

















Opening Keynote Panel - 10:00-12:00 am CET

10:00-10:30a CET	Welcome and Introduction from ESI Group
10:30-11:15a CET	Architecting CFD for the Industrial Scale , Sanjay Mathur, 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
12:00-12:05p CET	A word from our Gold Sponsor – Upstream CFD – Dr. Charles Mockett
<i>Networking Break</i>	

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 	
<i>Networking Break</i>			
03:10-03:50p CET	PANEL DISCUSSION		
Transportation		Multiphase II	
03:50-04:10p CET	Modelling of Hood Fluttering due to Aerodynamic Forces , Armando Perez Pena, ESI Group 	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 	
04:10-04:30p CET	Numerical Investigation of JAXA High-Lift Configuration using OpenFOAM , Baris Bicer, TURKISH AEROSPACE INDUSTRIES 	Modelling Gas and High-Viscous-Oil Slug Flow Regime to Estimate the Dispersed Phase Distribution Coefficient , Victor Pugliese, TEXAS TECH UNIVERSITY 	
04:30-04:50p CET	Good Practices in OpenFOAM for Ground Transportation: from External Aerodynamics to Aeroacoustics , Roger Almenar, ESI	Analysis of different multiphase CFD models for aerated stirred bioreactors , Stefan Seidel, ZHAW Zurich University of Applied Sciences 	
05:10-05:30p CET	Automotive Cabin Thermal Comfort Analysis Using a Pseudo-transient Thermal-CFD Coupling Methodology Between TAItherm and OpenFOAM , Denis Hinz, THERMOANALYTICS 	Numerical investigation on the microfluidic droplet coalescence under the influence of capillary-wettability interaction , Rakesh Majumder, NATIONAL INSTITUTE OF TECHNOLOGY 	

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
<i>Networking Break</i>	

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 Benchmark 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 , Bernhard Peters, UNIVERSITY OF LUXEMBOURG		GPU enabling of OpenFOAM by the use of PETSc4FOAM library , Stefano Zampini, KAUST	

Networking Break

01:40-02:20p

PANEL DISCUSSION

Environment		Turbulence and Combustion II		Technology		High Performance Comp. II	
02:20-02:40p CET	New developments for numerical wave tanks for coastal 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		OpenFOAM does not scale , Mattijs 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 Forchungszenrum 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	<p>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.</p> <p>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.</p>	<p>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.</p> <p>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).</p>
COVID-19		Visual-CFD
1:00-4.00p CET	<p>The COVID-19 pandemic is still present! In the phase of returning to a new “normal” our everyday 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.</p> <p>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.</p>	<p>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.</p> <p>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.</p>