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

<table>
<thead>
<tr>
<th>Value and benefit</th>
<th>Return</th>
</tr>
</thead>
<tbody>
<tr>
<td>Development</td>
<td>Customized based on need and application</td>
</tr>
<tr>
<td>Consulting</td>
<td>Knowledge transfer and on-site services</td>
</tr>
<tr>
<td>Support</td>
<td>Best practices, ticket based, guaranteed response time</td>
</tr>
<tr>
<td>Training</td>
<td>Basic users, Advanced users. Application based. Accredited</td>
</tr>
<tr>
<td>Visual Work Flow</td>
<td>Customizable, scriptable GUI. Basic and advanced physics (e.g. CHT, VoF, OverSet)</td>
</tr>
</tbody>
</table>

ESI Value Proposition

Enterprise-level engagement with the OpenFOAM Originator; Developer; Release, Test, Maintenance and Support authority

ESI Global Resources

Strong partner. Technology expert. Flexible Licensing and Service models. End-to-end Process and Physics

License-based RoI

Typically at least 60% cost redeployment based on software ownership

Technology RoI

- Partner Programmes
- Process automation
- Full support and maintenance
- Custom developments and captured best practices
- Worldwide provider and technology experts
CFD Democretisation through OpenFOAM

Why OpenFOAM? Status 2012

Costs / Place of employment - CAE

- Costs / Place, CFD
- Costs / Place, OpenFOAM
- Costs / Place, other

High Performance Computing - available hardware at Wolfsburg

- Actual storage size:
  - 700 TB Workstations
  - 100 TB Backup
  - 170 TB CAE-Bench
  - 70 TB NAS

average use (7 x 24 h) of HPC Platform (Wolfsburg)

Efficiency Improvement

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

Wind Noise
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
  ‣ Overhauled 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
OpenFOAM

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

- OpenFOAM-v1606+ (June 2016)
  - 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
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](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](http://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

Overview

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

Solvers and Applications (1/3)

- overLaplacianDyM Foam
  - Eulerian flows
  - $\text{FOAM\_TUTORIALS/basic/overLaplacianDyM Foam/heat\_Transfer}$
- overSimpleFoam
  - steady state parametric studies of incompressible flow
  - $\text{FOAM\_TUTORIALS/incompressible/overSimpleFoam/aero\_Foil}$
Overset 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)

- `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
OVERSET  ORIGINAL
OVERSET  ORIGINAL
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%
Overset: Verification and Validation
Parallel Performance

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

![Scalability: OpenFOAM-v1706 (overInterDyMFoam)](chart.png)
Lumped point FSI

lumpedPointMotion library

- **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

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:
- Tutorial: $\$FOAM\_TUTORIALS/\$compressible/rhoPimpleAdiabaticFoam/rutlandVortex2D
Interface capturing - isoAdvector

**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.
- Tutorial: $FOAM_TUTORIALS/multiphase/interIsoFoam/damBreak
Solitary wave generation models

- 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

New source term in fvOptions: jouleHeatingSource

- solves an equation for the electrical potential $V$

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

- Where $\sigma$ is electric conductivity.
- The source is given by:

$$Q = \sigma \nabla V \cdot \nabla V$$

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

- `$FOAM_TUTORIALS/heatTransfer/chtMultiRegionSimpleFoam/jouleHeatingSolid$`
OpenFOAM

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

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
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

<table>
<thead>
<tr>
<th>Platform</th>
<th>Download</th>
<th>MD5 sum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Source</td>
<td>OpenFOAM-1.7.1.gtz</td>
<td>2454728d946ee773c963fac19be9ca84</td>
</tr>
<tr>
<td>Source</td>
<td>ThirdParty-1.7.1.gtz</td>
<td>22877bc0b1f2640cd09f175cfc5729b5</td>
</tr>
</tbody>
</table>
Command line completion

- 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

• 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 - Challenging on Multi-scale simulation

Multi-scale simulation from meteorology to building with several billion cell meshes
OpenFOAM
Collectively delivering a Professional OpenFOAM

• Shimizu Coorp.
  ‣ October 2016

100 Billion cells
(98,304 parallels)
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
Thank you

Questions / Comments