CENTRAL TOOL ROOM & TRAINING CENTRE, BHUBANESWAR

ANSYS (Computational fluid dynamics)

Course Content Details

1. FUNDAMENTAL OF CFD
   - Fluid flow and heat transfer review
   - Navier-stokes equation
   - Introduction to FDM (finite difference method)
   - Introduction to FVM (finite volume method)

2. MODELING & MESHING USING ANSYS ICEM CFD-HEXA
   - GUI of ICEM CFD
   - Geometric modeling
   - Structured hexahedral meshing

3. CFD SIMULATION WITH ANSYS FLUENT
   - Solver basics
   - Fluid flow modeling (pressure & velocity)
   - Turbulence modeling
   - Heat transfer modeling

4. POST PROCESSING USING CFD-POST