



CONSULTING SERVICES WITH OpenFOAM®

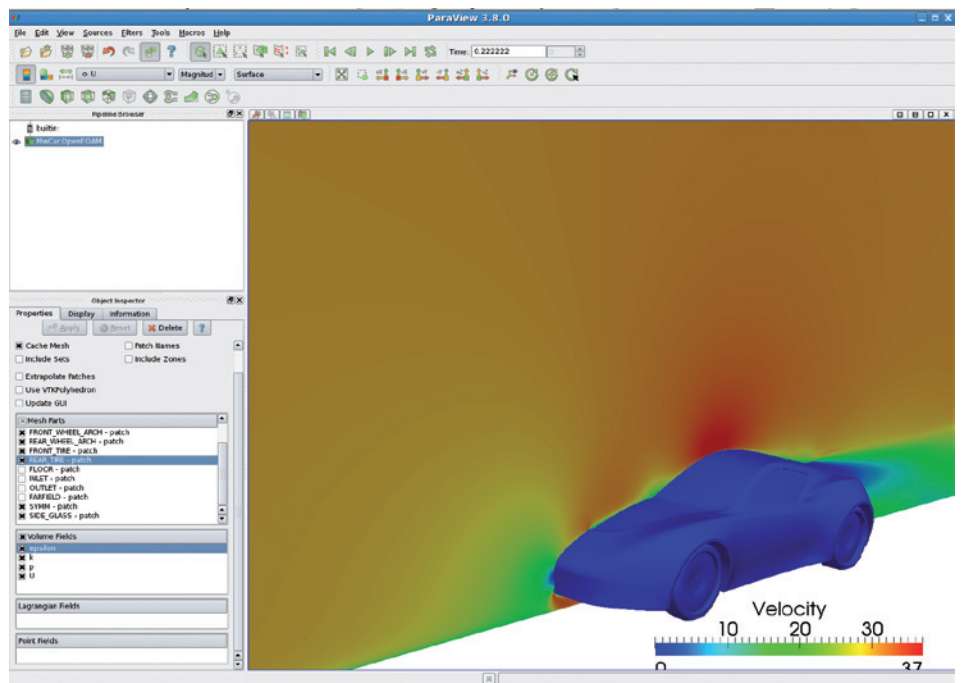
How ESI complements OpenFOAM®

- Industry experts in fluid dynamics and multiphysics simulation
- Hands-on experience with a variety of solvers and post processing tools
- Customized application development with Finite Element Analysis and Finite Volume Methods
- Cost effective solutions
- High flexibility of solvers, utilities and libraries with improved GUI
- Latest computing technology to minimize the solver's processing time
- Integration with other CFD applications and software
- Technical and maintenance support with access to dynamic developer community

Delivering CFD and Multiphysics solutions for more than 20 years

ESI CFD Consulting Services have been addressing CFD/Multiphysics challenges of industries like aerospace, thin film, automotive & transportation and energy for over 20 years. With on shore and off shore engineering support, customers get the required flexibility and access to expert resources. The team is backed by state-of-the-art computing infrastructure and support services.

OpenFOAM® is an industry recognized open source CFD code. A powerful, object-oriented, and customizable simulation platform, it can be adapted to customer needs by undergoing specific development. Utilization of this software requires numerical expertise to choose the most robust and efficient solvers, utilities and libraries. This is where ESI, with more than 20 years of CFD and FEA expertise, steps in. A partnership with ESI's consulting team can help you navigate and customize OpenFOAM® to meet your analysis requirements.

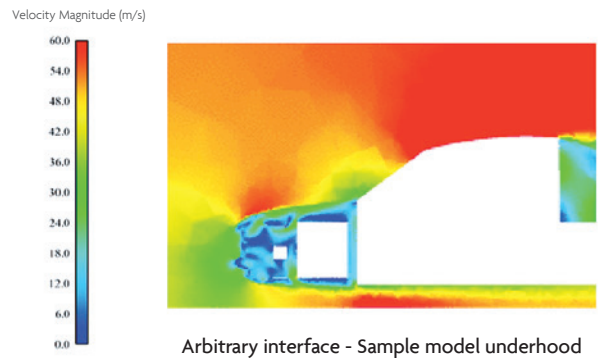
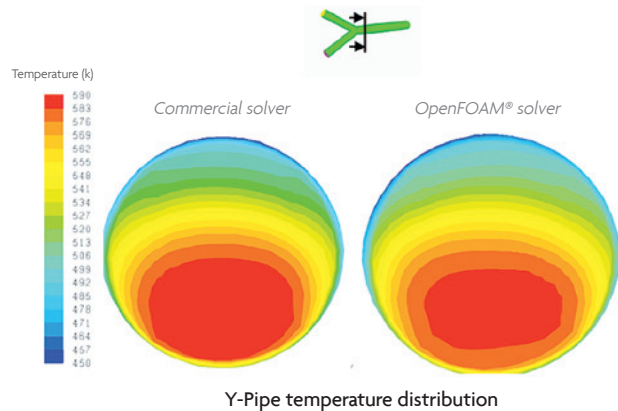
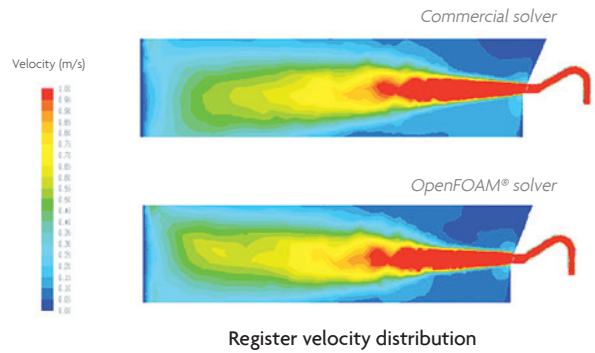
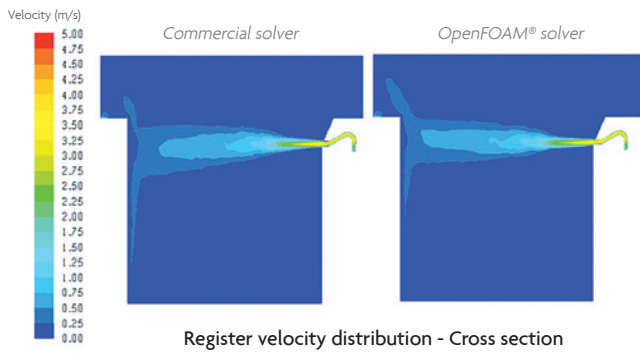
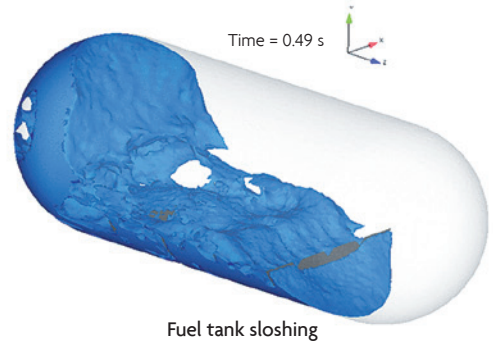
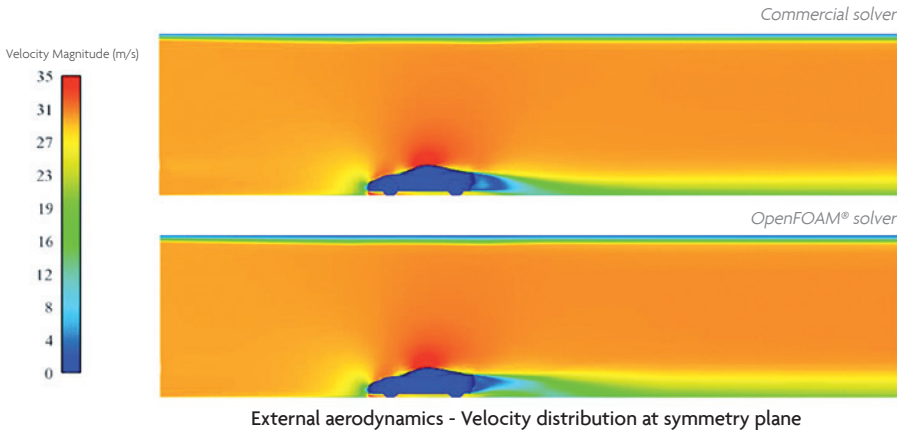


ESI's worldwide team of CFD engineers includes specialists with several years of experience in utilization and customization of OpenFOAM®.

What ESI's team offers:

- Customization of GUIs, solver code, utilities, and libraries for customer specific applications in numerous industries;
- Handling of complex geometry, utilizing body fitted Cartesian mesh using for instance ESI's mesh generation tool CFD-VisCART with automated wrapping capabilities for improved efficiency;
- Providing robust and efficient OpenFOAM® simulations through numerical expertise and in-house HPC resources.

Optimal results by ESI's consulting team using OpenFOAM®



Exposure to OpenFOAM® solvers:

- simpleFOAM
- rhoSimpleFOAM
- Oodles
- potentialFOAM
- buoyantSimpleFOAM
- rhoPorousSimpleFOAM
- CHTMultiRegionFOAM

Boundary conditions handled:

- Basic: fixed value, fixed gradient, time varying fixed value
- Derived: velocity, pressure inlet, turbulence intensity and mixing length
- Others: fan and porous media
- ESI's addition: radiative losses from wall

Other features:

- Parallel processing
- Output control capabilities
- Systematic file/directory structure

ABOUT ESI GROUP

ESI is a pioneer and world-leading provider in virtual prototyping that takes into account the physics of materials. ESI has developed an extensive suite of coherent, industry-oriented applications to realistically simulate a product's behavior during testing, to fine-tune manufacturing processes in accordance with desired product performance, and to evaluate the environment's impact on performance. ESI's solutions fit into a single collaborative and open environment for End-to-End Virtual Prototyping, thus eliminating the need for physical prototypes during product development. The company employs over 800 high-level specialists worldwide covering more than 30 countries.



ESI Group Headquarters | 100-102 Avenue de Suffren | 75015 Paris | FRANCE | T. +33 (0)1 53 65 14 14 | F. +33 (0)1 53 65 14 12 | info@esi-group.com

All PAM- and SYS- product names as well as other products belonging to ESI's portfolio are trademarks or trademarks of ESI Group, except specified proprietary mention. All other trademarks are the property of their respective owners - Specifications are subject to change without notice. OpenFOAM® is a registered trademark belonging to OpenCFD Ltd, UK