Computational Fluid Dynamics (6018-M) 5 ECTS

Lecturer: **B. Šarler**

Lectures: 30 h | Tutorials: 6 h | Labs: 16 h | Project: 8 h | Lang. :

Objectives

To acquaint students with the basic principles of computer programs for fluid flow simulation (liquids and gases), accompanied by various other phenomena (heat transfer, species transfer, solid mechanics, electromagnetic field), based on appropriate partial differential equations (PDEs).
Teach students the in-depth use of at least one computational fluid dynamics (CFD) simulation system through practical hands-on training in a well-equipped CFD lab.
To inspire students for CFD and encourage them to study the presented fundamentals in more detail.

Programme

1. Overview of the governing equations describing different types of fluid flow.
2. Essential elements and historical developments of CFD.
3. Space and time discretisation of transport phenomena fields.
4. Elaboration of the finite volume method for CFD.
5. Industrial CFD examples from the fields of materials processing, aerodynamics, biomedicine, environmental sciences etc.

Prerequisites

To achieve the objectives successfully, the students have to possess:
- Knowledge of the basics of thermofluid sciences (thermodynamics, heat transfer and fluid flow).
- Knowledge of the fundamentals of numerical methods.

Learning outcomes

After attending this course, the student will:
- Understand the basic principles and structure of CFD simulation systems.
- Understand discretisation problems: consistency, stability, convergence, discretisation order, numerical diffusion etc.
- To be aware of the peculiarities when solving the fluid flow equations and the problems regarding numerical modelling of turbulent flow.
- To be able to choose the appropriate formulation and numerical approach for the given physical problem and to assess the correctness of the obtained results.
- To be equipped with the practical skill of solving a spectrum of problems by a CFD simulation system.

Assessment

Theory - from lectures and exercise problems (50 %); Individual/group work at exercises (25 %); Project with a CFD code (ANSYS FLUENT and/or OpenFOAM) (25 %)

Literature

1. F. Moukalled, L. Mangani, M. Darwish, The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM and Matlab, Springer Verlag, Cham, 2016.
2. H. K. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics, The Finite Volume Method, 2nd Edition, Pearson, Harlow, 2007.
3. G.R. Liu, Mesh Free Methods: Moving Beyond Finite Element Method, CRC Press, Boca Raton, 2nd Edition, 2009.