28/01/2025

Computational Fluid Dynamics (CFD) Short Course

Purpose

The purpose of this short course is to;

1. equip learners with fundamental knowledge for using CFD in engineering applications
2. train learners on technical approach and CFD process in analyzing fluid flow phenomena
3. equip learners with knowledge of different computational methods necessary in analyzing fluid flow and heat transfer
4. enable learners appreciate the knowledge in analyzing the fluid solid interaction of engineering systems

Learning outcomes:

At the end of the course the learner should be able to:

1. explain the governing equations for fluid flow and how to solve them computationally
2. understand CFD simulation process, including physical model creation, mesh generation, numerical discretization schemes and solver methods in analyzing fluid flow
3. develop, and analyze successful simulations involving fluid flows, heat transfer and chemical reaction
4. interpret, discuss and draw conclusion from obtained simulation results

Course description

- Introduction to basics of CFD and application, types of flows, internal and external flow modeling
- Fluid flow governing equations (continuity, momentum, energy and species conservation) in Cartesian and cylindrical coordinates
- Introduction to turbulence; turbulent flows and turbulence modeling methods
- Introduction to computational methods; the finite volume method, finite deference method, and finite element method
- Solution techniques for discretized equations
- **CFD modelling** of **fluid flow**, **heat transfer** and **chemical reaction** phenomena

Reference text books:

1. Versteeg H.K, Malalasekera W. (1995) An introduction to Computational Fluid Dynamics, Prentice.

2. Anderson J.D. (1995) Computational Fluid Dynamics, MCGraw Hill.

Teaching methodology:

- **3 hours' practice per week** [lecture + practical]; for **four** weeks

Requirement:

- PC with ANSYS simulation software

PRACTICAL EXERCISES

Internal flow

1. Practical Exercise **#1: Entry flow in a pipe**
2. Practical Exercise **#2: Fluid flow and heat transfer in a mixing elbow**
3. Practical Exercise **#3: Fluid flow in harbor**
4. Practical Exercise **#4: Hydraulic jump/jet**

External flow (Aerodynamics)

5. Practical Exercise **#5: Flow round a sphere**
6. Practical Exercise **#6: Flow round an aerofoil**

Heat transfer

7. Practical Exercise **#7: Cooling (Electronic devices)**

Combustion simulations

8. Practical Exercise **#8: Species transport and gaseous combustion**

Discrete particle simulations

9. Practical Exercise **#9: Spray simulation using discrete particle model (DPM)**

Optimization

10. Practical Exercise **#10: Parametric study in ANSYS Fluent/** Optimization

Registration Link: https://forms.gle/YQSCzracb2rh13cv5