Computational Fluid Dynamics (CFD) for energy technologies

Course Nr.:	2130910
Course Title:	Computational Fluid Dynamics (CFD) for energy technologies
Course Credits:	4
Semester Start Date:	April 25 2019
Class Schedule:	Th. 9:45-11:15
Classroom :	Building 20.30; Room SR -1.012 (UG)

Instructor: Dr. Ivan Otic

Email: ivan.otic@kit.edu

Website: http://www.ifrt.kit.edu/english/21_270.php

Office Location: Geb. 07.08, Vincenz-Prießnitz-Straße 3, Room 327

Office Hours: Th 14:00-15:00 and by appointment

Course Description:

The course is aimed of giving the fundamental of Computational Fluid Dynamics (CFD) for energy technologies. Computational techniques for solving Navier-Stokes and Energy equations with emphasis on turbulent heat and mass transfer are introduced. Finite volume method and solution of systems of linear algebraic equations are discussed. Error control, accuracy and stability are discussed and demonstrated. Reynolds-Averaged-Navier-Stokes (RANS) equations and computation of turbulent flows are discussed and demonstrated. Explicit vs. implicit time stepping methods. The course consists of both, a theoretical and a practical component. The former will deal with the derivations and properties of the methods and models for CFD. The practical part will make use of open source CFD computer program OpenFOAM to give a "hands on" insight into the simulation of turbulent flows.

Prerequisites:

Undergraduate numerical analysis. Graduate-level fluid mechanics. Basic computer skills (Linux, C++)

Course Objectives:

At the completion of this course, students

- are able to understand fundamentals of non-linear partial differential equations.
- get working knowledge of computational techniques that can be used for solving engineering heat and mass transfer problems
- are able to understand fundamentals of statistical fluid mechanics and to derive RANS transport equations
- have learned how to computationally solve turbulent heat and mass transfer problems using OpenFOAM software
- are able to present their results in form of technical report.

CFD Project:

- Part of this class is performing CFD simulations of turbulent heat and mass transfer using open-source CFD software OpenFOAM.
- After CFD analysis is completed students have to write a technical report.
- Projects are to be performed individually or in teams of two but every student writes his own report.
- The CFD analysis technical report is part of the final examination.

Reference texts:

- Lecture Notes (Presentation slides)
- Project tutorial
- Simulation files required to perform the project
- Report example

Lecture and lecture material are in english.

Grading:

The final grade is based on performance in:

Project	30%
Final Exam	70%

Recommended Books:

- *An Introduction to Computational Fluid Dynamics: The Finite Volume Method, H. Versteeg and W. Malalasekra, 2007.*

- *Computational Methods for Fluid Dynamics, J.H. Fereziger and M. Peric, 2002.*

Course Policies:

1. **Project** Technical reports on student projects are expected to be submitted electronically by 12pm on the due date.

2. **Attendance and Absences** Class attendance is expected. Students are responsible for all missed work, regardless of the reason for absence. It is also the absentee's responsibility to get all missing notes or materials.

3. **Unauthorized/Excessive Assistance** The student may not get any unauthorized or excessive assistance in the preparation of any work.

4. **Authorship** The student must clearly establish authorship of a work. Referenced work must be clearly documented, cited, and attributed, regardless of media or distribution. Also for work licensed as public domain or Copyleft, (See: http://creativecommons.org/) the student must provide attribution of that work.

Tentative Course Outline:

The weekly coverage might change as it depends on the progress of the class.

	Content
1	• Introduction: What is Computational Fluid Dynamics?
2	• Governing Equations
3	• Numerical Methods: Introduction
4	• Numerical Methods: Finite Volume
5	• Numerical Methods: Solution of ordinary differential equations
6	• Numerical Methods: Convergence and numerical stability
7	• Turbulence and Turbulence Modelling
8	• Reynolds Averaged Navier-Stokes Simulation Approach
9	• Heat Transfer