Numerical simulation of industrial flows

Academic Year: (2023 / 2024) **Review date: 20-12-2022**

Department assigned to the subject: Thermal and Fluids Engineering Department

Coordinating teacher: COENEN , WILFRIED ROMAIN STEFAN

Type: Electives ECTS Credits : 3.0

Year : 4 Semester : 2

REQUIREMENTS (SUBJECTS THAT ARE ASSUMED TO BE KNOWN)

- Engineering Fluid Mechanics

OBJECTIVES

Upon successful completion of this course, students will:

1. have a systematic understanding of the key aspects and concepts of computational fluid mechanics;
2. have a coherent knowledge of numerical simulation of industrial fluid flow, including novel aspects that lie at the forefront of the current mechanical engineering state-of-the art;
3. have the ability to apply their knowledge and understanding to identify, formulate, and solve problems of numerical simulation of industrial fluid flow using established methods;
4. have the ability to select and apply different methods of numerical discretization relevant to computational fluid mechanics;
5. have the ability to design and set up numerical simulations of realistic industrial flow problems, interpret the data, and draw conclusions;
6. have the ability to select and use the most appropiate computational fluid mechanics software and hardware to solve a certain industrial fluid flow problem;
7. have the ability to combine theoretical knowledge of fluid mechanics with computational methods to solve industrial fluid flow problems;
8. have a thorough understanding of the range of applicability and the limitations of the computational techniques used for the numerical simulation of industrial fluid flows.

DESCRIPTION OF CONTENTS: PROGRAMME

- Introduction to CFD.
- Basic equations of fluid mechanics. Levels of approximation.
- Finite Volume method.
- Turbulence modeling.
- Applications: use of a commercial code (ANSYS FLUENT) to solve a real-life industrial problem.

LEARNING ACTIVITIES AND METHODOLOGY

The development of the course includes lectures where the theoretical concepts are exposed, combined with practical application classes in a computer room.
Students will also work on a final project.

ASSESSMENT SYSTEM

- Mid-term theory exam (50%)
- Attendance and participation in class (15%)
- Implementation of a problem of industrial interest in a general purpose commercial CFD code (35%)

% end-of-term-examination:	0
% of continuous assessment (assigments, laboratory, practicals…):	100

BASIC BIBLIOGRAPHY

- Ansys Ansys user manual, Ansys.
- Jiyuan Tu, Guan Heng Yeoh y Chaoquin Liu. Computational Fluid Dymamics. A practical approach., Elsevier, 2008

ADDITIONAL BIBLIOGRAPHY

- Hirsch, C. Numerical Computation of Internal and External Flows (Second Edition) , Elsevier, 2007