Free-of-charge online course „Computational fluid dynamics with OpenFOAM“

The format of the course allows a very flexible participation!

The course consists of four parts, each extending over two days. During day1, participants receive videos of the lectures, slides and exercises. During day2, teachers are available at any time for a one-to-one discussion via Zoom meeting to solve issues related to exercises and answer questions. The course will be held in English.

Objective. The purpose of the course is to acquire the knowledge and practical skills necessary to use the software OpenFOAM.

You will learn to

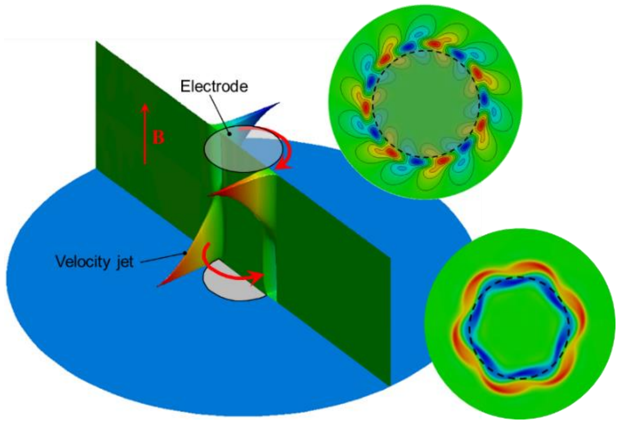
- set up and run cases in OpenFoam
  (create geometry and mesh, define boundary and initial conditions)
- use different mesh generation tools (structured and unstructured grids)
- simulate stationary, transient and turbulent flows
- analyze a log file (output file) and residuals to evaluate solution convergence
- post-process the solution: Paraview, extract solution data,…
- implement new equations and boundary conditions

Course organization. Participants undertake exercises using a virtual machine accessed via a ssh connection from participant’s PCs, applicable to all operating systems.

Target audience. The course is suitable for beginners and users seeking to broaden their basic knowledge of OpenFoam, particularly in programming, solving different types of flow problems, generating suitable meshes. No prior knowledge of OpenFOAM, C++ or Linux is required, but a basic knowledge of Linux and C++ is beneficial.

Please note: during day2 of each part of the course, a short individual discussion with participants is foreseen in order to discuss the progresses of the work. This is a requirement to get the final certificate and the full documentation of the course.

In order to register to the course please fill in the application form!
Send it to **dunja.rosanowitsch@kit.edu** and **chiara.mistrangelo@kit.edu**.



Schedule of the course

Mo 10.10	Tue 11.10 9^{30} - 14^{00}	Wed 12.10	Thu 13.10 9^{30} - 14^{00}	Fri 14.10

Introduction, what is OpenFoam, Linux commands
 	
Overview of OpenFOAM
Documentation, code structure, directory organization of a case (fundamentals of case structure)
 	
Finite volume method
 	
First OpenFOAM exercise
Lid-driven cavity
(setting a case, boundary and initial conditions, solver and control parameters)
 	
Check list for cavity
(what we learnt!)
	Zoom Meeting for questions and discussion	**Introduction to 3D laminar flow in a circular pipe**
		
		Meshing tools:
		○ **blockMesh + m4-script** Mesh generation with parameters
		○ **Snappy Hex Mesh** (SHM)
		
		Exercises
		• 3D flow in a circular pipe
		  • Study of various BCs
		  • Post-processing
		• Generation of geometry and mesh (pipe) with m4 parametrization
		• Example for SHM
		
		Check lists
		(what we learnt!)
			Zoom Meeting for questions and discussion	**Open issues/ questions can be sent per email**

Part1		Part2		
Day1-P1	Day2-P1	Day1-P2	Day2-P2	

Mo 17.10	Tue 18.10 9^{30} - 14^{00}	Wed 19.10	Thu 20.10 9^{30} - 14^{00}	Fri 21.10
Meshing tools: ○ Adaptive mesh refinement (AMR) ○ External tools and conversion to OF format **Kármán vortex street** (theory) **functionObjects** (e.g. time averaging, calculation of forces) **Exercises** Kármán vortex street (transient flow) ○ Structured mesh (blockMesh) ○ AMR → Comparison of results with two grid generation methods **Check list for Kármán vortex street** (what we learnt!)	**Zoom Meeting for questions and discussion**	**Programming** • Implementing the temperature equation in a solver • Implementing a new time-dependent boundary condition **Exercises** Application of new BC and solver with T-equation **Turbulent flow** (theory and exercises) **Summary & Best practice guidelines**	**Zoom Meeting for questions and discussion**	**Further questions can be sent per email**
Part3		Part4		
Day1-P3	Day2-P3	Day1-P4	Day2-P4	