Agenda

- Introduction to ESI
- CFD services at ESI
- OpenFOAM at ESI
- OpenFOAM parallelisation
- FAQ
Introduction to ESI

CFD services at ESI

OpenFOAM at ESI

OpenFOAM parallelisation

FAQ
ESI Worldwide Operations

- Headquarters in Paris, France
- Offices in more than 30 countries
- Average Headcount: +900 people
- Over 187 Scientists
- WW Revenue 2010: 117 M$
Office in Bologna
- Direct sales and technical team
- 7 people
- Domain covered
  - CFD, FEM, Crash, RTM, Welding, Vibroacoustic
- Indirect sales and technical team
- 3 people
- Domain covered
  - Casting

Major customers
- FIAT
- General Electric
- Alenia Aeronautica
- Agusta Westland
- Lamborghini
- CNH
- IVECO
- Valeo
- Denso Systems
- Electrolux
FY 2011 Key Figures Revenue

- FY11 Revenue of 94.2 Million Euros (≈ $130M)
- 11.9% revenue growth vs. FY 2010
Diversified Business Sectors
Licensing and Engineering Services

**Aeronautics/Aerospace**


**Automotive**


**Semiconductor/Electronics**


**Heavy Industries/Energy**


**Defense**

General Dynamics Land Systems, Lawrence Livemore National Laboratory, Lockheed Martin, NASA, Naval Surface Warfare Center, Norfolk Naval Shipyard Detachment, NUWC (Naval Undersea Warfare Center), Raytheon Integrated Defense Systems, Sandia National Laboratories, SECAT. U.S. Army RDECOM-TARDEC, U.S. Army-Redstone, Wright Patterson AFB …

**Other**

Kellog, Brown and Root, Foster Miller, BP Solar, Crystal Solar, GE Healthcare, American Standard, MiaSole, …
Virtual Try-Out Space

VTOS

Virtual Human
Casting
Stamping
Composites & Plastics
Welding & Heat Treatment
Advanced CFD
Vibro-Acoustics
Simulation Process Management
Crash & Safety
NVH & Dynamics
Multi-Physics
Electromagnetism
Agenda

- Introduction to ESI
- CFD services at ESI
- OpenFOAM at ESI
- OpenFOAM parallelisation
- FAQ
Offices for Engineering Services in
- NA
- Europe
- East Asia
- India
  - Therefore, it is very competitive for cost

Over 100 CFD engineers (40% Ph.D.’s, 50% MS’s) specialized in variety aspects of Computational Fluid Dynamics (CFD), such as:
- Automotive - Aerospace
- Fuel Cell & Batteries - Plasma & Thin Film
- Powertrain - Solar & Green Energy
- Medical - Vibroacoustics
- Ocean Waves (SPH) - EMAG
  - Therefore, no challenge is considered too big

Access to major commercial solvers
- Therefore, it can handle any type of projects and develop customized solutions
ESI CFD Experience and Resources

- CFD experience for a wide range of applications since 1990
- OpenFOAM experience since 2006
- Extensive hardware resources based in Detroit and Europe

Tools in **ORANGE** denote ESI licensed software

- Proprietary
  - ACE+
  - PAM-FLOW
  - UH3D
  - STAR-CCM+
  - FLUENT
  - CFX
  - PowerFlow
  - RadTherm

- Open Source
  - OpenFOAM
  - Dakota
  - FDS

- PRE
  - CFD-GEOM
  - CFD-VisCART
  - ANSA
  - T-GRID
  - ICEM

- POST
  - CFD-View
  - CFD-POST
  - Ensight
  - ParaView

Tools
- **FASTRAN**
- KULI
- Flowmaster
- Dymola
- GT-Suite
- AMESIM
Agenda

- Introduction to ESI
- CFD services at ESI
- OpenFOAM at ESI
- OpenFOAM parallelisation
- FAQ
Free, Open Source Software

- **Open source**: the software is distributed with source code
- **Free**: no license fee and you are free to modify the software
- OpenFOAM is **licensed under the GNU GPL** (General Public License)
- If GPL software is redistributed, the source code must be made available
Open Source Code

- Open source is a software feature
- Control is the benefit
- If benefits are significant, open source delivers a competitive advantage
- “Our unique value proposition . . . is to cater to our customers’ need to gain control” Bob Young, founder of Red Hat, 1999
- ESI designs the OpenFOAM framework to provide ultimate control
Benefits of Control

- Users and system integrators (like ESI) of OpenFOAM can:
  - modify the software freely
  - assess code quality
  - know what the software is doing
  - deploy OpenFOAM however they wish, e.g. own hardware, cloud, etc.
  - influence development priorities
  - manage costs, i.e. choose when and how to get something for their money
OpenFOAM Software

- OpenFOAM is software primarily for computational fluid dynamics (CFD)
- Used for aerodynamics, fire simulation, chemical processing, ship design, etc.

- OpenFOAM is open source software of ESI, i.e.
  - ESI produces the software
  - and owns the OpenFOAM trademark

- OpenFOAM is distributed via the OpenFOAM Foundation
  - to which ESI assigns the copyright of OpenFOAM code
  - to ensure code base is licensed always open source only
  - using the General Public Licence (GPL)
Features of OpenFOAM

- **Meshing tools** (computational geometry): generation, conversion, manipulation
- **CFD solvers**: incompressible; multiphase; heat transfer; combustion, compressible (high speed); electromagnetics; particle
- **Physical modelling**: turbulence, transport/rheology, thermophysical, particle tracking, reaction kinetics/chemistry
- **Core technology**: numerics, linear solvers, parallelisation, dynamic mesh
- **Post-processing**: ParaView (open source), VTK, run-time post-processing, data manipulation tools, third-party
OpenFOAM pre-processor
OpenFOAM pre-processor

- Customized GUI for both Windows and Linux
- Entire model setup including mesh generation and post-processing
- Can also import meshes from other commercial codes such as Fluent, StarCD, CFD-VisCART
- One can run GUI with and without graphics
  - without graphics hardly takes 70Mb RAM irrespective of mesh size
- Intelligent handling of inputs on the fly
- Allows multiple OpenFOAM branches
  - OpenFOAM, OpenFOAM-ext, customized …
OpenFOAM includes snappyHexMesh mesh generator for complex geometries.

Automated, produces no bad cells.

Runs in parallel, can produce meshes of 100s millions cells.
Interface-tracking uses leading-edge methods developed by OpenCFD

Guarantees interface is preserved

One example of technology pioneered by OpenCFD
Introduction to ESI

CFD services at ESI

OpenFOAM at ESI

OpenFOAM parallelisation

FAQ
Core technology example: parallelisation

- Use **domain decomposition** to split mesh/fields into sub-domains, allocated to separate processors
- Applications run in parallel on sub-domains with communication by MPI-protocol software
- OpenFOAM has **an interface layer** (PStream) into which MPI libraries, e.g. OpenMPI, SGI MPT, Shmem, etc. can be plugged
- Applications generally require no ‘parallel-specific’ coding
  - so everything runs in parallel (unlike other codes)
With a suitable set up, OpenFOAM scales well to at least 1000 CPUs

- Cray XT, CSC Finland: “On Cray XT OpenFOAM exhibits super-linear scalability”
  - (64-1024 CPUs)

- Cray XT, HECToR UK: “OpenFOAM scales well on HECToR for both simple tutorial cases and for complex industrial cases”

- SGI Altix ICE 8400 2x6 core Intel Xeon X5680 3.3GHz SGI MPT: 95% efficiency from 12-768 cores

On performance in general. . .

- OpenFOAM is one of a family of finite volume unstructured CFD codes

  - . . .whose performance is broadly similar

  - Performance is different with certain algorithms
Introduction to ESI

CFD services at ESI

OpenFOAM at ESI

OpenFOAM parallelisation

FAQ
Why has ESI acquired OpenCFD?

- The virtual engineering market continues to show an increase in the demand for open source software, specifically in CFD domain.
- ESI Group already has OpenFOAM expertise via its earlier acquisition of Mindware Engineering, which will be a natural complement to the OpenCFD team and ensure a successful downstream integration.
- ESI believes in the openness of OpenFOAM and will be investing in OpenFOAM to scale it to reach more users.
How do I get professional support to use OpenFOAM?

- ESI provides a range of support services to OpenFOAM users
- Scheduled and on-site training courses at a range of locations worldwide;
- Support package including CFD assistance and code customization for OpenFOAM;
- Contracted code development projects for OpenFOAM;
- Large enterprise subscription including software support, platform services (tuning, benchmarking, porting), collaborative functionality integration and strategic planning;
- CFD consultancy, system and process integration.
What guarantee do I have that OpenFOAM will be always free and open source?

- The OpenFOAM software is copyright to OpenFOAM Foundation
- OpenFOAM Foundation is a non-profit, non-stock corporation which was established to distribute OpenFOAM exclusively under a no-cost open source license to the general public
- The “Bylaws” of the Foundation ensures that OpenFOAM will only be distributed free and open source.
Who is in charge of developing OpenFOAM?

- ESI-OpenCFD develops and maintains the OpenFOAM software and releases it through the OpenFOAM Foundation.
- Through the Foundation's reporting system, users can contribute:
  - Bug fixes
  - Feature additions
- The Foundation is also planning a contributions repository to allow larger contributions to be made available for download.
- Over time, prioritized contributions may be integrated by OpenCFD into the OpenFOAM distribution.