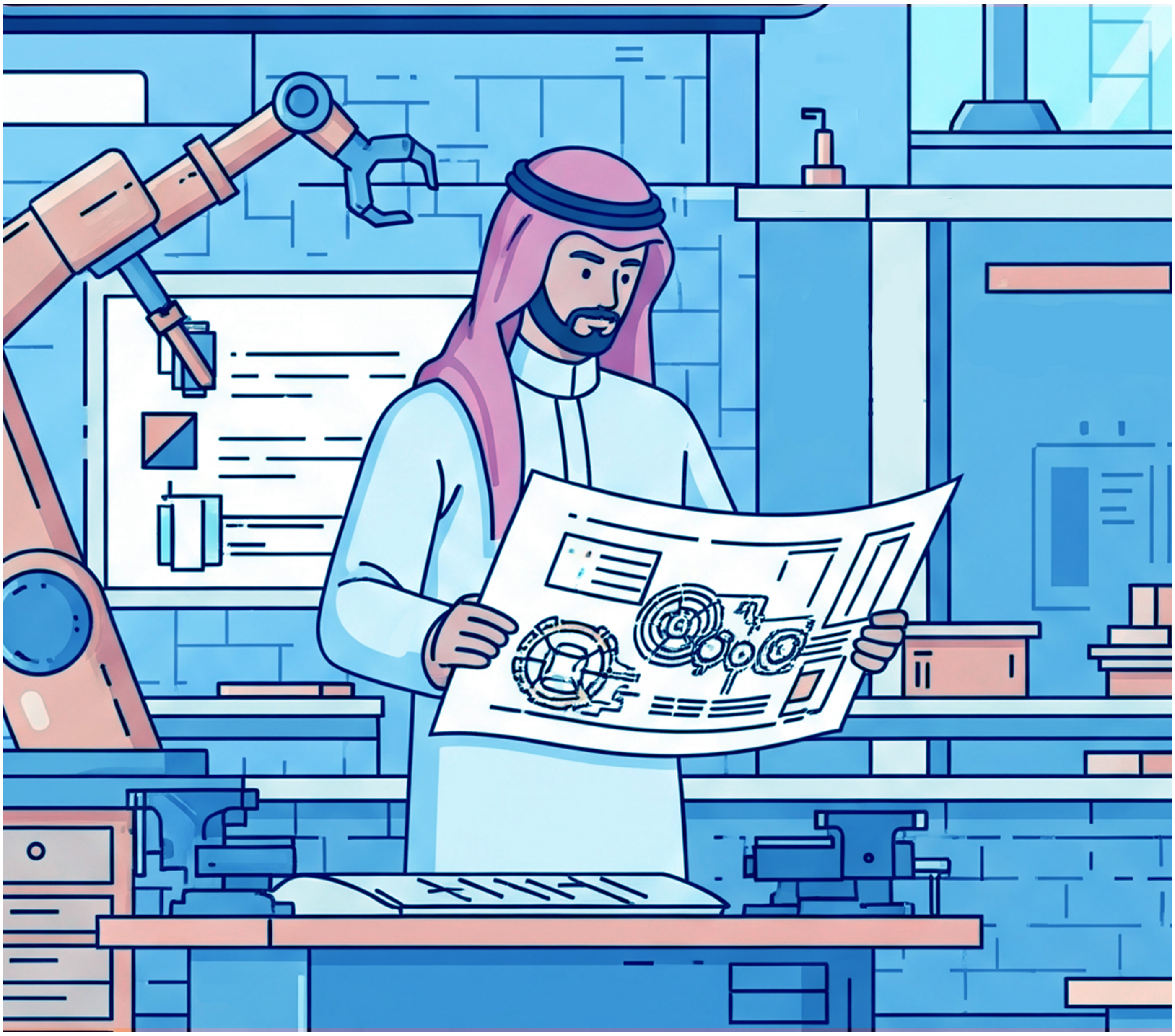

Mechanical Engineering

Advanced Computational Fluid Dynamics (CFD)

Course Introduction

Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flow as well as heat and mass transfer. The advent of high power desktop computers resulted in a number of commercial packages becoming available. The availability of commercial CFD packages helps industrial companies benefit from the powerful capabilities of CFD in designing and improving their products.

CFD enables engineers to simulate a new product allowing 'What if' scenarios to be tried out without the need for expensive prototypes and testing. The use of CFD helps design better and faster, as well as save money, meet increasingly stringent environmental regulations and ensure industry compliance. CFD analysis leads to shorter design cycles and consequently products get to market faster.

This training course covers the basic knowledge about CFD, geometry modeling, meshing and fluid domain extraction. It stipulates how to setup the boundary and initial conditions, how to solve and post-process the results.

Target Audience

- R&D Engineers – For learning basics and avoid committing costly mistakes in simulation.
- Designers (Not using CFD themselves) – To understand and appreciate the CFD output received and also to understand the limitations of CFD
- Techno-Managers – Who wish to start a CFD division in their organization and to get an idea of future CFD uses
- Consultants – Solving various CFD techniques & understanding special projects
- Academia– Those wishing to train themselves in CFD for research or jobs

Learning Objectives

- Fundamentals of Computational Fluid Dynamics (CFD)
- How to build geometry and computational domain
- How to build computational mesh
- How to apply boundary and initial conditions
- How to prepare and solve the simulation
- How to post process and interpret the output data
- Latest techniques in CFD and their possible applications in their areas of interest

Course Outline

- **01 DAY ONE**

  Module (1) Basics of Fluid Mechanics and heat transfer

  - 1.1 Governing equations
  - 1.2 Types of fluid flow
  - 1.3 Industrial applications involving fluid mechanics
  - 1.4 Heat transfer mechanisms
- **02 DAY TWO**

  Module (2) Fundamentals of Computational Fluid Dynamics (CFD)

  - 2.1 Problem formulation
  - 2.2 Basic elements of CFD simulation
  - 2.3 Computational domain and simulation types
  - 2.4 Boundary and initial conditions
  - 2.5 CFD limitations
- **03 DAY THREE**

  Module (3) ANSYS-CFX software

- 3.1 Software elements and interfaces
- 3.2 Design Modular (Geometry generation)
- 3.3 Building geometry elements
- 3.4 Building geometry (Hands on session)

- **04 DAY FOUR**

Module (4) Mesh generation

- 4.1 Mesh generation module interface
- 4.2 Different types of mesh
- 4.3 Boundary layer mesh generation
- 4.4 Mesh generation (Hands on session)

- **05 DAY FIVE**

Module (5) Problem setup and solution

- 5.1 Boundary and initial conditions
- 5.2 Solver options
- 5.3 Solution an data post-processing
- 5.4 Results interpretation and validation

Confirmed Sessions

FROM	TO	DURATION	FEES	LOCATION
April 14, 2025	April 18, 2025	5 days	2150.00 $	Virtual - Online
Aug. 25, 2025	Aug. 29, 2025	5 days	5950.00 $	switzerland - Geneva
Dec. 22, 2025	Dec. 26, 2025	5 days	4250.00 $	UAE - Dubai