Course overview

Open-source software offers several advantages in terms of code customization and work in a collaborative environment, making it an ideal tool to be applied for research purposes.

The OpenFOAM® code is a consolidated tool that is extensively adopted for Computational Fluid Dynamics modeling (CFD) both in industry and academia. Thanks to the wide community of users and developers, the code includes the state of the art of numerical and physical models; also, its object-oriented structure allows to easily implement new libraries and models in it.

This course is thought for motivated PhD students, whose research project mainly involves application and development of CFD models. The main focus will be on:

- CFD: basics, turbulence, incompressible, compressible and multiphase flows.

- Object oriented programming techniques applied to CFD modeling;

- Pre and post-processing techniques.

Who should attend

PhD students and engineers working in the field of Computational Fluid Dynamics who are interested in applying and developing OpenFOAM in academic or industrial context. The course is open to partecipants from companies and research centers.
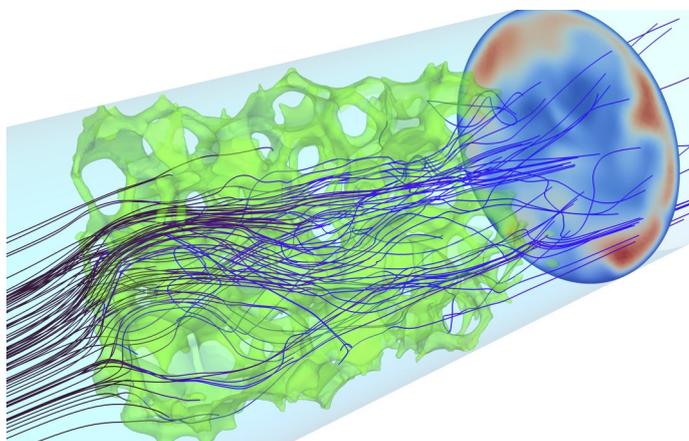
Program

Wednesday, 22th January 2020
Prof. Tommaso Lucchini

Overview of the OpenFOAM structure: case setup, mesh generation, post-processing. Hands-on session.

Thursday, 23th January 2020
Dr. Augusto Della Torre

Mesh generation in OpenFOAM: block-structured grids, use of open-source tools (`snappyHexMesh`, `cfMesh`), mesh conversion from Third Party software. Hands-on session.

Friday, 24th January 2020
Prof. T. Lucchini & Dr. A. Della Torre

Overview of OpenFOAM solvers for compressible flows, heat transfer, lagrangian tracking, reacting flows. Hands-on session.

Tuesday, 17th March 2020
Prof. Gavin Tabor (Exeter University)

Introduction to turbulence and its modeling. RANS, LES and DES turbulence modeling. Hands-on session.

Wednesday, 18th March 2020
Prof. Hrvoje Jasak (University of Zagreb)

Object oriented programming language applied to CFD modeling. Mesh management techniques: deformation, topological changes, generalized grid interface, immersed boundary.

Friday 19th March 2020
Prof. Holger Marschall (TU Darmstadt)

Executive summary of simulation methods for multiphase flows. High Resolution Schemes in OpenFOAM and specialisation to interface/shock capturing schemes. Hands-on session.

POLITECNICO
MILANO 1863

Computational Fluid Dynamics with Open-source Software

22-24 January 2020 / 17-19 March 2020

Course organization

The course is organized by Politecnico di Milano, involving also teachers from three other international universities: University of Exeter, University of Zagreb, Technical University of Darmstadt.

- Prof. Tommaso Lucchini (director)

- Dr. Augusto Della Torre (co-director)

- Prof. Gavin Tabor (University of Exeter)

- Prof. Hrvoje Jasak (University of Zagreb)

- Prof. Holger Marschall (TU Darmstadt)

The main course topics will be presented by slides and by running OpenFOAM on site. To maximize learning, participants are strongly encouraged to attend the course with a laptop where both Linux and OpenFOAM were already installed.

Teaching Material

Attendees will be provided with the following material:

- Slides of the lessons

- OpenFOAM cases and files presented during the course.

- USB Stick with Linux Live Distribution to be run with Virtual Box. OpenFOAM versions included.

Location

Lessons will be held at Bovisa Campus of Politecnico di Milano. Course participants will receive the detailed time table with lesson rooms two weeks before the start of the course.

The course is free for students of the Ph.D. school of Politecnico of Milano. The course is also open to all interested external candidates upon payment of a registration fee. Scholarships are available for external PhD students to cover the registration fee.
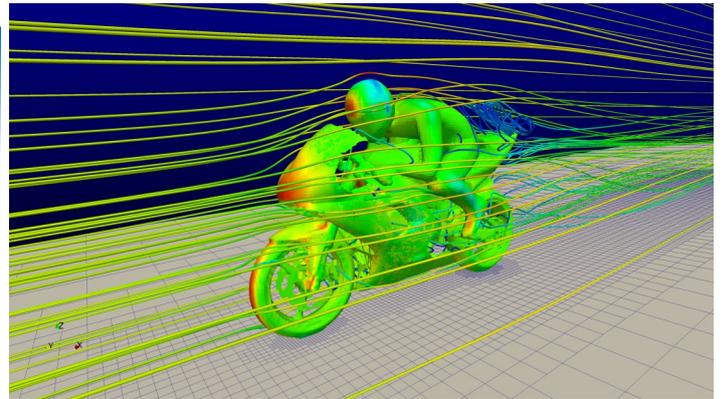
Registration for external candidates

To be enrolled, external candidates must register at the POLIMI website (www.polimi.it/onlineservices -> **"Register"**) and then they will find in their online services webpage the function **"Application for doctoral single courses"**. By selecting it, candidates will be asked to choose the course(s) of interest (up to 10 credits per year) and to upload a CV and a motivation letter.

Their request will be evaluated promptly. In case of positive evaluation (which depends also on the course slots availability), candidates will proceed with the registration and may start attending the course. After successful completion of the course training, the candidates will be able to download from the same website a certificate attesting the earned credits.

The registration fee amounts to 500,00 € . Selected categories (including Visiting PhDs, PhD candidates of Institutions with which agreements have been signed, and employees of companies which have signed a "Joint Research Center" agreement with POLIMI) will receive a fee waiver, and will be requested to pay only the cost of the stamp and the insurance: for the academic year 2018/2019 the total amount is 32.20€.

More information also available at this website: http://www.dottorato.polimi.it/en/during-your-phd/

Please contact Prof. Tommaso Lucchini and Dr. Augusto Della Torre if further information is needed.

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks

POLITECNICO
MILANO 1863