Supplementary Course (EVA) at ZHAW School of Engineering

Title: Thermo Fluid Dynamic Model Development
Short Code: rEVA_OpenFOAM2

ECTS Credits	3
Profile	Energy & Environment (EnEn)
Responsible Institute /Centre	nstitute of Computational Physics (ICP)
Responsible lecturer and contact informtion	Prof. Dr. Gernot Boiger
Type and duration of examinations	Students prepare semester project; Exam: final presentation & Q&A of semester project; oral, 20 min per student.
Start date and duration	Semester: Spring Detail:
Location	Winterthur & active streaming
Course type	held in 4-5 blocks + 1 block final presentation of projects • Contact hours: 40 (hrs) • Guided self-study: 10 (hrs) • Independent self-study: 40 (hrs)
Language of instruction	English
Short description (max. 300 characters)	This EVA builds on EVA OpenFOAM 1: knowledge about basic thermo-, fluid-, dynamic simulation model application, development, extension and adaption within the CFD toolbox OpenFOAM is extended.
Contents and Learning Objectives	**Learning Objectives EVA OpenFOAM 2:** Pushing the limits of EVA OpenFoam1 further, at the end of EVAOpenFoam2, the student knows considerably more: • About the actual character of OpenFoam® in contrast to commercial CFD tools • How to apply OpenFoam® from meshing over pre-processing to post-processing (including the use of blockMesh, snappy hex Mesh, paraview, Matlab in combination) • The main features of OpenFoam® (e.g.: tutorial cases, solvers, utilities) • How to understand and/or find his/her way through the basic software structure (e.g.: Finding, using) • How to choose, modify, recompile and apply his/her first, self written OpenFoam® application (e.g.: solver, utility, boundary condition...)

Supplementary Course (EVA) at ZHAW School of Engineering

	• About numerical background about the main solution algorithms within OpenFoam (e.g.: PISO, SIMPLE loop). **Contents EVA OpenFOAM 2:** • Implementation of pre- and post- processing utilities • Basics of turbulence modelling • Applications: a.) icoFoam/cavity b.) Channel Flow c.) Karman – Eddies d) Heat Transfer & Radiation modelling e) Multi- Reference Frame (MRF) modelling (e.g.: Mixing, pump) g) Buoyant flow (Boussinesq-Approx.) • Introduction to "non-standard" OpenFoam® tools such as a) SWAK (=Swiss Army Knife) for FOAM to implement function based, flexible boundary conditions; b) Snappy Hex Mesh (Meshing Tool) • Chose, plan, modify/program, recompile, apply and verify your first own "boundary condition" • „Update an older solver“: We will try to update the ancient "icoLagrangianFoam" (OF version 1.6) to the latest OF version; The solver is about particle tracking of simple, spherical hard ball particles within a transient, laminar, incompressible flow. A simple feature like that does not exist anymore as a stand alone piece of code in OF… but can be very useful.			
Prerequisites	Basic knowledge of CFD; - Installed and working version of OpenFOAM; - Interest in thermo- fluid dynamic modelling			
Literature	OpenFoam® User guide: http://www.openfoam.org/docs/user/ OpenFoam® programmer's guide: http://www.foamcfd.org/Nabla/guides/ProgrammersGuide.html			
Special requirements	any installed version of OpenFOAM (e.g.: in virtual machine Linux Ubuntu) e.g. from openfoam.org			
Offer for profiles	Aviation (Avi)	☒	Business Engineering (BE)	☐
	Computer Science (CS)	☒	Data Science (DS)	☒
	Electrical Engineering (ElE)	☐	Energy & Environment (EnEn)	☒
	Mechanical Engineering (ME)	☒	Mechatronics & Automation (MA	☒
	Medical Engineering (Med)	☐	Photonics and Laser Engineering (Pho)	☒
	Information and Cyber Security (ICS)	☐	Civil Engineering (CE)	☐