1: Computational Fluid Dynamics (CFD)

❖ **Course Outlines :-**

❖ **2D & 3D using DESIGN MODELER**

- Intro to CFD
- Intro to ANSYS workbench
- Intro to DM
- Launching DesignModeler
- Set up Working Units
- DesignModeler Interface
- Menu Toolbar
- Graphics Option Toolbar
- Selection Toolbar
- Details View
- Concept of Plane and Sketch
- How to create Planes and Sketches
- Sketching interface
- Concepts in Design Modeler
- Frozen, Active, Suppressed, Hidden
- How to create 3D geometry
- How to modify geometry
- Concepts in DM from meshing perspective
- Multi-body parts , Share topology
- Rest of 3D operation
- How to repair bodies

❖ **CFD Meshing**

- Intro to Theory of meshing
- Intro to ANSYS CFD Meshing
- Global Mesh Settings and Applications
- Application (Global Mesh Settings)
- Local mesh Settings
- Application
- Application (Global Mesh Settings)
- Mesh quality
- Local mesh Settings
- Application

❖ **CFD Fundamentals**

- Mathematical review
- Fluid mechanics review

❖ **Theory of CFD**

- Intro to governing equations
- Turbulence and its modeling
- Selection criteria
- LES and DNS

❖ **CFD Applications**

- CFD analysis for Fluid Flow and HeatTransfer in a pipe (2D &3D)
- CFD analysis for Transonic Flow Over a Airfoil
- CFD analysis for HVAC
- CFD analysis for heat exchangers
- CFD analysis for wind turbine
- CFD analysis for Fire fighting (one app)
- CFD analysis for water hammering
- User Defined Functions(UDFs)(introduction)