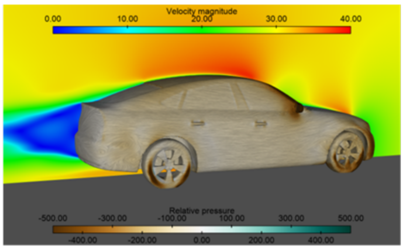
Computational Fluid Dynamics with OpenFOAM: An Introduction

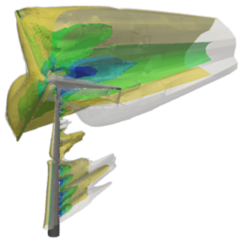
Summer Semester 2024

Course Information

- Module ID: 107720
- Credit hours: 2 SWS
- Lectures: April 02 – 04, 2024 (9:00 – 15:30)
- Exercises: April 15, 22, 29 and May 6, 2024 (17:30 – 19:00)
- Language: All course materials will be presented in English.

OpenFOAM is an open-source library for solving PDEs with wide applications in computational fluid dynamics (CFD) and beyond. This seminar will introduce students to the basic concepts of computational fluid dynamics and enable students to set up flow simulations in OpenFOAM, including generating grids, setting initial/boundary conditions, running simulations, post-processing, and developing customized solvers.

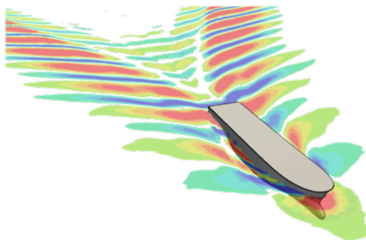


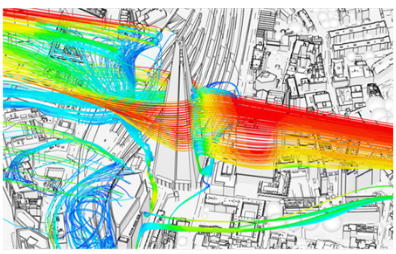


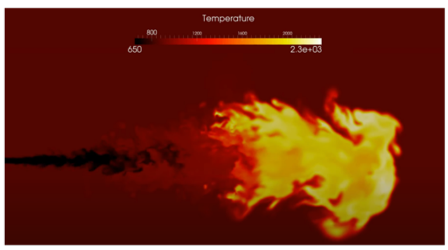
$$\frac{\partial T}{\partial t} + \nabla \cdot (\boldsymbol{u}T) = \nabla \cdot (\nu \nabla T)$$

OpenFOAM

```
fvm::ddt(T) + fvm::div(u, T)
== fvm::laplacian(nu, T)
```







Images courtesy of SimScale, Wolf Dynamics and Wehrfritz et al.

Instructors

Prof. Dr. Heng Xiao (ITLR-DDSim, University of Stuttgart),
Dr.-Ing. Xu Chu (SimTech, University of Stuttgart).

Teaching assistants

Zhuoran Liu (Ph.D. Student, ITLR-DDSim, University of Stuttgart),
Haochen Wang (Ph.D. Student, ITLR-DDSim, University of Stuttgart).

Learning Objectives

After the training, students will be able to:

1) familiarize with Linux command lines and C++ usage.
2) establish the concepts of OpenFOAM workflow.
3) set up and run simulations in OpenFOAM.
4) post-process the solutions: evaluate solution convergence, visualize by ParaView, extract quantities of interest, …
5) master high-level programming in OpenFOAM: develop customized solvers.

Agenda

Lectures

- **1st Day: April 02**

1) Basics of Linux command line
2) Concepts and software architecture of OpenFOAM
3) Basic workflow of CFD in OpenFOAM (mesh generation to postprocessing)

- **2nd Day: April 03**

4) Intermediate workflow in OpenFOAM (change mesh, run parallelly, plot residuals, etc.)
5) Additional post-processing techniques: sampling, etc.

- **3rd Day: April 04**

6) Basics of C++
7) Writing customized solvers
8) Detail about the implementation of finite volume method and source term

Exercises

1) **April 15:** Tips and tricks in OpenFOAM
2) **April 22:** Intermediate C++ (i.e., class)
3) **April 29:** “codeStream” for implementation of initial/boundary conditions
4) **May 06:** Simple non-premixed combustion solver

Project

Development of a simple combustion solver

Requirements

Before the course, the students are required to ***install OpenFOAM on their own laptops, which will be used in the class***. The course will be run in an interactive manner, alternating between lectures and exercises throughout these days.

Examination / Assessment

The students' academic performance will be evaluated based on only the final project. The assessment includes writing a report and delivering a presentation.