Delivering a professional Open∇FOAM®

esi
OpenFOAM
CONFERENCE
in Germany | OCT 17-19, 2017

We stand on the shoulders of giants (2015)

Custodians of OpenFOAM® (2016)

…

esi
get it right®

www.esi-group.com

OpenFOAM – foreword

Greetings from, and thanks to the OpenFOAM Team

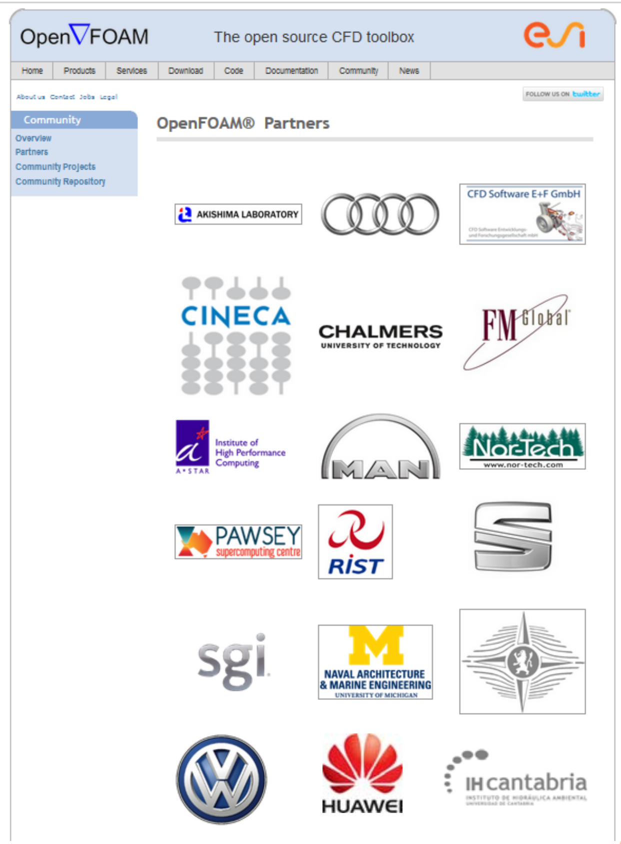
- **OpenCFD Core Development and Supporting Teams**
  - *Andrew Heather*
  - *Mattijs Janssens*
  - *Sergio Ferraris*
  - *Mark Olesen*
  - *Prashant Sonakar*
  - *Roger Almenar*
  - *Pawan Ghildiyal*
  - *Fred Mendonca*
  - *Karen Kettle*
  - *Takashi Minabe (Japan)*
  - *Mohsen Battoei (North America)*
  - *Ravi Ajjampudi (India)*
  - *Bjorn Landmann, Sebastien Vilfayeau (Germany)*
  - *Matej Forman (Training Coordinator)*

OpenCFD – Commitment to OpenFOAM Users

Development and Release Schedule

- OpenCFD owns the OpenFOAM trademark
- Releasing OpenFOAM since 2004

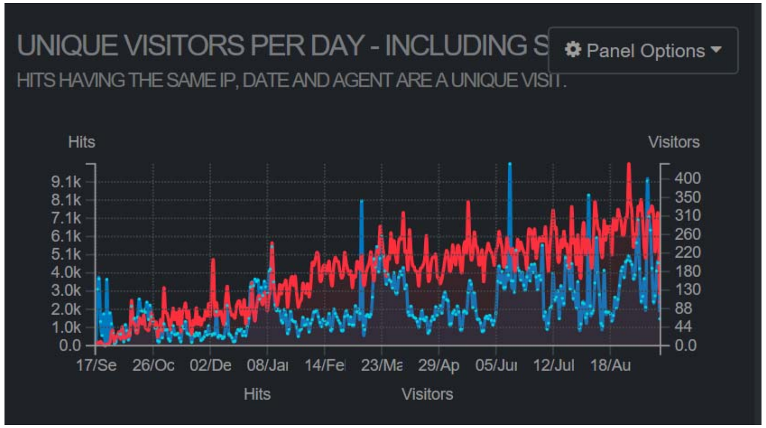
- Professional Six-monthly Development and Release cycle, including
  - New developments
  - Consolidated bug-fixes
  - Overhaulled testing procedure for Quality Assurance
  - Release and Development repositories in GitLab
    https://develop.openfoam.com
    - Master branch
    - Develop branch (includes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



OpenCFD – Commitment to OpenFOAM Users

Development and Release Schedule

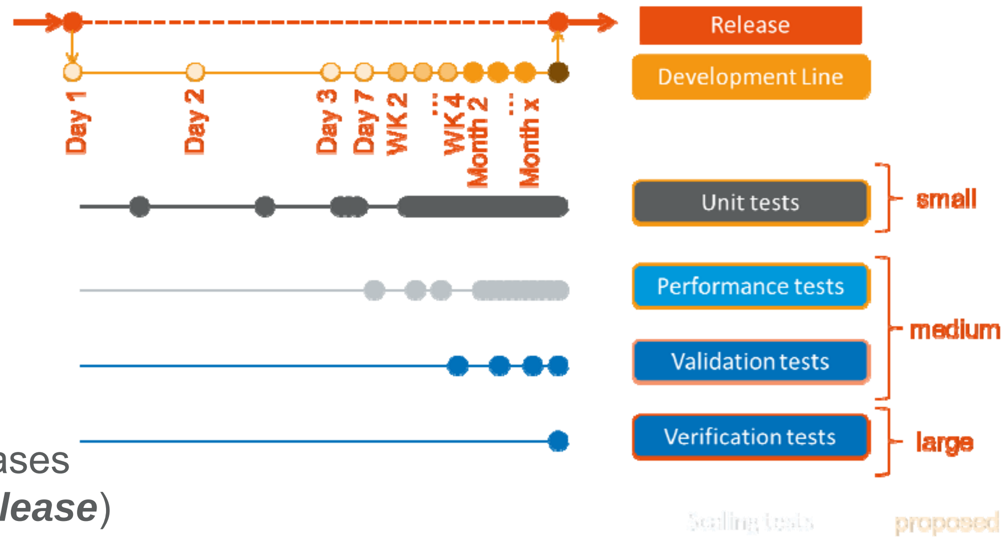
- OpenFOAM.com releases so far
  - OpenFOAM-v3.0+ on Jan 13th 2016
  - OpenFOAM-v1606+ on June 30th 2016
  - OpenFOAM-v1612+ on 23rd December 2016
  - OpenFOAM-v1706 on 30th June 2017



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***)

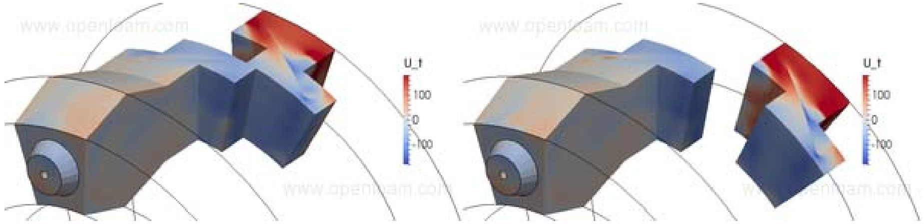
- Test loops grow with every new functionality released

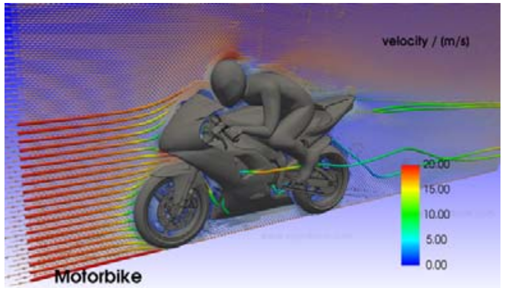
get it right®

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





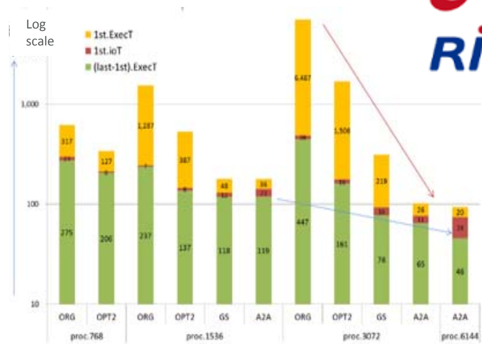
- ‘External‘ Contributors to OpenFOAM-v3.0+
  - DES and new family of k-ω-SST models
  - Inter-region heat transfer

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

CFD+engineering
Ingenieurbüro für Strömungsmechanik

OpenFOAM

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

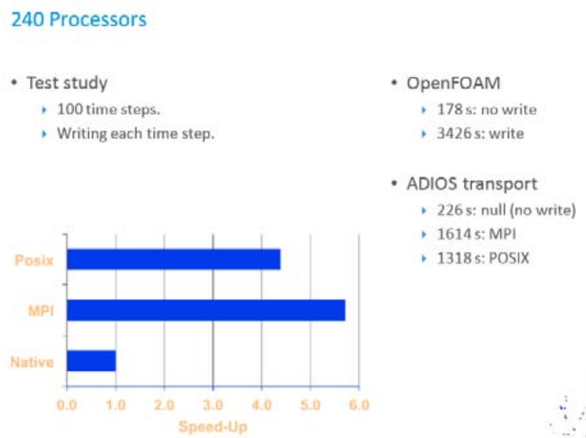


- OpenFOAM-v1606+ (June2016)
  - Message passing performance scaling
    - Gather-scatter order
    - All-to-all processor communications
  - Performance profiling (Bernhard Gschaider)
  - DFSEM (help from Ruggero Poletto)
  - Validated Aeroacoustics enhancements and coupling to Acoustic codes

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*



  - 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

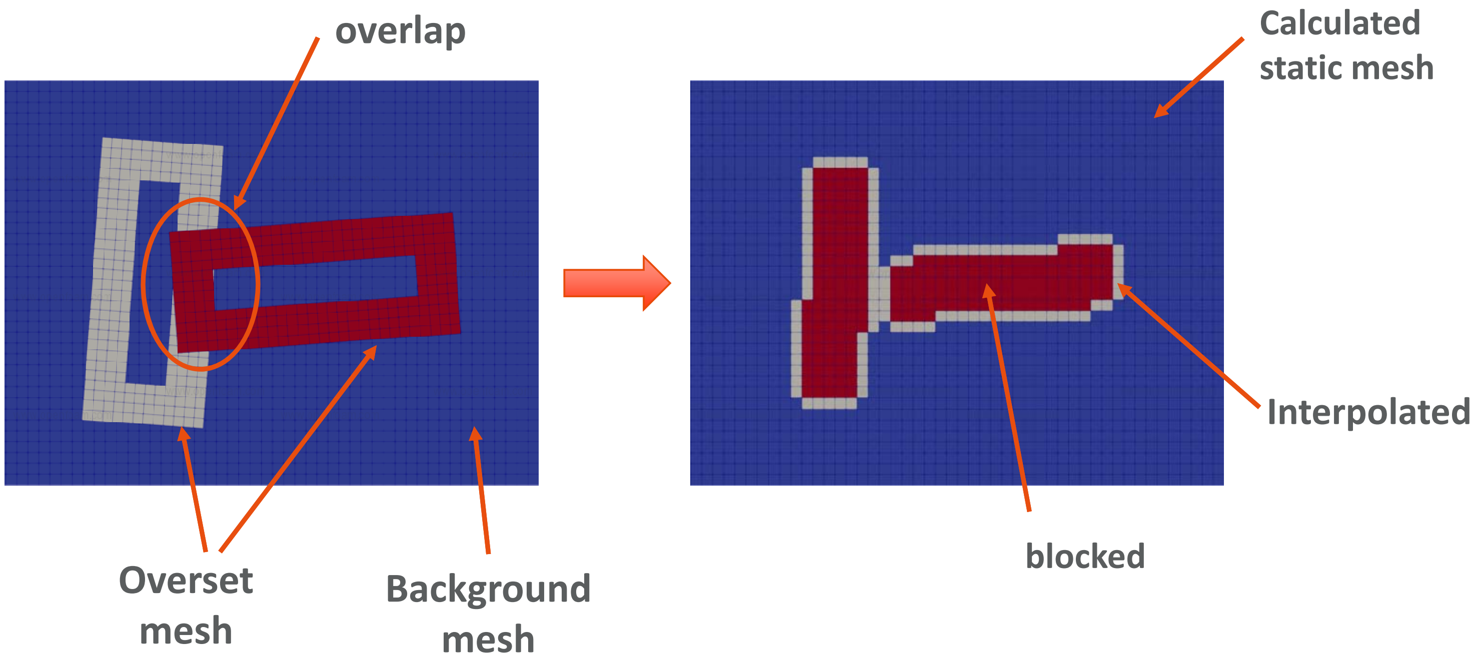
New development highlights and contributions in v1706

- **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

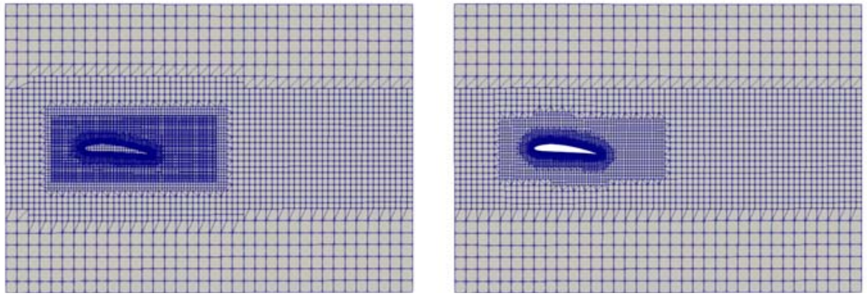


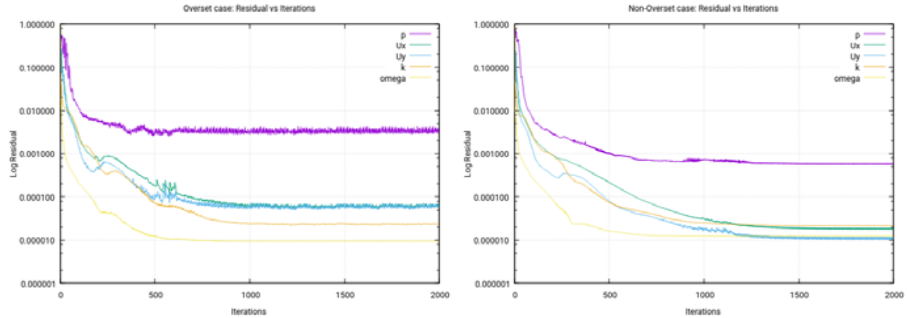
Overset: Verification and Validation

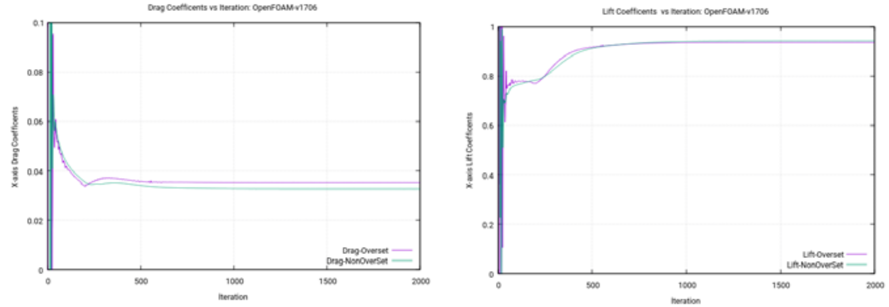
overSimpleFoam vs simpleFoam

- 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%

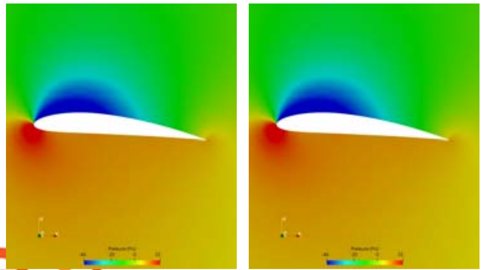
OVERSET ORIGINAL





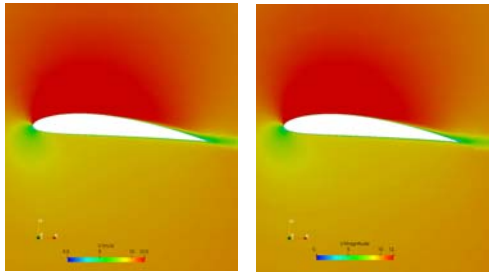


OVERSET ORIGINAL



Pressure

OVERSET ORIGINAL



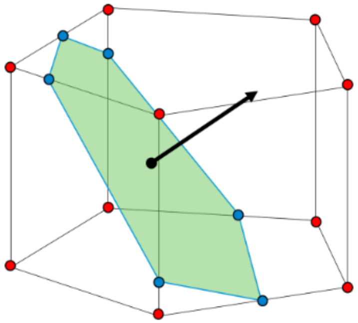
Velocity

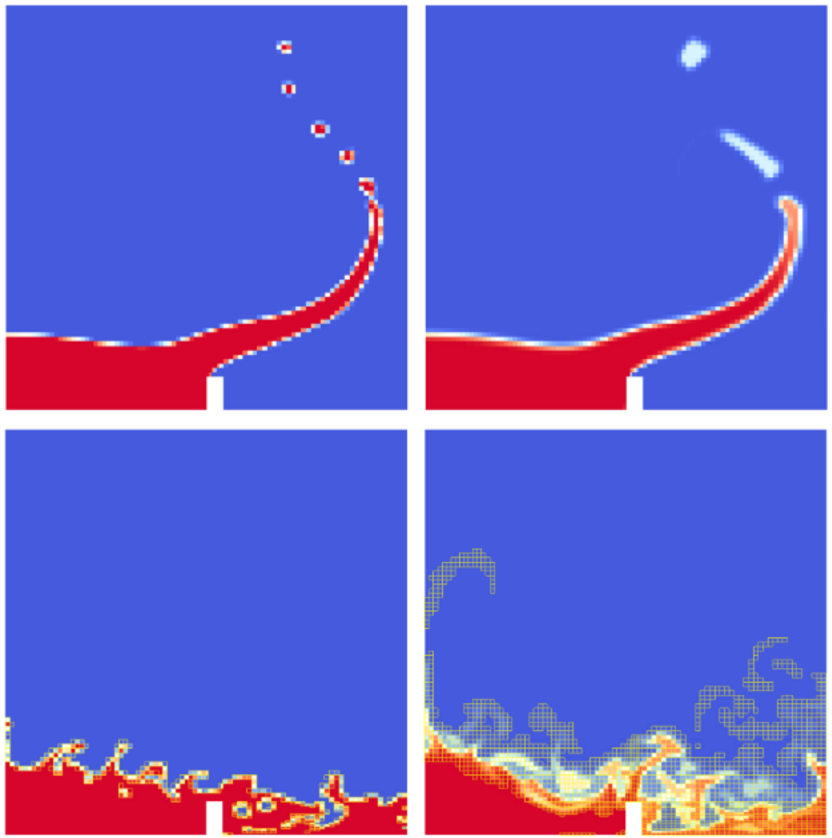
Interface capturing - isoAdvector

`interIsoFoam`

Solvers/numerical technique

- 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.



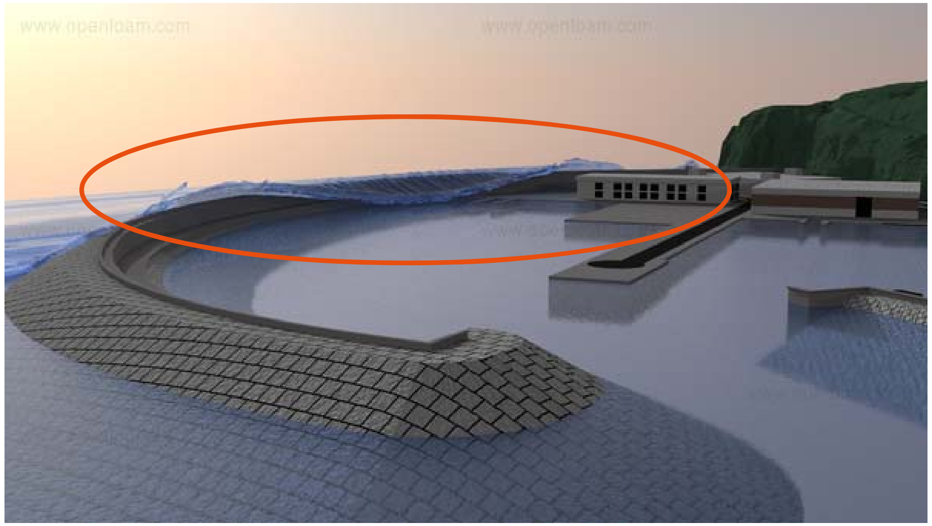


- Developed by Dr. Johan Roenby, DHI, Associate Prof. Henrik Bredmose at DTU Wind Energy and Prof. Hrvoje Jasak at University of Zagreb, Department Faculty of Mechanical Engineering and Naval Architecture.
- *Reference: Roenby J, Bredmose H, Jasak H. 2016 A computational method for sharp interface advection. 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

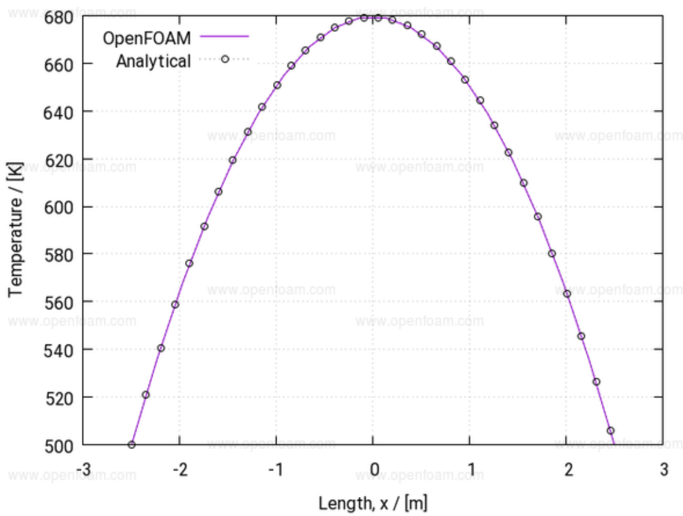
New source term in fvOptions: jouleHeatingSource

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

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

- 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

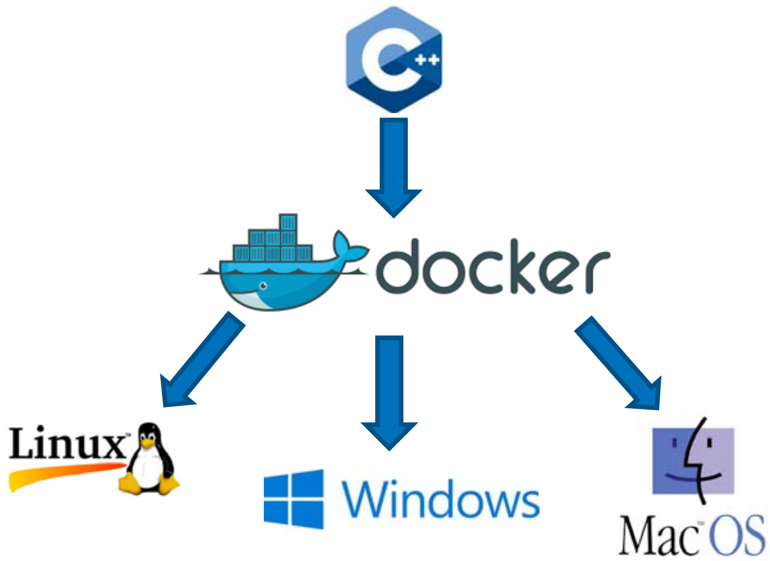
- `$FOAM_TUTORIALS/heatTransfer/chtMultiRegionSimpleFoam/jouleHeatingSolid`

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



installation

OpenFOAM

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`

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
```

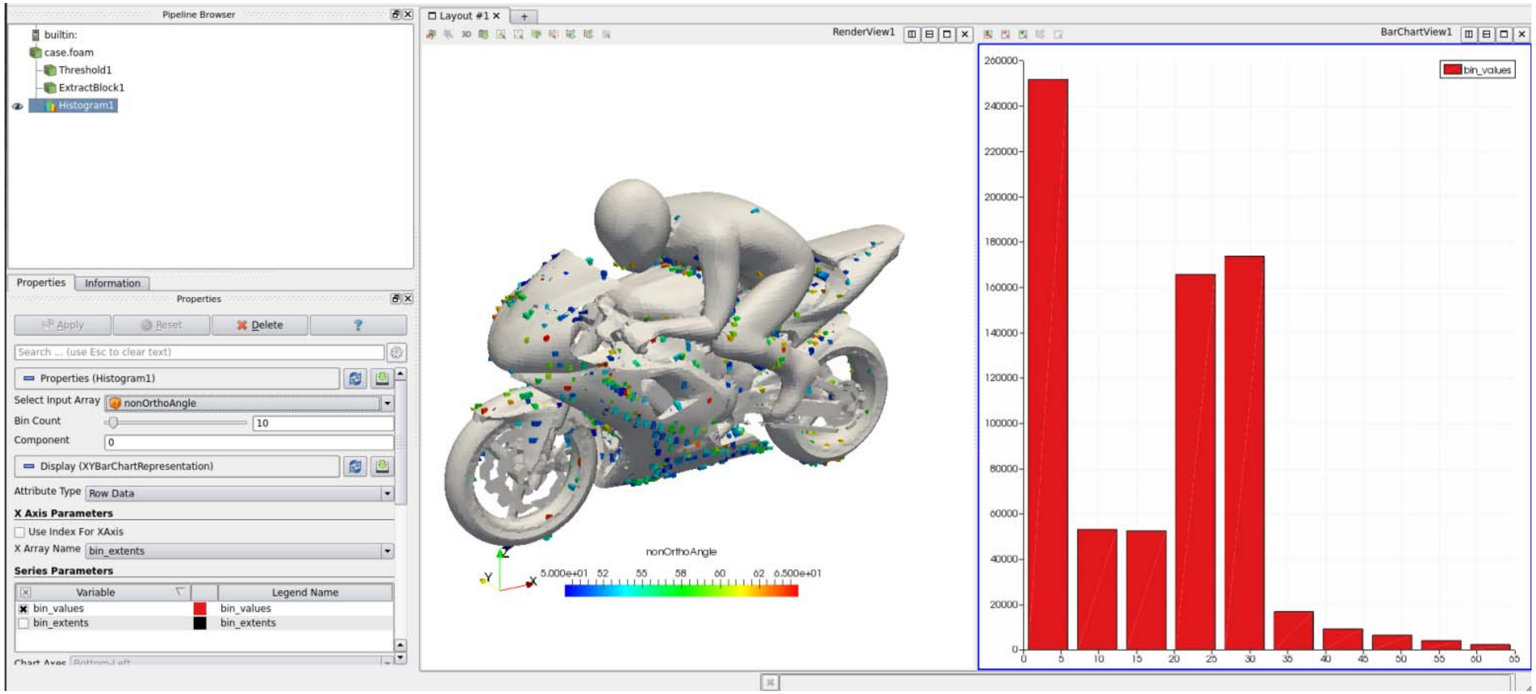
- Example 3:

```
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

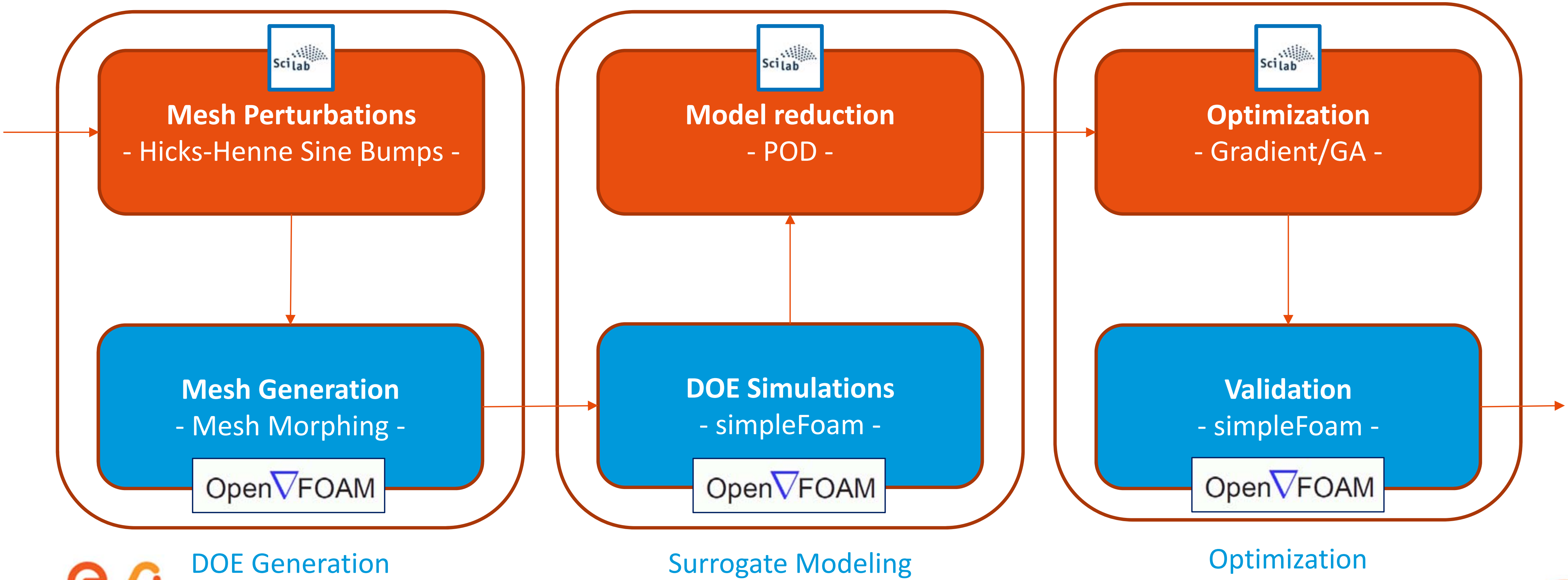
Some enhancements targeted for v1712 and beyond

- 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/marine > Gabi ***Process and Multiphase*** 4pm
- (COMM) Next phases of isoAdvector integration
- (COMM) Particle physics (Monte Carlo) > Borg ***OpenFOAM Technology*** 3pm
- Extended Theory and User guide documentation
- (COMM) Finite Area functionality > Hrvoje ***OpenFOAM Technology*** 11am
- (COMM) Extended Acoustics analogies
- (COMM) Third-party meshing integrations > Franjo ***Meshing*** 2.30pm
- (COMM) Third-party multiphase utilities > TUDarmstadt ***Process*** 2.30/5pm
- Optimization strategies > Vaggelis NTUA ***Optimisation*** 2pm

(COMM) means OpenFOAM contributions from the Community

OpenFOAM and SciLab - Conclusion

OpenFOAM + Scilab process : Automatable



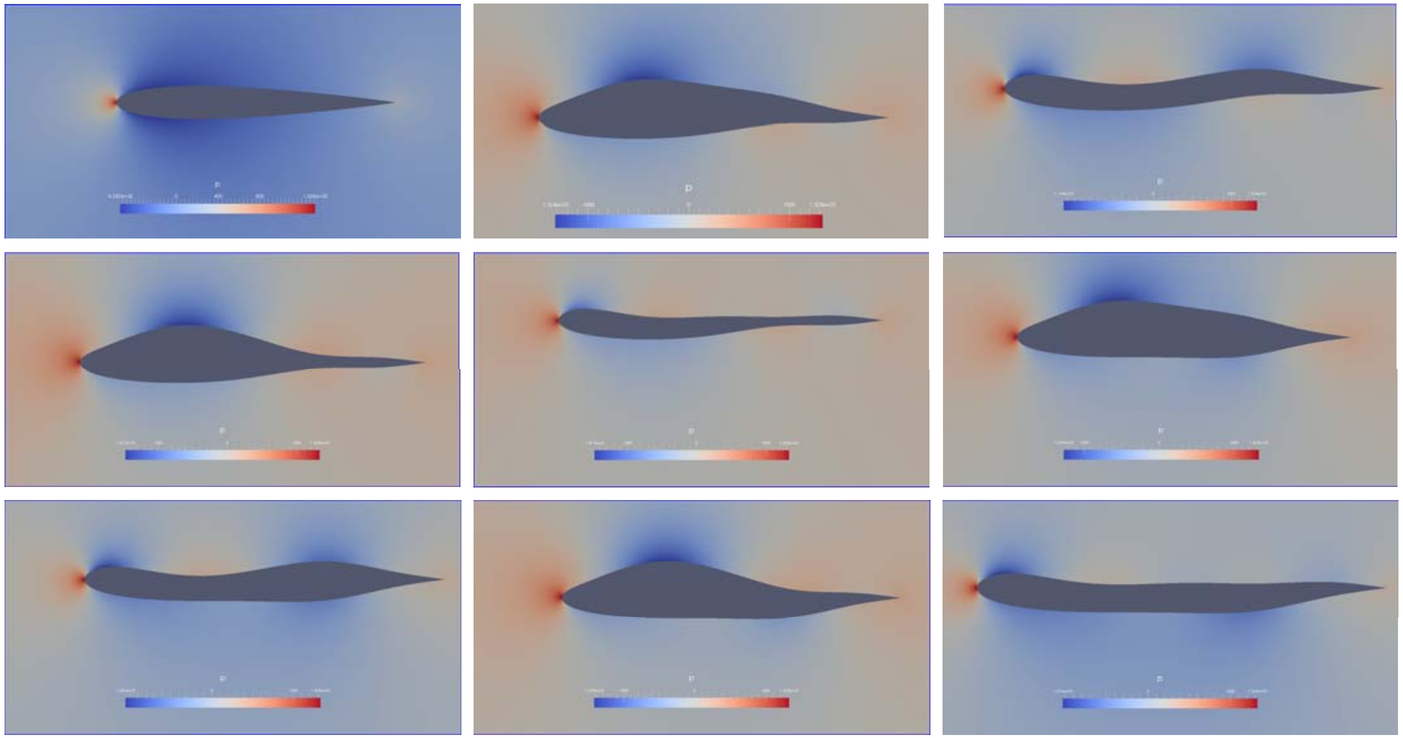
II Design Of Experiment and POD basis

OpenFOAM – DOE simulations

DOE set up

1] From template case, create new cases

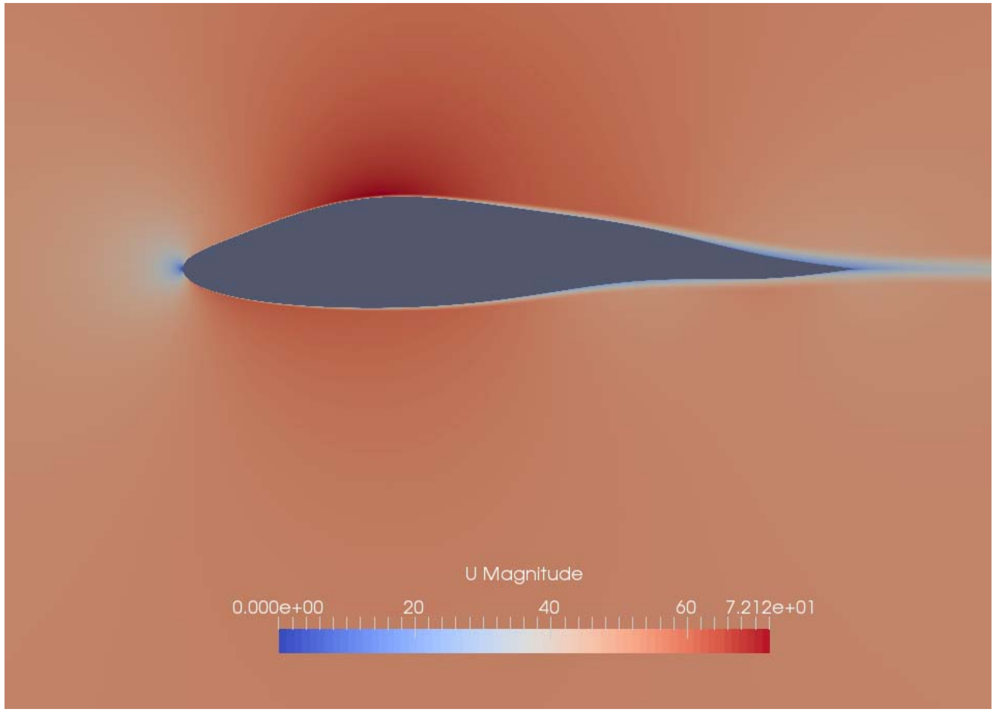
2] Change the *constant/polymesh/points* file

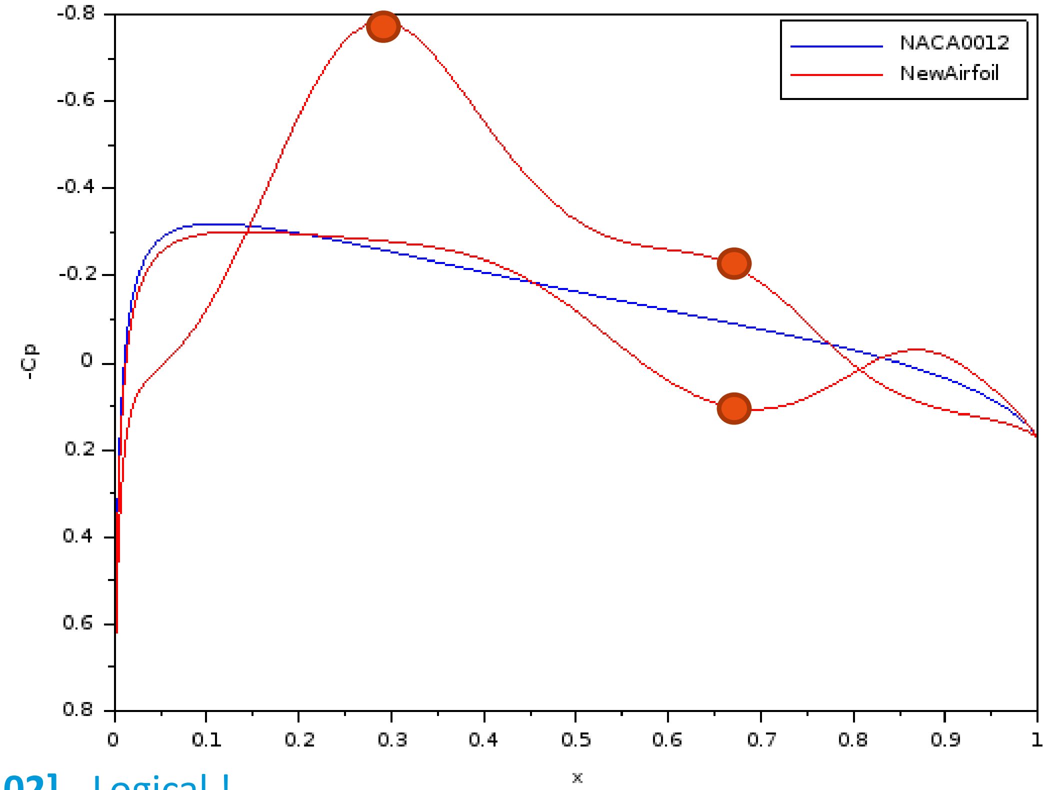


2-level full factorial + center point DOE for 3 parameters varying pressure

III Results: Global Optimum

Within DOE - Linear behavior of the pressure

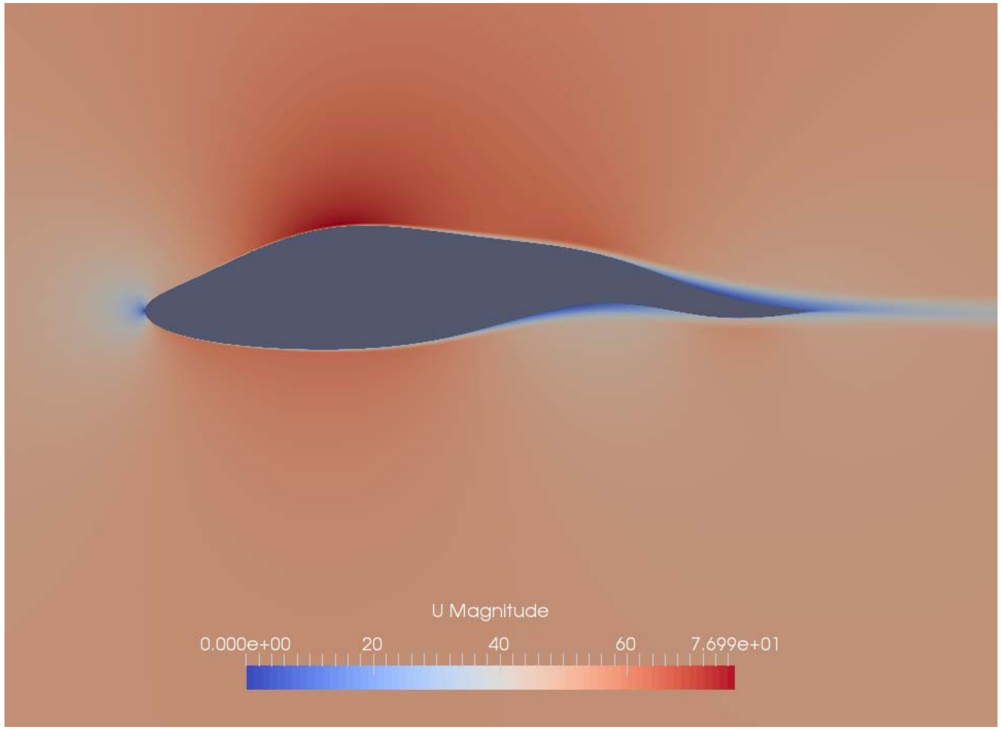


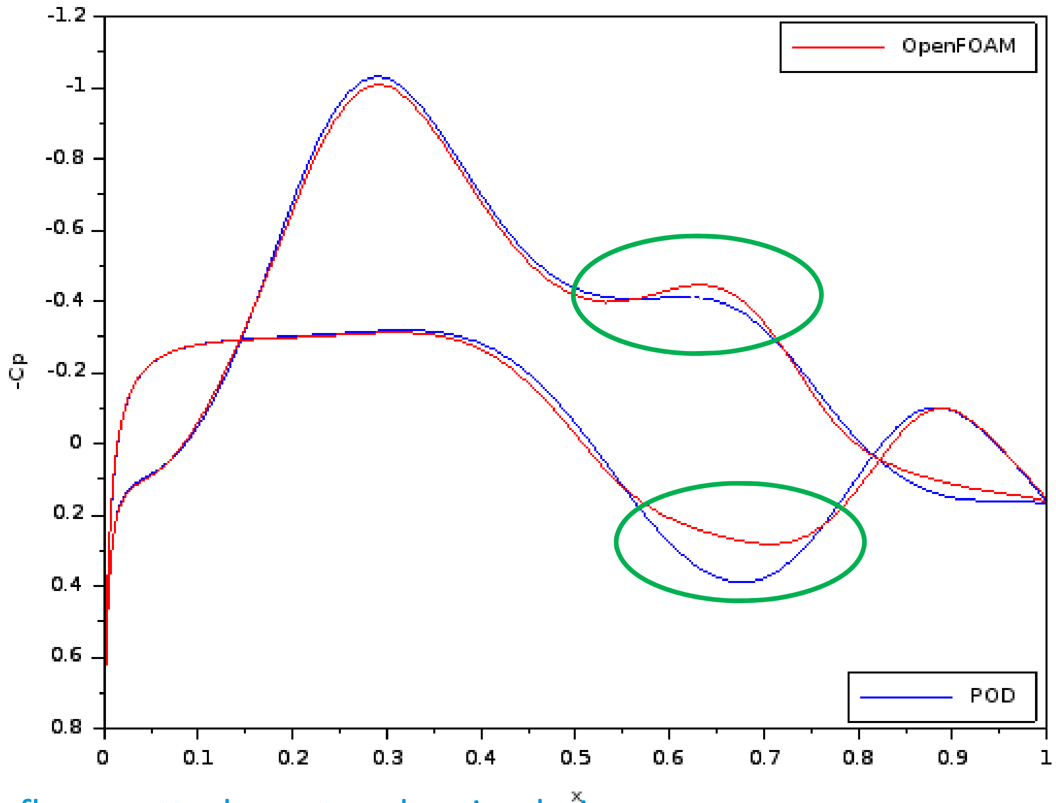


[0.05 0.02 0.02] - Logical !

IV Results: Reduced model limits

Out of DOE - Low error in lift prediction





[0.07 0.04 0.05] - Model not trained for flow reattachment and recirculation zone

OpenFOAM Virtual Training

- **OpenFOAM Virtual training**
  - Course material (common)
    - Course material on .ppt / .pdf
    - Printed manuals sent in advance
  - Delivery (web)

    - Trainer presents via **GoToMeeting / GoToTraining**
    - Trainee runs on their own Wi-Fi enabled laptop
  - Compute (bare-metal Cloud server)

    - ESI-Cloud server/terminal
    - ssh-connexion to server
    - OpenFOAM installed on ESI-Cloud
    - Paraview client on laptop connecting to server on ESI-Cloud
  - 2-day Foundation course
  - 2-day Advanced course
  - 1-day Application Training

- **Physical OpenFOAM training**
  - Course material (common)
    - Course material on .ppt / .pdf
    - Printed manuals hand-out
  - Delivery (physical)

    - Trainer presents via overhead screen projection
    - USB-bootable laptops with OpenFOAM and Paraview
  - Compute (laptop)

    - Laptop; OpenFOAM and Paraview on USB
  - 2-day Foundation course
  - 2-day Advanced course

OpenFOAM 2017 New Trainings

1-Day Virtual Applications and Process

Training

Overview

Courses

Core

1. Foundation
2. Advanced

Applications

1. Aeroacoustics
2. External Aerodynamics
3. Fire Modelling
4. Overset

GUI Driven

1. Visual-CFD

When and where

Schedule & Booking
Virtual Training
On-site Training
Locations

Other

Training Enquiries
Terms and Conditions

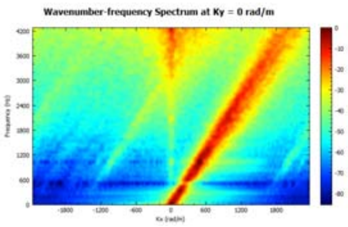
OpenFOAM® Aeroacoustics Course

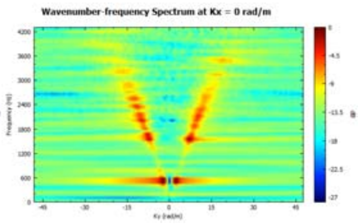
This course covers Theoretical and Applied concepts in Aeroacoustics (CAA) using OpenFOAM, touching on progressive Multiphysics design challenges, e.g. Aero-Vibro-acoustics (AVA). The course organisers are leading experts in flow-noise source prediction, propagation and noise abatement in all engineering sectors including transportation (Automotive, Aerospace, Marine, Rail), energy, power generation, building environment and turbomachinery. The training provides examples of process and best practice, which are useful to existing OpenFOAM users wishing to broaden their application knowledge of OpenFOAM towards Aeroacoustics, Hydroacoustics and Hydro/Aero-Vibroacoustics. All features demonstrated in this course are part of the Official release of OpenFOAM.

Topics Covered

- Basics - aeroacoustics source and propagation mechanisms, including a shallow dive into classical theory
- Narrowband (tonal) and broadband flow noise
- Spatial and Temporal considerations (mesh and time-step)
- Turbulence and wall-modelling
- Steady-state: fast approximations of source mechanisms and ratings
  - Surface based (shear-flow correlation)
  - Volume based (Lilley-Proudman correlation)
- Transient:
  - Appropriate physics modelling and boundary conditions for aeroacoustics
  - Numerical control settings in OpenFOAM
- Pure acoustics in OpenFOAM?
- Spectral postprocessing using the built-in OpenFOAM point-based and surface-based temporal-Fourier analysis tools
- Noise Propagation to the far-field using acoustics analogies, e.g. Curle
- General hints and tips
- Insights into coupling with Acoustic propagation codes for acoustics and vibroacoustics using SEA, FEA and BEM.

Nuggets





OpenFOAM® Overset Course

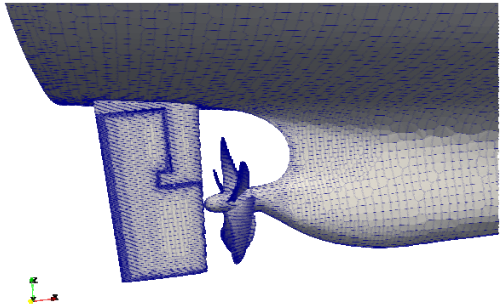
This course covers Theoretical and Applied concepts using overset (chimera) meshing in OpenFOAM. The course organisers are application experts in all engineering sectors, especially transportation: Automotive, Aerospace, Marine, Rail.

The training provides examples of process and best practice, which are useful to existing OpenFOAM users wishing to broaden their application knowledge of OpenFOAM towards using overset meshing as an alternative to mesh morphing or sliding interfaces.

All features demonstrated in this course are part of the Official release of OpenFOAM.

Topics Covered

- Meshing for overset
  - basics of background and foreground mesh generation, and assembly
  - interpolation controls, solver settings

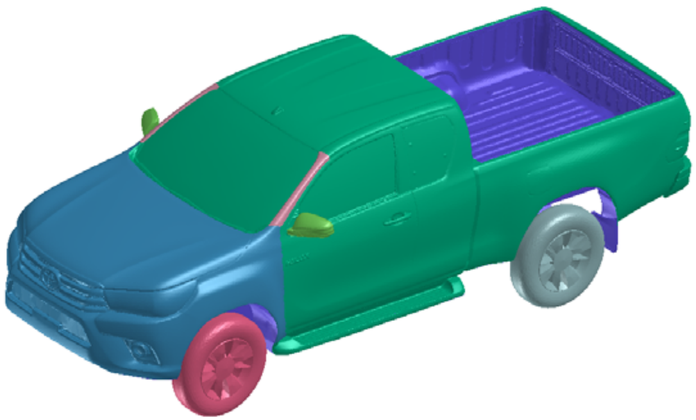


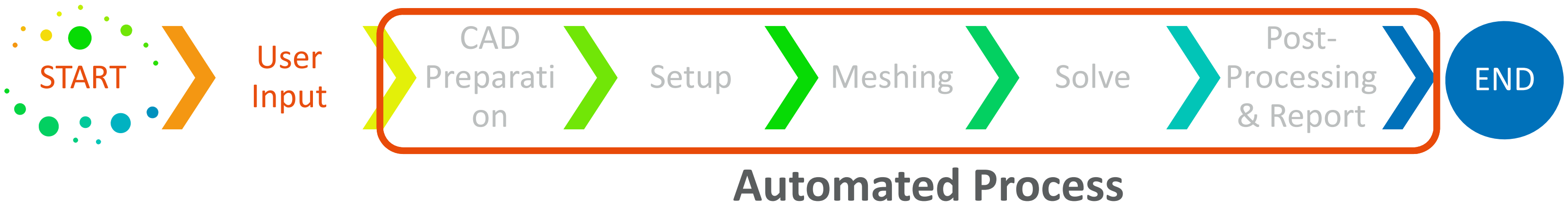
START
User Input
CAD Preparation
Setup
Meshing
Solve
Post-Processing & Report
END

Automated Process

External Aerodynamics

Executing the Automated Process





Simplified User Input (Such as Excel → CSV)

Editable Entry
Non-editable Entry
Comments

Simulation properties

Case name	sample	
STL File Name	vehicleGeometry_m.stl	
Freestream velocity	20	m/s
TargetMeshSize	50	In Millions
Simulation Type	Transient	Steady/Transient
No of iterations/time (s)	1	
No of partitions	64	
Save post file data	Y	
Field averaging over N last iterations	200	

Wind Tunnel Type

Wind tunnel data in STL file	N	tunnel_m.stl		
User Wind tunnel data	N	(-11.67, -2.644, -0.126)	(12, 2.64, 4.162)	(Y/N): If No, generic size:
Automated Wind tunnel	Y			

Ground Condition

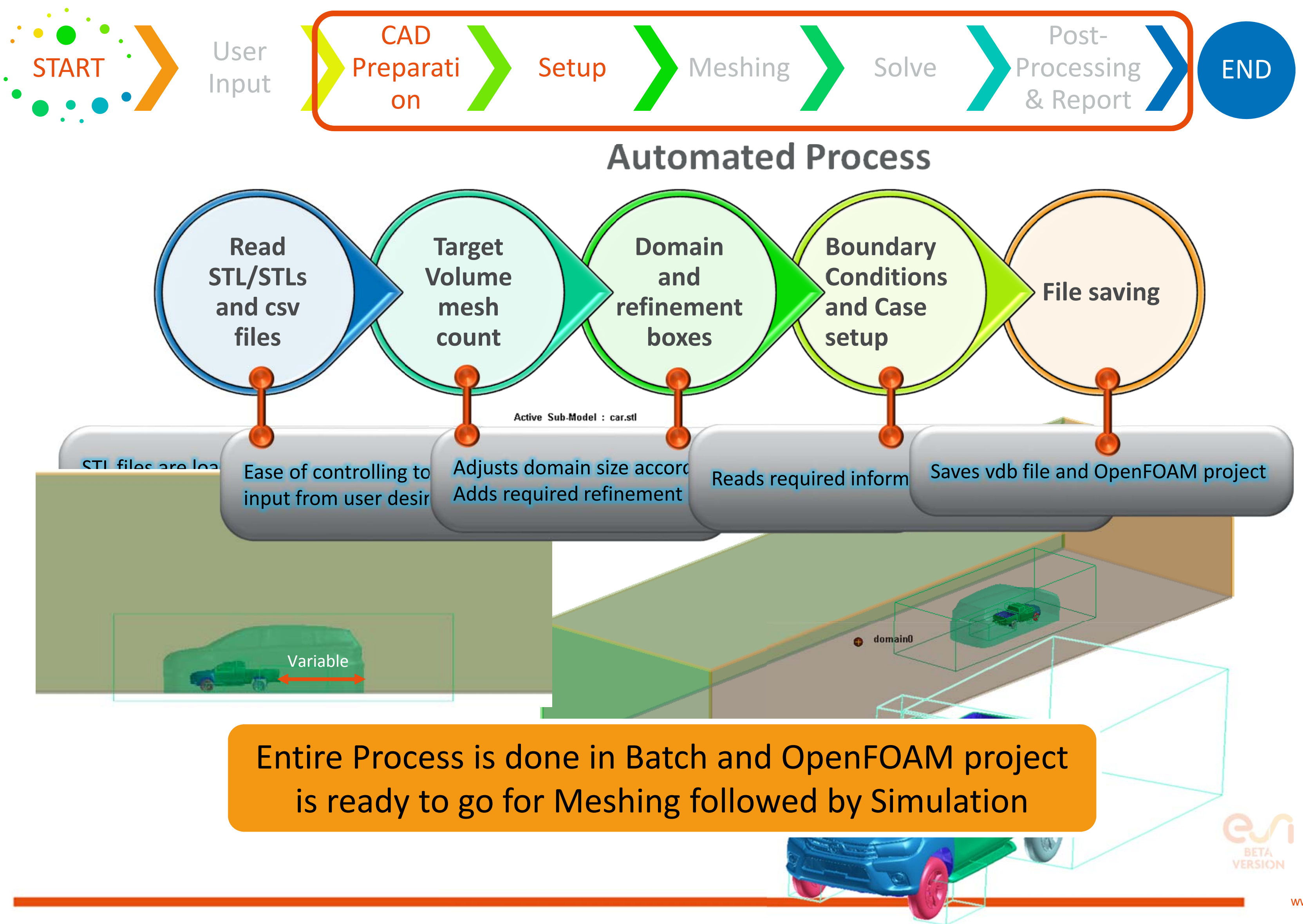
Moving ground/rotating wheels	N	(Y/N): Will activate motion on wheels and NO-SLIP part of tunnel floor	

Vehicle Information

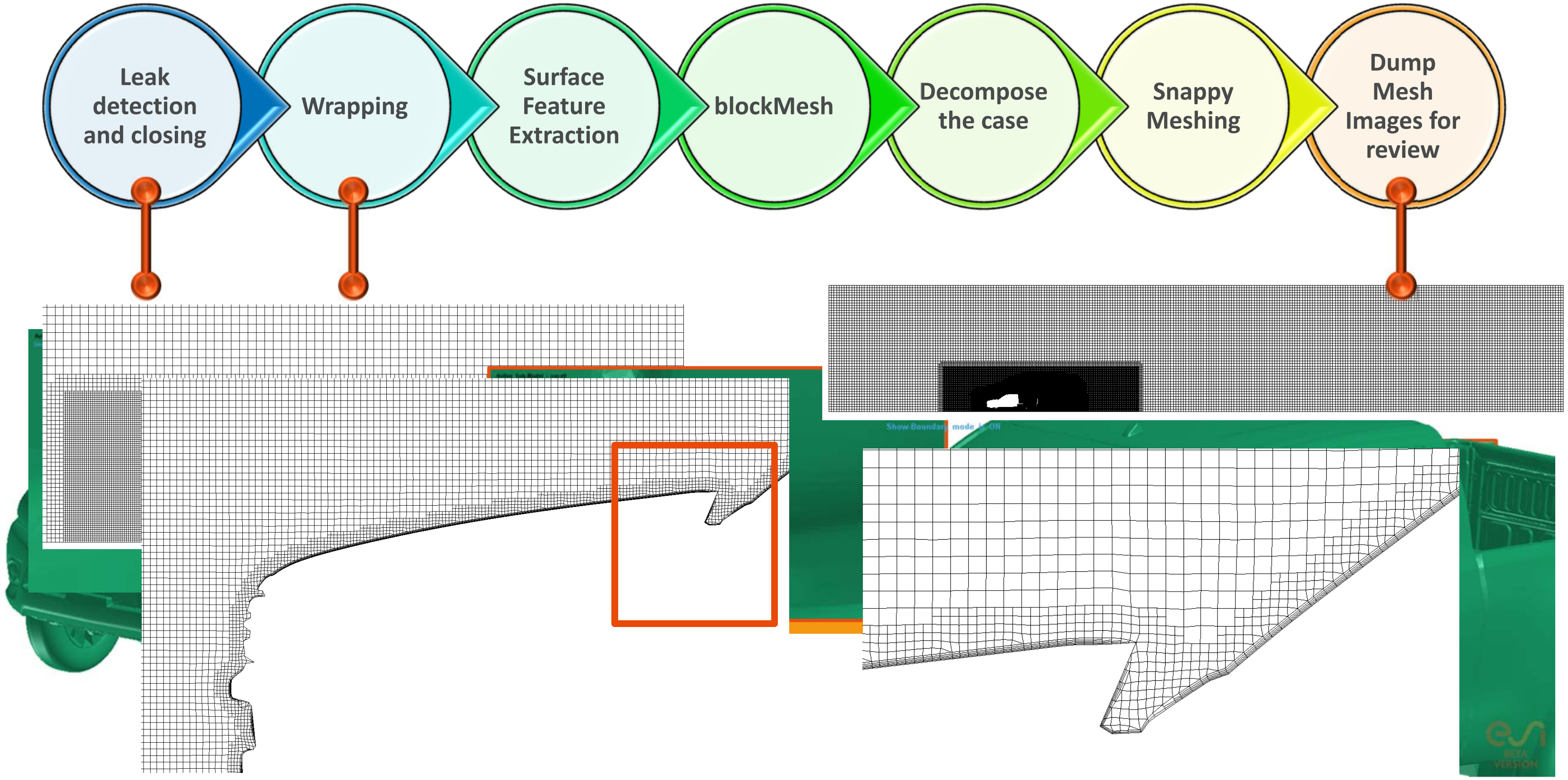
Vehicle frontal area	2.74835	m2
Wheel base	3.08999	m
Center of Rotation	(5.645, 0,- 0.485)	

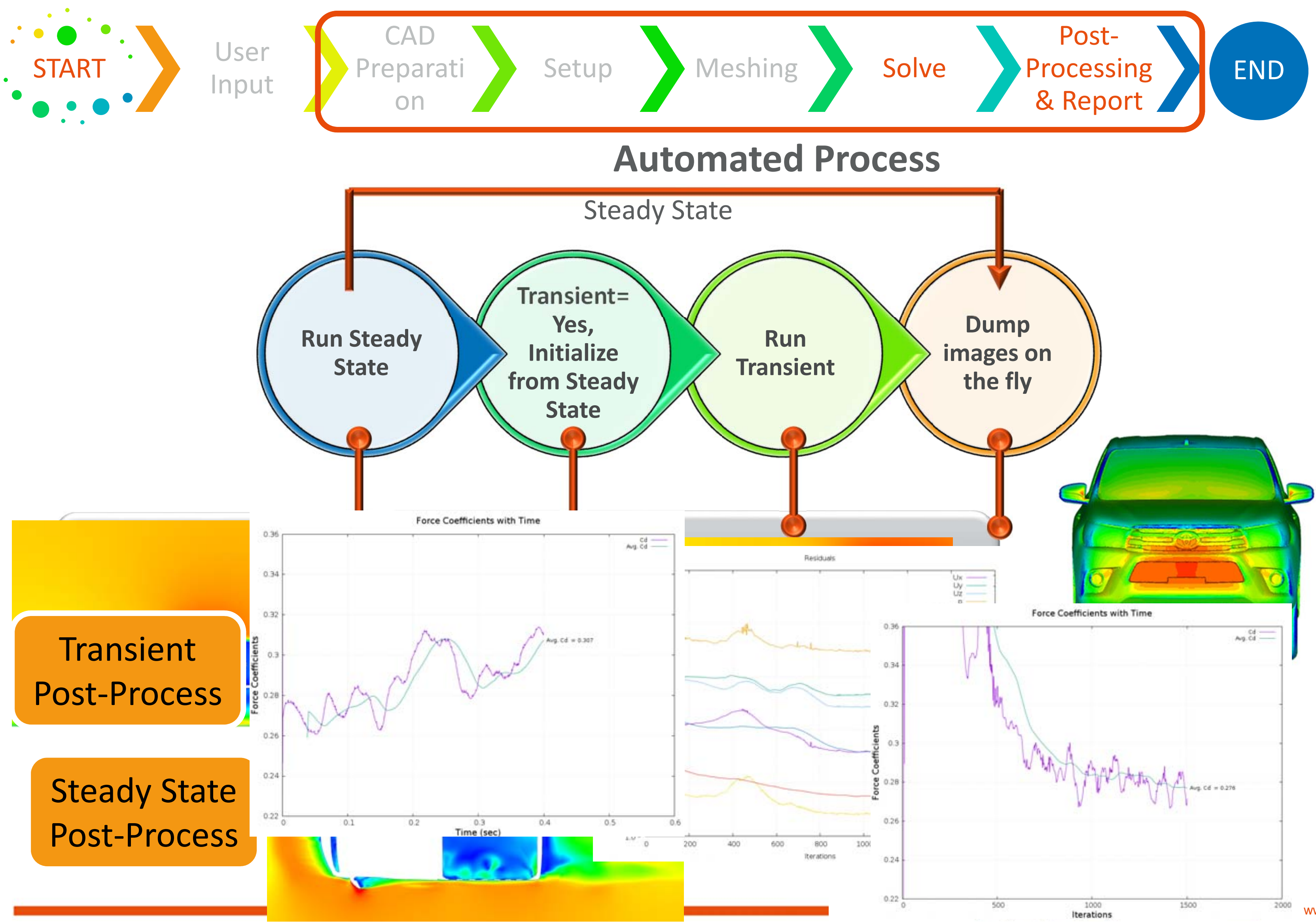
Modeling

Turbulence model	SST-K-Omega	(SST-K-Omega/realizable K Epsilon/kEpsilon)
Air density	1.205	kg/m3
Air dynamic viscosity	0.0000185	Pa-s


START
User Input
CAD Preparation
Setup
Meshing
Solve
Post-Processing & Report
END
Automated Process
Read STL/STLs and csv files
Target Volume mesh count
Domain and refinement boxes
Boundary Conditions and Case setup
File saving
STL files are loa
Ease of controlling to input from user desir
Adjusts domain size accord Adds required refinement
Reads required inform
Saves vdb file and OpenFOAM project
Active Sub-Model : car.stl
Variable
domain0
Entire Process is done in Batch and OpenFOAM project is ready to go for Meshing followed by Simulation

Automated Process

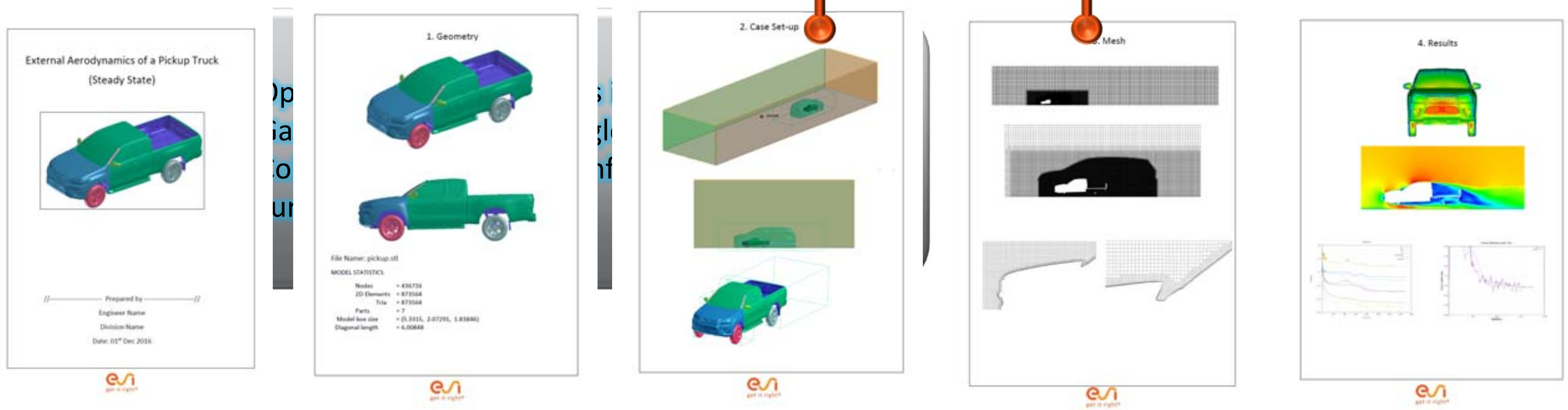



START
User Input
CAD Preparation
Setup
Meshing
Solve
Post-Processing & Report
END
Automated Process
Steady State
Run Steady State
Transient= Yes, Initialize from Steady State
Run Transient
Dump images on the fly
Transient Post-Process
Steady State Post-Process
Force Coefficients with Time
Time (sec)
Force Coefficients
Residuals
Iterations
Force Coefficients with Time
Iterations
Force Coefficients

START › User Input › CAD Preparation › Setup › Meshing › Solve › Post-Processing & Report › END

Automated Process

Collection of Images and case information › Assembling report in PDF format › File Saving



OpenFOAM

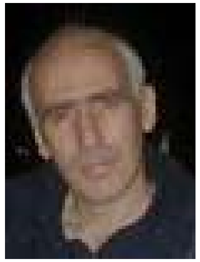
... in 2017, see us at

- **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

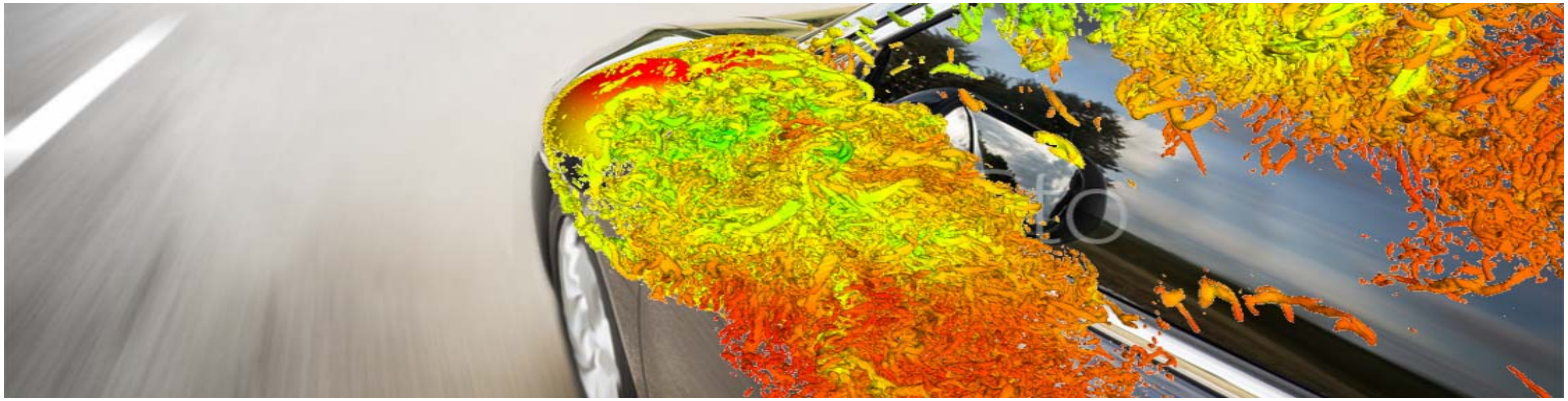


KYRIAKOS GIANNAKOGLOU
Adjoint Optimisation

CHRIS BEALE
Fuel cells

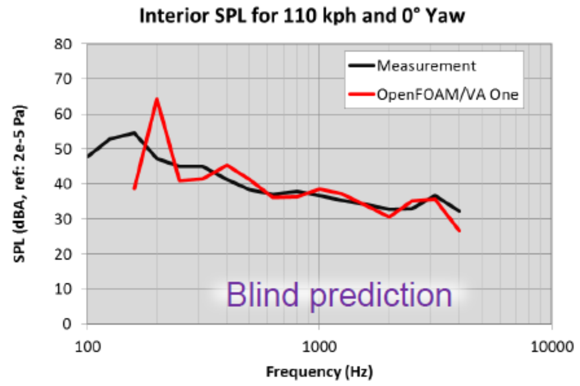
KARL MEREDITH
Fire modelling and Suppression

THANK YOU



Acoustics Solution

- Full frequency Acoustics Simulation during the entire vehicle development
  - Propagation and transmission modeling with
  ESI VA-One Solution)
  - Mechanical vibration (vibro-acoustics) with
  ESI VPS Solution
  - Flow induced noise (aero-acoustics) – **OpenFOAM**



Thank you

Questions / Comments