Introduction to Icepak

Introduction to ANSYS Icepak in Workbench

an ANSYS Tutorial

Click Here to Start Tutorial

Introduction to ANSYS Icepak in Workbench

Topics Covered in this tutorial
- An introduction to the new ANSYS Workbench window
- Adding systems to the Project Schematic
- Creating and solving a project in ANSYS Icepak
- Transferring results to the Static Structural analysis system

Click Here to Continue

Welcome to ANSYS Icepak 12.1!

Click Here to Continue

This tutorial is an introduction to ANSYS Icepak in Workbench. Using the ANSYS Workbench framework, a project is created and solved in ANSYS Icepak. The results are then transferred to Static Structural where further structural analysis can be performed.

This tutorial is intended for those who have used previous versions of ANSYS Icepak. We hope this tutorial will help you quickly get started using ANSYS Icepak in ANSYS Workbench.
Drag the **Geometry** system from the Toolbox and drop it onto the **Project Schematic**.
In this case, we will import a recently loaded geometry.
Drag and drop an Icepak system onto the Geometry cell in the Project Schematic.

Double-click the Setup cell to launch Icepak.
Create *Icepak* objects for all CAD objects in the model.

Place CPU and HEAT_SINK in an assembly. We will now convert this assembly into a non-conformal assembly.

Double-click assembly.1 to display the object panel.

Enable Mesh separately and enter Slack values to define the assembly bounds.

- Min X = 0.005
- Min Y = 0.0015
- Min Z = 0.001
- Max X = 0.005
- Max Y = 0
- Max Z = 0.005
Enter the values shown in the Mesh control panel.
Note: Mesher-HD is used due to CAD objects in the model.

Click Generate mesh.
Display mesh on the HEAT_SINK.

Notice that the mesh is non-conformal at the assembly bounds.
Click Basic parameters.

For Variables solved, enable Flow and Temperature.
Set Radiation to OFF.
Set Flow regime to Turbulent (Zero equation).
Introduction to Icepak

Cell states are up to date.

Save as ice_wb
Drag a Static Structural system onto the Icepak Solution cell to transfer the Icepak solution as a load for the Static Structural setup.
Double-click Setup to launch ANSYS Mechanical.
Select the geometry to which the (Icepak imported) Body Temperature will be scoped.
Select the Icepak bodies from which temperatures will be transferred. In this case, we will select All.
The Icepak solution is being imported into the Mechanical application and applied as a temperature load onto the scoped bodies.
Further structural analysis can be done, clicking the **Solve** button will initiate the solver.

---

**What have we learned?**

- How to work with your ANSYS Icepak projects in ANSYS Workbench
- How to add and connect systems to the Project Schematic
- How to create and solve a project in ANSYS Icepak
- How to transfer results to the Static Structural analysis system