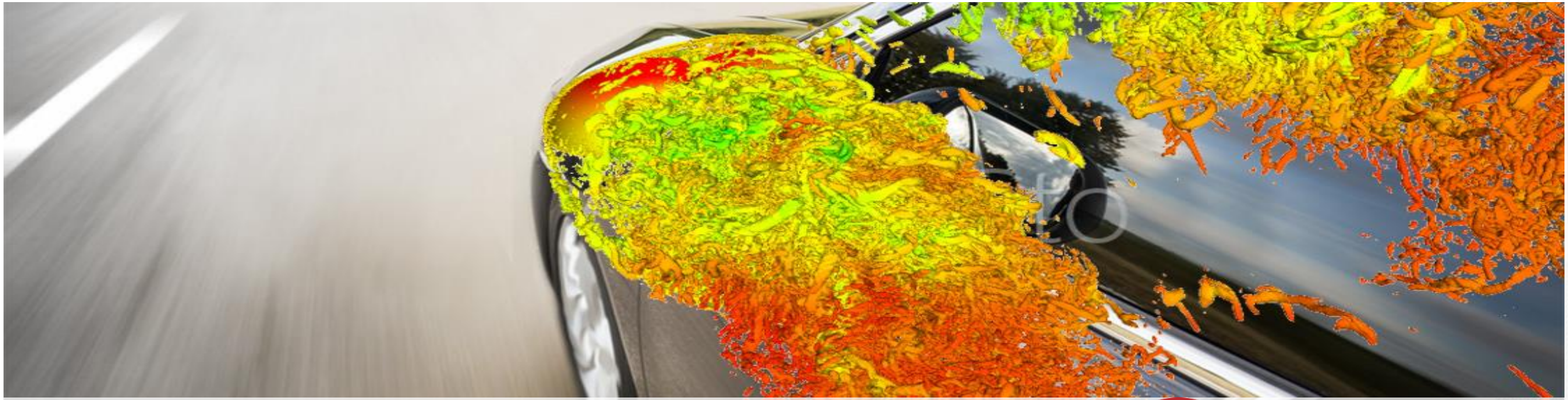
Delivering a professional OpenFOAM®

We stand on the shoulders of giants (2015)
Custodians of OpenFOAM® (2016)



Fred Mendonça, ESI-OpenCFD

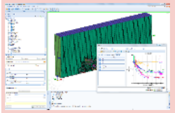
ARCHER Users Webinar, 11th October 2017

OpenFOAM

OpenFOAM is the registered trademark owned by ESI / OpenCFD

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

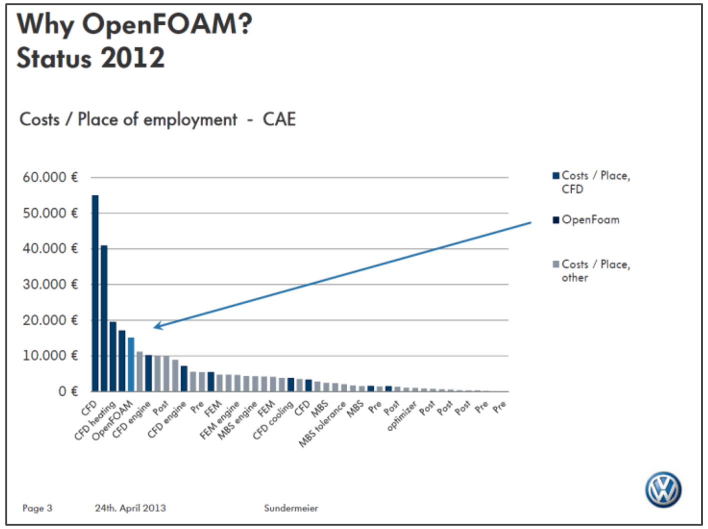
OpenFOAM Support

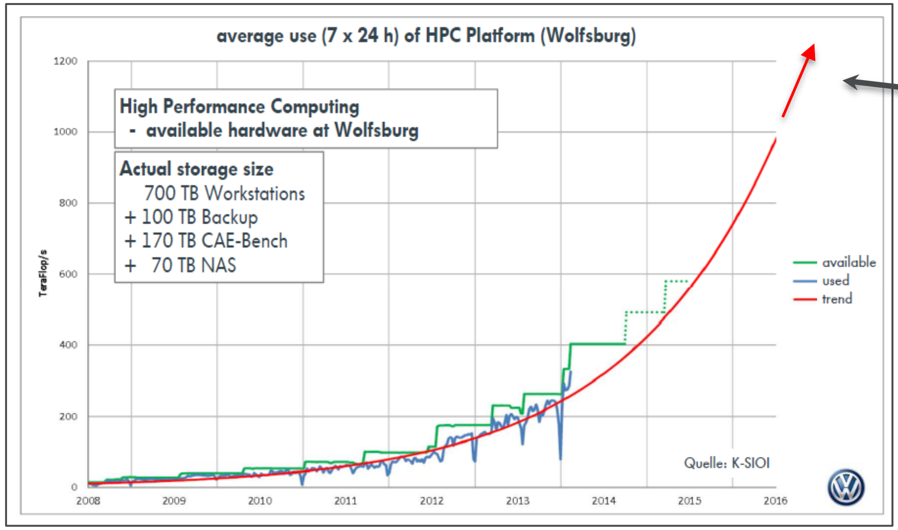
	Value and benefit	Return
Development 	**Customized** based on **need** and **application**	Sponsor only the **developments you need** Benefit from others’
Consulting 	**Knowledge transfer** and **on-site** services	**Additional expertise** whenever required
Support 	**Best practices**, ticket based, **guaranteed response time**	**Global standard** consistent across borders **Experienced, dedicated**
 Training	**Basic** users, **Advanced** users **Application** based **Accredited**	**Standardised material** worldwide, updated based on **latest release**
Visual Work Flow Process Management 	**Customizable, scriptable** GUI **Basic** and **advanced** physics (e.g. CHT, VoF, OverSet)	**Fast learning**, easily **adaptable to process**

ESI Value Proposition	Value and Benefit
	**Enterprise-level engagement** with the OpenFOAM **Originator**; **Developer**; **Release, Test, Maintenance** and **Support authority**

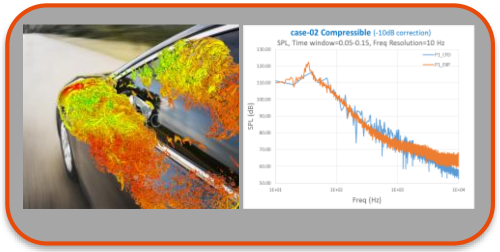
ESI Global Resources	License-based RoI	Technology RoI
Strong partner **Technology expert** **Flexible *Licensing* and *Service* models** **End-to-end *Process* and *Physics***	**Typically at least 60% cost redeployment based on software ownership**	**Partner Programmes** **Process automation** **Full support and maintenance** **Custom developments and captured best practices** **Worldwide provider and technology experts**

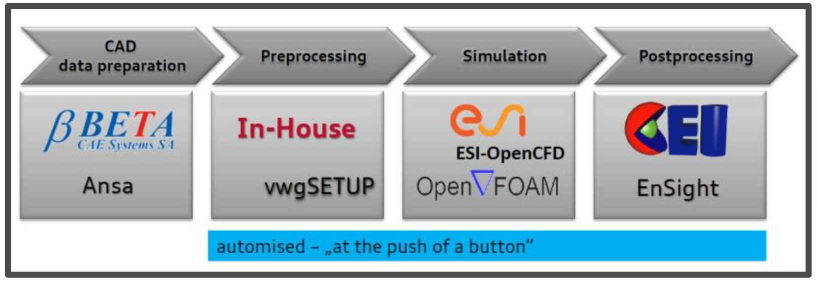
CFD Democretisation through OpenFOAM





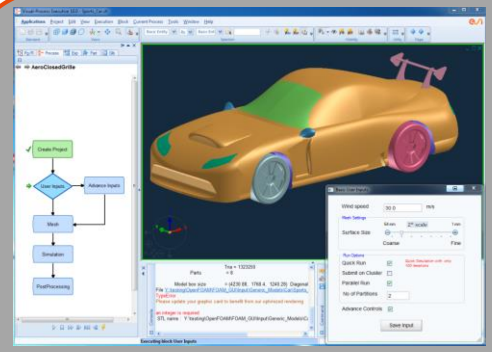
Wind Noise





Efficiency Improvement

Visual-CFD



Process Automation/Customization based on ESI Visual developed for all ESI customers WW

OpenCFD – Commitment to OpenFOAM Users

Development and Release Schedule

- OpenCFD owns the OpenFOAM trademark

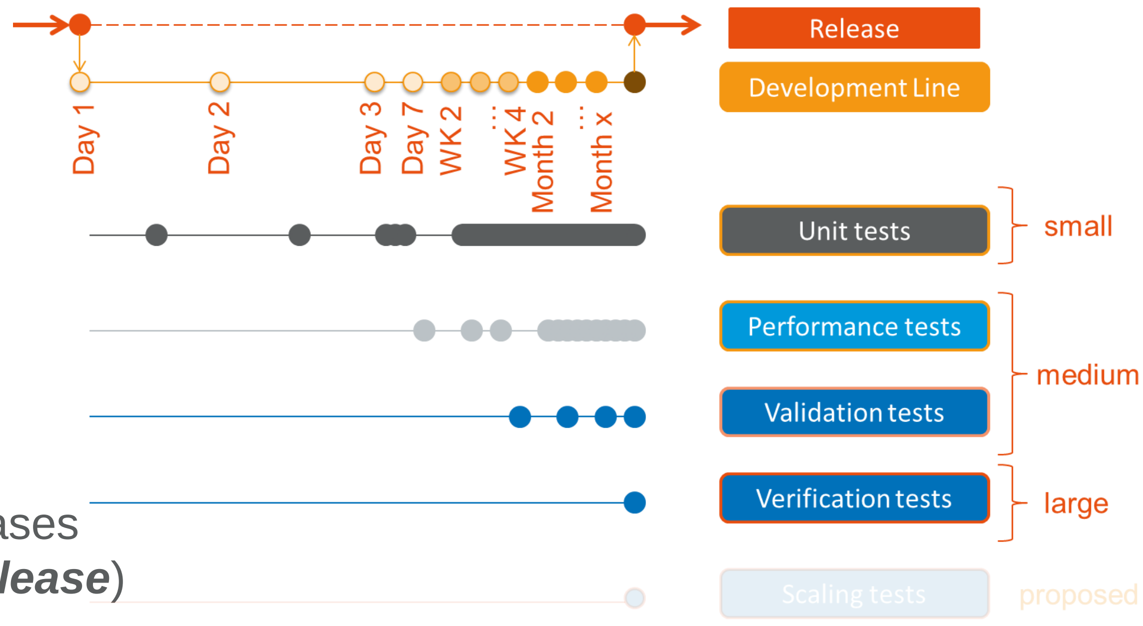
- Professional Six-monthly Development and Release cycle, including
  - New developments
  - Consolidated bug-fixes
  - Overhaulled testing procedure for Quality Assurance
  - Release and Development repositories in GitLab https://develop.openfoam.com
    - Master branch
    - Develop branch (includes .org version merge) > Master > Release
    - Community Repositories > Develop

- OpenFOAM-plus releases so far
  - OpenFOAM-v3.0+ on Jan 13th 2016
  - OpenFOAM-v1606+ on June 30th 2016
  - OpenFOAM-v1612+ on 23rd December 2016
  - OpenFOAM-v1706 on 30th June 2017

Quality Assurance testing

Release-cycle test battery

- Small (unit) test loop
  - ***Nightly*** tests to ensure no cross-feature breakage
  - Approximately 550 feature-by-feature tests
  - Execution time ~ 4 hours (nightly)
- Medium test loop
  - Tutorials and small validation tests
  - Approximately 300 tests
  - Execution time ~ 2 days (***weekly***)
- Large test loop
  - ~20 Client cases
  - Several million steady and transient cases
  - Execution time ~ 1 week (***once per release***)

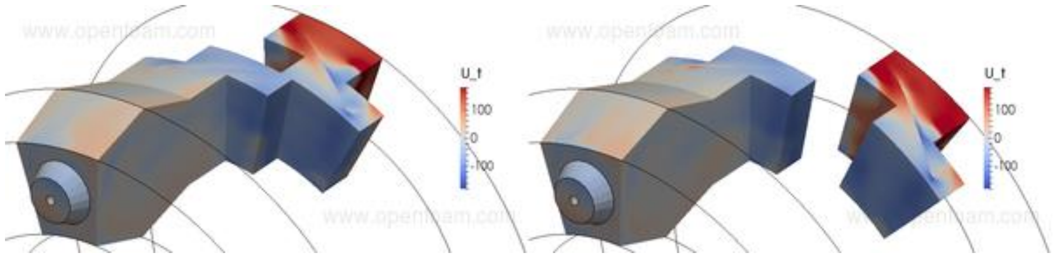


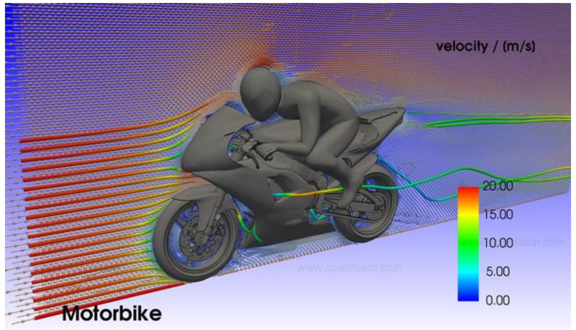
- Test loops grow with every new functionality released

OpenFOAM

Updates in OpenFOAM: OpenFOAM-v3.0+ (Jan 2016)

- Features developed in 2014-2015 released in v3.0+
  - Pre-processing
  - Meshing
  - Solver
    - Initialisation
    - Heat transfer / CHT
    - Boundary conditions
    - Turbulence
    - Run-time controls
  - Post-processing





- ‘External‘ Contributors to OpenFOAM-v3.0+
  - DES and new family of k-ω-SST models
  - Inter-region heat transfer

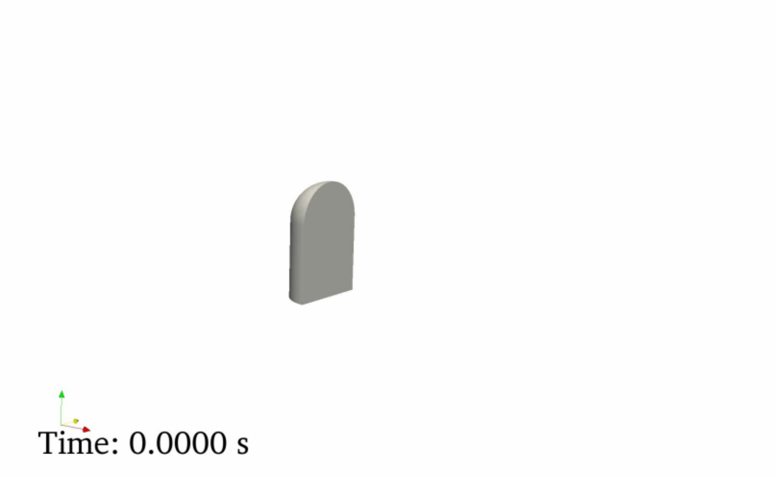
CFD Software E+F GmbH
CFD Software Entwicklungs- und Forschungsgesellschaft mbH

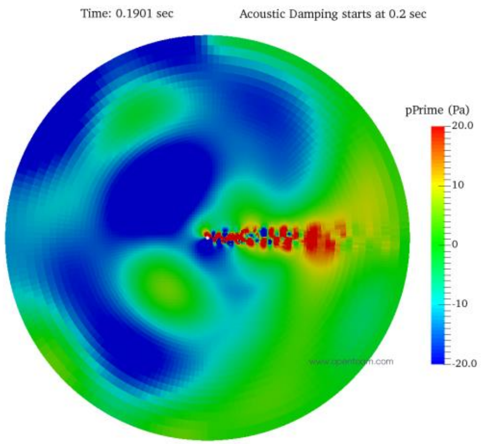
CFD+engineering
Ingenieurbüro für Strömungsmechanik

OpenFOAM

Updates in OpenFOAM: OpenFOAM-v1606+ (June 2016)

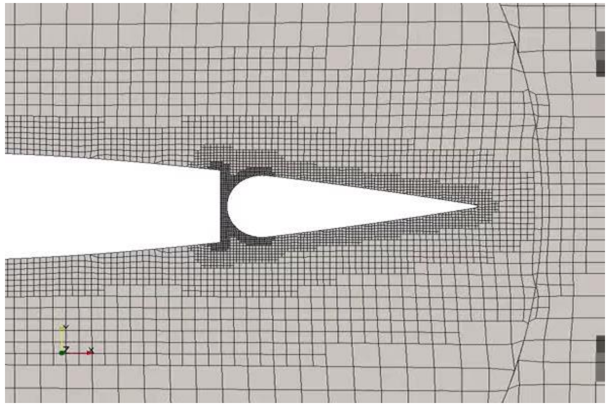
- OpenFOAM-v1606+ (June2016)
  - Message passing performance scaling
    - Gather-scatter order
    - All-to-all processor communications
  - Performance profiling (Bernhard Gschaider)
  - DFSEM (help from Ruggero Poletto)
  - Validated Aeroacoustics enhancements and coupling to Acoustic codes



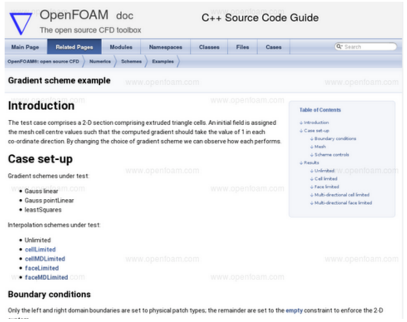


OpenFOAM

Updates in OpenFOAM: OpenFOAM-v1612+ (Dec 2016)

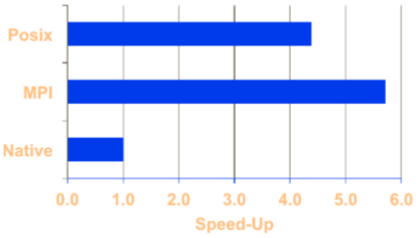


- OpenFOAM-v1612+ (Dec2016)
  - VoF sampling and Lagrangian particle injection
  - Eddy-Dissipation concept combustion model
  - Wave modelling and damping (contribution from IH Cantabria)
  - Meshing improvements to AMI and morphing
  - Documentation improvements
  - Community Repository
    - isoAdvector
    - Efficient I/O for HPC – Adios libraries
    - … to contribute, please register on the GitLab site *https://develop.openfoam.com/Development/OpenFOAM-plus*



240 Processors

- Test study
  - 100 time steps.
  - Writing each time step.
- OpenFOAM
  - 178 s: no write
  - 3426 s: write
- ADIOS transport
  - 226 s: null (no write)
  - 1614 s: MPI
  - 1318 s: POSIX



  - Community-assembled on-line tutorials
    - Thanks to initiative from Joszef Nagy supported by Andy Heather
  - Significant enhancements to the online Documentation on www.openfoam.com

Overview

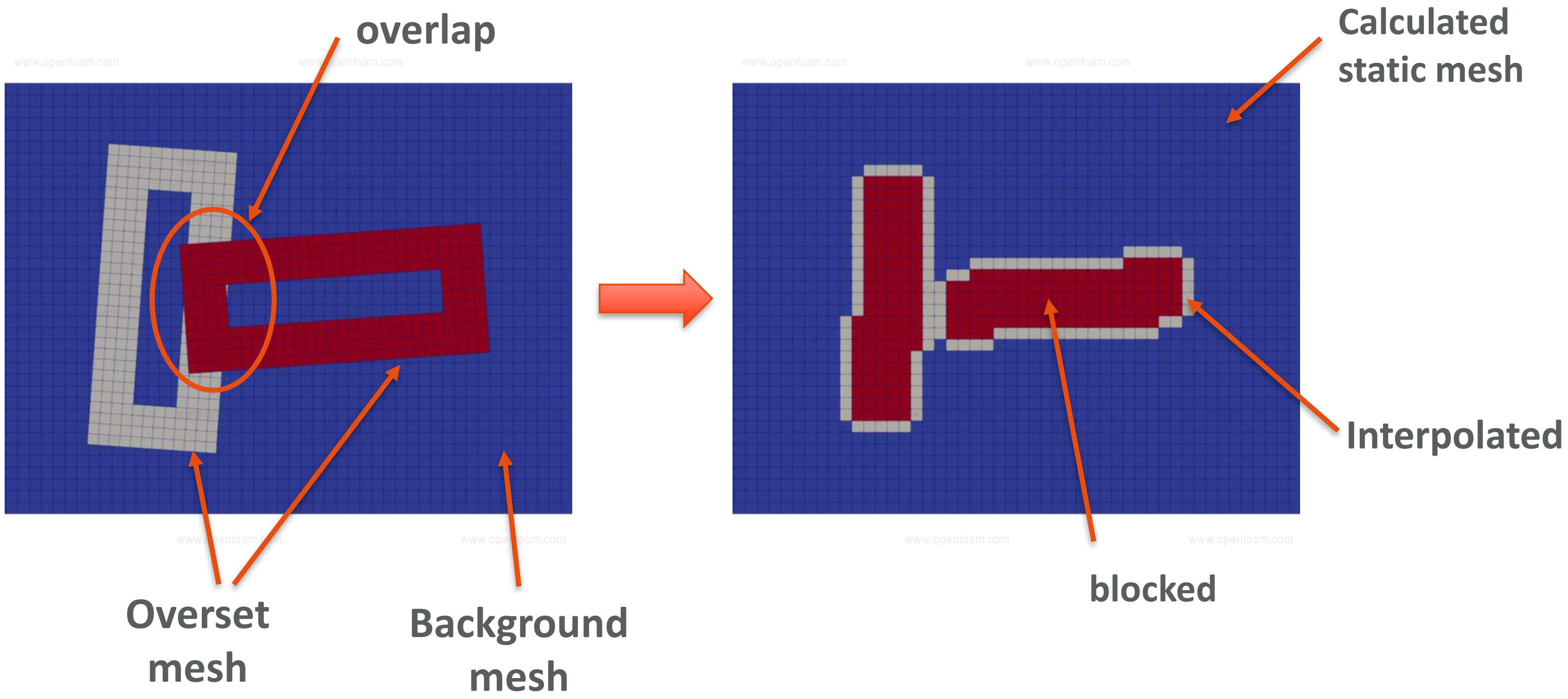
New development highlights and contributions in v1706

- **Meshing**
  - Overset mesh functionality (Chimera grids)
- **Physical models**
  - Joule heating source term
  - Lumped point FSI
- **Solvers**
  - Solver for low Mach number flows
  - Iso-surface-based interface capturing for VOF
- **Boundary conditions**
  - New wave generation models
- **Numerics**
  - Improved second order restart
  - Updated time step control
- **Installation**
- **Usability improvements**
  - Command-line bash completion

Overset mesh

overview

- First release of the Overset mesh
  - *Cell-to-cell mapping* between disconnected meshes

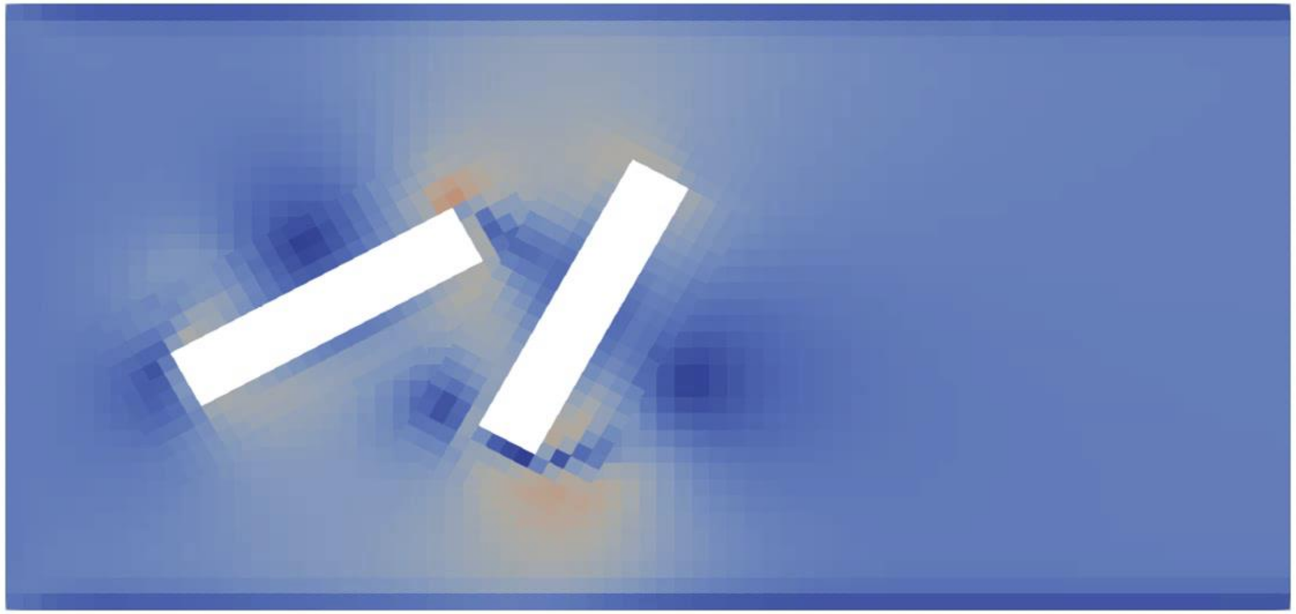


Overset mesh

mesh

overview

- First release of the Overset mesh
  - Released for specific solvers with *"over"* in its name

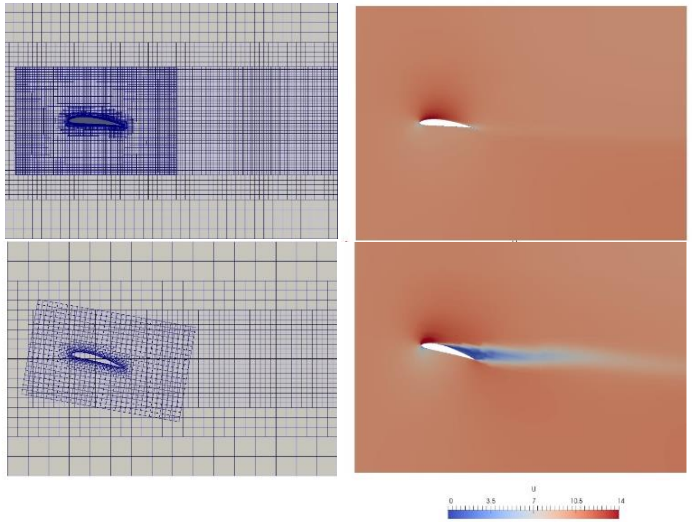


Overset mesh

Solvers and Applications (1/3)

mesh

- overLaplacianDyMFoam
  - Eulerian flows
  - $FOAM_TUTORIALS/basic/overLaplacianDyMFoam/heatTransfer
- overSimpleFoam
  - steady state parametric studies of incompressible flow
  - $FOAM_TUTORIALS/incompressible/overSimpleFoam/aeroFoil

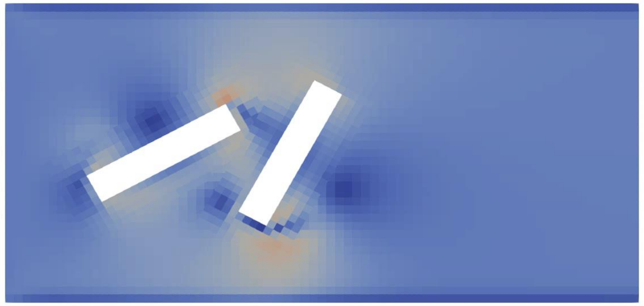


Overset mesh

mesh

Solvers and Applications (2/3)

- overPimpleDyMFoam
  - transient incompressible flows (moving mesh)
  - $FOAM_TUTORIALS/incompressible/overPimpleDyMFoam/twoSimpleRotors



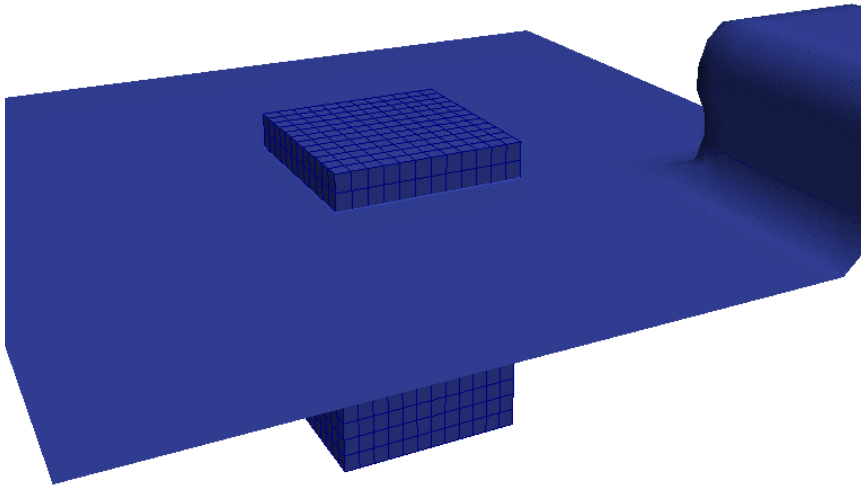
- overRhoPimpleDyMFoam
  - transient compressible flows (moving mesh)
  - Best practices tutorial in preparation

Overset mesh

Solvers and Applications (2/3)

mesh

- *overInterDyMFoam*
  - multiphase incompressible VOF (moving mesh)
  - `$FOAM_TUTORIALS/multiphase/overInterDyMFoam/floatingBody`

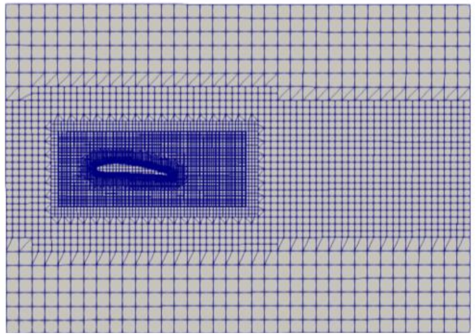


Overset: Verification and Validation

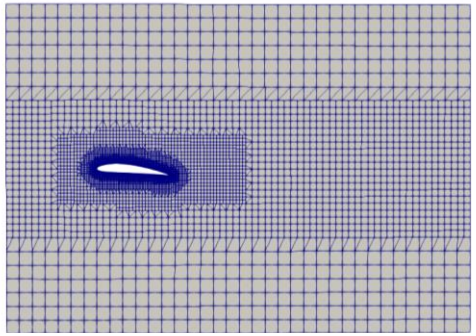
overSimpleFoam vs simpleFoam

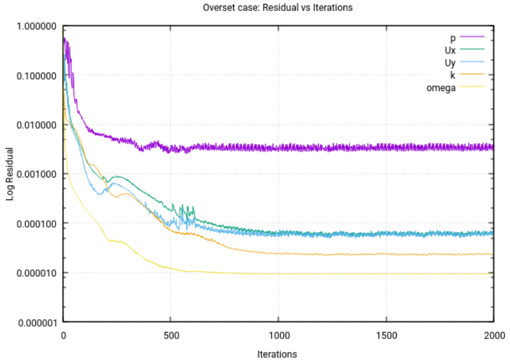
- Mesh
  - Slight difference in background refinement
  - Overset is skewed (not 100% overlap)
  - Overset: 112.4% Original: 100%
- Residuals
  - Overset: equiv. stability Original: equiv. stability
  - Overset: Resid. < 5.0e^{-3} Original: Resid. < 1.0e^{-3}
- Forces
  - Lift = 0.5% difference Drag = (0.002) counts
- CPU Performance (for 2000 iterations)
  - Overset: 180% Original: 100%

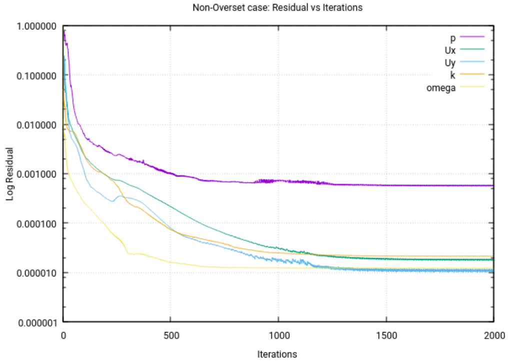
OVERSET

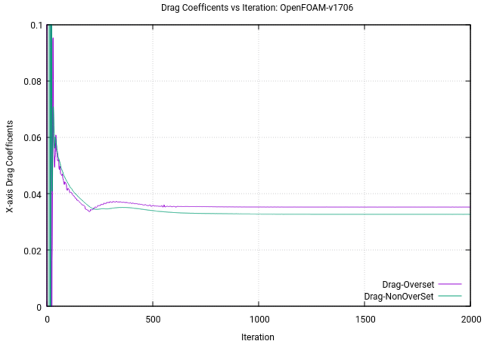


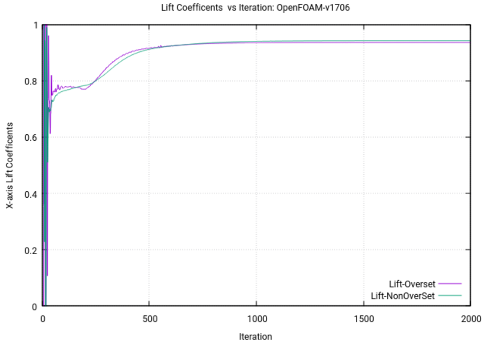
ORIGINAL



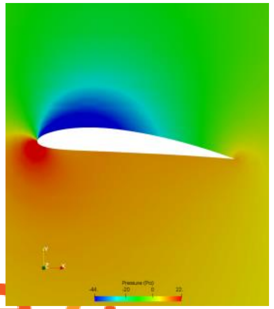




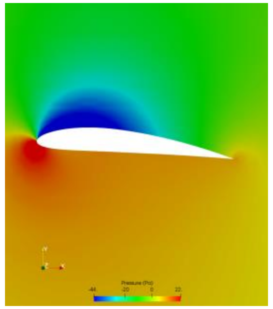




OVERSET

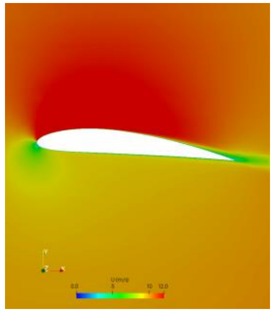


ORIGINAL

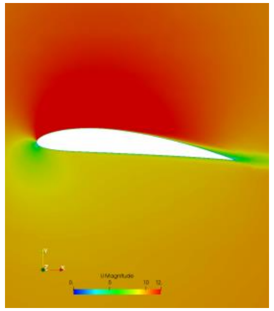


Pressure

OVERSET



ORIGINAL



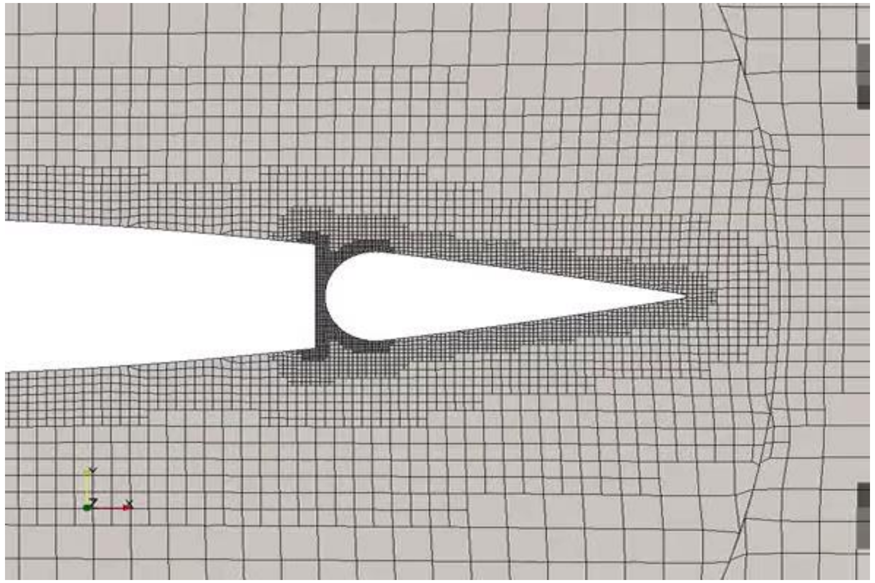
Velocity

Overset: Verification and Validation

overPimpleDyMFoam vs PimpleDyMFoam

- Mesh
  - Overset: 101.3% Original: 100%
- Residuals
  - Overset: equiv. stability Original: equiv. stability
- Forces
  - Lift = t.b.d Drag = t.b.d
- Performance
  - Overset: 195% Original: 100%

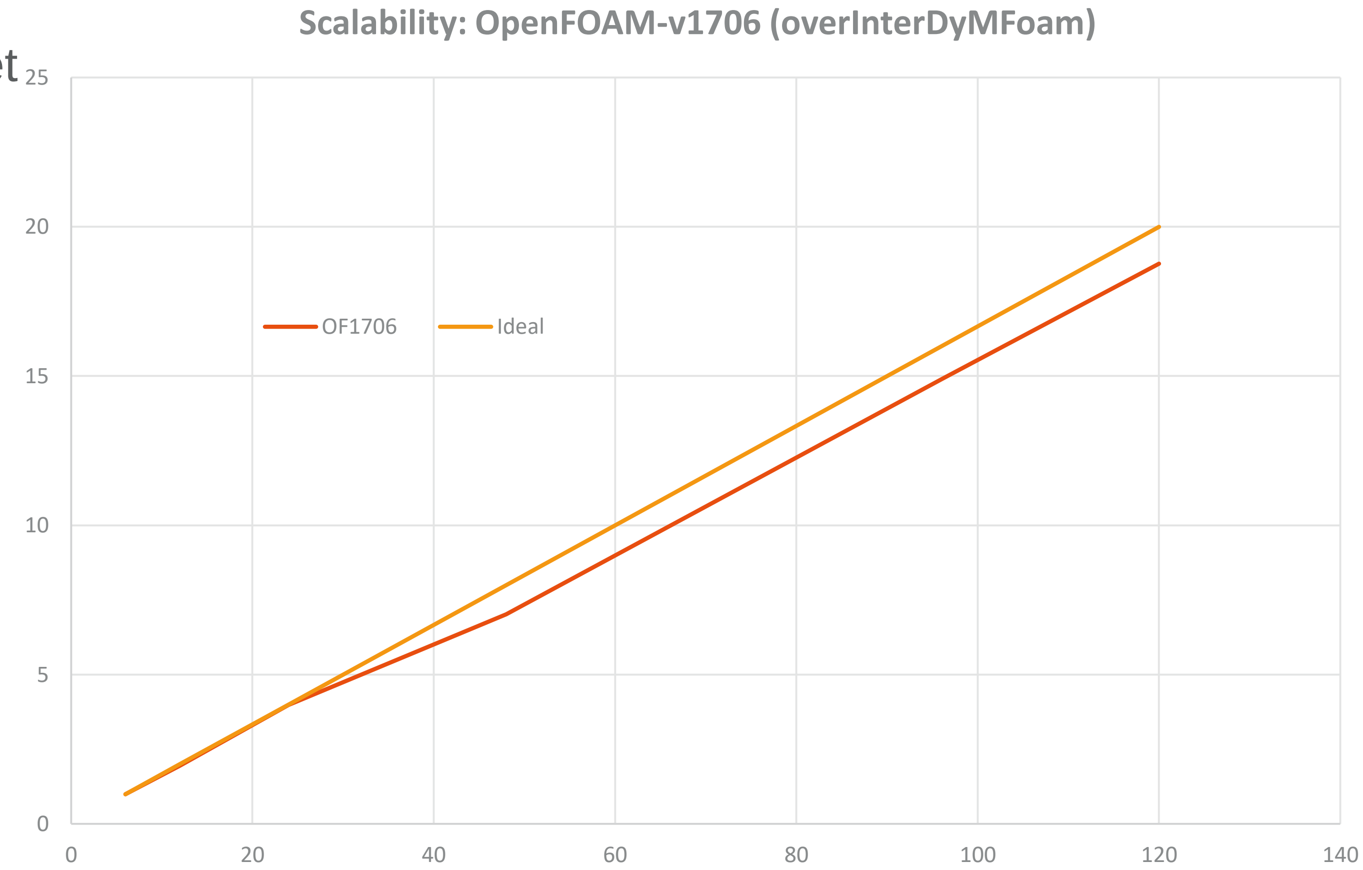
COMPARING TYPICAL AMI vs OVERSET



Overset: Verification and Validation

Parallel Performance

- overInterDyMFoam
  - 10 time-steps of the floating body overset tutorial
  - Scaling based on 6-proc datum
  - Scaling reported up to 120 processors
    - >90% efficiency



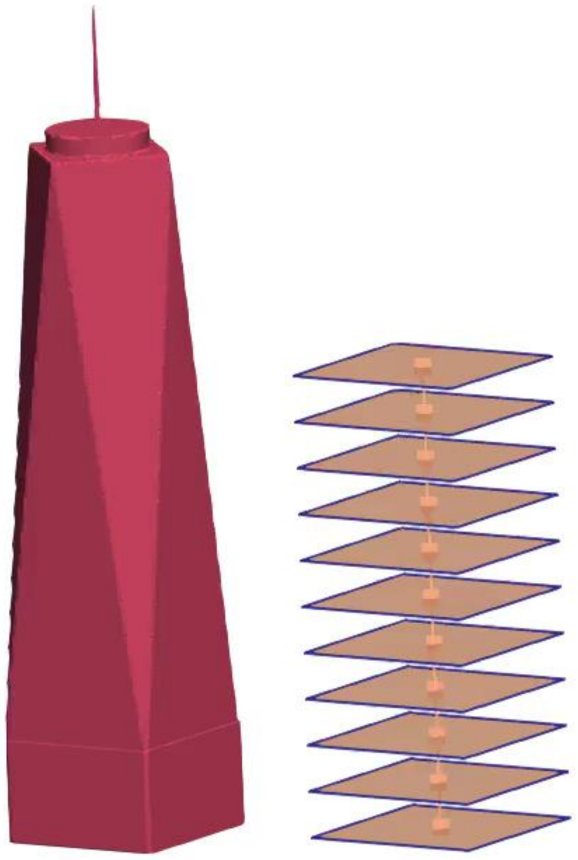
Lumped point FSI

lumpedPointMotion library

Physical model

- ***lumpedPointDisplacement*** - point displacement **boundary condition** is responsible for movement of patch points based on a **coarse representation** of the model using **lumped points**
- Integrated forces and moments acting on the patch are transferred to an external application
- Typical application is a structure loads by passing fluid

- Tutorial: `$FOAM_TUTORIALS/incompressible/lumpedPointMotion/building`



Low Mach number flows

Solvers

`rhoPimpleAdiabaticFoam`

- New approach to low-Mach number compressible flows
  - Temperature is treated assuming an idealised adiabatic process to comply with $\gamma = C_p/C_v$ (=1.4)
  - Modified Rhie-Chow interpolation results in insensitivity to time-step size and under-relaxation factor

  - Most useful in acoustic simulations, reducing spurious pressure wave generation at mesh interfaces
- Original contribution from CFD Software E+F GmbH
- *Reference:*
  - *Knacke, T. (2013). Potential effects of Rhie & Chow type interpolations in airframe noise simulations. In: Schram, C., Dnos, R., Lecomte E. (ed): Accurate and efficient aeroacoustic prediction approaches for airframe noise, VKI LS 2013-03.*
- Tutorial: `$FOAM_TUTORIALS/compressible/rhoPimpleAdiabaticFoam/rutlandVortex2D`

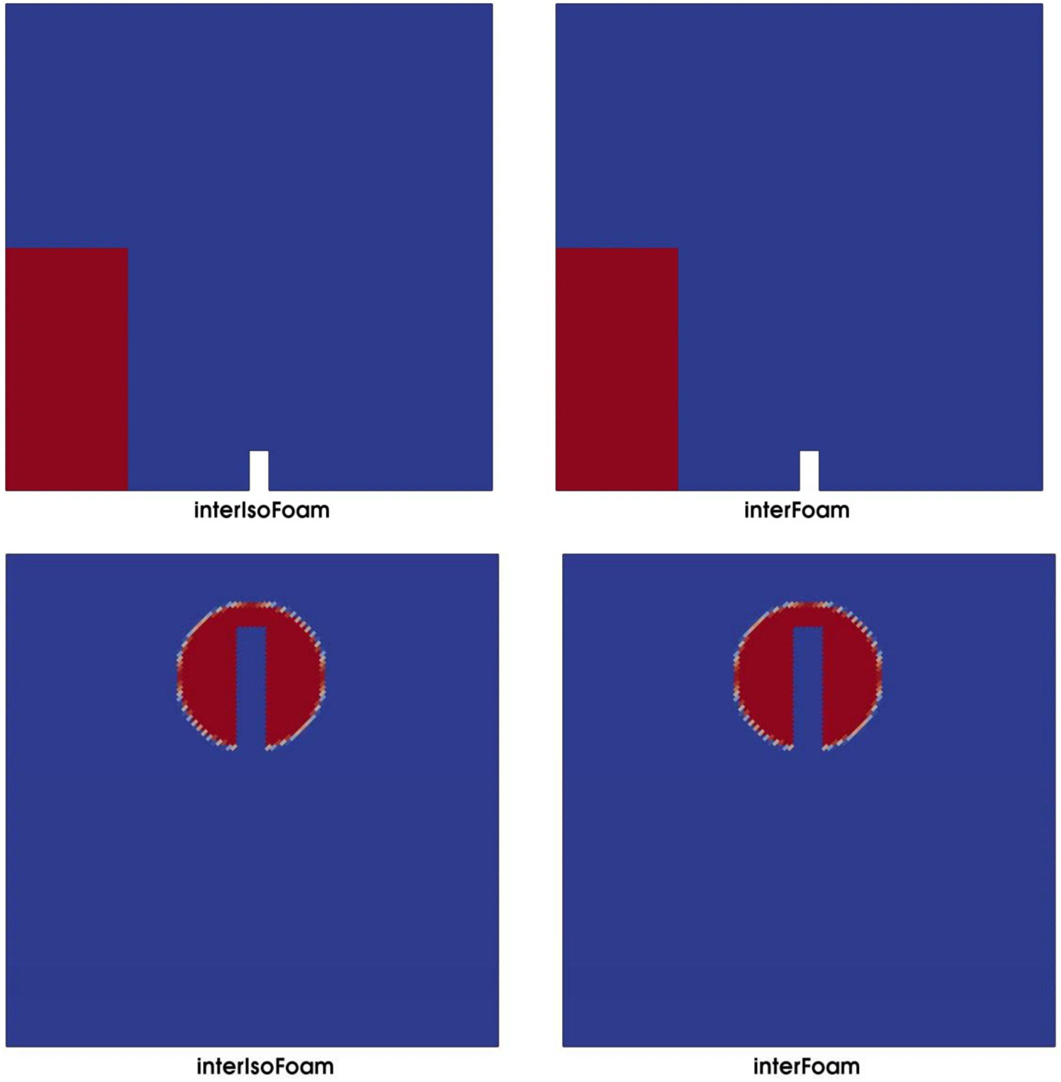
Interface capturing - isoAdvector

Solvers/numerical technique

`interIsoFoam`

- Alternative method to existing MULES algorithm
- Implemented for isothermal, immiscible incompressible fluids
- offers more accurate interface advection and a sharper interface representation
- Works well on structured and unstructured meshes.

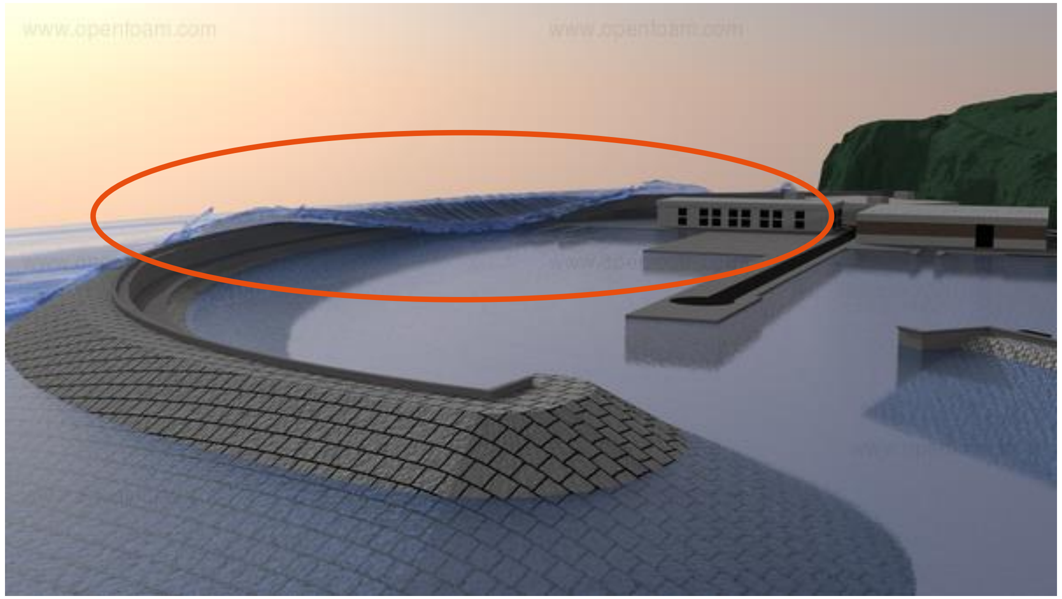
- Developed by Dr. Johan Roenby, DHI, Associate Prof. Henrik Bredmose at DTU Wind Energy and Prof. Hrvoje Jasak at University of Zagreb, Department Faculty of Mechanical Engineering and Naval Architecture.
- *Reference: Roenby J, Bredmose H, Jasak H. 2016 A computational method for sharp interface advection. R. Soc. open sci. 3: 160405. http://dx.doi.org/10.1098/rsos.160405*
- Tutorial: `$FOAM_TUTORIALS/multiphase/interIsoFoam/damBreak`



Solitary wave generation models

Boundary Conditions

- Populating implementation of the wave modelling introduced in 1612+ release
- New solitary wave generation for:
  - Grimshaw model
  - McCowan model



- supplied by:
The Environmental Hydraulics Institute IHCantabria

- Author: Gabriel Barajas

- Tutorial:
  $FOAM_TUTORIALS/multiphase/interFoam/laminar/waveExampleSolitaryGrimshaw
  $FOAM_TUTORIALS/multiphase/interFoam/laminar/waveExampleSolitaryMcCowan

Joule heating

Physical model

New source term in fvOptions: jouleHeatingSource

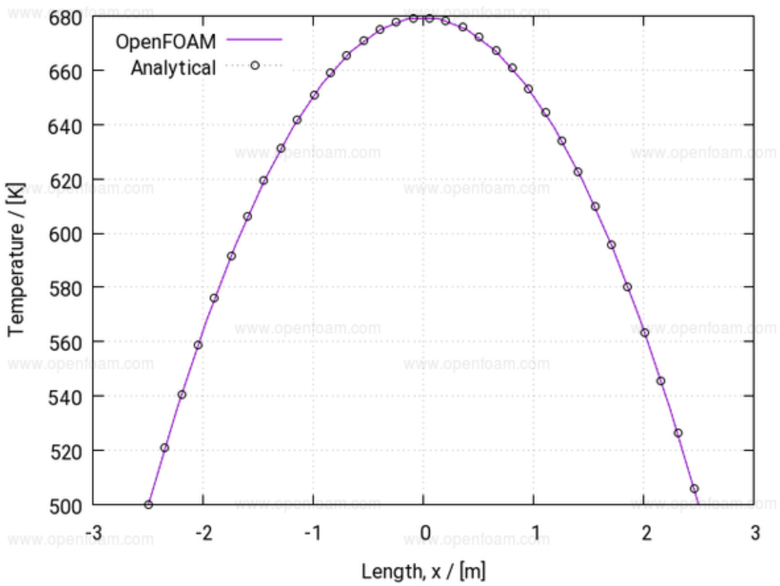
- solves an equation for the electrical potential **V**

$$\nabla \bullet (\sigma \nabla V) = 0$$

- Where σ is electric conductivity.
- The source is given by:

$$\dot{Q} = \sigma \nabla V \cdot \nabla V$$

- Conductivity
  - isotropical function of temperature
  - Anisotropical function of temperature
    - Prescribed by a vector



- $FOAM_TUTORIALS/heatTransfer/chtMultiRegionSimpleFoam/jouleHeatingSolid

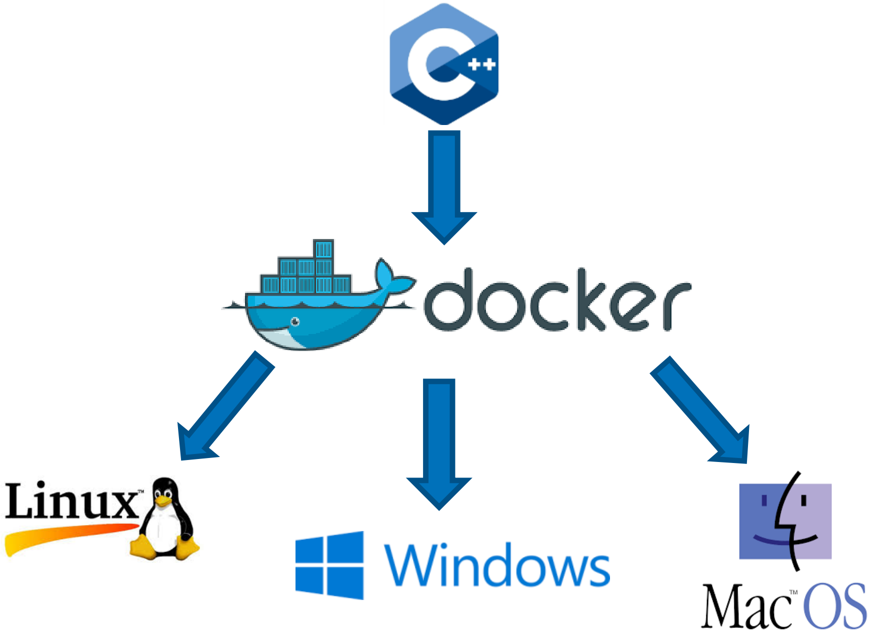
get it right®

OpenFOAM

installation

No restrictions on user OS system

- Same version of OpenFOAM runs on any platform (Linux, Windows, Mac OS)
  - Using Docker containers running OpenFOAM on CentOS 7
  - Easy MS Windows installer



OpenFOAM

installation

Windows Subsystem for Linux (WSL) and OpenFOAM v1706

- Users may use **native Windows 10 Bash on Ubuntu on Windows**
- Using a genuine **Ubuntu image of 16.04** from Canonical
- Precompiled version of **OpenFOAM-v1706 from OpenCFD**

- **DOWNLOAD – UNPACK - USE**

- `http://openfoam.com/download/install-windows-10.php` installation

OpenFOAM

Release history

- Brings previous releases of OpenFOAM down to version 1.0

- **Download** source code
- **Read** release notes

- Download → Release History

26/08/2010: OpenFOAM 1.7.1 README

Platform	Download	MD5 sum
Source	OpenFOAM-1.7.1.gtgz	2454728d946ee773c963fac15be9ca84
Source	ThirdParty-1.7.1.gtgz	22877bc0b1f2640cd09f175cfc5729b5

Command line completion

Usability

- Command line completion for all OpenFOAM utilities and applications
- Using TAB key will expand possible options:
- Example 1:

```
checkMesh <TAB> <TAB>

  -allGeometry        -help                -noZero       -writeAllFields
  -allTopology        -latestTime          -parallel     -writeFields
  -case               -meshQuality         -region       -writeSets
  -constant           -newTimes            -roots
  -decomposeParDict   -noFunctionObjects   -srcDoc
  -doc                -noTopology          -time
```

- Example 2:

```
checkMesh -time <TAB> <TAB>

  0    0.1  0.2  0.3  0.4  0.5
```

- Example 3:

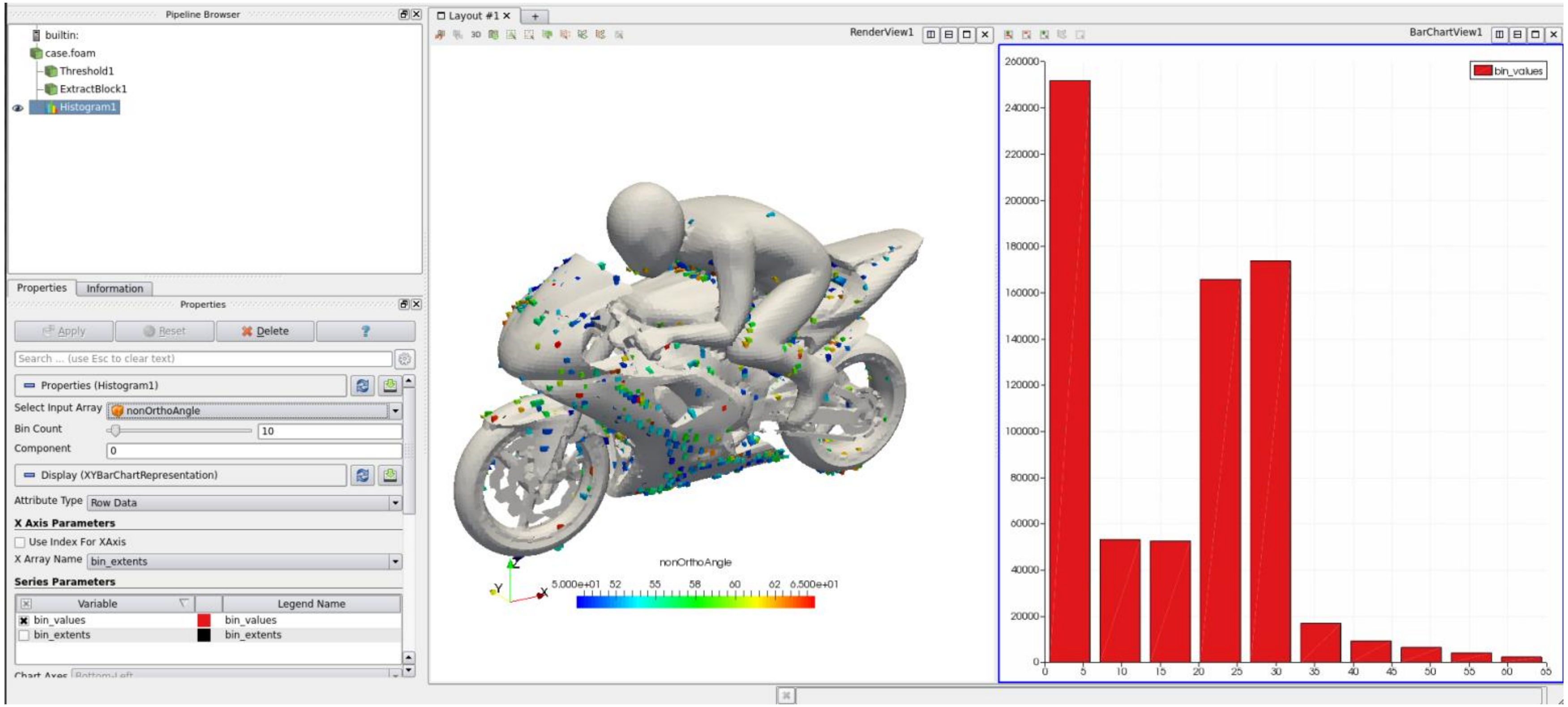
```
checkMesh -region <TAB> <TAB>

  air     porous
```

Mesh quality visualisation

Usability - checkMesh

- New option for checkMesh:
  - `writeAllFields` – will write all quality parameters as volumetric fields
  - `writeFields '(skew)'` – will write only listed fields



OpenFOAM 2017 Roadmap

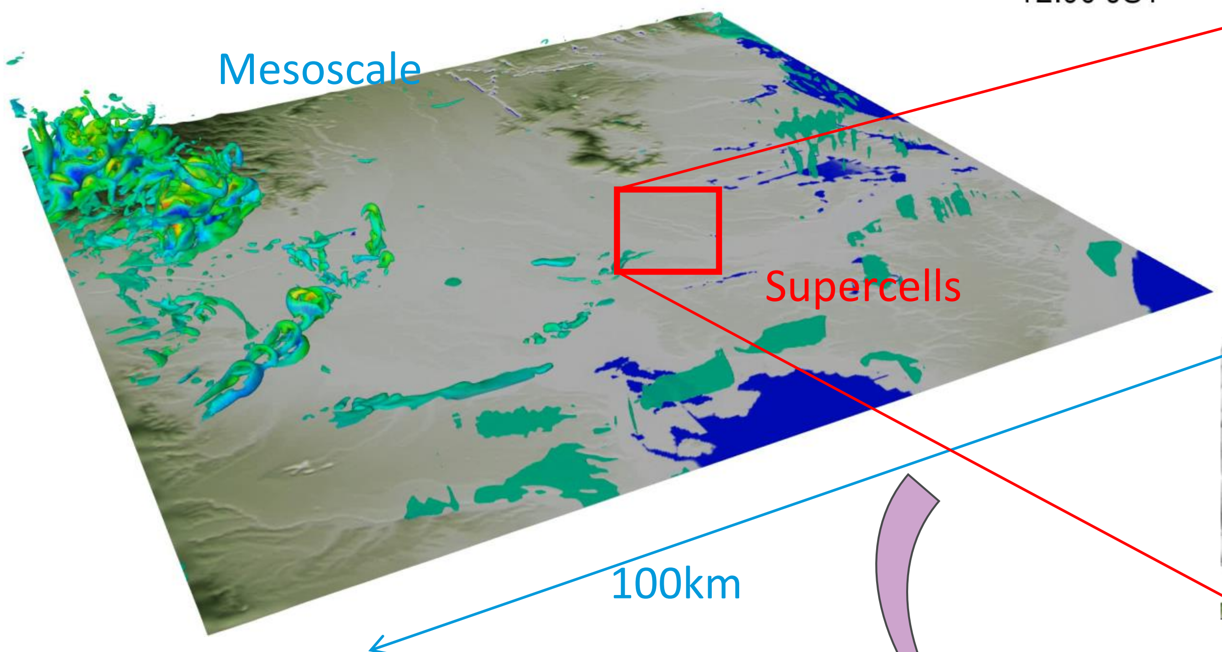
Some enhancements targeted for v1712

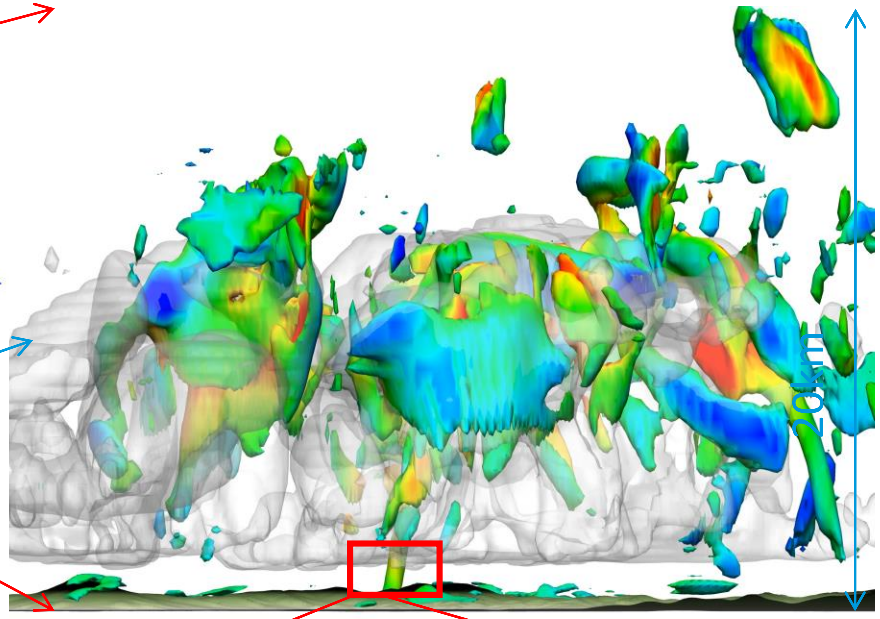
- Overset mesh release
    - Best practises for applications in marine and ground transportation
- Extensions to FSI lumped mass interaction
- Continuing Parallel I/O scaling and operation improvements
- Continuing Improvements in mesh generation
- Multiphase exchange (melting and evaporation)
- CHT enhancements, underhood and heat-transfer
- (COMM) Next phase of integration of wave modelling and marine solutions
- (COMM) Next phase of isoAdvector integration
- (COMM) Particle physics (Monte Carlo) with advanced physics
- Extended Theory and User guide documentation
- (COMM) Finite Area functionality
- (COMM) Extended Acoustics analogies
- (COMM) Third-party meshing integrations
- (COMM) Third-party chemistry utilities
- Optimisation strategies

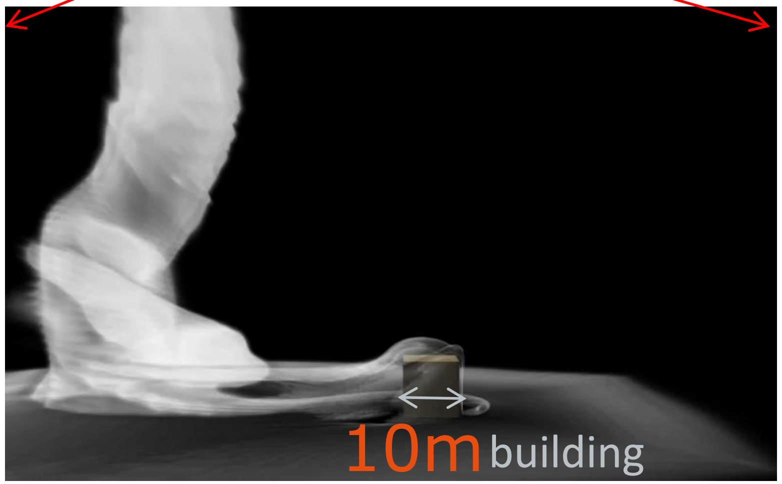
(COMM) means OpenFOAM interactions with the wider Community

SHIMIZU Corporation - An application (2011 Japanese Tsunami)

SHIMIZU Corporation - Challenging on Multi-scale simulation







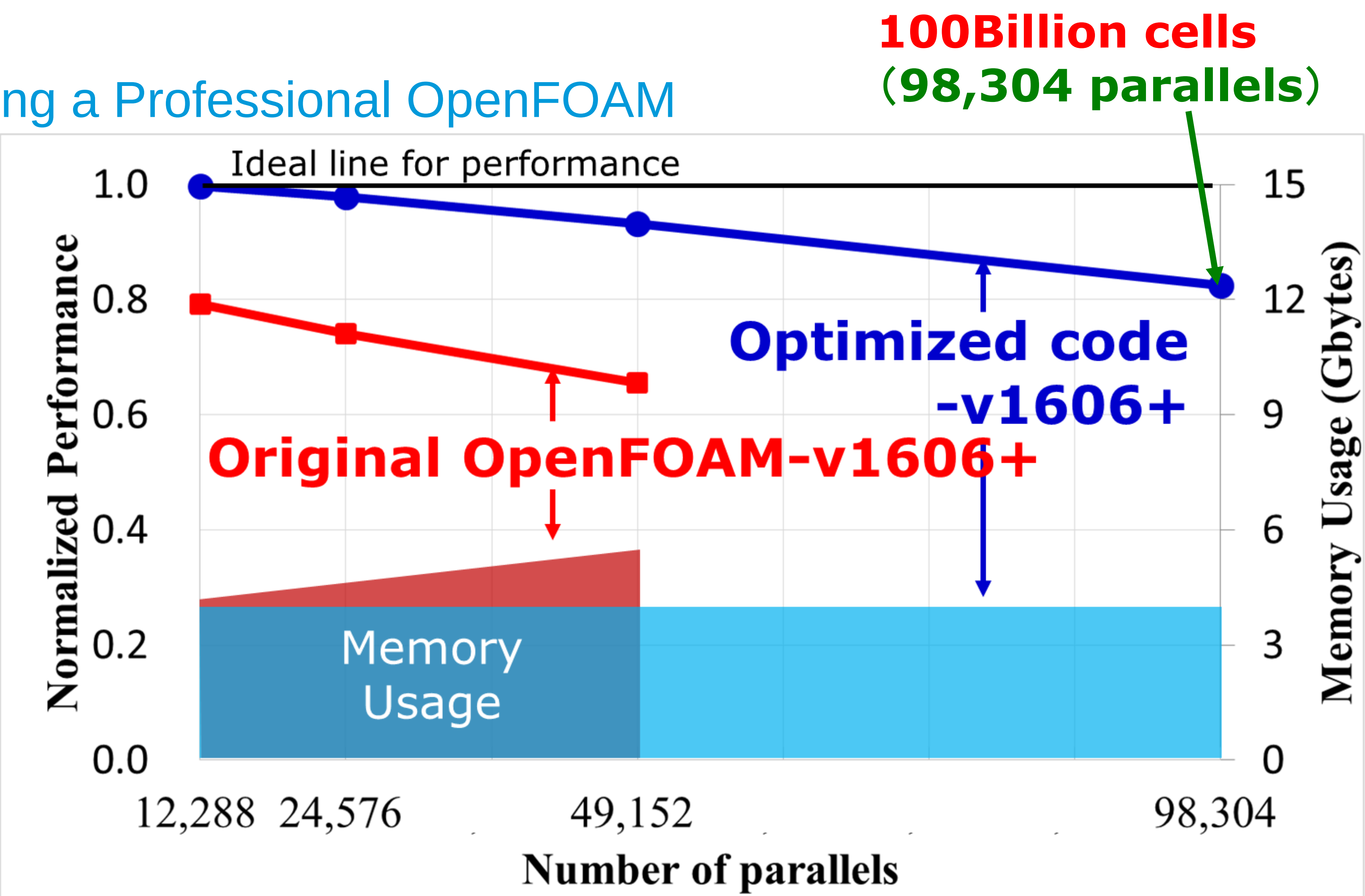
Multi-scale simulation from meteorology to building with several billion cell meshes

31

OpenFOAM

Collectively delivering a Professional OpenFOAM

- Shimizu Coorp.
  - October 2016



OpenFOAM

... in 2017, see us at

- **Events**
  - Workshops in Asia for OpenFOAM in AeroVibroAcoustics
    - China –12-13th July
    - Japan – 19-20th July
    - North American Forum – 26-27th September
    - India – t.b.a (Nov/Dec)
  - Conference Europe - 17-19th October 2017
    - Wiesbaden, nr. Frankfurt, Germany

    - Workshop for OpenFOAM in AeroVibroAcoustics
      - 19-20th October, Frankfurt
    - **REGISTRATIONS STILL OPEN**
  - Release webinars
    - for v1712: January 2018
- **Next releases**
  - v1712 in December 2017
  - v1806 in June 2018

OPENFOAM CONFERENCE KEYNOTES

PHIL ROE
Upwind methods

KYRIAKOS GIANNAKOGLOU
Adjoint Optimisation

CHRIS BEALE
Fuel cells

KARL MEREDITH
Fire modelling and Suppression

get it right®

Thank you

Questions / Comments

OpenFOAM

get it right®