

## Basic Usage of OpenFOAM, 2 ECTS, 2022

Start date: 2<sup>nd</sup> May 2022

End date: 19<sup>th</sup> June 2022

Maximum number of participants: ~30

MAXIMUM NUMBER OF PARTICIPANTS HAS BEEN REACHED. YOU MIGHT BE PUT IN A WAITING LIST, OR REJECTED IMMEDIATELY.

### Syllabus:

- Purpose:
  - To learn the basic skills of using the [OpenFOAM open source CFD tool](#)
  - To learn efficient use of Linux (scripting), ParaView and plotting software.
- Learning outcomes:
  - The student will be able to efficiently set up, run, post-process, and validate CFD simulations in OpenFOAM
- Contents:
  - We go through the basic settings and structure of OpenFOAM cases.
  - We practice the use of OpenFOAM solvers, utilities and functionObjects.
  - We validate the results of academic test-cases.
  - We script the work processes to make them efficient and reproducible.
  - The examination is by hand-in of assignments.
- Prerequisites:
  - You are enrolled as a PhD student the entire duration of the course.
  - You should have a background in Fluid Dynamics, and ideally some CFD experience and/or a course in CFD. The present course does not teach the basics of Fluid Dynamics nor CFD. It is focusing on tools for CFD, assuming some background knowledge.
  - Linux skills are preferable, or will have to be gained by additional efforts at the start of the course.
- Preparations (before the start of the course):
  - Installation of required software (a specific OpenFOAM version, which may not necessarily be the latest one).
  - Get CID account and set up mail address in Canvas.
  - Make sure to have good Internet connection for the on-line gatherings.
  - Set aside time for the course! It is indeed a lot of work! See the requested hand-ins below, under Examination.
- Flipped-classroom arrangement:
  - Learning platform: <https://canvas.chalmers.se/> (log in with CID)
  - Approx. 13h recorded lectures, with distributed lecture notes.
  - Scheduled on-line discussion time (on the material in the lectures).
  - Discussion group for active sharing of student knowledge.
- Examination, with a sharp deadline:
  - Linux quiz
  - Basic usage quiz
  - Set-up, simulation, and validation of two academic cases. To be handed in perfectly cleaned up, and with scripts that can be used by us to check that everything has been correctly done. The scripts should work at first attempt, and it is thus important that it has been made sure that a change of user or installation directory does not break the script.

### Registration:

Registration starts in December the year before the course, and the course is usually full after February (but can happen earlier).

Information requested to show interest to register (send to [hakan.nilsson@chalmers.se](mailto:hakan.nilsson@chalmers.se)):

- Full name (first, middle, family)
- Birth date (YYYY-MM-DD, or Swedish personal number, or ideally CID if you know what that is – since that is what I need this for)
- Preferred email address
- University of PhD student enrolment
- Start and end dates of PhD studies (YYYY-MM)
- Name of supervisor
- Email address to supervisor
- A scan of a signed letter from supervisor, showing that you are a PhD student during the duration of the course, that she/he wants you to participate in the course “Basic Usage of OpenFOAM”, 2 ECTS, and that you will have time to fulfil the requirements of the course.

### Approximate time spent on the course, reported by student(s):

- Preparations: 3h
- Linux tests: 3h
- Lectures, discussions and quizzes: 12h
- Assignment: 40h
- Total: 58h (approx. 1.45 weeks full time)  
Highly dependent on student background!

### Certificate

- PhD students at Chalmers will have their result reported in Ladok, under course code [FMMS055](#)
- Other PhD students will get a signed certificate in scanned pdf (and sent by regular mail if requested). The certificate may contain statements of how the work in the course has progressed.

### FAQ

- “I already have basic OpenFOAM knowledge. Can I then skip this first course and only register to the second course?”
  - No, if you already have basic OpenFOAM knowledge you should easily complete this first course. In addition to making sure that all who pass this first course have enough knowledge to participate in the second course, I also want to see how you are working and that I will not get any requests for exceptions...
- “Can I hand in other assignments than the ones requested in the course, to prove that I already have basic OpenFOAM knowledge?”
  - No, we have to make this as efficient as possible for us. Therefore, all students have to hand in the same assignments. We are also building on those assignments in the second course.