

Course guide

240NU217 - 240NU217 - Computational Fluid-Dynamics Applied to Nuclear Technology

Last modified: 13/06/2023

Unit in charge: Barcelona School of Industrial Engineering
Teaching unit: 748 - FIS - Department of Physics.

Degree: MASTER'S DEGREE IN NUCLEAR ENGINEERING (Syllabus 2012). (Optional subject).
MASTER'S DEGREE IN INDUSTRIAL ENGINEERING (Syllabus 2014). (Optional subject).

Academic year: 2023 **ECTS Credits:** 4.5 **Languages:** English

LECTURER

Coordinating lecturer: Mas De Les Valls Ortiz, Elisabet

Others:

PRIOR SKILLS

Basic fluid dynamic concepts (a brief summary will be provided at the beginning of the course)

DEGREE COMPETENCES TO WHICH THE SUBJECT CONTRIBUTES

Specific:

CEN10. Ability to use effectively, understand the operation and validity ranges, and interpret the results of thermal-hydraulic codes and fluid dynamic calculation.

Generical:

CGN3. Ability to lead, plan and monitor multidisciplinary teams.

CGN2. Ability to design, calculate and design processes, equipment, facilities and plants related to the procurement of nuclear energy and the use of ionizing radiation.

CGN1. Have adequate knowledge of mathematical aspects, analytical, scientific, instrumental, technological and management.

CGN4. Ability to conduct research, development and innovation in relation to nuclear technology.

Transversal:

CT5. FOREIGN LANGUAGE: Achieving a level of spoken and written proficiency in a foreign language, preferably English, that meets the needs of the profession and the labour market.

CT2. SUSTAINABILITY AND SOCIAL COMMITMENT: Being aware of and understanding the complexity of the economic and social phenomena typical of a welfare society, and being able to relate social welfare to globalisation and sustainability and to use technique, technology, economics and sustainability in a balanced and compatible manner.

CT3. TEAMWORK: Being able to work in an interdisciplinary team, whether as a member or as a leader, with the aim of contributing to projects pragmatically and responsibly and making commitments in view of the resources that are available.

CT4. EFFECTIVE USE OF INFORMATION RESOURCES: Managing the acquisition, structuring, analysis and display of data and information in the chosen area of specialisation and critically assessing the results obtained.

04 COE. EFFICIENT ORAL AND WRITTEN COMMUNICATION. Communicating verbally and in writing about learning outcomes, thought-building and decision-making. Taking part in debates about issues related to the own field of specialization.

07 AAT. SELF-DIRECTED LEARNING. Detecting gaps in one's knowledge and overcoming them through critical self-appraisal. Choosing the best path for broadening one's knowledge.

Basic:

CB7. (ENG) Que els estudiants sàpiguen aplicar els coneixements adquirits i la seva capacitat de resolució de problemes en entorns nous o poc coneguts dintre de contextos més amplis (o multidisciplinars) relacionats amb la seva àrea d'estudi.

CB10. (ENG) Que els estudiants poseixin les habilitats d'aprenentatge que els permetin continuar estudiant d'una manera d'una forma que haurà de ser en gran mesura autodirigit o autònom.

CB9. (ENG) Que els estudiants sàpiguen comunicar les seves conclusions i coneixements (i darrers raonaments que els sustentin), a públics especialitzats i no especialitzats de manera clara i sense ambigüitats.

TEACHING METHODOLOGY

The computational Fluid Dynamics (CFD) course is designed so as the students acquire the CFD fundamentals that enable them to accurately use an advanced CFD tool to solve real problems of their interest. To reach so, both theoretical and practical sessions are provided.

Two main parts are defined: the fundamentals study and the applied CFD project. In both parts, a PBL methodology is used.

In the fundamentals part the energy equation is discretized and numerically solved for a simple 1D scenario. A basic code is delivered by the teacher and students implement some aspects and study the code performance under some scenarios. A final research paper is written.

In the applied CFD project, the students choose the project's subject. A state of the art is built by the whole team by the reading of reference research papers and the analysis of the involved research groups. Step-by-step, the case of study is set up and simulations carried out in teams. Combining all teams' results, a parametric study is carried out what allows a deeper insight in the studied phenomenon.

Weekly work is encouraged by small deliverables that support the learnings and become the starting point of the following session.

At the end of the course, students will be able to autonomously use OpenFOAM CFD toolkit and prepare research papers.

LEARNING OBJECTIVES OF THE SUBJECT

This course has the following objectives:

- 1.- Provide an overview of the state-of-the-art CFD code and studies applied to Nuclear Technology and, in a more generalised manner, in Engineering.
- 2.- Provide an overview of CFD methodology including the spatial discretization (i.e. FDM, FVM and FEM), the temporal discretization, the Navier-Stokes algorithm, the stability constraints and the associated errors.
- 3.- Provide the guidelines for an accurate CFD study: from mesh analysis to validation and verification steps.
- 4.- Present the numerical strategies to assess thermal-hydraulic phenomena of interest in nuclear technology as heat and mass transfer, turbulence and multi-phase flows. Being all of the relevant aspects in other Engineering fields.
- 5.- Increase the expertise in the usage of any basic numerical tool (i.e. Python, Matlab, Excel, ...) for the implementation of simplified unidimensional models.
- 6.- Show how to use the state-of-the-art literature on CFD studies to define simplified cases with the appropriate hypothesis.
- 7.- Offer knowledge on the main research groups and researchers within the field of study. Analyse and discuss aspects such as their leadership and gender distribution.
- 8.- Provide experience contextualized in a role-play of a professional engineering meeting where leadership and strategies are analysed from an ethical point of view.
- 9.- Provide the guidelines to efficiently use OpenFOAM CFD code and to analyze the obtained results.
- 10.- Raise awareness about all the potential cause of discrepancies between a CFD result and an experimental one.
- 11.- Qualify students for the preparation of scientific reports on CFD studies.

STUDY LOAD

Type	Hours	Percentage
Hours large group	36,0	88.89
Hours small group	4,5	11.11

Total learning time: 40.5 h

CONTENTS

Introduction

Description:

The course starts with the introduction of different approaches on the analysis of a system of study including a 0D balance, a 1D analysis using a system code, other quasi-3D approaches and, finally, the CFD approach. Within this presentation, the utility and limitations of a CFD study is exposed using real cases.

In parallel, the project carried out all over the course will be defined according to students' interests.

Specific objectives:

- Provide an overview to the students on the utility and limitations of the CFD studies applied to engineering.
- Define the course project.
- Familiarize the student with the CFD research in the field of the course project (subject, methodologies and social aspects of the research).

Related activities:

- 1) Definition of the working teams and the subject of the project.
- 2) Reading of some actual CFD papers related with the project's subject.
- 3) Build-up of a shared database with the most relevant information from the read papers.
- 4) Analyse the database including the technical CFD aspects and the social aspects such as the authorship and its gender distribution, the involved research groups and its relevance, the leaders of such research groups, etc.
- 5) Oral presentation of the main aspects of the papers and proposal of a case of study for the project, including the hypothesis, boundary conditions and main goals.

Full-or-part-time: 10h 30m

Theory classes: 3h

Practical classes: 3h

Guided activities: 3h 30m

Self study : 1h

Numerical Methods

Description:

In this module the fundamentals of the 3 main discretization methods of the transport equations are described: finite differences – FDM, finite volumes – FVM and finite elements – FEM. Its application is shown on the energy transport equation of an incompressible fluid under steady flow conditions. The advantages and disadvantages of the most relevant spatial numerical schemes and mesh effects are made explained and shown. All along this explanation, the student will partially implement some discretization in an already built 1D code provided by the teacher (either in Python, Matlab or Excel). The student will be asked to show the results of such implementation and, at the end of the module, will prepare a detailed CFD paper with all the information. In a second part of the module, the different strategies for the temporal discretization will be shown and discussed, with special focus on stability constraints (CFL).

Specific objectives:

- Indicate the main differences among the 3 methods (FDM, FVM and FEM).
- Increase the expertise of the students in the implementation of algorithms and their verification.
- Provide the required indications for the preparation of a research paper.

Related activities:

- 1) To partially implement the discretization of the unidimensional temperature equation using Python, Matlab or Excel.
- 2) To prepare a research paper with the development, the results and the analysis that show the acquisition of the learnings.
- 3) To identify a bug in the transient code provided by the teacher.

Full-or-part-time: 23h

Theory classes: 4h

Practical classes: 3h

Guided activities: 14h

Self study : 2h

Navier-Stokes algorithms

Description:

In this module, an overview of the SIMPLE and the PISO algorithms is provided, being such algorithms the basis for most of the algorithms used in CFD codes. Also, the fundamentals and the advantages and disadvantages of other algorithms (i.e. fully implicit and explicit ones) are provided.

Specific objectives:

- Provide an overview on the main Navier-Stokes algorithms.

Full-or-part-time: 4h

Theory classes: 3h

Self study : 1h

Validation and Verification

Description:

All CFD codes must go through a verification and validation (V&V) process before being applied as a feasible tool for engineering. In this module, a revision of the main V&V cases is provided. Also, it is shown how to realize a mesh analysis. The acquired knowledge will be applied to the project, using OpenFOAM toolkit.

Specific objectives:

- To provide an overview of the main V&V cases.
- To explain in detail how to realize a mesh analysis and its relevance.
- To introduce OpenFOAM's fundamentals and how to build a new case set up.
- To realize a mesh analysis of the project, using OpenFOAM.

Related activities:

- 1) To build-up the base case of the project including the geometry, the mesh, the physical properties and the boundary conditions.
- 2) To perform a mesh analysis of the base case.

Full-or-part-time: 16h

Theory classes: 3h

Practical classes: 2h

Guided activities: 10h

Self study : 1h

Heat transfer

Description:

In this module, the 3 main heat transfer modelling methods in fluids are shown: (1) passive scalar, (2) Boussinesq hypothesis and (3) compressible fluid. Both advantages and disadvantages are shown for each method, together with its range of validity. Its implementation in a Navier-Stokes algorithm is presented and some examples within OpenFOAM toolkit shown. Each method will be applied to the project using OpenFOAM.

Specific objectives:

- To provide a detailed knowledge of the different heat transfer modelling strategies in fluids.
- To show the OpenFOAM's algorithm with these heat transfer modelling.
- To use such OpenFOAM algorithms to study heat transfer phenomenon within the project.

Related activities:

- 1) To apply the passive scalar method to the project and discuss the results.
- 2) To apply the Boussinesq hypothesis to the project and discuss the results. Perform a comparative study.
- 3) To apply the compressible fluid model to the project and discuss the results. Perform a comparative study.

Full-or-part-time: 33h

Theory classes: 5h

Practical classes: 6h

Guided activities: 20h

Self study : 2h

Turbulence

Description:

In this module a description of the nature of a turbulent flow is provided, together with the challenge of its numerical modelling. The 3 main turbulence modelling strategies are described: (1) the Direct Numerical Simulation or DNS, (2) the Large Eddy Simulation or LES, and (3) the Reynolds Averaged Navier-Stokes or RANS. The particularities of each strategy are commented together with their mesh requirements. The Wall Function concept is introduced. The learned concepts are applied to the project.

Specific objectives:

- To provide with an overview of the main turbulence modelling strategies.
- To use OpenFOAM to study the effect of turbulence on the case of study of the project.

Related activities:

- 1) To simulate the case of study within the project using a low Reynolds – RANS model.
- 2) To perform a mesh study based on the y^+ .

Full-or-part-time: 15h

Theory classes: 2h

Practical classes: 3h

Guided activities: 9h

Self study : 1h

Other phenomena

Description:

The phenomenology involved in the Project can include multiphase flows, thermal radiation and/or fluid-solid interaction. According to the students' interests, some hours will be dedicated to provide basic knowledge on any of these phenomena and its modelling using CFD tools. Extra simulations including such phenomena will be performed in order to improve the project carried out all along the course.

The closing of the project will imply a final oral defence and role-play where leadership, strategy and ethical aspects will be analyzed

Specific objectives:

- To provide with basic knowledge on the numerical simulation using OpenFOAM of any other relevant phenomenon within the case of study of the subject's project.
- To train students to get a critical thinking that helps them to identify the limitations and the weak point in the realized projects.
- To train students on the oral defense of CFD projects.
- To train student to keep an ethical point of view in the definition of strategies and relationships within a professional engineering meeting.

Related activities:

- 1) Simulation of a phenomenon not included in the previous modules and related to the project.
- 2) Oral defense of the project.
- 3) Role-play.

Full-or-part-time: 10h 30m

Theory classes: 1h 30m

Practical classes: 2h

Guided activities: 6h

Self study : 1h

GRADING SYSTEM

Final qualification FQ will be calculated as $FQ = 0.6 CE + 0.4 FE$, being CE and FE the qualifications of the continuous evaluation and of the final exam, respectively.

The qualification of the continuous evaluation will be computed taking into account both the active participation in the in-person lessons and the relative weight of each assignment. The information relative to the assessment criteria for each task will be discussed during the lectures and uploaded in Atenea. The qualifications will be provided through Atenea.

The final exam qualification includes two indicators: the role-play and the written individual exam. In the role-play, the gained know how will be assessed in a simulation of a real scenario where both engineers and contractors are discussing the results obtained in the CFD project.

The delivery of all reports is a requirement to pass the course.

BIBLIOGRAPHY

Complementary:

- Holzmann, T. ResearchGate [on line]. 7th ed. Augsburg: Holzmann-CFD, 2015 [Consultation: 26/04/2023]. Available on: https://www.researchgate.net/publication/307546712_Mathematics_Numerics_Derivations_and_OpenFOAMR.
- Ferziger, Joel H. [et al.]. Computational Methods for Fluid Dynamics [on line]. 4th ed. Cham: Springer, 2019 [Consultation: 16/11/2022]. Available on: <https://link-springer-com.recursos.biblioteca.upc.edu/book/10.1007/978-3-319-99693-6>. ISBN 3540420746.
- Patankar, Suhas V. Numerical heat Transfer and Fluid Flow [on line]. Washington: Hemisphere, 1980 [Consultation: 16/11/2022]. Available on: <https://www-taylorfrancis-com.recursos.biblioteca.upc.edu/books/mono/10.1201/9781482234213/numerical-heat-transfer-fluid-flow-suhas-patankar>. ISBN 0891165223.
- Wilcox, David C. Turbulence Modeling for CFD. 3rd ed. La Cañada, Calif.: DCW Industries, 2006. ISBN 9781928729082.



RESOURCES

Other resources:

To correctly carry out all activities, a laptop with linux and OpenFOAM installed are necessary