OpenFOAM for Industry

ESI - custodians of a CFD legacy

HPC enabling of OpenFOAM® for CFD applications 6th April 2016

ESI-OpenCFD / Fred Mendonca

get it right®

Fred Mendonça

Director, ESI-OpenCFD Global Operations

- Undergraduate and MSc at Imperial College, London
- 3yrs CFD Methods Development at BritishAerospace/Airbus
  - Euler and N-S multi-block, multigrid
  - Jameson-based, coupled flow-energy compressible flow solver algorithm
- 5yrs Support and Applications Manager at CHAM
  - the 1st commercial CFD commercial company
  - PHOENICS
- 20yrs Director of strategic applications at CD-adapco
  - Turbomachinery
  - Aeroacoustics
- Director of Operations at ESI-OpenCFD since Feb2015
  - OpenFOAM operations at ESI worldwide
  - Development, Support, Release, Maintenance

OpenFOAM for Industry

Agenda

- Introduction
- Updates in OpenFOAM
- Next steps
- Q&A

OpenFOAM for Industry

Agenda

- Introduction
  - Backdrop
  - 2015 Highlights
- Updates in OpenFOAM
- Next steps
- Q&A

At our annual conference

2015 - A Transitional Year

- Notable developments in this year
  - ESI world
    - Acquisitions and New technologies
    - Vision for CAE
  - OpenFOAM
    - Widening outreach, customer growth and welcoming new names
    - Global presence and Growing team at OpenCFD

- We stand on the shoulders of giants
  - A very brief history of open-source CFD

- Moving forward together
  - Developing a stronger community

6 recent ESI strategic acquisitions

CIVITEC

Virtual simulation of automated driver assistance (ADAS)

March 2015

OpenFOAM

Open Source Fluid Dynamics Software

September 2012

MODELICA

CyDesign™

New 0D/1D Technology

October 2013

PICVIZ® LABS LIGHTING THE UNKNOWN

INENDI

Visual Analysis of Big Data

April 2015

Ciespace

Technology to accelerate CAE delivery, including meshing, in the cloud

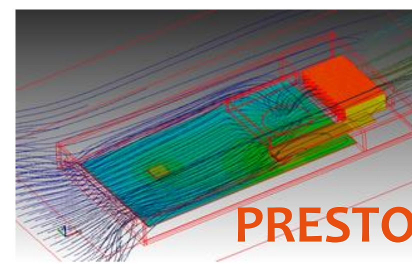
April 2015

IC.IDO

The Visual Decision Company

Real Time immersive virtual reality

September 2011



Disruptive solution dedicated to the electronics cooling market

May 2015

Integration of the recent acquisitions
Positioning in a global offer

Virtual Manufacturing


Casting


Composites


Sheet Metal Forming


Welding & Assembly

Virtual Reality


IC.IDO Immersive Experience

IC.IDO
The Visual Decision Company

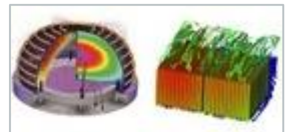
Virtual Performance

Virtual Performance Solution


Virtual Seat Solution

Vibro-Acoustics

Virtual Environment


Electronics, CFD & Multiphysics

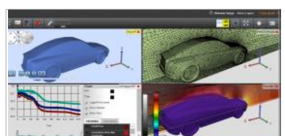
OpenFOAM


Electromagnetics

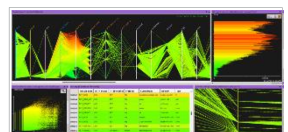
Virtual Integration Platform


Multi-Domain Simulation


Decision Support


Modeling on demand Cloud/SaaS

Ciespace


Data analytics

PICVIZ® LABS LIGHTING THE UNKNOWN


Virtual Systems & Controls

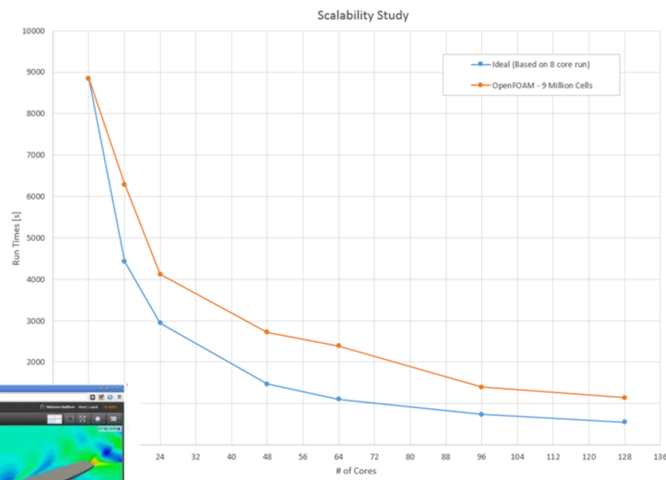
CIVITEC CyDesign™

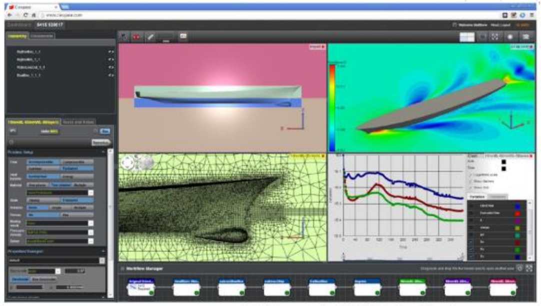
ESI World – Vision for CAE

Democratize Virtual Prototyping – Enablers

- Cloud/SaaS Architecture
- Open Application Framework
- Collaboration
- Analytics

- SaaS Cloud Offering
  - HPC Platform Accessibility through a Browser
    - Application Framework
      - CAD Import, Geometry repair, Meshing, Solver, Results processing
      - Highly scalable
    - Client & Server Side rendering
      - 3D remote visualisation of large data-sets
      - Parallel rendering
      - Simultaneous viewing of animations
    - Collaboration
      - Real-time sharing





- ~7 Million Cells, Multi-Region Mesh
- Steady State Resistance
- Waves with Moving Mesh (30s simulation time)

OpenFOAM

Global presence Growing Team at OpenCFD

- **OpenFOAM Development in Europe and NA**
- **Support in Europe, NA, Asia**
- **Services and Training in Europe, NA and Asia**

get it right®

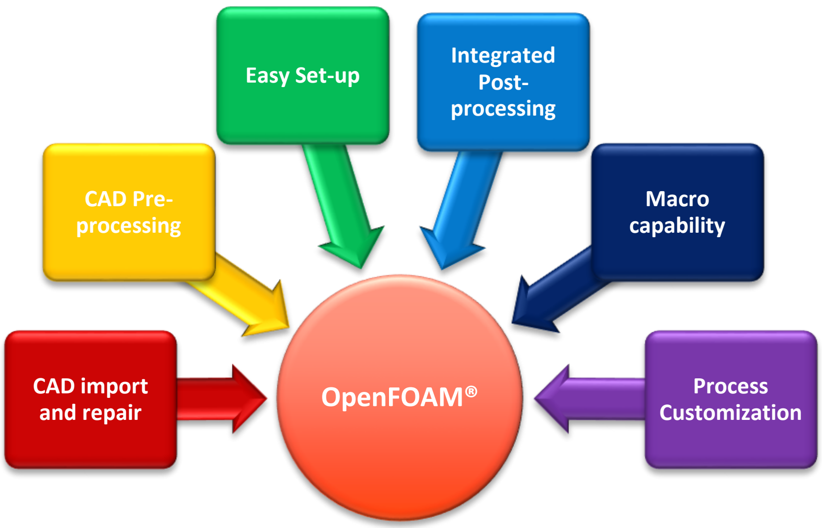
OpenFOAM

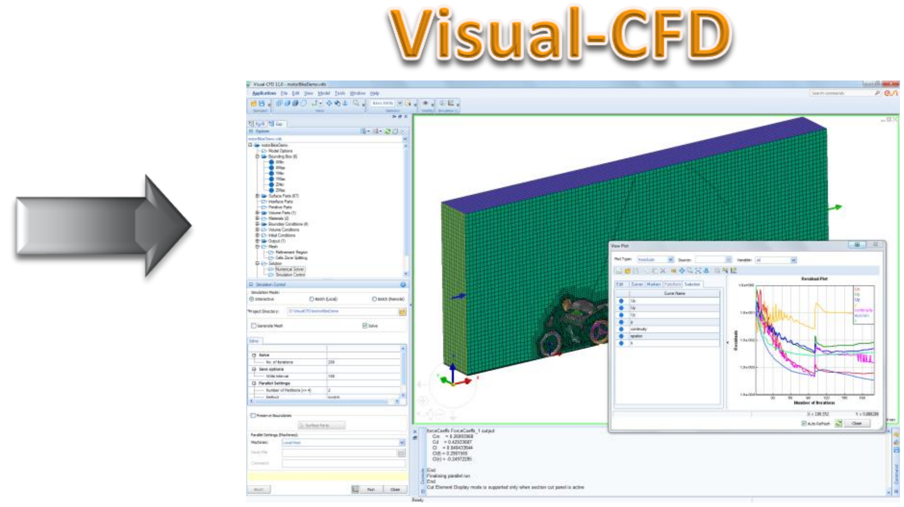
Widening Outreach

- Increased accessibility, applications and multi-domain CAE
  - Windows (containerised) accessibility
  - Graphical environment and process control
  - Multi-physics
    - Aero Vibro Acoustics (details shown later)
    - Fluid-Structures Interaction
    - Thermal and passenger comfort

OpenFOAM

Graphical Environment and Process control



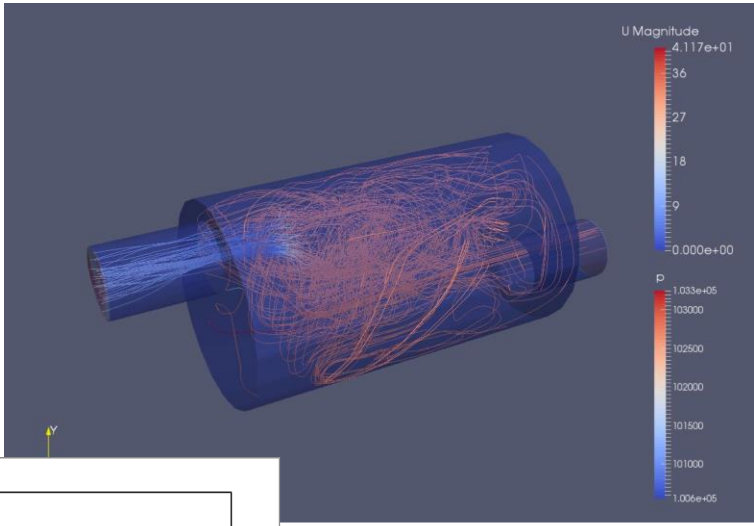


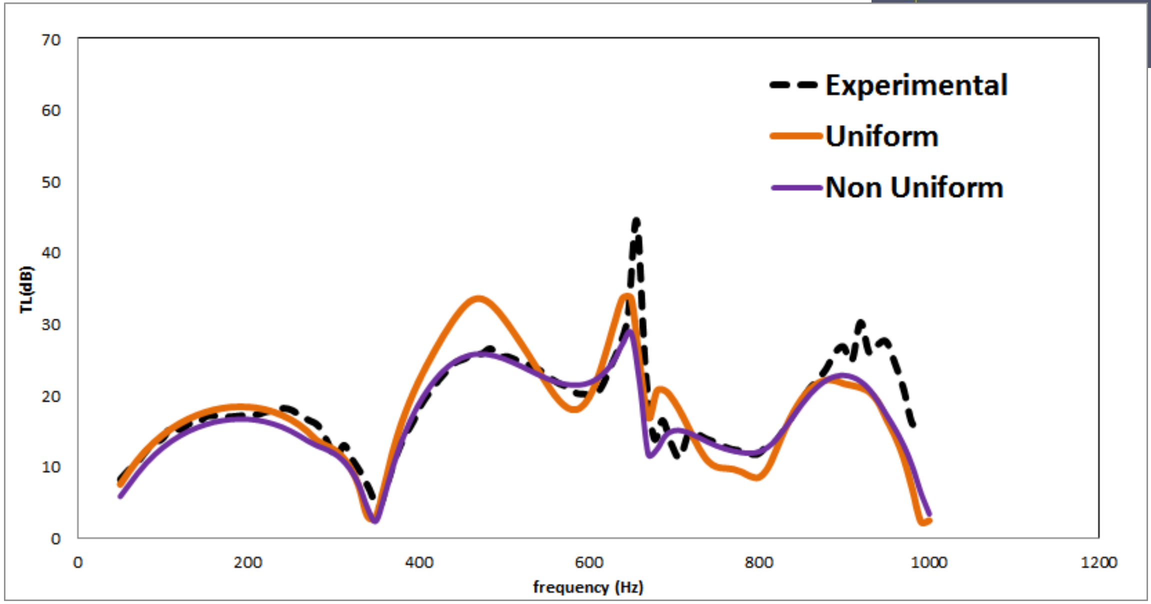
- Training course offered monthly since May 2015
- 2016 (Feb) relese supports conjugate heat transfer (CHT)
- Developing Applications Process controls for
  - External aerodynamics, including aeroacoustics
  - Marine hydrodynamics and stability
  - HVAC systems

OpenFOAM

Multi-physics – Aero Vibro Acoustics

- Acoustics coupled with meanflow
  - *OpenFOAM background process* provides non-uniform flow in a perforated-pipe muffler
  - VA One predicts the transmission loss
  - *Data courtesy of Caterpillar Inc.*

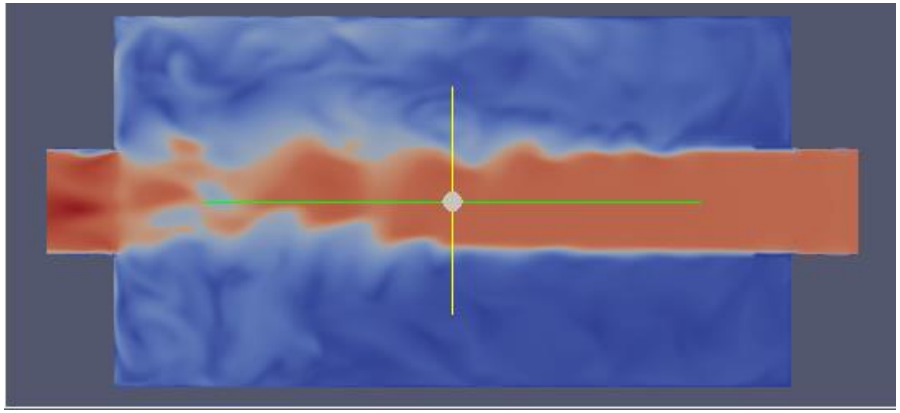




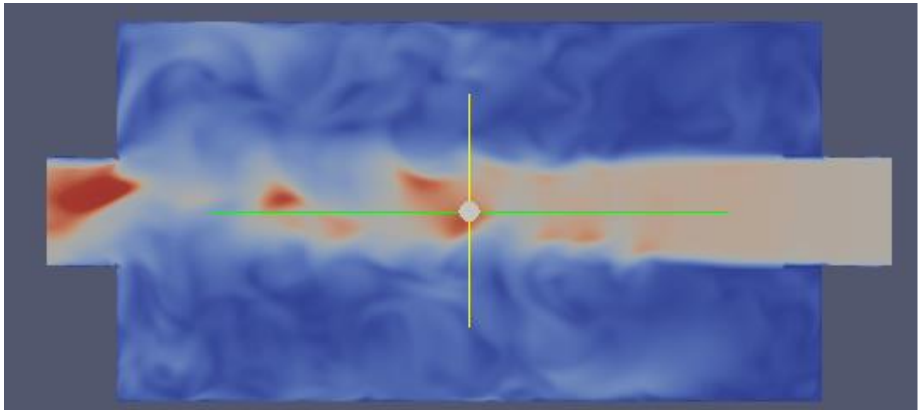
OpenFOAM

Multi-physics – Aero Vibro Acoustics

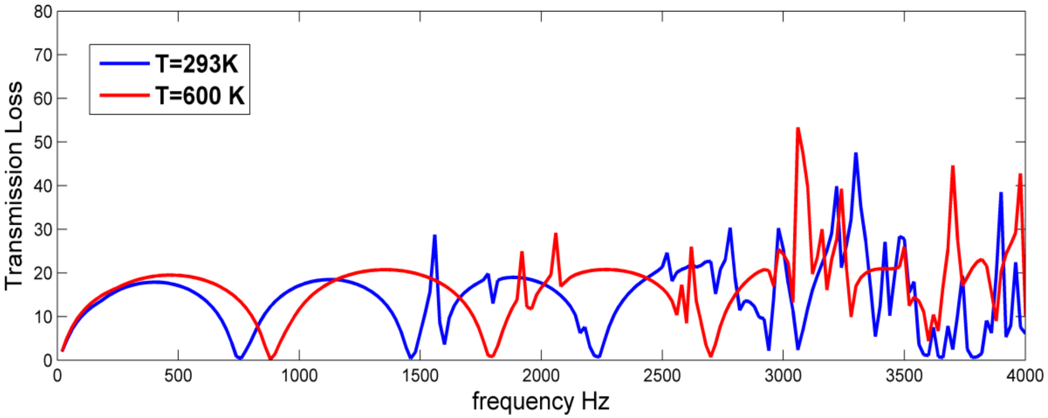
- Acoustics coupled with flow-thermal effects from OpenFOAM



Mach number at T=600K



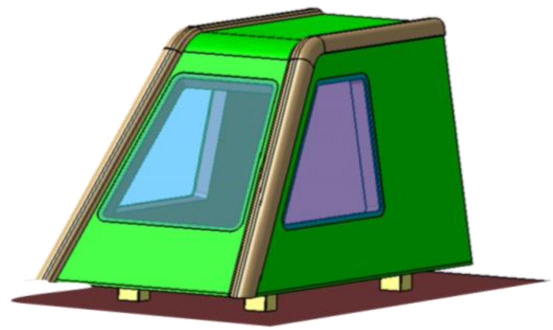
Mach number at T=600K

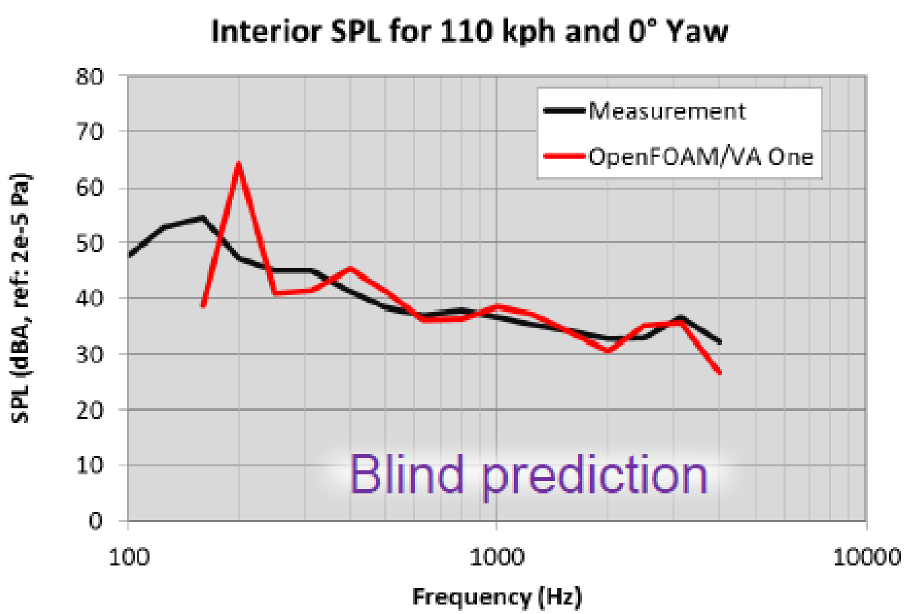


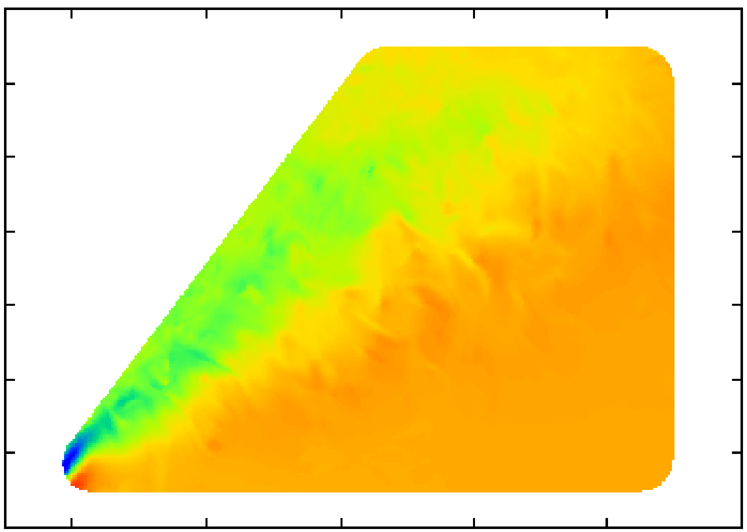
OpenFOAM

Multi-physics – Aero Vibro Acoustics

- Case study with Hyundai
  - Progressive studies on AVA since 2011
  - Flow and acoustic excitation on side-glass
  - OpenFOAM predicts the flow/aocutics sources
  - VA One transmits the noise internally to the drivers ear



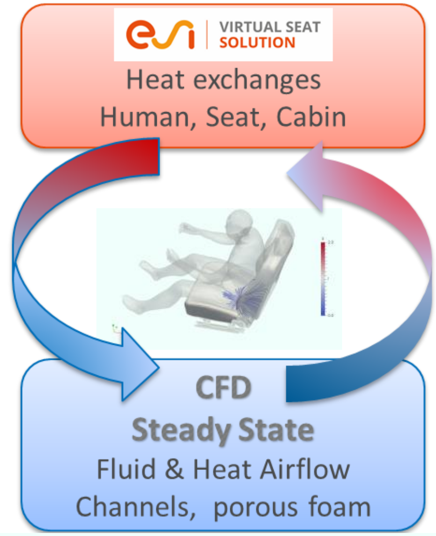


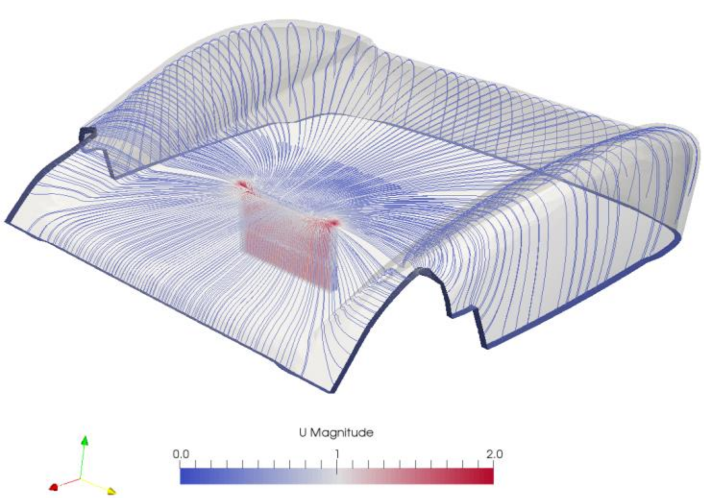


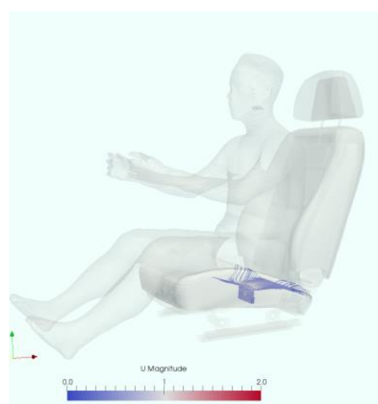
Extension towards seat comfort

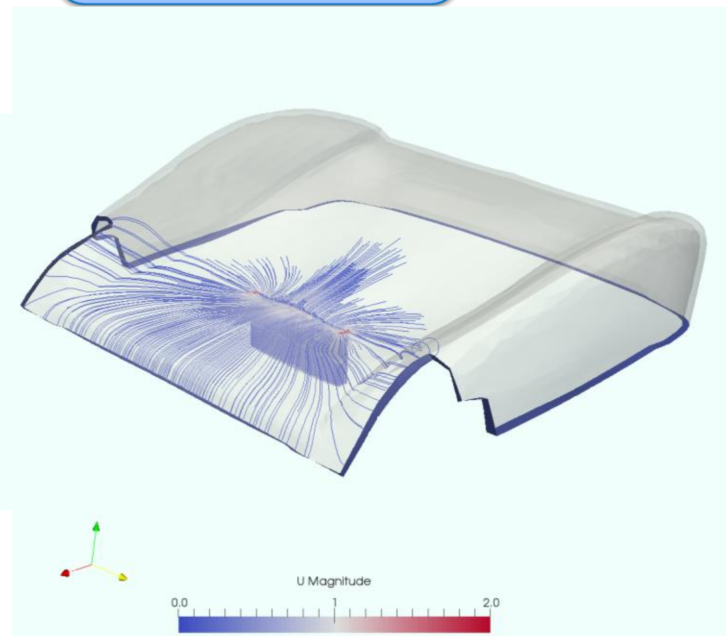
Thermal Comfort - Ventilated seat

Manufacturing -> Seating → Thermal, Heat, Transient, Comfort and Stress Analysis









OpenFOAM

Code and Copyright transferred to the OpenFOAM Foundation

- November 2014 transfer of OpenCFD bug-fix branch
  - Leading to v2.3.1 public release
- March 2015 transfer of OpenCFD concurrent development branch
- May 2015 transfer of OpenCFD chemistry modules
- June 2015 transfer of OpenCFD essential bug-fixes
- September 2015 transfer of OpenCFD concurrent development branch
  - Enhancements since March 2015
  - Pre-publication to secure OpenCFD client‘s investment

- Lack of public release led OpenCFD‘s customers to demand
  - Frequently released new functionality
  - Progressive envorinment for 3rd-party contributions
  - Proper quality control
  - Due accreditations to contributors

We stand on the shoulders of giants

A brief history of open source CFD

- Notable candidates
  - ANSWER (Gosman et al, 1969 *Heat and Mass Transfer in Recirculating Flows*, Academic Press, London.)
  - TEACH code (Imperial college CFD course 1980s)
  - OpenFOAM, 2004 (OpenCFD, OpenFOAM Foundation)
  - Code Saturn, 2007 (EDF)

- Long history of Corporate and National labs/administrations development, maintenance and wider user-base
  - NASA codes (since 1970s)
  - BAe/Airbus, DLR (Tau), Onera (Elsa) developments with collaborative circulation and interaction (prevalent since the 1980s, and ongoing …)

We stand on the shoulders of giants

A brief history of open source CFD

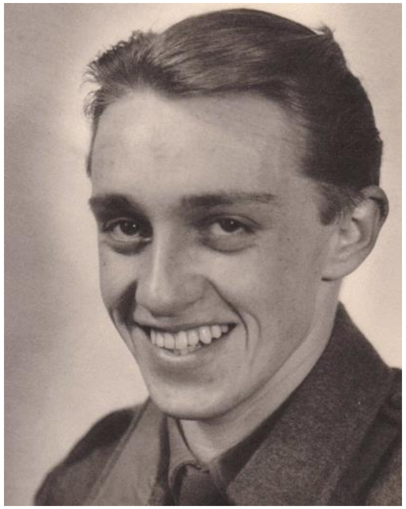
- Enablers
  - Segregated pressure-velocity solvers
    - Patankar, Runchal 1960-70s (Staggered grids)
    - Peric, Demirdzic 1980s (Co-located algorithm)
    - Issa, Gosman, 1980s (PISO)
  - Coupled equations solvers
    - MacCormack, 1969, Jameson, 1981, (Multi-step explicit and semi-implicit)
    - Roe, 1980s, (compressible flux schemes)
    - Weiss and Smith, 1990s (Preconditioning for low-Mach number flows)

We stand on the shoulders of giants

A brief history of open source CFD

- Amidst a seminal contribution
  - D.Brian Spalding
    - With Patankar, SIMPLE algorithm, turbulence two-equation model
    - Supposedly passed his codes (2D, 3D …) to fellow researchers in UK and wider
    - Formed the basis of well known commercial codes
      - PHOENICS
      - FLOW3D/CFX (before Taskflow)
      - Fluent (before Rampant)
        - One Fluent developer is reported to have said in the late 1990s

      "We've finally written Spalding out of Fluent"



  - …and a continuing legacy at Imperial College
    - from where Weller, Tabor, Jasak, Fureby published, in 1998

    "A tensorial approach to computational continuum mechanics using object-oriented techniques."

We stand on the shoulders of giants

Accrediting all past contributors for where CFD finds itself today

- More than 10 years after the first release of OpenFOAM
  - Hundreds of contributors
  - Thousands of users
  - 10s of thousands of contributions
  - 100s of thousands of downloads per year
  - Millions of simulations
  - Billions of CPU hours

- ESI, as proud owner of the trademark, commits to the next exciting phase of OpenFOAM with the community

OpenFOAM

Windows version

- Widening OpenFOAM's footprint
- About our containerised (Docker) Windows version
  - OpenCFD installer wizard
  - Guarantees the same results as the original Linux executable
  - New releases on diverse platforms will be virtually immediate
  - Portable across any linux distribution which support containerisation
  - Performance benefits are
    - Average 25% scalar performance gain versus available native builds
    - Parallel Scalablility demonstrated
  - Lightweight virtual box

- Field-tested during July-Sept 2015
- Freely Available from Oct 2015
  - See openfoam.com

Moving forward together

Building a stronger community

- OpenCFD-led Collaborations
  - New functionality releases
    - Quality assured code development and testing
    - Regular release schedule
- Community led Collaborations
  - More regular new-functionality releases
  - Tell us where you think improvements are needed and how you can help

- Collaborative topics
  - Overset mesh
  - Conjugate Heat Transfer modifications/handling
  - Coupled solver
  - Meshing enhancements (sHM and fHM)
  - … community suggestions for new features/improvements

OpenFOAM for Industry

Perspective

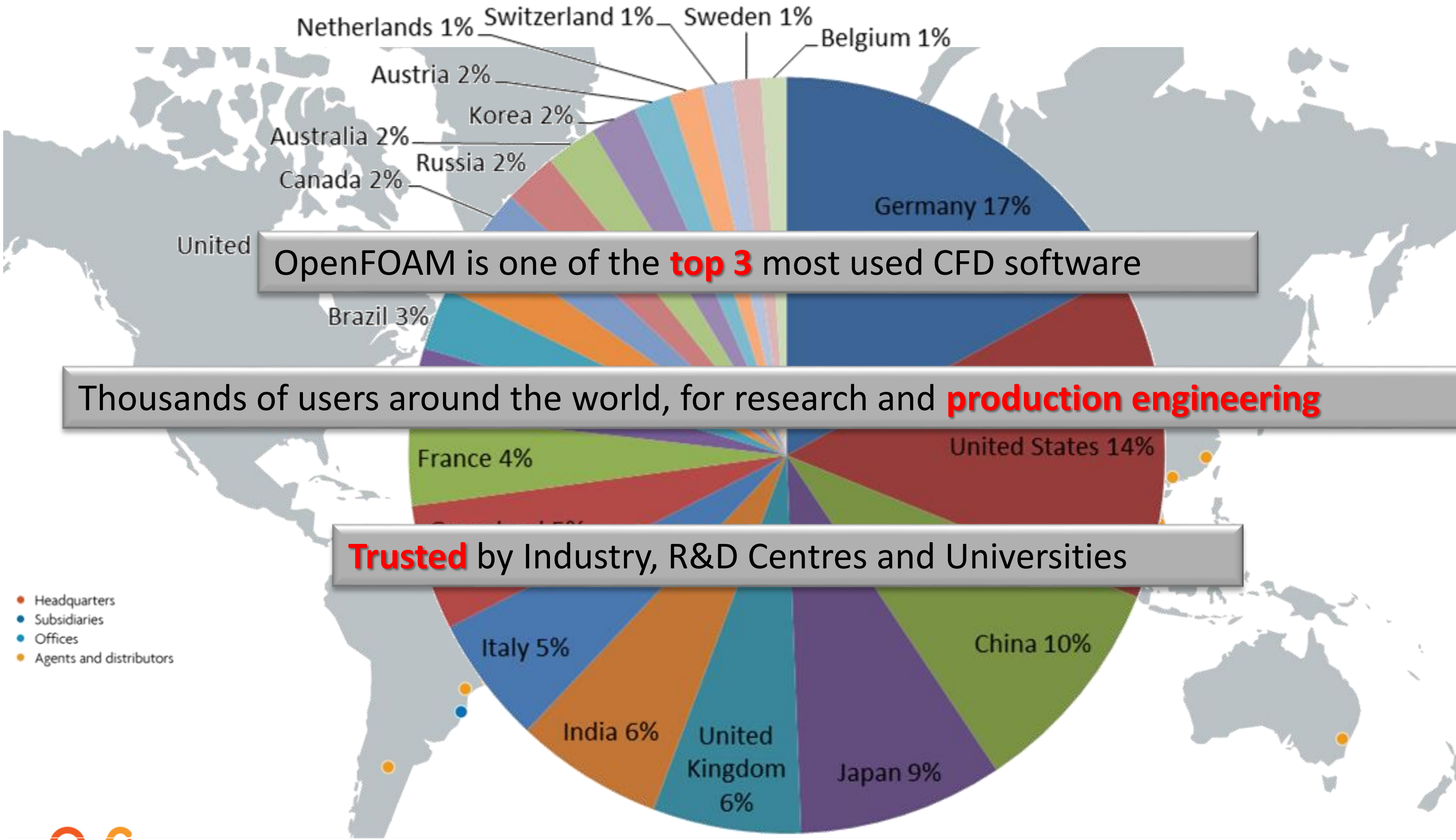
- OpenFOAM is an extremely powerful toolset, with many different applications from automotive to multiphase and chemical processing.
  - Users need to have quite a bit of control to be able to make use of OpenFOAM robustly.

- OpenFOAM Customers as well as Community have requested further enhancement of OpenFOAM to easier the usage at industry level.
  - Target: Increase easiness of use and robustness, to make it comparable to commercial tools.

- ESI Group is responding to these requests, to make OpenFOAM the leading CFD software tool.
  - Hence the need for a "transitional year" in 2015

OpenFOAM v2.3 downloads

More than 150k downloads of
OpenFOAM v2.3 in 2014



OpenFOAM is one of the **top 3** most used CFD software

Thousands of users around the world, for research and **production engineering**

Trusted by Industry, R&D Centres and Universities

get it right®

OpenFOAM for Industry

Agenda

- Introduction
- Updates in OpenFOAM
- Next steps
- Q&A

OpenFOAM for Industry

Updates in OpenFOAM: OpenFOAM+

- New features from 2014-2015 developments
  - Pre-processing
  - Meshing
  - Solver
    - Initialisation
    - Heat transfer / CHT
    - Boundary conditions
    - Turbulence
    - Run-time controls
  - Post-processing

OpenFOAM for Industry

Updates in OpenFOAM: OpenFOAM+

- ESI OpenCFD released OpenFOAM+
  - OpenFOAM+ uses the OpenFOAM Foundation version as a common code base, and offers wider functionality and platform support. Its purpose is to accelerate the public availability of new features which are sponsored by OpenCFD's customers and contributed by the OpenFOAM community
  - OpenFOAM+ is distributed by OpenCFD under the GPL license as:
    - Source code to be compiled on any Linux system
    - Pre-compiled binary installations for Linux and Mac OS X systems
    - MS Windows installer
  - More than 35k downloads since release in January'16.
  - 2 releases planned per year.
    - June
    - December

OpenFOAM for Industry

Updates in OpenFOAM: OpenFOAM+

- Added value in all areas of OpenFOAM: meshing, case set-up, turbulence models, multi-physics, solvers and post-processing.
  - Includes contribution of renown companies in specific areas.
  - Enhanced interaction with other ESI tools (VA One, VPS).
  - Community integration

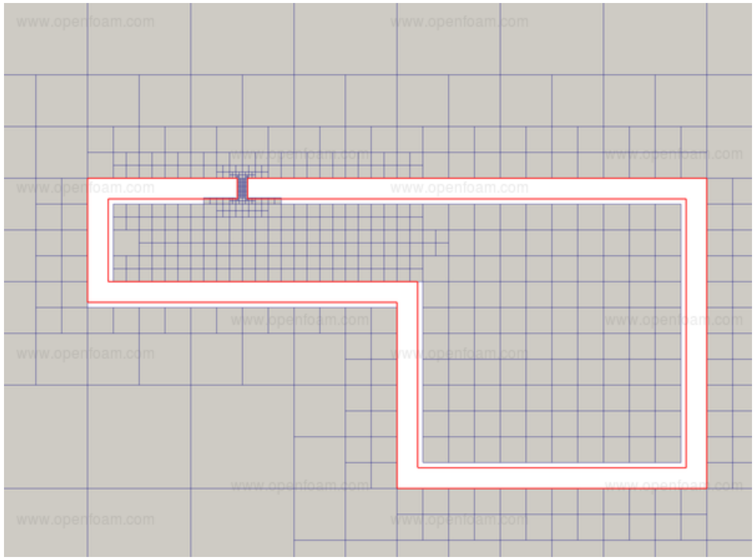
- Quality assurance, with thorough testing in the development process, industry tests and customer regression cases.
  - Worldwide ESI teams participate in this process, from Japan to North America.

- Supported by ESI values and infrastructure.
  - International technical, sales, marketing and administrative support
  - Customer focus

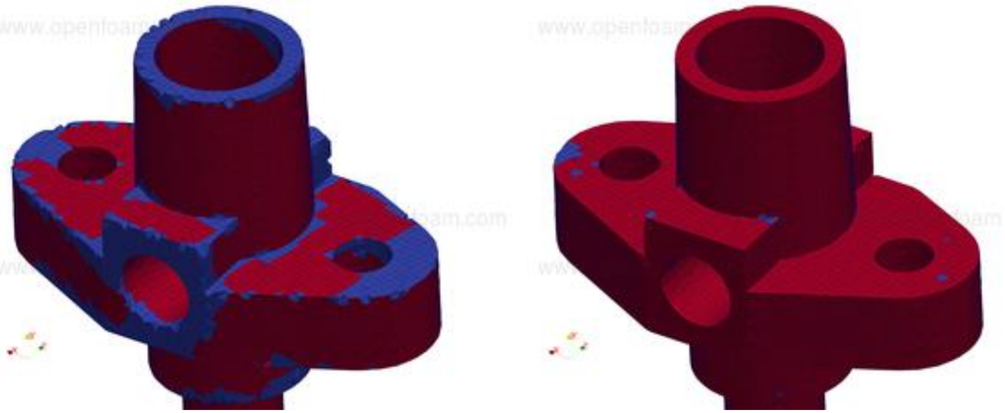
OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- New features in meshing:
  - Automatic gap finding and refinement



  - Improved conformance to feature lines for boundary layer mesh generation



OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

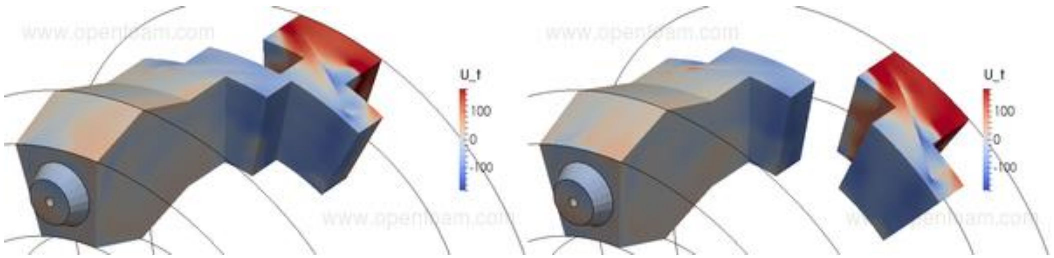
- New features in case set-up:
  - createZeroDirectory, utility, to ease the set-up of OpenFOAM cases by creating a complete time zero directory for any solvers, including multi-region cases (like conjugate heat transfer).
    - This will be a strong feature to simplify OpenFOAM case preparation.

  - Improvements in parallelization:
    - Decomposition of surface STL data for meshing.
    - New redistributePar utility, to decompose, reconstruct and re-distribute parallel cases reducing memory requirements.

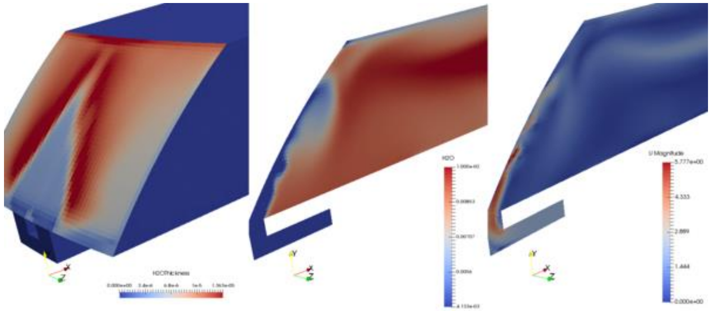
OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- New features in boundary conditions:
  - PeriodicAMI for repeating, non-conformal geometries



  - Humidity boundary condition

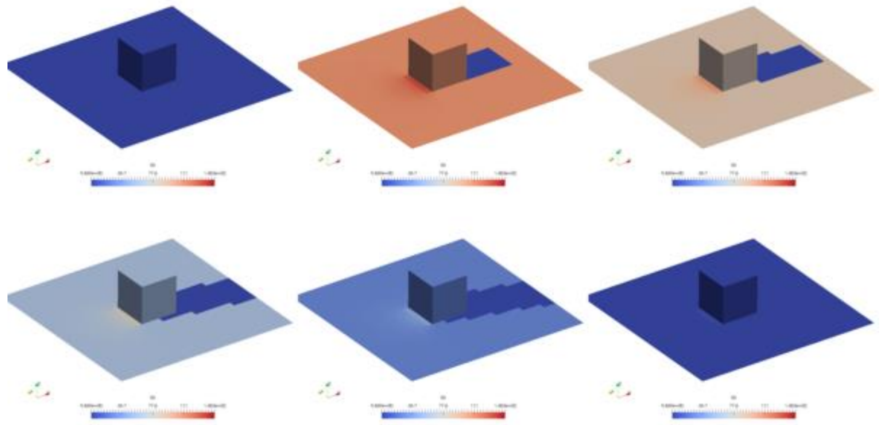


  - Co-simulation between OpenFOAM and external codes

OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- New features in physics:
  - Solar model

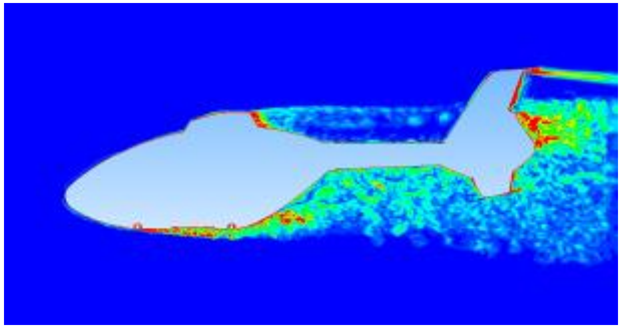


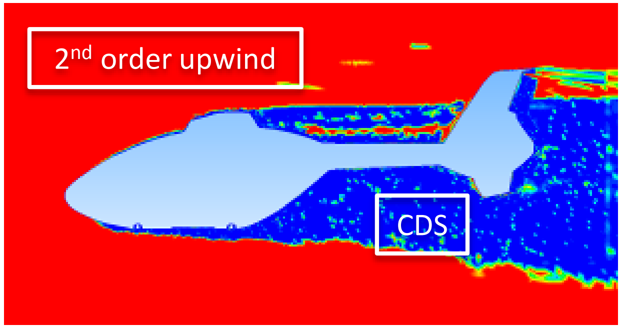
  - Diffusion combustion model
  - New DES models (based on kOmegaSST and updated Spalart-Allmaras)
  - New combustion models and solvers (moving mesh, lagrangian, etc).

OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- New features in numerics:
  - New velocity damping to aid solver sobustness during start-up of the solution process.
  - New Hybrid discretisation scheme for DES/LES applications, enabling the solver to switch between different numerical schemes.
    (acknowledging CFD Software E+F GmbH)

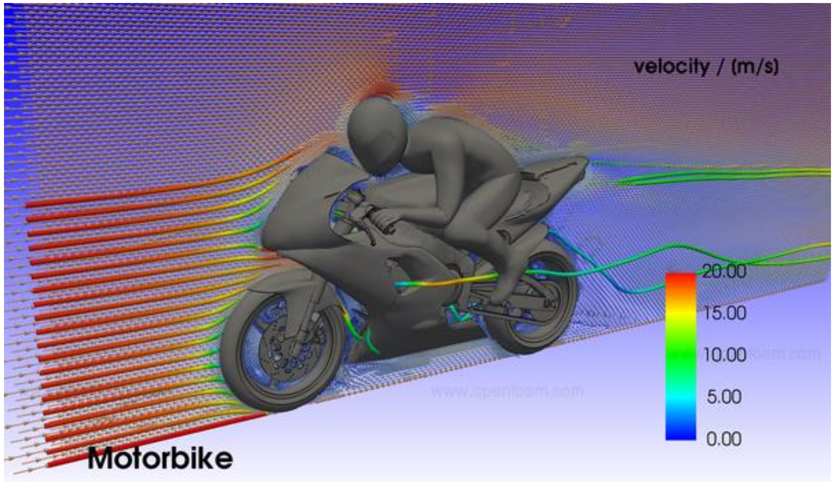




OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- New features in post-processing:
  - Automated picture extraction
    - No need to download complete model&results files.



  - Run-time case termination based on user statistics.
  - Enhanced EnSight export in transient runs
  - Enhanced options to report forces&moments, as well as summary of fluxes through components.

OpenFOAM for Industry

Updates in OpenFOAM (OpenFOAM+)

- Quality assurance is done by means of consistent development process and thorough testing:
  - Unit tests, to ensure that the new developments did not break any other functionality.
  - Medium size tests, monitoring performance and resources required.
  - Industrial regression tests, to support the release process.

OpenFOAM for Industry

Agenda

- Introduction
- Updates in OpenFOAM
- Next steps
  - OpenFOAM enhancements and community projects
  - OpenFOAM partnership programme
  - OpenFOAM User Conference
  - Features in the next release
- Q&A

OpenFOAM for Industry

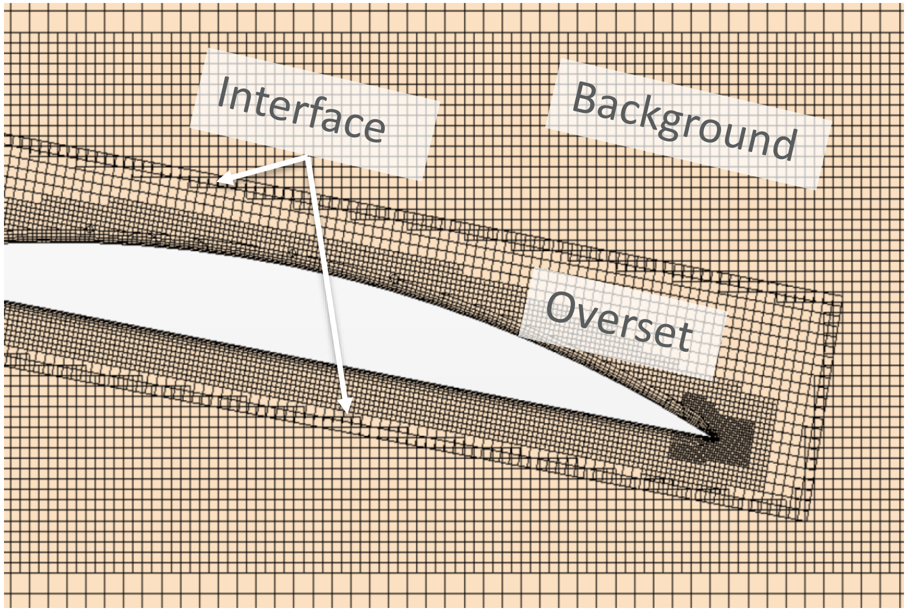
Next steps

- OpenFOAM enhancements and community projects:
  - Aero-Vibro-Acoustic technology: enhancement and validation of OpenFOAM for aeroacoustic simulations.
  - Overset mesh project (collaborative project: KHI, Audi, SLR, Shimizu and others).
  - Enhancements around Multiphase physics.
  - Conjugate Heat Transfer (collaborative project).
  - Documentation (collaborative project).
  - Meshing (collaborative project).
  - HPC initiatives (MPI, I/O).

OpenFOAM for Industry

Next steps

- OpenFOAM enhancements:
  - Overset mesh project
  - Partners in
    - Marine
    - External aero
    - Heavy industry
    - Building services
    - HPC communications
  - More partners welcome
    - Consolidate wide-ranging applications
    - Ensure robust release in June 2017



OpenFOAM for Industry

Next steps

- Other OpenFOAM enhancements:
  - Conjugate Heat Transfer (collaborative project)
  - Documentation (collaborative project)
  - Meshing (collaborative project)
  - HPC initiatives (MPI, I/O)

OpenFOAM for Industry

Next steps

- OpenFOAM Partnership Program
  - OpenFOAM is a big project with strong potential for collaboration from many entities.
  - Partnership program established to acknowledge the support in the growth and maintenance of OpenFOAM.
  - 4 levels of partnership.
    - Bronze (Website & Events)
    - Silver (UGM & Workshop Sponsors)
    - Gold (Development partners)
    - Platinum (Governance members)

All contributions towards maintenance and release of OpenFOAM+

OpenFOAM for Industry

Agenda

- Introduction
- Updates in OpenFOAM
- Next steps
- Q&A

OpenFOAM for Industry

Next steps

- 4th OpenFOAM Users Conference:
  - Dates: 11-13th October 2016
    - 2 days conference
    - 1 day parallel OpenFOAM workshops.
  - Place: Cologne (Germany)

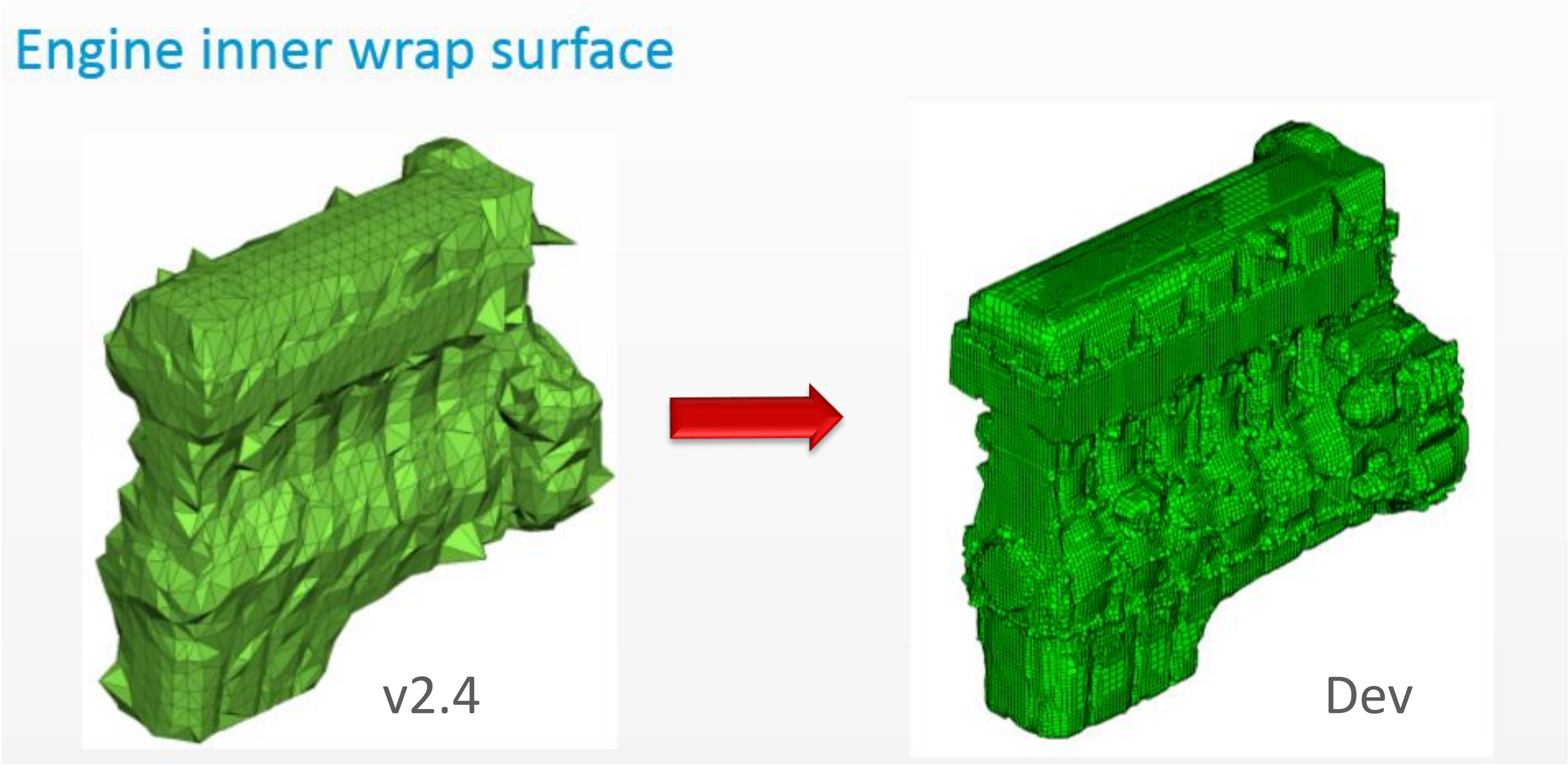
  - Abstracts welcomed:
    - Like last year, the best student presentation will be awarded.

  - Book the dates!!

OpenFOAM for Industry

Next steps – Upcoming features

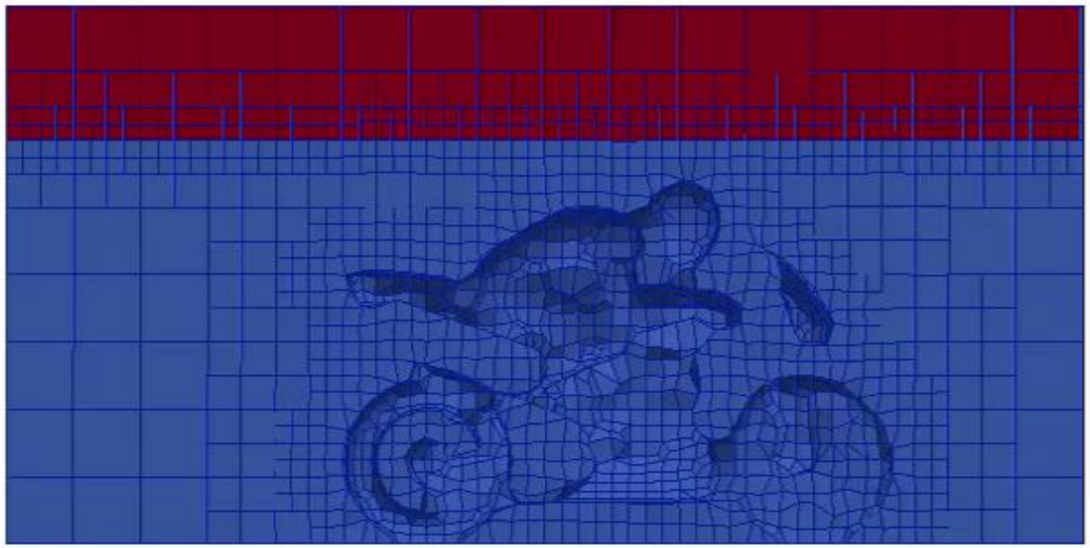
- Wrapping with snappyHexMesh !!! ?
  - v2.4 permits some level of coarse-grain wrapping
  - Improvements allow for an additional sequence to refine the coarse-grain



OpenFOAM for Industry

Next steps – Upcoming features

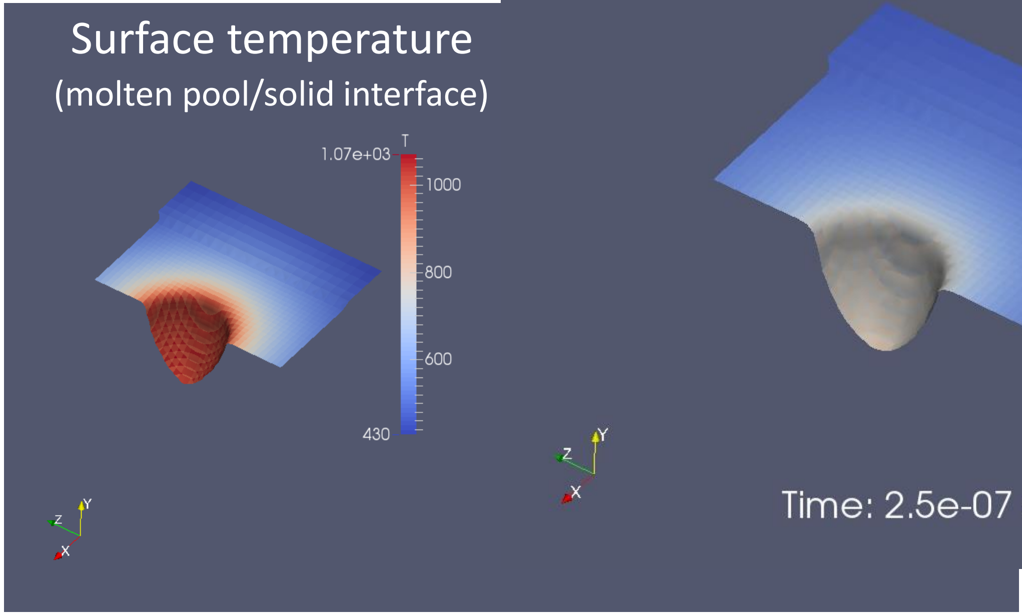
- Compatibility with AMR (arbitrary mesh refinement)

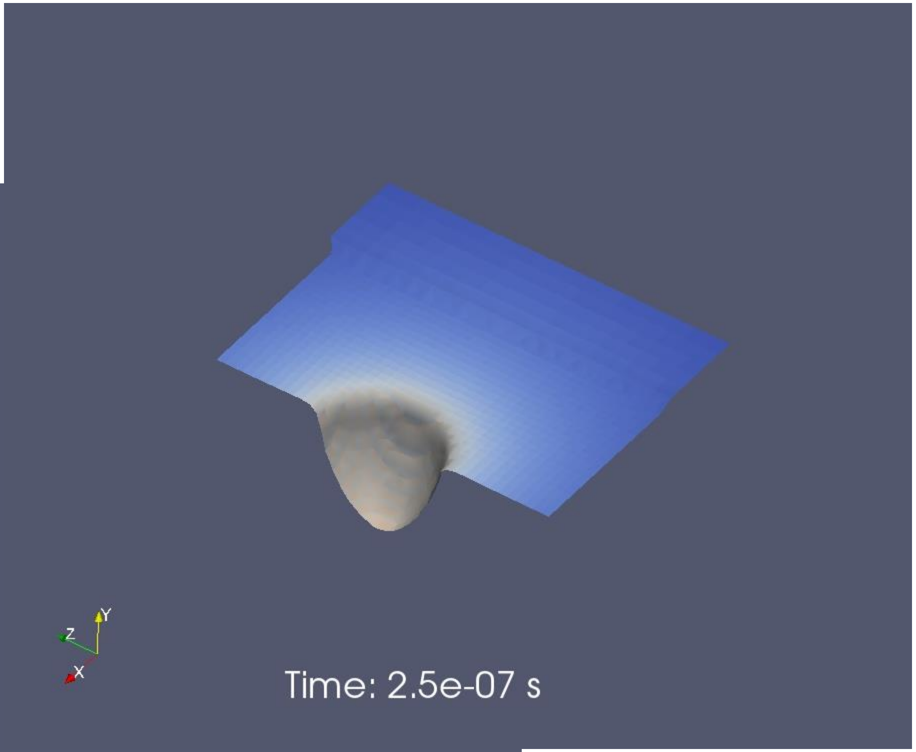


OpenFOAM for Industry

Next steps – Upcoming features

- Laser ablation/melting
  - Laser energy incident on surface
  - Surface ablation arisies from
    - Pool melting
    - Evaporation
  - Application to surface coatings



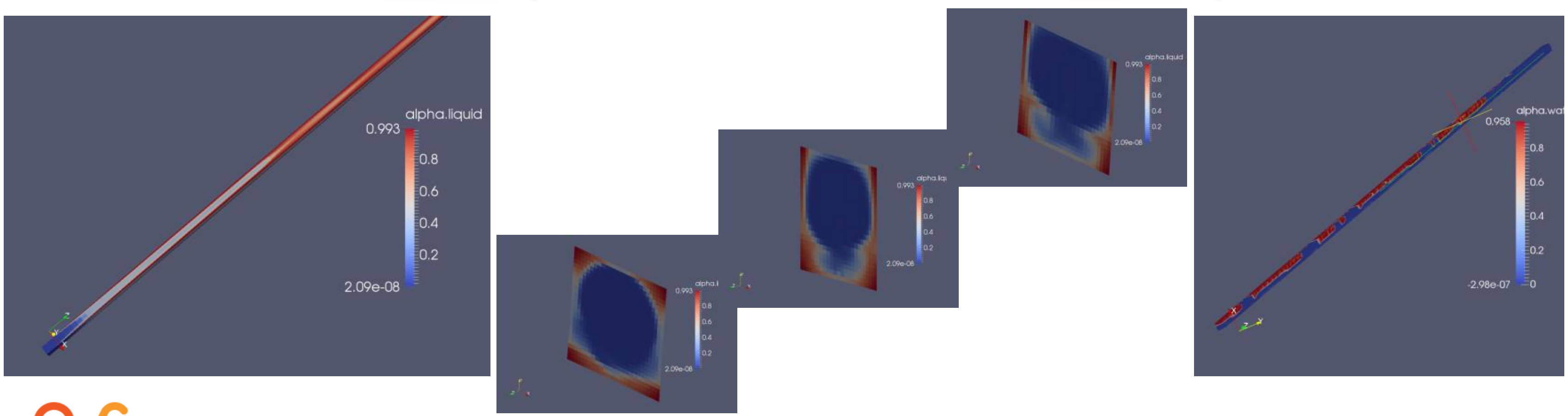


OpenFOAM for Industry

Next steps – Upcoming features

- VoF condensation
  - VoF extended to deal with heat exchanger thin channel condensation
    - Wall heat-transfer
    - Condensate heat and mass transfer
  - Transitional channel condensation; film > wavey > slug > bubbly
  - Also deals with reverse process (evaporation/boiling)

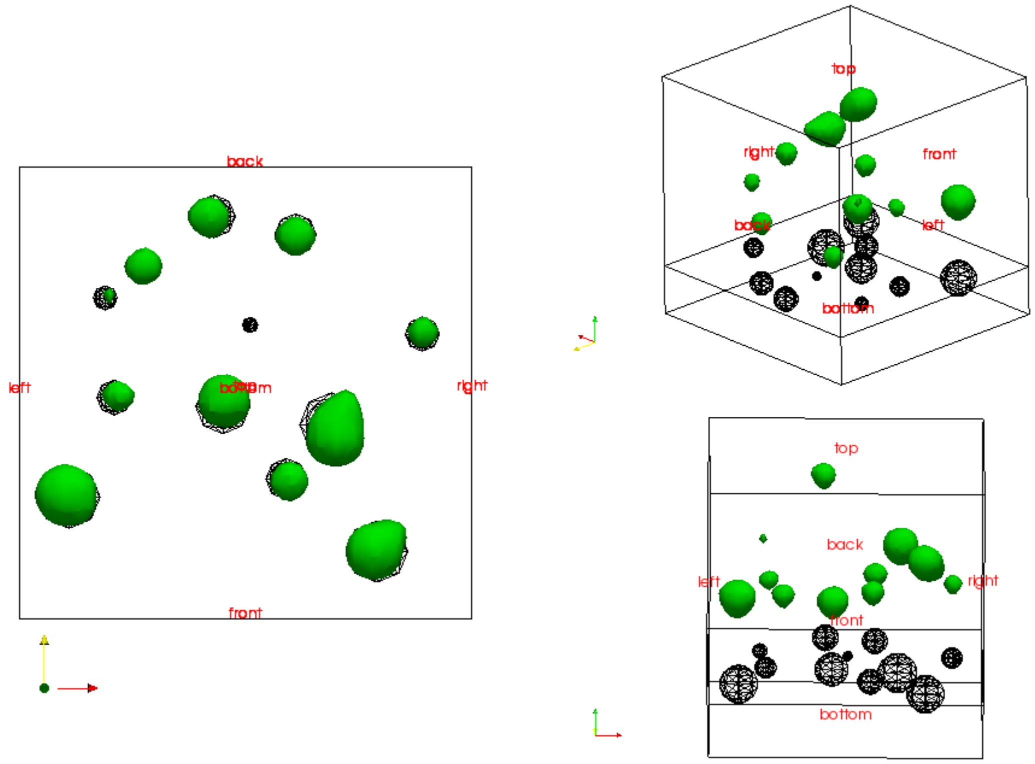
FILM TRANSITIONAL BUBBLY



OpenFOAM for Industry

Next steps – Upcoming features

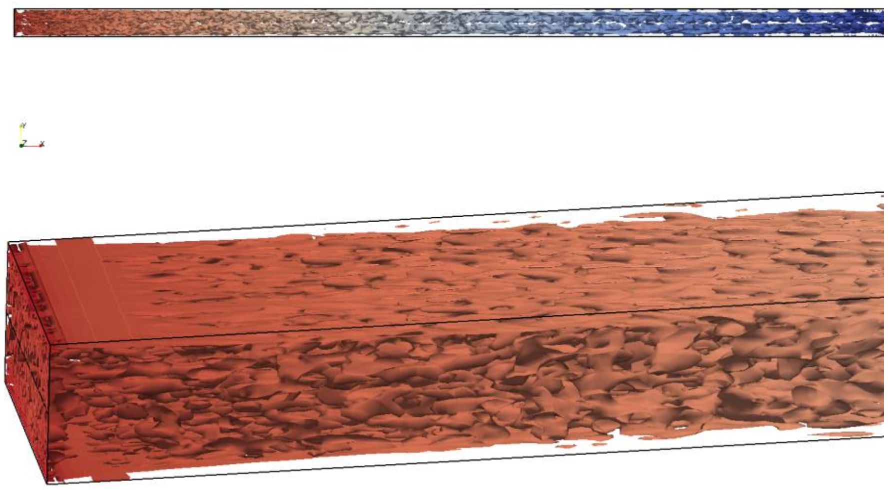
- Extract Lagrangian from VoF
  - Gathering of VoF „blob“ statistics – size, velocity, position – at interface
  - Represent as equivalent lagrangian particles



OpenFOAM for Industry

Next steps – Upcoming features

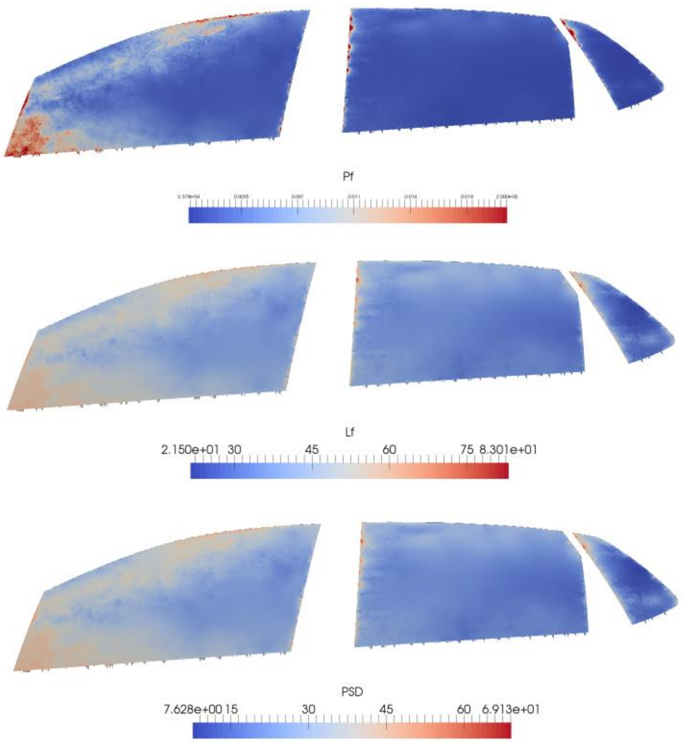
- Turbulence inflow generator
  - Meanflow turbulence tanslated into correlated turbulent structures
  - Divergence-free (compressible) formulation
  - Turbulent structures become „self-sustaining“ after just 2-3 hydraulic diameters



OpenFOAM for Industry

Next steps – Upcoming features

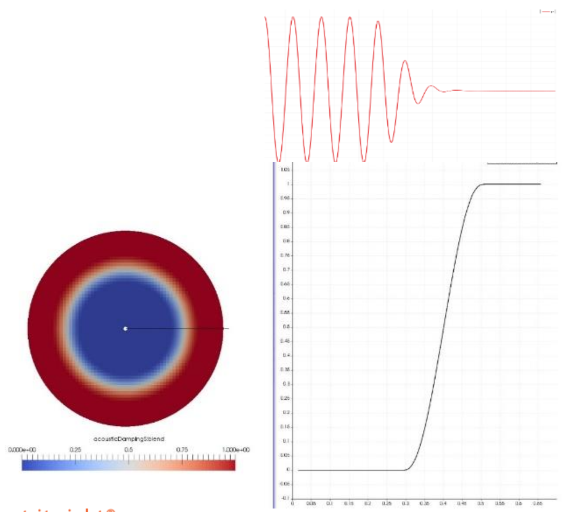
- Surface FFTs
  - At discrete frequencies, frequency bands (e.g. 3rd octave)
  - Fluctuating P_{rms} (Pa), Overall sound (dB), Power specra (Pa^2/Hz)

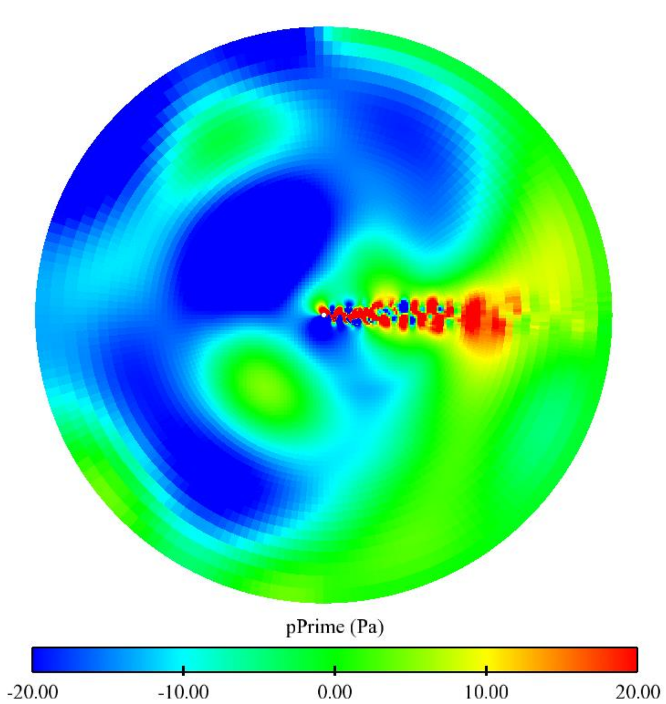


OpenFOAM for Industry

Next steps – Upcoming features

- Compressible scaled dimensions and velocities
  - 40m/s inflow (Mach 0.114)
  - Diameter = 0.025m
  - (vs incompressible, normalised to 1m/s inflow and 1m diam)
  - Damping
    - starts at 0.2sec
  - ~1million cells

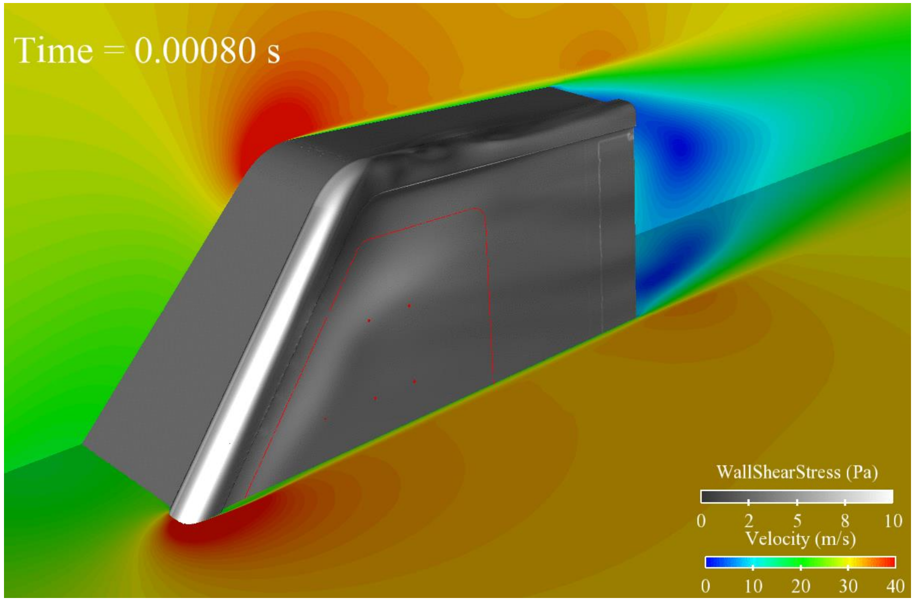


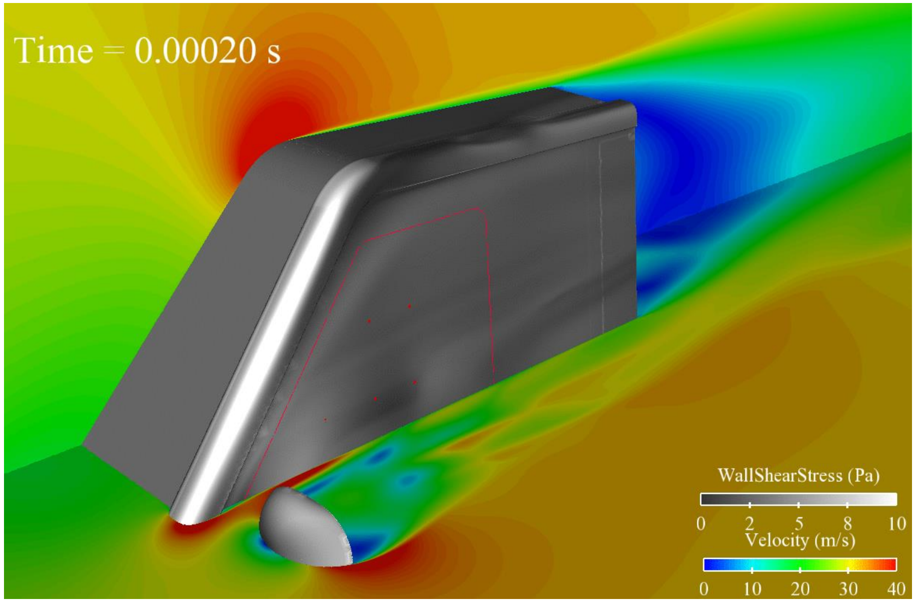


OpenFOAM for Industry

Next steps – Upcoming features

- OpenFOAM enhancements for AVA:
  - Aero-Vibro-Acoustic technology: enhancement and validation of OpenFOAM for aeroacoustic simulations.
    - Including boundary conditions and acoustic treatment.





OpenFOAM for Industry

Next steps – Upcoming features

- HPC
  - Improvements to performance and memory (linear solver)
    - Gather-scatter
    - All-to-all communication
    - On-the-fly communication schedule
  - Applies to
    - Geometry initialisation calculations
    - Dynamic mesh calculations
    - Lagrangian

  - Parallel I/O – with a high priority
  - Scalability of
    - conjugate heat transfer simulations
    - Lagrangian
    - All solvers (GAMG)
  - … watching your work with interest…

OpenFOAM for Industry

Next steps – Upcoming features

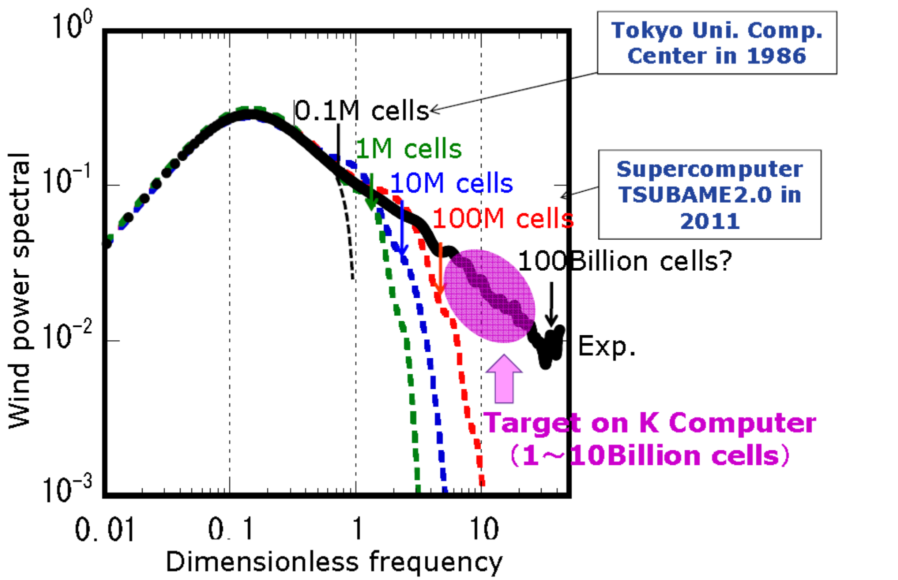
- HPC
  - Strongly accredit the efforts of research institutes and forums like here today
  - In particular:
    - RIST (Japanese Research Organisation for Information Science and Technology)

RIST

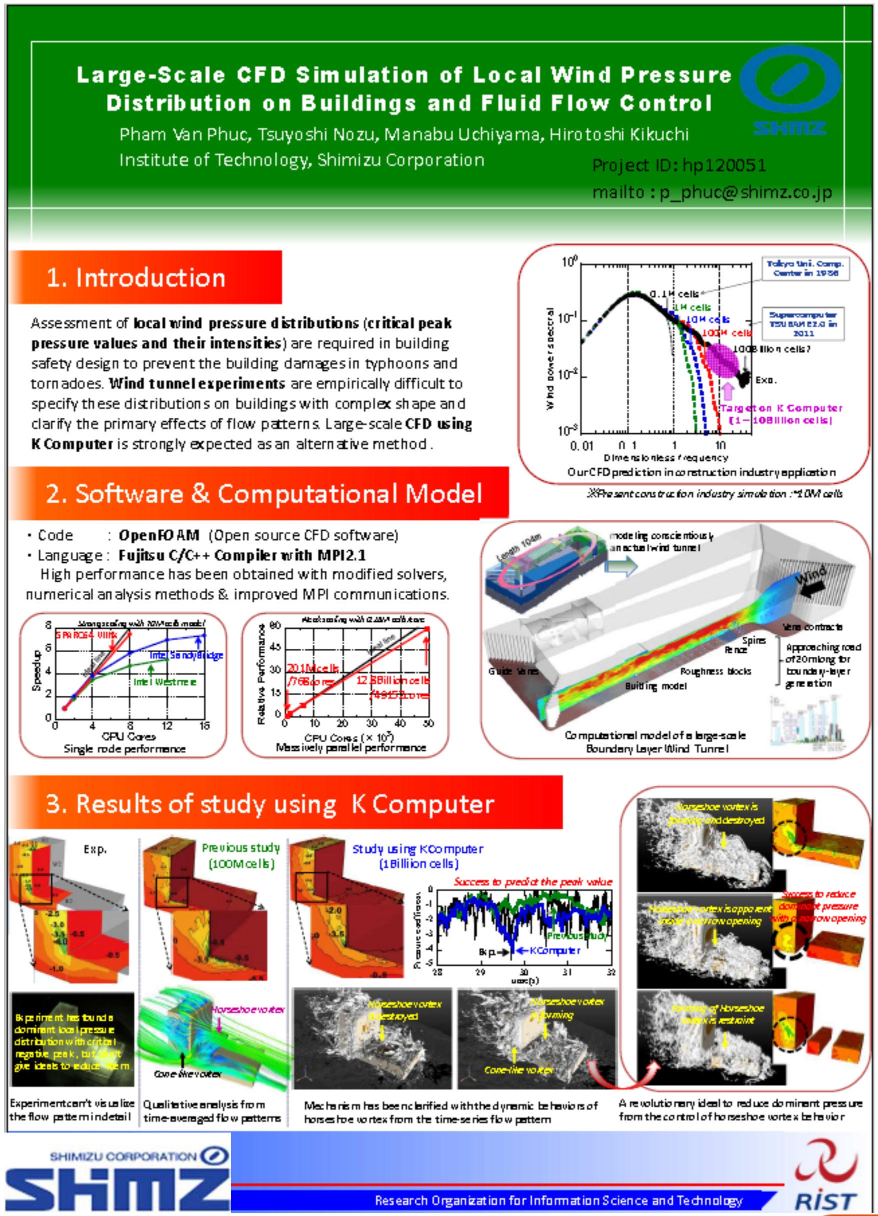
OpenFOAM for Industry

Next steps – Upcoming features

- HPC
  - Stationary - transient



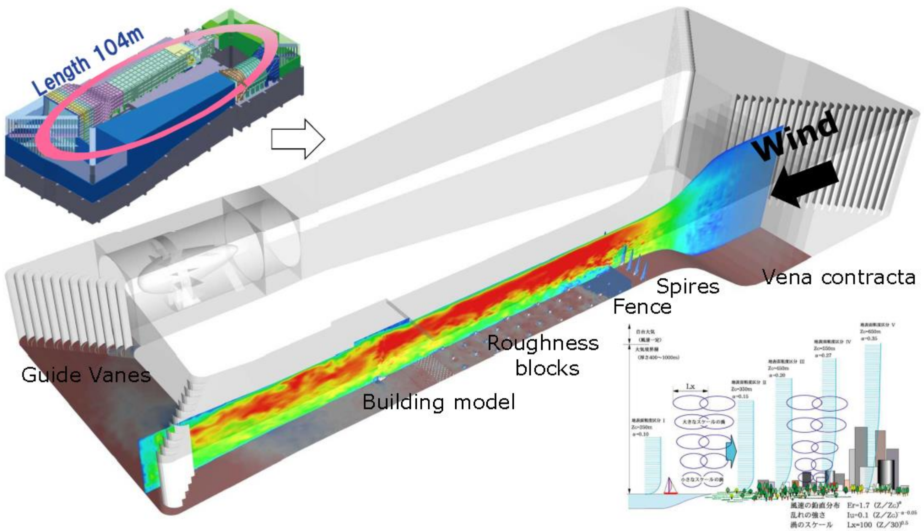
※Present construction industry simulation:~10M cells



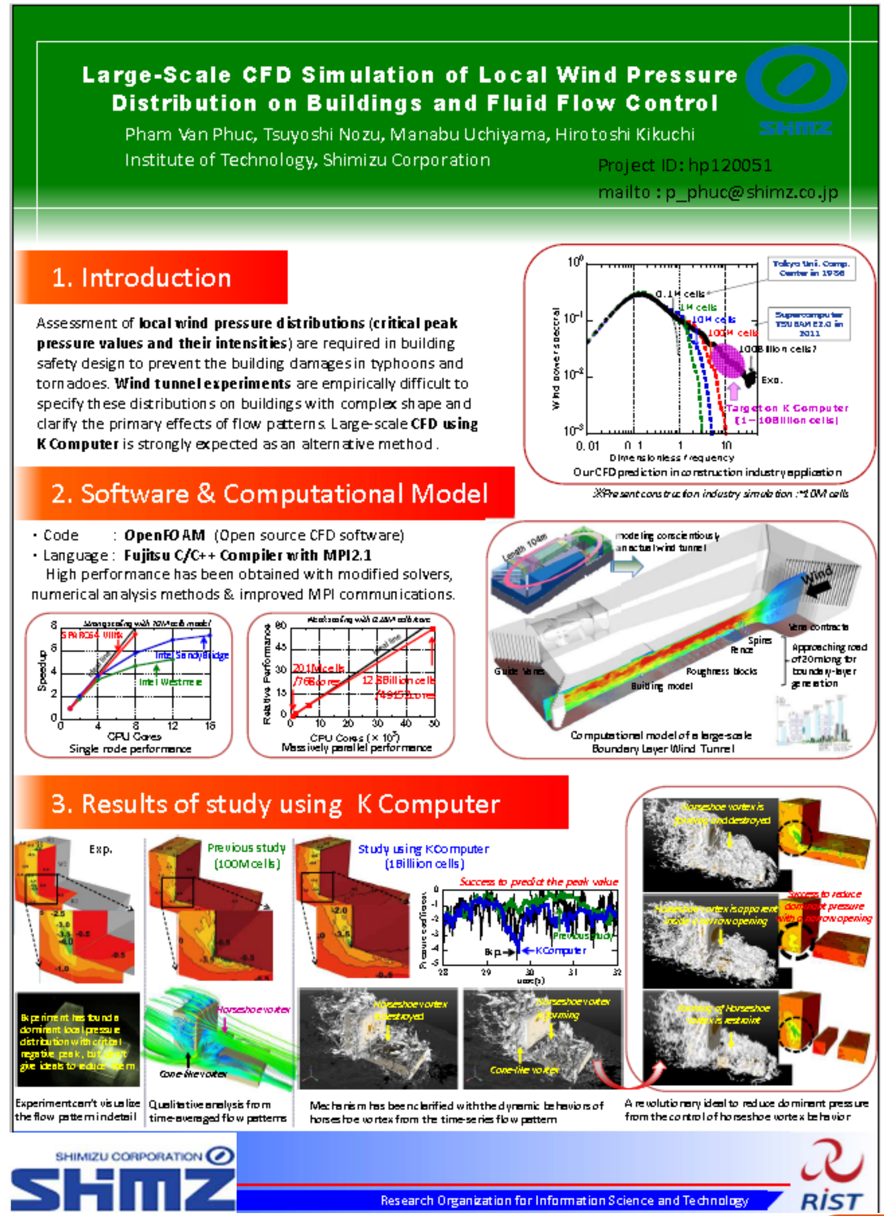
OpenFOAM for Industry

Next steps – Upcoming features

- HPC
  - Stationary - transient



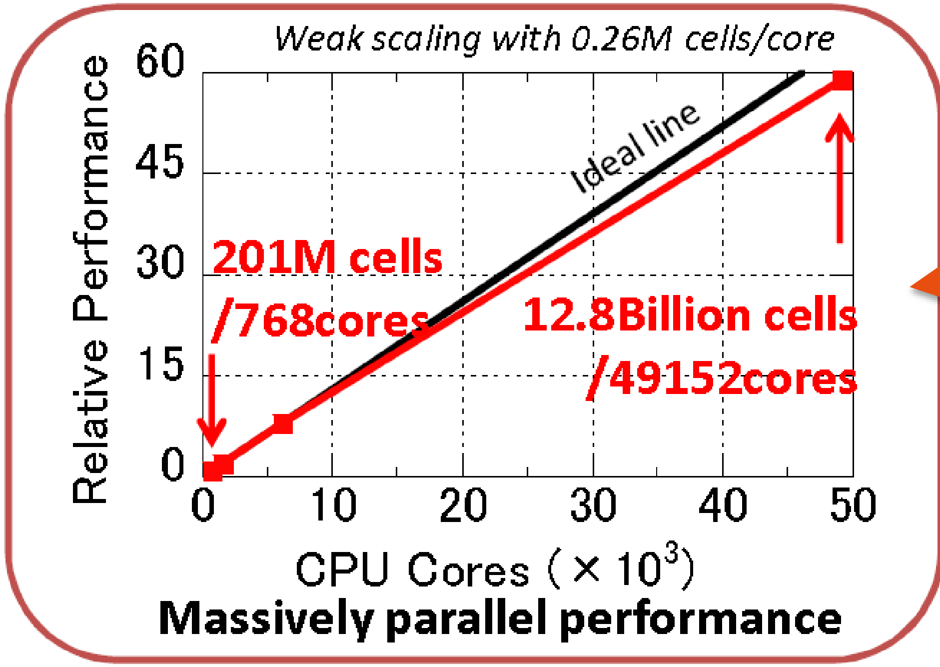
A large-scale simulation of a 6.4billion cells using Large Eddy Simulation (LES)

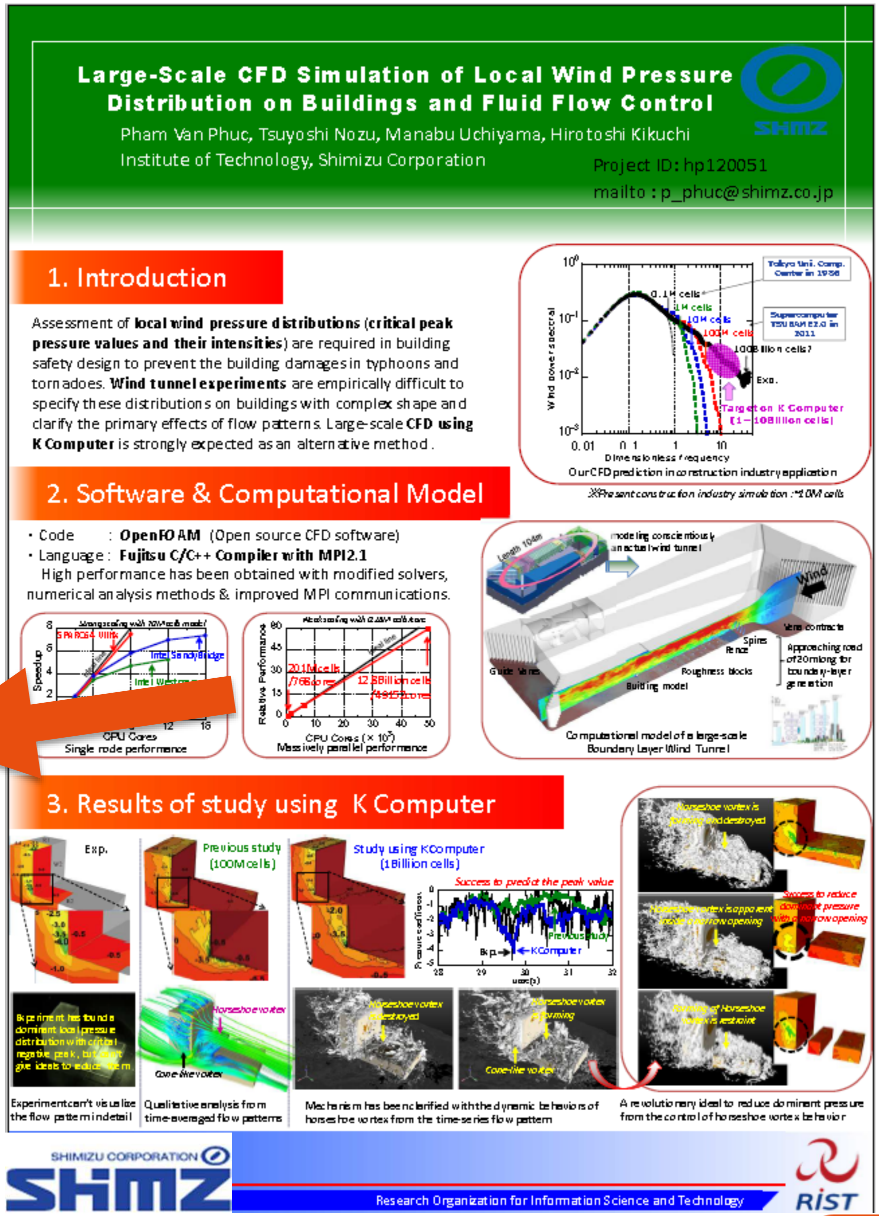


OpenFOAM for Industry

Next steps – Upcoming features

- HPC
  - Stationary - transient

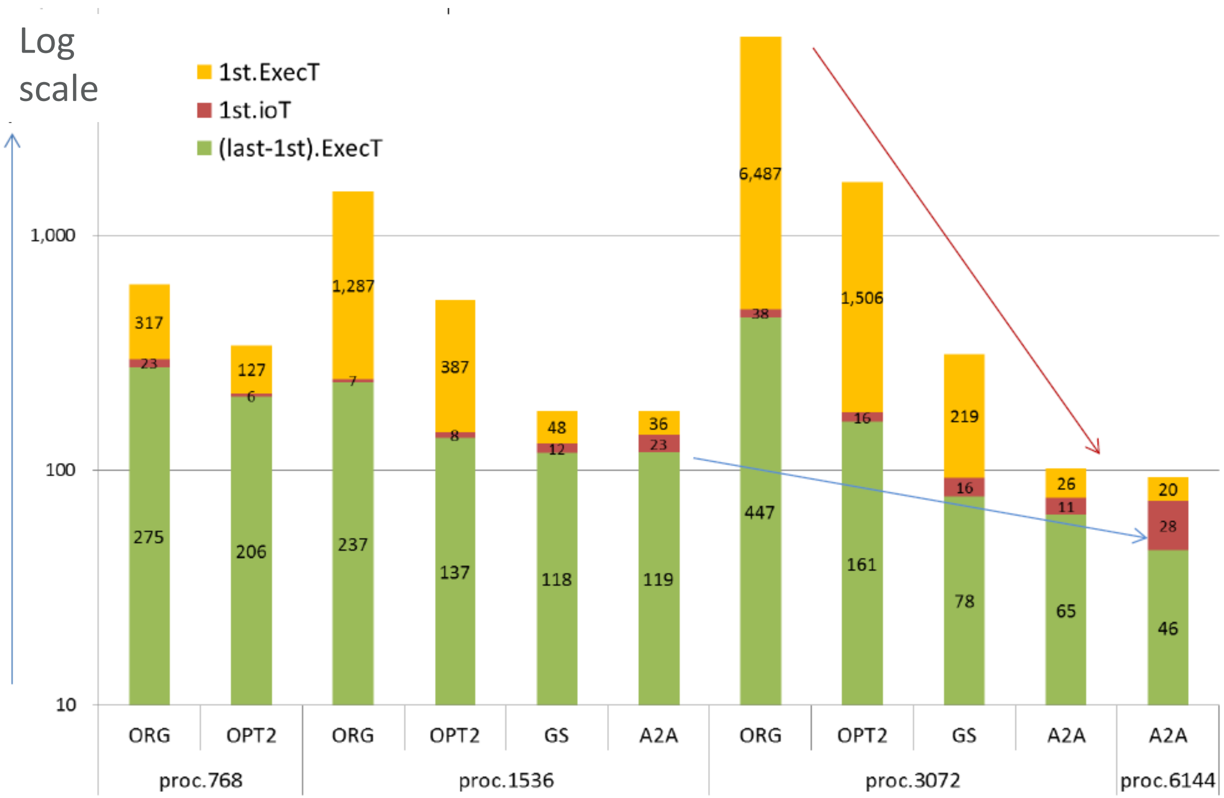




OpenFOAM for Industry

Next steps – Upcoming features

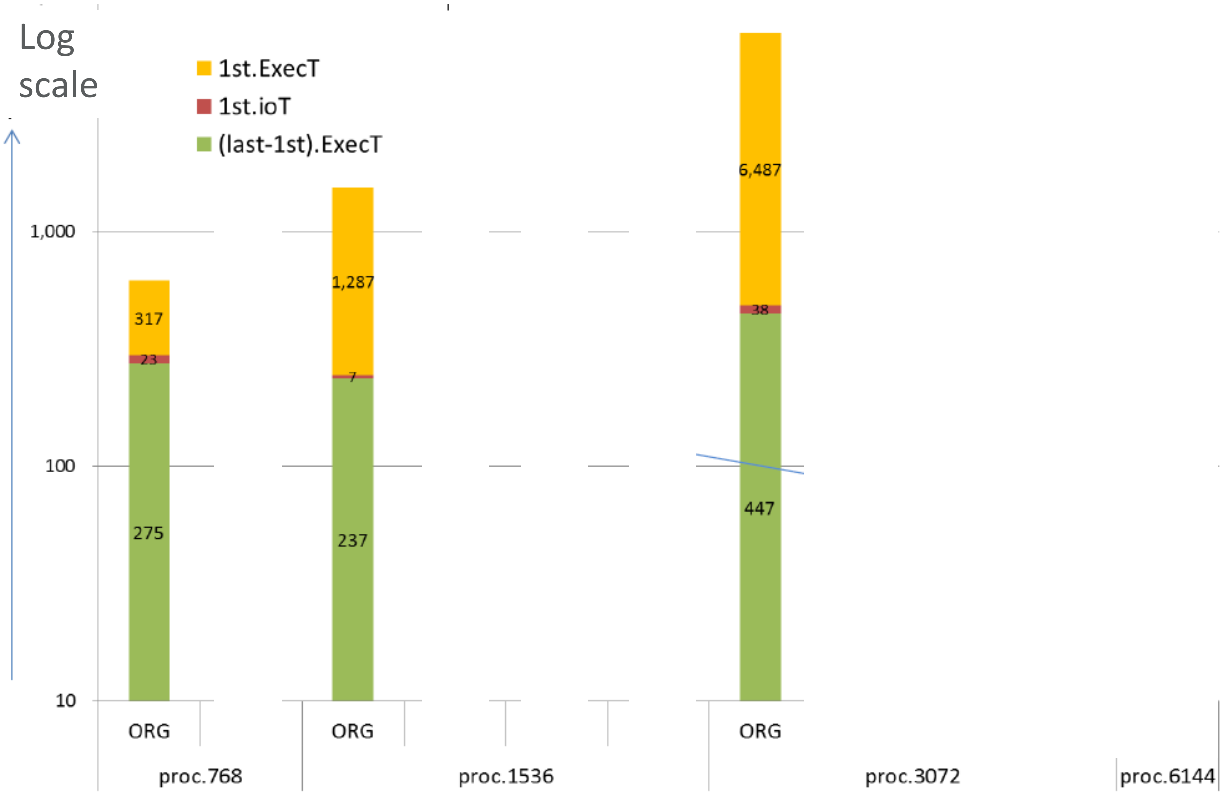
- HPC : Dynamic Mesh performance – pimpleDyMFoam (500million cells)



OpenFOAM for Industry

Next steps – Upcoming features

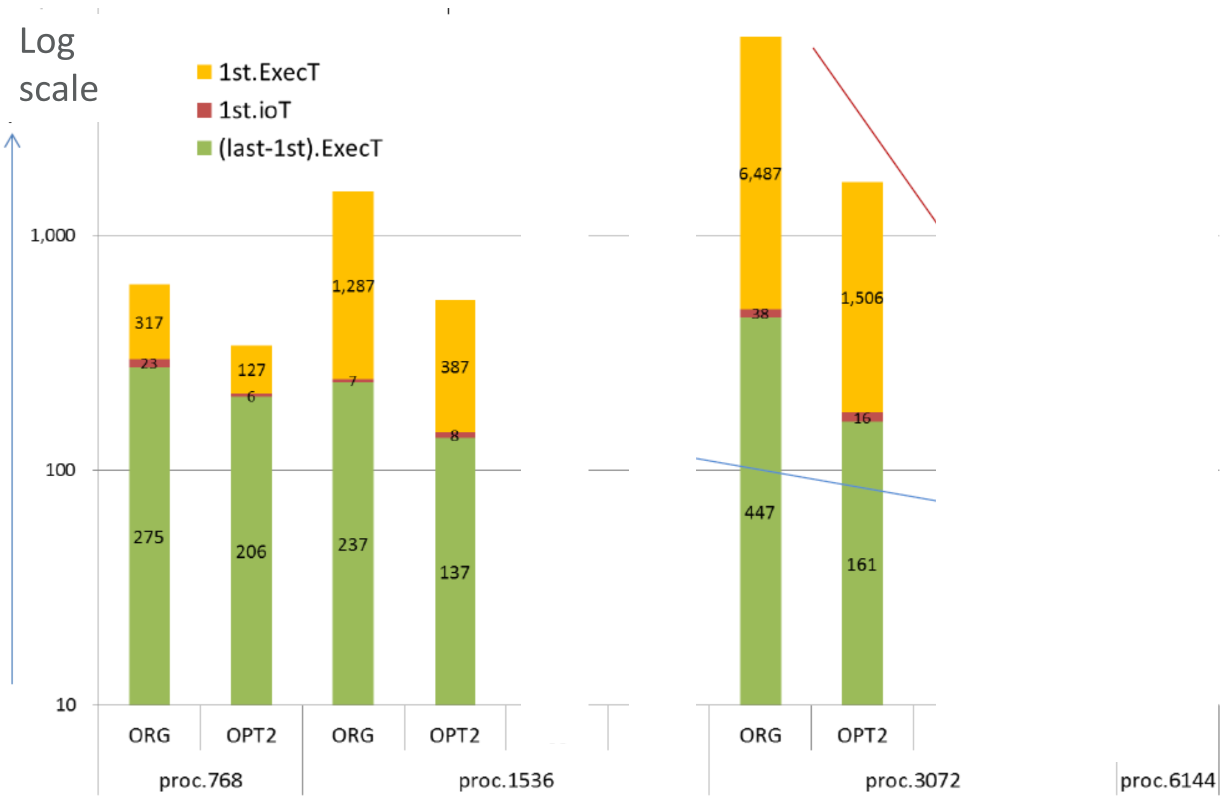
- HPC : Dynamic Mesh performance – pimpleDyMFoam (500million cells)
  - Original code compilation on K-computer



OpenFOAM for Industry

Next steps – Upcoming features

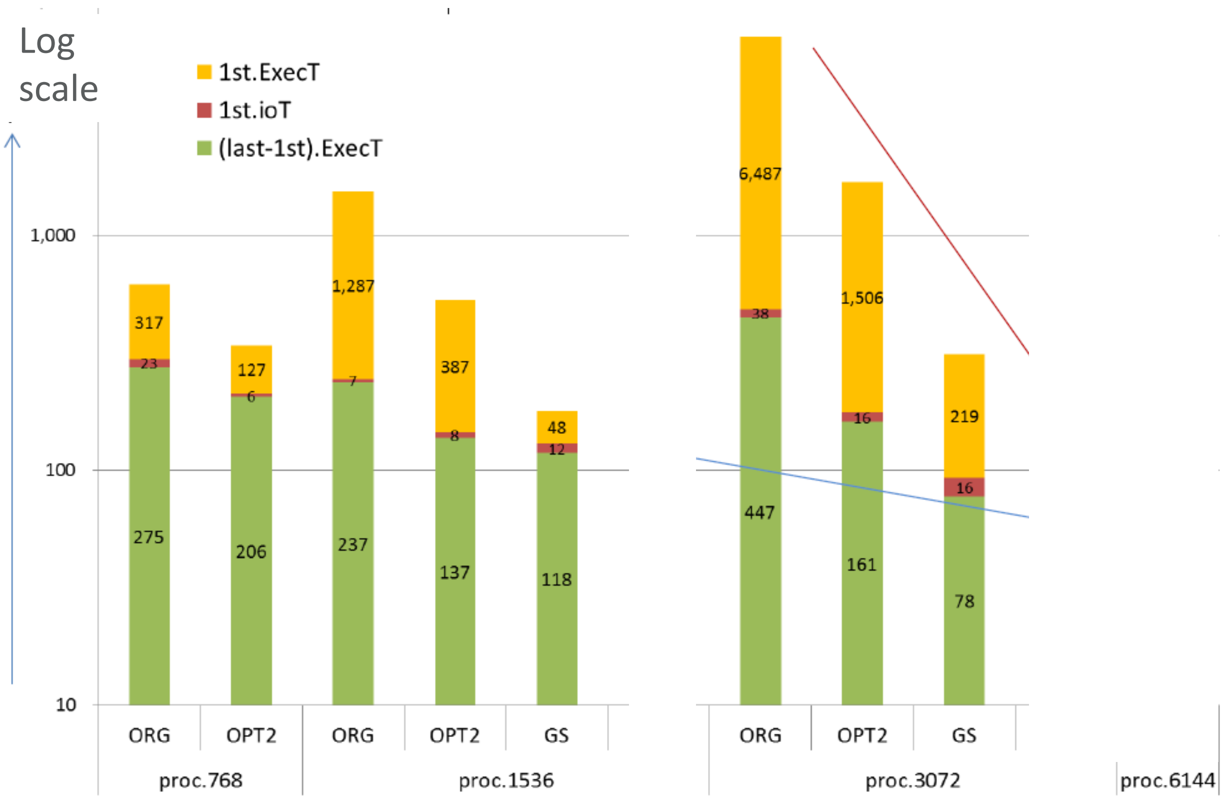
- HPC : Dynamic Mesh performance – pimpleDyMFoam (500million cells)
  - Compilation Optimisations on K-computer



OpenFOAM for Industry

Next steps – Upcoming features

- HPC : Dynamic Mesh performance – pimpleDyMFoam (500million cells)
  - Multi-pass “block-communications” - gather-scatter reverse ordering



<u>e.g. 0-7proc gather</u>

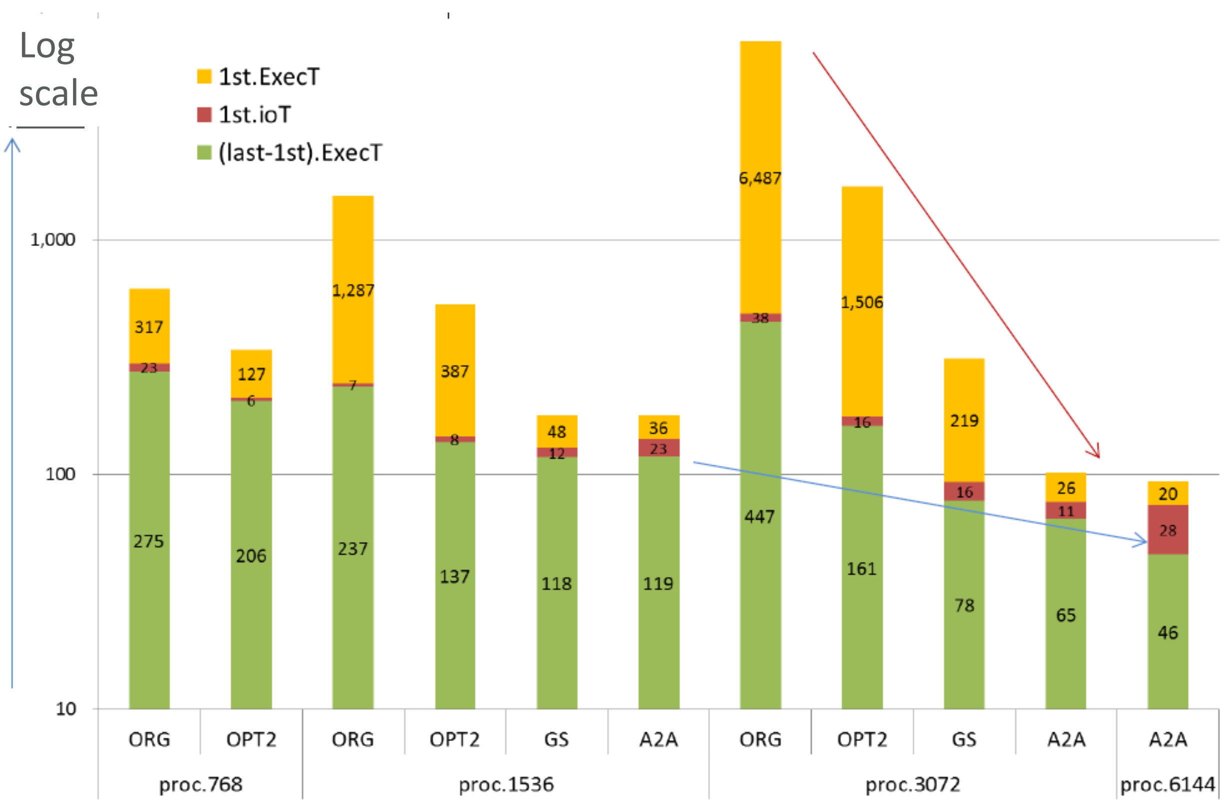
Pass 1: 1 > 0
3 > 2
5 > 4
7 > 6
Pass 2: 2 > 0
6 > 4
Pass 3: 4 > 0

<u>Scatter</u>
in reverse

OpenFOAM for Industry

Next steps – Upcoming features

- HPC : Dynamic Mesh performance – pimpleDyMFoam (500million cells)
  - Tuning of all-to-all communications



e.g.

- "nxn" proc comms via master process

- reduced to "n" comms via local processor data exchange

OpenFOAM for Industry

Summary

- ESI Group and OpenCFD are committed to support OpenFOAM's growth as required by the community:
  - Target: Increase usability as well as robustness, to make it comparable to commercial tools.

- ESI Group supported, supports, and will continue supporting OpenFOAM.

… watching your work with great interest…

OpenFOAM for Industry

Agenda

- Introduction
- Updates in OpenFOAM
- Next steps
- Q&A

Thank you