Course Description:

The course is designed for Students with no or little CFD knowledge who want to learn CFD application to solve engineering problems. The course will provide a general perspective to the CFD and its application to fluid flow and heat transfer and it will teach the use of a popular CFD packages and provides them with the necessary tool to use CFD in specific applications.

Ansys software will be the commercial package that will be used in this course. Ansys Fluent is the most common commercial CFD code available and most of the engineering companies use this code for their research & development and product analysis.

Course Objectives:

Students will be familiar with the following at the end of this class:

- Basics of computational fluid dynamics
- General CFD simulation process
- Various solution methods and their suitability for different engineering applications
- Accuracy & Convergence
- CFD Simulation of:
  - Basic fluid flows
  - 2D\3D modeling
  - Heat transfer modeling
  - Turbulent modeling
  - Multiphase flows

Course Outline:

Week 1

- Basic CFD
  - Introduction
    - History & background
    - What is CFD
    - Real world Application examples
  - Governing equations –NS

Week 2

- Discretization Methods - Review
  - Finite volume
- Solution methods
  - Overview
  - Pressure based solver
  - Solution limits

Week 3

- Geometrical Modeling:
  - Basics & Intro
  - Boundary conditions in geometrical modeling
- Optimum geometry\\Negative volume

Week 4

- Mesh Generation
  - Basics & Intro

Solution independency – meshing & discretization considerations

Week 5

- Boundary conditions, their physical meaning and how to identify and apply them
  - Wall conditions
  - Inlet/outlet
  - Pressure

Week 6

- Application based geometry selection: 2D, 3D, symmetric or axi-symmetric?
- Geometry selection & simplification:

- Accuracy vs. Efficiency

Week 7

- Modeling basic flows
  - Steady
  - Viscous laminar flow

Week 8

- Solution Accuracy & Convergence
  - Convergence criteria
  - Initialization
- Boundary condition based local refinements

Week 9

- Solution validation & verification
  - Definition
  - Quality metrics
  - Application specific methods

Week 10

- Unsteady flow simulation
  - Initial conditions
- Example: **Unsteady flow past a cylinder**

Week 11

- Heat Transfer
  - Introduction
  - Energy equation

- Solution strategies – Review

Week 12

- Conduction,
- Convection
  - Natural convection

Week 13

- Overview of other topics in applied CFD:
  - Turbulence modeling
    - κ-ε model
    - Turbulence boundary conditions
  - Multiphase flows