PhD program in Civil, Chemical and Environmental Engineering

Curriculum in Fluid Dynamics and Environmental Engineering

Academic year 2019/2020

1. Title of the course

Introductory CFD course using Open Source software with an overview to numerical optimization

2. Contents

This course introduces the open source Multiphysics/CFD solver OpenFOAM® (Open Field Operation and Manipulation) and the numerical optimization toolkit Dakota (Design and Analysis toolkit for Optimization and Terascale Applications). The course is aimed to those with little or no experience in the use of OpenFOAM® and Dakota, and with a basic understanding of CFD, FVM, and numerical optimization. During the lectures, the attendees will be introduced to all aspects of OpenFOAM® as a general-purpose Multiphysics/CFD solver, from structure and organization of the library, to running cases, to basic programming in OpenFOAM®. The attendees will also be introduced to Dakota, from how to couple Dakota with OpenFOAM® (or any other black box solver), to design of experiments and optimization methods, to single and multi-objective optimization, to surrogate based optimization. Hands-on sessions will be organized, where the participants will have the opportunity to solve pre-defined cases (or their own cases) with the assistance and guidance of the instructors. Additionally, a few sessions will be dedicated to geometry generation, mesh generation, post-processing and visualization by using Open Source tools. To ease the learning process, a short introduction to the finite volume method (FVM), C++ programming language, and numerical optimization and design of experiments will also be provided. No prior knowledge of OpenFOAM®, Dakota, C++ or Linux is required. However, a basic knowledge of Linux is beneficial. A basic knowledge in CFD/FVM is also desirably. For hands-on examples, attendees can use the workstations specially configured for the course or if they want, attendees are free to bring their own computer with a working installation of Linux, OpenFOAM® (version 2.4.0 or 2.4.x), and Dakota (version 6.2).

3. Structure of the course

Lecture sessions and hands-on sessions.

4. Lecturers

Dr. Joel Guerrero.

5. Duration and credits

6 days (40 hours) spread in 3 consecutive weeks, plus at least 20 hours for the final project. 5 CFU.

6. Period and registration procedure

The course will take place with at least 5 students registered. Enroll by sending an e-mail to joel.guerrero@unige.it or vittori@diam.unige.it.
The course will take place in July 2020.

7. Deadline for registration

1 month before the beginning of the course.

8. Final exam

The course examination will consist of a final project to be delivered after the course. The final project will be made up of a tutorial/report that may be defined according to the student personal interest. The final outcome should be a detailed tutorial where the student must use advance OpenFOAM® or Dakota features, or it might be a validation case. The tutorial must be accompanied by a report explaining in detail the physics involved, discussion of the results, and the steps to follow to reproduce the case. The final tutorial/report will be reviewed by the course instructor, TAs, and the other students. Additionally, there will be some compulsory minor homework to be delivered during week 2 and week 3.