EN608 – Computational Fluid Dynamics (CFD) Laboratory

CFD Laboratory Description

This Course will provide core knowledge of the fundamentals of CFD for engineers, and an introduction to the methods and analysis techniques used in CFD. It also provides an introduction to the use of commercial CFD codes to analyzeInternal and External flow heat transfer, Multiphase and Combustion problems of practical engineering interest.

Objective:

- To introduce student to applied computational fluid dynamics and to teach them how to solve a fluid flow problem using commercially available CFDsoftware
- Equip the Participant with the Computational Fluid Dynamics Fundamentals.
- Enable the student formulate the design problems into CFD/FEA.

Laboratory Outcome:

After successfully completing this laboratorystudent will be able to:

- Have a working knowledge of a variety of computational techniques that can be used for solving engineering problems
- To develop an understanding for the major theories, approaches and methodologies used in CFD;
- To build up the skills in the actual implementation of CFD methods (e.g. boundary conditions, turbulence modelling etc.) in using commercial CFD codes;
- To gain experience in the application of CFD analysis to real engineering designs.
- Proficiency in engineering design

List of Exercises:

- Geometric Creation – Session01
- Geometric Creation – Session02
- Fluid Flow inside a bend pipe
- Interpreting the results
- Case Study
- Governing Equations / Mathematical Models
- Flow inside a cyclone / scrubber
- External flow over a airfoil

- External flow over a 2D/3D car
- Case Study
- Turbulent models
- Pressure drop analysis in a valve
- Heat Transfer mechanism
- Heat transfer analysis in heat exchanger
- Heat transfer analysis in solar flat plate collector
- Heat transfer analysis in solar air heater
- Multi-phase flow / species
- Multi-phase fluidization
- Multi-phase cyclone
- Species (Combustion experiment)