Ph.D. program in Civil, Chemical and Environmental Engineering

Curriculum in Fluid Dynamics and Environmental Engineering

Academic year 2020/2021

1. Title of the course

Introduction to General CFD Using OpenFOAM Technology with an Overview of Turbulence Modeling and Multiphase Flows.

2. Contents

This course is aimed at those with little or no experience in the use of OpenFOAM or those who want to improve their skills using OpenFOAM as a general-purpose CFD and Multiphysics solver. During the lectures, the attendees will be introduced to all aspects of OpenFOAM as a general-purpose CFD solver, from structure and organization of the library to setting cases from scratch, to assessing the convergence, to post-processing and analyzing the results, to basic programming. Additionally, we will also address solid modeling and geometry preparation. During this course, we also briefly explore the following advanced physical models: turbulence modeling, multiphase flows, compressible flows and heat transfer, moving bodies, and source terms. Hands-on sessions will be organized, where the participants will have the opportunity to solve predefined cases or their cases with the assistance and guidance of the course instructors. To ease the learning process, a short introduction to the finite volume method will also be provided.

This course prepares the attendees to conduct end-to-end CFD studies using the numerical library OpenFOAM, that is,

- Solid modeling and geometry preparation.
- Meshing and mesh quality assessment.
- Case setup (including advanced physical models).
- Launching and monitoring the simulations.
- Qualitative and quantitative post-processing.
- Critical assessment of the results.

Nevertheless, the general knowledge acquired can be used with any other CFD solver.

3. Structure of the course

- Lectures and hands-on sessions to validate the acquired knowledge.

4. Instructors

- Joel Guerrero – Main instructor.
- Mattia Cavaiola – Teaching assistant.

5. Duration and credits

- This is a fully immersive course that lasts one week (40 hours).
- The teaching hours are from 9:00 to 17:00, plus half an hour of Q&A.
- Additionally, the student should dedicate at least 20 hours to prepare the final project.
- This course is worth four credits (4 CFU).

6. Period and registration procedure

- By sending an email to joel.guerrero@unige.it
- The course usually takes place during the month of June or July.

7. Deadline for registration

- One month before the beginning of the course.

8. Final exam

The final exam consists of writing a report addressing a CFD validation case or a case related to the student's research activity. The topic chosen must use advanced concepts and physical models explained during the lectures. The case to be developed must be agreed upon between the examiner and the student.

In the final report, the student must provide a detailed description of the problem, boundary conditions, initial conditions, solution method, convergence details, quantitative and qualitative analysis of the outcome, and a critical assessment of the results and sources of uncertainties. In addition, the case input files and steps to follow to replicate the results must be provided to the instructors for review.

Based on the instructors' feedback, the student will have a one-time opportunity to improve his/her report. After acceptance of the final report by the instructor, the student must deliver a short presentation. The course is graded on a pass/fail basis.

9. Additional notes

- This course is based on the latest version of OpenFOAM available when the course takes place.
- The course is based on the version released by the OpenFOAM Foundation (www.openfoam.org).
- However, the version released by OpenCFD (ESI) can also be used (www.openfoam.com).
- The solid modeling and geometry preparation session is delivered using Onshape.
    - A free account can be created at the following link: www.onshape.com.
- No prior knowledge of OpenFOAM®, C++, or Linux is required. However, a basic understanding of Linux is beneficial. Basic knowledge of CFD is also desirable.