8th OpenFOAM Conference | Agenda Oct. 13, 2020

Opening Keynote Panel - 10:45-12:035am CET

10:45-11:15a CET	**Welcome and Introduction from ESI Group**
11:15-12:00a CET	**Collaboration across the CFD community and continuing the V&V journey**, Althea de Souza, Chair of NAFEMS CFD working group
12:00-12:30p CET	**Scalable and optimized HPC in the Cloud for your OpenFOAM simulations,** Romain Klein, Rescale
12:30-12:35p CET	**A word from our Gold Sponsor – Upstream CFD – Dr. Charles Mockett**
	Networking Break

Industry Sessions – 01:00 pm – 14:20 pm CET

	Optimization	Multiphase I
01:00-01:20p CET	**An Adjoint-based Topology Optimization Framework for Fluid Mechanics and Conjugate Heat Transfer in OpenFOAM**, Vaggelis Papoutsis, NTUA	**CFD Modelling of Fuel-Air mixture formation in a GDI engine using OpenFOAM,** Andrea Pati, TU DARMSTADT
01:20-01:40p CET	**Real time visualization of parametrized turbulent flow,** Amine Ammar, ENSAM	**Development of an eularian solver for fluidized beds under deactivation conditions,** Aitor Atxutegi, UNIVERSITY OF THE BASQUE COUNTRY
01:40-02:00p CET	**Creating data-driven CFD workflows using OpenFOAM and PyTorch, André Weiner,** TU Braunschweig	**Industrial simulation of multiphase and moving body flow using OpenFOAM and Visual-CFD,** Nima Abbaspour, WAGNER
02:00-02:20p CET	**On the efficiency and robustness of the adjoint method: Applications in steady and unsteady shape optimization in fluid mechanics,** Themis Skamagkis, NTUA	**On the simulation of the Filling Stage of Thermoplastic Injection Molding using the Open-Source Solver openInjMoldSim,** Célio Fernandes, UNIVERSITY MINHO
02:20-02:40p CET	**Machine-learning based approach to global optimization and interactive design,** Matthias Bauer, NAVASTO	**Modelling the Extrusion Phase of Extrusion Blow Molding**, Miguel Nóbrega, UNIVERSITY MINHO
	Networking Break	
03:10-03:50p CET	**Technical Committees Panel Discussion: HPC, Marine, Nuclear, Turbulence, Multiphase**	
03:50-04:35p CET	**Architecting CFD for the Industrial Scale**, Sanjay Mathur, ESI Group	

	Transportation	Multiphase II
04:40-05:00p CET	**Numerical Investigation of JAXA High-Lift Configuration using OpenFOAM,** Baris Bicer, TURKISH AEROSPACE INDUSTRIES	**An injection head to generate a stable falling liquid lm within a circular duct,** Luis Alberto Borraz Jonapa, UNIVERSITY OF SCIENCE AND ARTS OF CHIAPAS
05:00-05:20p CET	**Modelling of Hood Fluttering due to Aerodynamic Forces,** Armando Perez Pena, ESI Group	**Modelling Gas and High-Viscous-Oil Slug Flow Regime to Estimate the Dispersed Phase Distribution Coefficient,** Victor Pugliese, TEXAS TECH UNIVERSITY
05:20-05:40p CET	**Automotive Cabin Thermal Comfort Analysis Using a Pseudo-transient Thermal-CFD Coupling Methodology Between TAITherm and OpenFOAM,** Denis Hinz, THERMOANALYTICS	**Analysis of different multiphase CFD models for aerated stirred bioreactors,** Stefan Seidel, ZHAW Zurich University of Applied Sciences
05:40-06:00p CET		**Numerical investigation on the microfluidic droplet coalescence under the influence of capillary-wettability interaction,** Rakesh Majumder, NATIONAL INSTITUTE OF TECHNOLOGY

Agenda subject to change. All times listed in Central European Time Zone (CET)

8th OpenFOAM Conference | Agenda Oct. 14, 2020

Opening Keynote Panel - 09:00-10:15 am CET

09:00-09:45a CET	**OpenFOAM: Governance and Recent Highlights**, Andrew Heather, Technical Director, Open CFD
09:45-10:15a CET	**Large-eddy simulations of airflow and aerosol transport on a London bus during the Covid-19 pandemic**, Prof. Thorsten Stoesser, University College London
10:15-10:20a CET	**A word from our Gold Sponsor – Amazon Web Services – Dr. Neil Ashton**
	Networking Break

Industry Sessions – 11:00 am – 03:40 pm CET

	Heat Transfer and Energy	Turbulence and Combustion I	High Performance Computing
11:00-11:20a CET	**Heat transfers in fixed beds made with wood chips**, Lionel Gamet, IFP ENERGIES NOUVELLES	**Turbulence modelling investigation for 3.5:1 prolate spheroid using adaptive-mesh refinement in OpenFOAM®,** Marian Fuchs, UPSTREAM CFD	**HPC Benchmarck Project: how to use and initial test-case(s),** Ivan Spisso, CINECA
11:20-11:40a	**Towards modeling of MHD effects on imploding liners in context of Magnetized Target Fusion approach,** Victoria Suponitsky, GENERALFUSION	**Simulation of a catalytically assisted burner using a simplified combustion model,** Henrik Rusche, WIKKI	**AmgX GPU Solver Developments for OpenFOAM,** Matt Martineau, NVIDIA
11:40-12:00a	**An overview on electrochemical simulation with OpenFOAM,** Norbert Weber, HELMHOLTZ-ZENTRUM DRESDEN	**Comprehensive model for blast furnace with two way coupling of raceway model using OpenFOAM,** Prakash Abhale, TATA STEEL	**A CPU-GPU paradigm to accelerate turbulent combustion and reactive-flow CFD simulations,** Federico Ghioldi, POLITECNICO MILANO
12:00-12:20p	**Wall-Modeled Large-Eddy Simulations of Airfoil Trailing Edge Noise,** Thomas Malkus, OHIO STATE UNIVERSITY	**Development of optimisation strategies to enhance the performance of NOx Postprocessor**, Senthilathiban Swaminathan, MONTAN UNIVERSITY LEOBEN	**The Effect of HDR InfiniBand and In-Network Computing on OpenFOAM Simulations,** Ophir Maor, HPC Council
12:20-12:40p	**External Core Catcher Cooling,** Samyak Darshan, COLLEGE OF ENGINEERING BENGALURU	**Lowering the obstacles for SMEs to adopt multi-physics biomass furnace simulations by providing a cloud-based solution,** Henrik Rusche, WIKKI	**GPU enabling of OpenFOAM by the use of PETSc4FOAM library,** Stefano Zampini, KAUST
	Networking Break		
01:40-02:20p	**Technical Committees Panel Discussion: Numerics, Documentation&Tutorial, Optimization - Special focus on OpenFOAM Journal**		

	Environment	Turbulence and Combustion II	Technology	High Performance Comp. II
02:20-02:40p CET	**New developments for numerical wave tanks for coastel and offshore applications,** Gabriel Barajas Ojeda, IH CANTABRIA	**Modeling large-scale thermoplastic fires,** Alex Krisman, FM GLOBAL	**Modeling hyperelastic solids in OpenFOAM,** Dr. József Nagy, EULERIAN SOLUTIONS	**Does OpenFOAM scale?** Mattjis Janssens, ESI
02:40-03:00p	**Modelling of H2O2 flotation for removing microplastics from waste water,** Emmanuel Thom, UNIVERSITY FREIBERG	**Dynamic Zone Flamelet Model-an efficient yet accurate turbulent combustion model implemented in OpenFOAM,** Dr. Wei Yao, CHINESE ACADEMY OF SCIENCES	**Three-Dimensional Simulation of Flow-Field around a Flapping Foil using Immersed Boundary Solvers of OpenFOAM,** Chandan Bose, UNI LIEGE	**Performance Evaluation of OpenFOAM on Juelich Supercomputing Facilities (JURECA, JUWELS and JUSUF),** Abouzar Ghasemi Forchungszentrum Juelich
03:00-03:20p	**High resolution urban air quality modeling using a multi-scale approach,** Rakesh Kadaverugu, CSIR	**Models for Turbulence and Thermodynamics in Simulating \\ Non-Premixed Combustion in a Cement Kiln,** Domenico Lahaye, TECHNICAL UNIVERSITY DELFT	**Nonlinear Response Analysis of a Chord-Wise Flexible Flapper in the Wake of a Bluff Body using OpenFOAM,** Rajanya Chatterjee, IIT MADRAS	
03:20-03:40p	**OpenFOAM computational performance:double vs mixed precision,** Federico Brogi, Istituto Nazionale di Geofisica e Vulcanologia		**Workflow Development for CFD Analysis on an Aerospace S-Duct,** Ishan Nande, Beta CAE	

	OpenFOAM Best Practices & Meshing	OpenFOAM Adjoint Optimization
9:00-12.00a CET	Aimed at users with experience in OpenFOAM, who wish to improve the robustness, speed and accuracy of their simulations with best practice settings validated by OpenCFD. We will discuss new performance improvements and developments recently released in v1912 and v2006 on numerics and physical modelling and present latest best practices and insights.	Aimed at users who want to perform shape optimization with OpenFOAM and the Adjoint technology released in OpenFOAM. Discussing the definition of the optimization processes, understanding the simulation requirements and outputs, and making engineering decisions supported by tangible results. We will also discuss new functionality introduced in the latest release. This workshop is led by developers of the method from NTUA (National Technical University of Athens).
	COVID-19	**Visual-CFD**
1:00-4.00p CET	The COVID-19 pandemic is still present! In the phase of returning to a new “normal” our everday activities are linked to the question of how to minimize the risk of infection and how to increase confidence in our safety. Scientific research unveiled that the pathogen can remain airborne and active in aerosol form for several hours. The analysis of infection outbreaks have shown that keeping 1,5 m distance to other people does not always protect from becoming infected. Interventions like wearing masks or costly structural measures like acryl glass barriers are not always necessary or effective. Validated CFD-tools, like OpenFOAM, allow us to simulate and visualize the dispersion of aerosols under various conditions. In turn, digital simulation enables us to assess the effectiveness of interventions. The opensource nature of OpenFOAM provides the unique framework to generate knowledge about the containment of the spread of the virus in a broad community with increasing speed.	Visual-CFD is an advanced user interface created for OpenFOAM and made available within ESI’s multi-domain simulation platform, Visual-Environment. With Visual-CFD, users can import CAD Models, clean them up, setup, solve and post-process an OpenFOAM case in a fully customizable environment. Its familiar CFD User Interface and terminology makes it an invaluable tool for beginners and experienced users alike. Visual-CFD is an advanced user interface created for OpenFOAM and made available within ESI’s multi-domain simulation platform, Visual-Environment. With Visual-CFD, users can import CAD Models, clean them up, setup, solve and post-process an OpenFOAM case in a fully customizable environment. Its familiar CFD User Interface and terminology makes it an invaluable tool for beginners and experienced users alike.