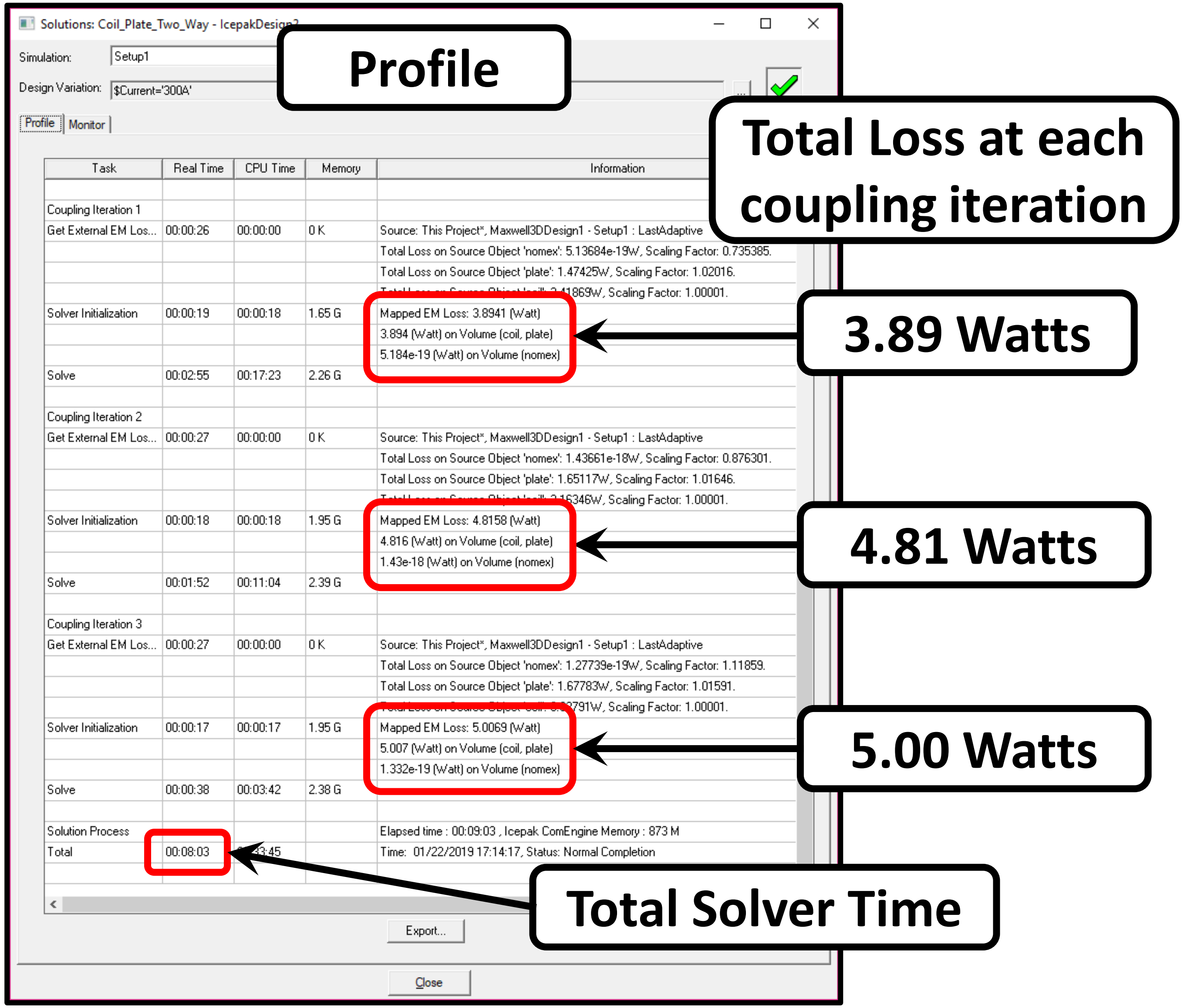


Ansys

Icepak – Eddy Current Setup – Solution Profile and Solution Monitors

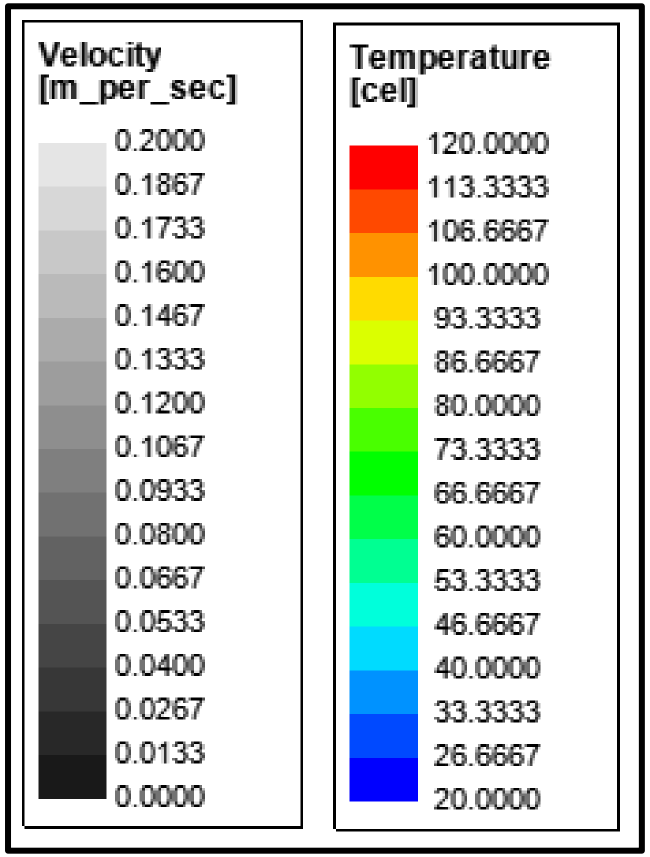


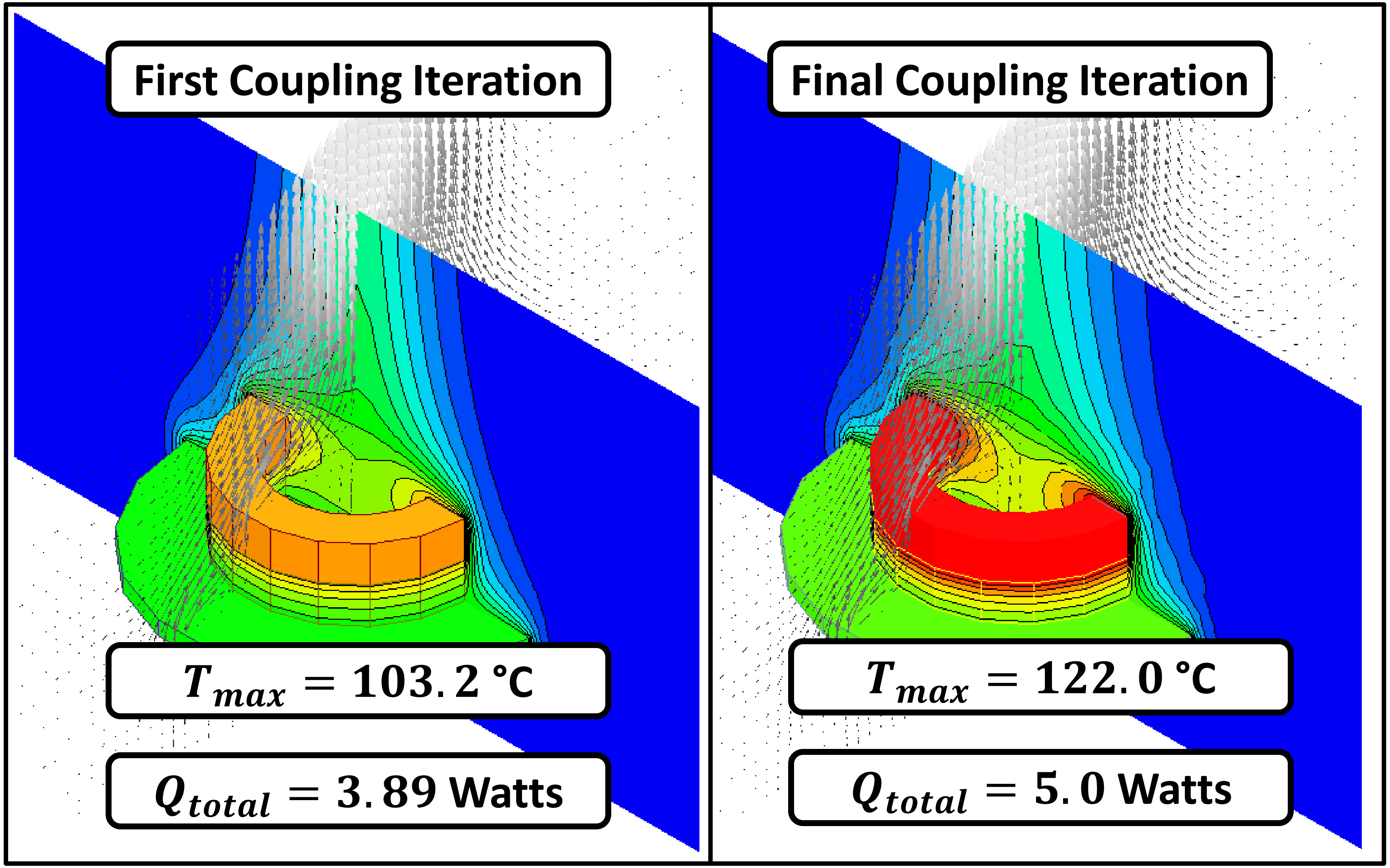




Ansys

Icepak – Eddy Current Setup – Results





Ansys

End of presentation