
















## 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	Keynote 2
<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	tbc	Development of an eularian solver for fluidized beds under deactivation conditions, Aitor Atxutegi, UNIVERSITY OF THE BASQUE COUNTRY 	
01:40-02:00p CET	Machine-learning based approach to global optimization and interactive design, Matthias Bauer, NAVASTO 	tbc	
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	Creating data-driven CFD workflows using OpenFOAM and PyTorch, André Weiner, TU BRAUNSCHWEIG 	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	tbc	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	<b>Collaboration across the CFD community and continuing the V&amp;V journey</b> , Althea de Souza, Chair of NAFEMS CFD working group
09:45-10:15a CET	<b>Large-eddy simulations of airflow and aerosol transport on a London bus during the Covid-19 pandemic</b> , Prof. Thorsten Stoesser, University College London
<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	<b>Heat transfers in fixed beds made with wood chips</b> , Lionel Gamet, IFP ENERGIES NOUVELLES 	<b>Turbulence modelling investigation for 3.5:1 prolate spheroid using adaptive-mesh refinement in OpenFOAM®</b> , Marian Fuchs, UPSTREAM CFD 	<b>HPC Benchmark Project: how to use and initial test-case(s)</b> , Ivan Spisso, CINECA 		
11:20-11:40a	<b>Towards modeling of MHD effects on imploding liners in context of Magnetized Target Fusion approach</b> , Victoria Suponitsky, GENERALFUSION 	<b>Simulation of a catalytically assisted burner using a simplified combustion model</b> , Henrik Rusche, WIKKI 	<b>AmgX GPU Solver Developments for OpenFOAM</b> , Matt Martineau, NVIDIA 		
11:40-12:00a	<b>An overview on electrochemical simulation with OpenFOAM</b> , Norbert Weber, HELMHOLTZ-ZENTRUM DRESDEN 	<b>Comprehensive model for blast furnace with two way coupling of raceway model using OpenFOAM</b> , Prakash Abhale, TATA STEEL 	<b>GPU enabling of OpenFOAM by the use of PETSc4FOAM library</b> , Stefano Zampini, KAUST 		
12:00-12:20p	<b>Wall-Modeled Large-Eddy Simulations of Airfoil Trailing Edge Noise</b> , Thomas Malkus, OHIO STATE UNIVERSITY 	<b>Development of optimisation strategies to enhance the performance of NOx Postprocessor</b> , Senthilathiban Swaminathan, MONTAN UNIVERSITY LEOBEN 	<b>The Effect of HDR InfiniBand and In-Network Computing on OpenFOAM Simulations</b> , Ophir Maor, HPC Council 		
12:20-12:40p	<b>External Core Catcher Cooling</b> , Samyak Darshan, COLLEGE OF ENGINEERING BENGALURU 	<b>Lowering the obstacles for SMEs to adopt multi-physics biomass furnace simulations by providing a cloud-based solution</b> , Bernhard Peters, UNIVERSITY OF LUXEMBOURG 	<b>A CPU-GPU paradigm to accelerate turbulent combustion and reactive-flow CFD simulations</b> , Federico Ghioldi, POLITECNICO DI MILANO 		
<i>Networking Break</i>					
01:40-02:20p	<b>PANEL DISCUSSION</b>				
Environment		Turbulence and Combustion II		Technology	
02:20-02:40p CET	<b>New developments for numerical wave tanks for coastel and offshore applications</b> , Gabriel Barajas, IH CANTABRIA 	<b>Modeling large-scale thermoplastic fires</b> , Alex Krisman, FM GLOBAL 		<b>Modeling hyperelastic solids in OpenFOAM</b> , Dr. József Nagy, EULERIAN SOLUTIONS 	
02:40-03:00p	<b>Modelling of H2O2 flotation for removing microplastics from waste water</b> , Emmanuel Thom, UNIVERSITY FREIBERG 	<b>Dynamic Zone Flamelet Model (DZFM) -an efficient yet accurate turbulent combustion model implemented in OpenFOAM</b> , Dr. Wei Yao, CHINESE ACADEMY OF SCIENCES 		<b>Three-Dimensional Simulation of Flow-Field around a Flapping Foil using Immersed Boundary Solvers of OpenFOAM</b> , Chandan Bose, UNIVERSITY OF LIEGE 	
03:00-03:20p	<b>High resolution urban air quality modeling using a multi-scale approach</b> , Rakesh Kadaverugu, CSIR 	<b>Models for Turbulence and Thermodynamics in Simulating \ Non-Premixed Combustion in a Cement Kiln</b> , Domenico Lahaye, TECHNICAL UNIVERSITY DELFT 		<b>Nonlinear Response Analysis of a Chord-Wise Flexible Flapper in the Wake of a Bluff Body using OpenFOAM</b> , Rajanya Chatterjee, IIT MADRAS 	
03:20-03:40p	<b>OpenFOAM computational performance:double vs mixed precision</b> , Federico Brogi, Istituto Nazionale di Geofisica e Vulcanologia 			<b>Workflow Development for CFD Analysis on An Aerospace S-Duct</b> , Ishan Nande, Beta CAE 