

# ANSYS DISCOVERY FUNDAMENTALS



Learn how to use Ansys Discovery to evaluate flow, thermal, and structural performance during the design process using simulation workflows—so you can iterate faster, reduce late-stage surprises, and make better engineering decisions earlier.

## **SIMULATION WORKFLOWS**

Intro to Discovery Modeler, CAD interaction, geometry cleanup, volume extraction, named selections, and meshing basics (fidelity, size, and accuracy)

## **THERMAL & STRUCTURAL MODELING**

Structural analysis (stress, deformation, safety factor), fluid flow analysis (internal/external flows, pressure drop evaluations), thermal analysis (heat transfer modeling, cooling performance), and design tradeoffs across physics domains

## **DISCOVERY BEST PRACTICES**

When to use Discovery vs advanced solvers, simplifying geometry for speed and stability, early-stage, high-impact insights, over-modeling pitfalls

## **HANDS-ON APPLICATION**

Demos and workshops focused on structural and flow-based problem solving from start to finish, full CFD/FEA simulation case study, team collaboration, and open Q&A.

Available for quarterly open-enrollment or for company exclusive sessions.

*\*Content may vary depending on time required for basic topics to be completed*