



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