Using OpenFOAM to model of complex industrial devices

Some examples of reacting and swirling flows

Dr. Christophe Duwig, chdu@topsoe.dk	Haldor Tospøe A/S (HTAS) DK-2800, Lyngby, Denmark
Dr. Henrik Hassing, hnh@force.dk **M.Sc. Elisabeth Akoh Hove,** eah@force.dk	Force Technology DK-2800, Lyngby, Denmark

Abstract

CFD simulations were traditionally limited by the extremely high cost of the hardware. This lack of computational power advocated the use of *cheap* models although better options were available and also shaped the organisation of CFD activities. However, the past decade has enabled access to large CPU resources at low cost which has started to impact on the way we do or will do CFD. Indeed, the cost balance in CFD has changed dramatically and now the limiting factor is commercial CFD-codes because of license cost and often poor flexibility. The question is then 'how to use wisely the large available CPU resources?' and the answer certainly lies in OpenSource CFD. The present contribution will highlight the progresses at Haldor Topsøe and more generally in Denmark within the OSFRI [1] effort.

The OSFRI effort consists of eight Danish companies and two universities that have combined their efforts in development using OpenFOAM for replacing commercial codes. However, greater resources are required before using open source programs compared with the commercially available programs (less user support, lack of manuals, ...). The OSFRI project establishes a forum of member companies and training institutions, each of which contributes to the open source CFD programs. The aim is to mutualise resources and ease the switch to OpenFOAM with the support of the Danish Agency for Science, Technology and Innovation. It consists of three subprojects focusing on modeling particulate flows, reacting flow and turbulent swirling flows. Some key results of the activities (new solvers, courses, seminars, …) as well as the interplay between the partners will be presented at the workshop.

Beyond replacing existing programs, OpenFOAM was also recognized to open new avenues in term of simulation capabilities. At HTAS, the flexibility and no-license cost has enabled to use state-of-the-art modeling tools for investigating reacting and swirling flows. Large Eddy Simulation (LES) often remains in academia despite of its outstanding prediction capabilities but, the past year, it has been applied to a variety of problems. Figure 1 shows a simulation of hydrogen auto-ignition in hot oxidant: the jet penetrates the domain and turbulent structures develop in the shear-layer. Reactive mixtures resulting from the mixing eventually auto-ignite: pool of precursor HO_2 are formed followed by OH and the heat-release. We stress that the use of detailed chemistry is necessary for this problem and figure 1 also highlights the complex interaction between reaction and turbulence, hence the benefits of using LES. Figure 2 shows a swirling jet discharging in a combustion chamber. Although round at the nozzle, the jet cross-section deforms and follows a non-trivial dynamics; here well captured by the LES. These examples will be presented at the workshop for demonstrating the potential improvements offered by using OpenFOAM for LES of industrial devices.

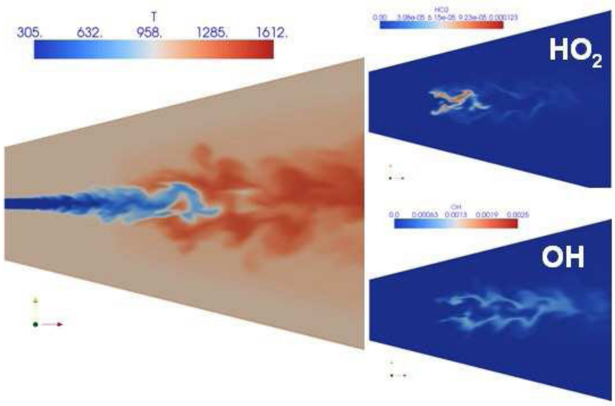

Figure 1: LES of auto-ignition of a high speed hydrogen jet discharging in hot oxidant using detailed chemistry.

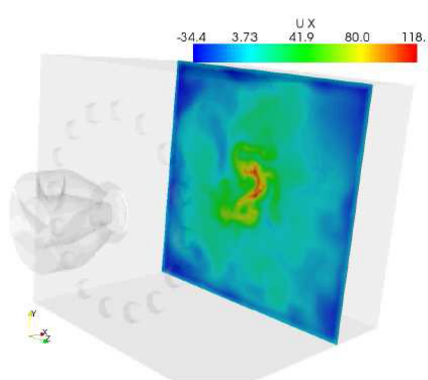

Figure 2: LES snapshot of the streamwise velocity field [m/s] in a swirling jet. The swirler geometry is represented in light grey.

Key words: OpenFOAM in the Danish industry, Combustion, Complex gas phase chemistry, Large Eddy Simulation.

References

[1] OSFRI project: http://www.osfri.dk