Master thesis topic:

Implementing low-dissipation high-order solver in OpenFOAM

Wojciech Sadowski[†] and Federico Lo Presti[†]

[†]Chair of Thermal Turbomachines and Aeroengines, Department of Mechanical Engineering

Mon 22nd Jan, 2024

Contact information: Email wojciech.sadowski@rub.de or room IC-E2/55.

What will You learn? Offered master thesis project allows for deepening of Your understanding of the numerical methods for computation of unsteady and turbulent flows, learning how to use OpenFOAM (an open source Computational Fluid Dynamics software) or improving Your proficiency with it. Furthermore, you will have the chance to gain valuable experience conducting larger scale computations of the high performance computing cluster.

Theoretical background OpenFOAM is usually used only with 2nd order accurate schemes, which are not accurate enough for detailed investigations of turbulent flows. Recently, a couple of works used OpenFOAM with the build in `cubic` interpolation scheme and self coded high-order time integration schemes (mostly Runge–Kutta). The codes used by these authors remain formally 2nd-order accurate in spatial dimensions, even if high order interpolation scheme is used, due to the way the face values are averaged over the faces (the leading order of the error is 2nd-order). Nevertheless, for small cell sizes, the solvers are shown to recover a 4th-order accurate solution in space. The dissipation properties of the `cubic` scheme are also shown to be much better than typically used `linear` scheme, which is 2nd-order accurate.

Project description The work will start from the base prepared by a student working at the chair previously, who has already developed a set of test cases and tooling for evaluation of errors[1] and testing of high-order discretization. The work will encompass

[1]The report of his work is available at request.

analysing the code already present in OpenFOAM, reading the papers describing the implementation of the time-integration schemes, programming and running the test cases. In particular, the work will consist of:

1. Analysis of how the `cubic` scheme is implemented.
2. Reproducing the results by other researches using the `cubic` scheme. In particular: **showing that the scheme leads to 4th-order convergence in space.**
3. Implementing a high-order Runge–Kutta scheme in the context of PISO pressure-velocity coupling. **Proving the convergence** in the three-dimensional Taylor-Green Vortex test case.
4. Testing the dissipative properties of the implemented time-discretisation scheme, **potentially in the inviscid case.**
5. Comparing the results of using both schemes for Homogenous Isotropic Turbulence and DNS of a channel flow.
6. *(Optional)* Including passive scalar transport in the software.
7. Writing the report on the work (master thesis).

Necessary skills To complete the tasks, previous experience in the following areas is required (*knowledge in all the mentioned areas is not required, however, You should have a little experience in at least some of them, feel free to contact us if you are unsure*):

- ability to use Linux operating system
- `python` and `bash` scription
- `C/C++` programming
- use of OpenFOAM or any other Computational Fluid Dynamics solver
- fundamentals of Finite Volume Method, or any other discretization scheme to treat PDEs
- foundation of fluid mechanics (knowledge about turbulent flows is helpful)