# 1st OpenFOAM HPC Challenge (OHC-1)

**JULY 1, 2025, VIENNA**

Mini-symposium to be held in the auspices of the
2025 OpenFOAM Workshop

*Organized by the OpenFOAM HPC Technical Committee (TC)*

*Presented by:*
*Sergey Lesnik, Gregor Olenik, Mark Wasserman*

# Organizing Committee

- **Mark Wasserman, Huawei**
- **Sergey Lesnik, Wikki GmbH**
- **Gregor Olenik, TUM**
- **Charles Mockett, Upstream CFD GmbH**
- **Fillipo Spiga, NVIDIA**
- **Neil Ashton, NVIDIA**
- **Elisabetta Boella, E4 Computer Engineering**
- **Fabrizio Magugliani, E4 Computer Engineering**
- **Simone Bna, CINECA**
- **Ivan Spisso, Leonardo Company**
- **Gavin Tabor, Univ. of Exeter**

# Objectives

- **Evaluate the efficiency and scalability of OpenFOAM on different hardware platforms (hardware track)**
- **Compare the performance of latest OpenFOAM code enhancements and flavors (software track)**
- **Propose unified metrics that will help guide the future development of OpenFOAM on next-generation hardware**
- **Bring together hardware vendors, industrial users and developers to discuss OpenFOAM performance**
- **Publish benchmark performance results of OpenFOAM on state-of-the-art hardware for guidance to users and developers**

# Scope

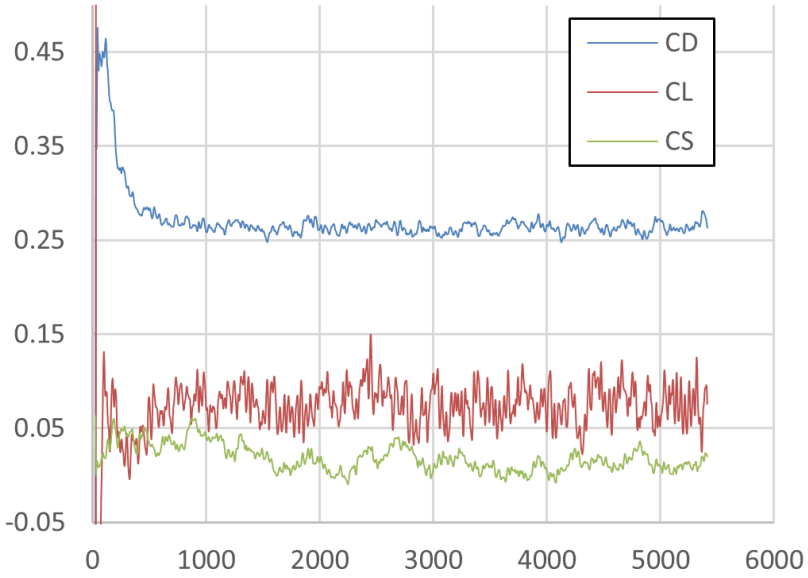
- **Industrial testcase of external flow over a static DrivAer automotive model, investigated as case 2 in the AutoCFD2 workshop, and as case 2a in the subsequent AutoCFD3 & AutoCFD4**
- **A full OpenFOAM setup* of the testcase, including meshes, physical model, and solver selection can be found here**
- **Reference numerical results are available in the submission sheets**
- **In the hardware track, participants are invited to submit and present results obtained with the pre-defined setup using an official OpenFOAM version, on any combination of hardware**
- **In the software track, participants are invited to submit and present results obtained with any OpenFOAM version and custom solver, on any hardware, so long as the mesh and physical modelling remains true to source, and accuracy is retained**
- **Participants are also invited to submit and present HPC enhancements related to pre- and post-processing of OpenFOAM data, applied to the above testcase**

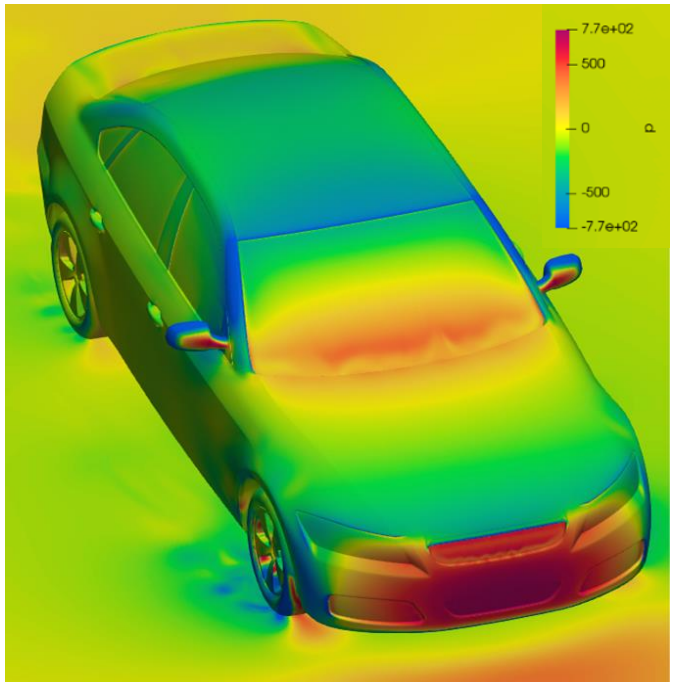
*Special thanks to Upstream CFD for providing the OpenFOAM DrivAer setup, which has been adapted for the OHC-1

# DrivAer Testcase (Case 2a)

- **occDrivAerStaticMesh**
  - Meshes generated with snappyHexMesh – 3 levels of refinement (65M, 120M, 236M)
  - Steady-state, incompressible flow
  - RANS with kw-SST turbulence model
  - SimpleFOAM solver
  - GAMG linear solver for pressure, Smooth solver for velocity
  - Hierarchical decomposition, RCM renumbering, potentialFoam init

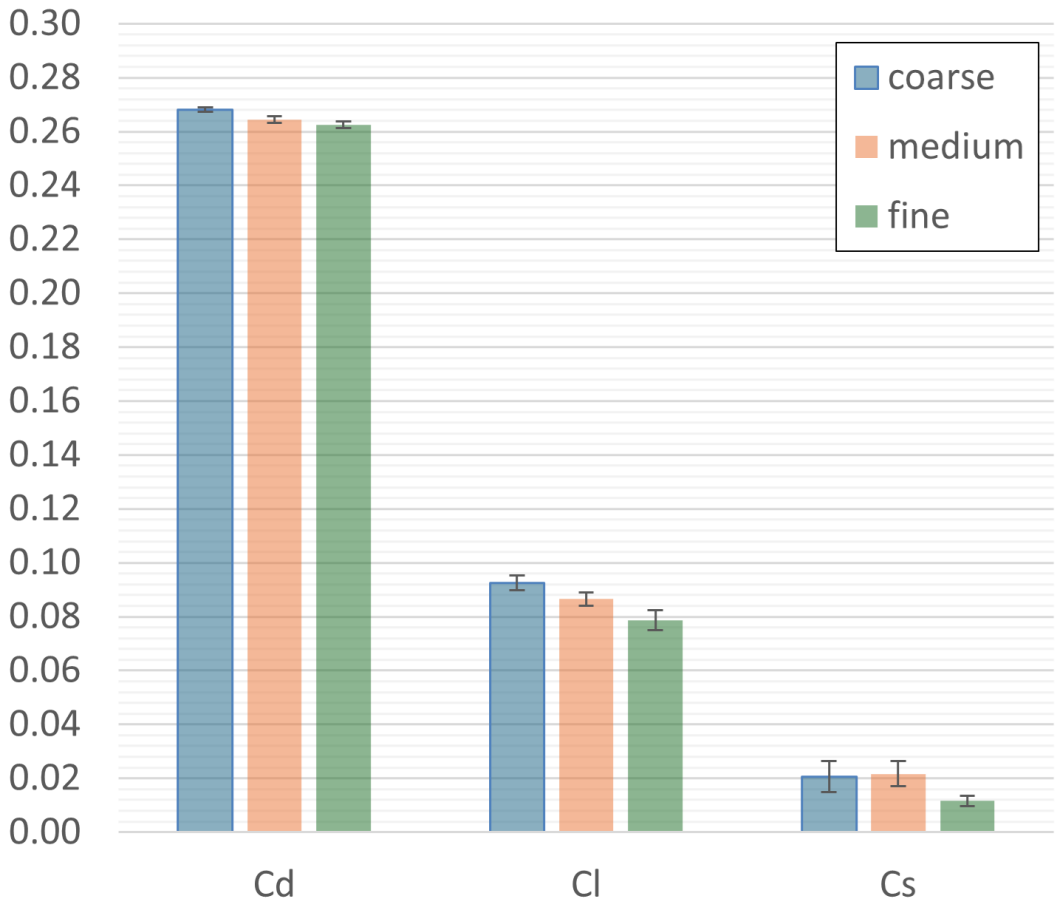
- **Simulations do not converge to a true steady state**
  - Typical for flows with massive separation
  - Adopted standard industry practice to produce mean values based on averaging
  - 4000 time steps (iterations) required to obtain sufficiently converged values (95% confidence interval on mean drag = +/- 1.5 counts)



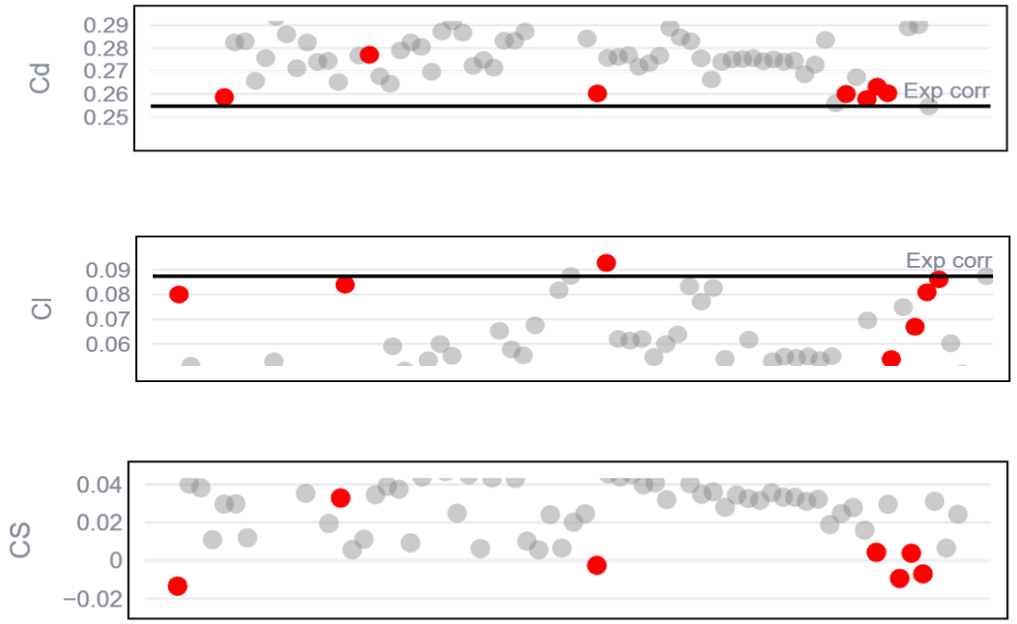



# DrivAer Testcase (Case 2a)

- occDrivAerStaticMesh (Baseline Setup)




## AutoCFD-4 RANS Results



# Rules (Hardware Track)

- **Objective: Demonstrate and compare the performance of OpenFOAM on various hardware configurations**

- **Hardware: unconstrained, so long as it supports standard OpenFOAM**
- **Software: OpenFOAM.com v2412**
- **Compiler: unconstrained**
- **MPI: unconstrained**
- **Case setup: occDrivAerStaticMesh, with fixed mesh, solver and numerical models (no changes allowed except for no. of processors for decomposition)**
- **Mandatory pre-processing stages (decomposition, renumbering, potentialFoam init)**

# Rules (Software/Open Track)

- **Objective: Showcase code/hardware optimizations that improve performance (time, energy), whilst retaining the basic level of accuracy as demonstrated with standard OpenFOAM**

- **Hardware: unconstrained**
- **Software: unconstrained, OpenFOAM-based code**
- **Case setup: occDrivAerStaticMesh, with fixed mesh and physical modelling (temporal and spatial discretization, turbulence model)**
  - Steady-state, incompressible flow
  - RANS with kw-SST turbulence model
  - Pre-defined fvSchemes (changes allowed but have to be reported)
- **Proposed topics for investigation: accelerators, pre-/post-processing (I/O), mixed-precision, linear solvers, renumbering/decomposition**

# Metrics

- **Hardware/Software Track**
  - Single- and Multi-node Scalability
  - Time-to-solution, excl. Pre-processing (decomposition, renumbering) and initialization (potentialFoam)
  - Energy-to-solution
  - Accuracy w.r.t reference solution obtained with standard OpenFOAM
    - Mean and variance of aerodynamic coefficients

# Data Submission Guidelines (General)

- **To maximize the outcome of the workshop for the OpenFOAM community, a structured simulation post-processing approach was adopted. The structured approach will ensure that:**
  - Simulation results can be compared to reference results
  - Solutions from different participants can be compared
  - The organizing committee will be able to process all incoming data
- **To enable a structured data processing all participants are requested to use the provided spreadsheet to submit their results:**
  - Detailed hardware/software setup
  - Wall-clock measurements
  - Aerodynamic forces (mean + history)
- **In case results are not provided in the required format, the submission will not be considered for comparison, and a speaker slot can not be guaranteed**

# Data Submission Guidelines (General)

- **Participants are required to submit one spreadsheet per mesh and track**
  - Please don't mix data from multiple mesh resolutions or HW/SW track in one submission
- **Multiple submissions are allowed**
- **Structure of the Excel template***
  - META Data: Description of simulation environment (SW/HW)
  - Simulations: Timings of multiple simulations for the same mesh/track
  - Aero Forces: Mean Aerodynamic Forces + time/iteration history
  - Force Plots: Generated plots of aerodynamic forces + reference results
  - Convergence Plots: Generated plots of convergence histories for selected simulations
- **How to extract data required for submission?**
- **Example (full) submission**
- ***Special thanks to Burkhard Hupertz for providing the original (AutoCFD4) data submission template, which has been adapted for OHC-1**

# Data Submission Guidelines (Continued)

- **What details need to be reported regarding custom solvers in the software track?**
  - Numerical schemes
  - Linear solver
  - Statistical analysis of aerodynamic forces (Meancalc)
  - (optional) input files
  - Precision
- **All data submitted will be made available to participants after the workshop via a dedicated repository (opt-out possible)**
- **Submitted data will be used for comparison, and compilation of a manuscript to be submitted to the OpenFOAM journal**