



Course Name: Introduction to ANSYS CFX
Duration: 4 days

ANSYS CFX is a leading software code which has been used to solve a wide variety of fluid flow related problems encountered across a vast range of industries.

This introductory course is intended for new and intermediate users and will provide attendees with an understanding of the basics and fundamentals of Computational Fluid Dynamics (CFD) using CFX within the ANSYS Workbench.

Specifically, the course covers essential CFD workflow steps, including geometry creation and editing, meshing methods and diagnostics, problem setup including physics and solver settings, results interpretation and post processing techniques.

The course format consists of a series of lectures followed by a set of comprehensive 'hands-on' workshops.

Course Agenda:

DAY 1

Introduction to ANSYS DesignModeler

- Introduction to ANSYS Workbench
- Introduction to DesignModeler
- Planes and Sketches
- Geometry Cleanup and Repair
- 3D Operations
- CAD Connections and Parametrization

DAY 2

Introduction to ANSYS Meshing Application

- Introduction to ANSYS Meshing
- Meshing Methods
- Global Mesh Settings
- Local Mesh Settings
- Mesh Quality Checking and Diagnostics

DAY 3:

Introduction to ANSYS CFX (Part 1)

- Introduction to CFD
- Overview of ANSYS CFX Interface
- CFX-Pre: Domains and Boundary Conditions
- CFX-Pre: Solver Settings
- CFD-Post: Results Post Processing

DAY 4:

Introduction to ANSYS CFX (Part 2)

- Interfaces, Sources and Additional Variables
- Transient Flows
- Turbulence Modelling
- Heat Transfer
- CFX Expression Language
- Moving Zones
- CFX Output Control and CFX Command Language
- Overview of Advanced Physics

User-Specific Problems or Additional Topics (if time permits)