Development of a Framework for Internal Combustion Engine Simulations in OpenFOAM

Clemens Gößnitzer[1,*] and Stefan Posch[1]

[1] LEC GmbH, Inffeldgasse 19, 8010 Graz, Austria,
{clemens.goessnitzer,stefan.posch}@lec.tugraz.at

Keywords: *OpenFOAM, CFD, Methodology Development, Internal Combustion Engine*

High-accuracy simulations of internal combustion engines (ICE) allow a deep insight into the physical processes of the different phases of the engine cycle: gas exchange, mixture formation, compression, combustion and emission formation. The commercial solvers for ICE simulations provide a full package which covers these areas. However, the user of such software is unable to look into the source code, making it impossible to implement new models or investigate possible implementation errors in the code. Additionally, costs arise due to licensing requirements for commercial solvers.

The open source OpenFOAM framework[1] already provides a class and two solvers which are dedicated to internal combustion engines. However, there is no way to move engine valves and piston simultaneously with the standard tools that OpenFOAM provides. Thus, a new engine library was written for the open-source OpenFOAM CFD framework.

The new framework is capable of simulating a whole fired engine cycle. Piston and valves are moved simultaneously. Due to large deformations of the mesh, a methodology to avoid insufficient mesh quality was developed. Furthermore, the modelling of ignition and combustion is described in detail. To validate the method, the simulation results are compared with experimental data in terms of the averaged in-cylinder quantities pressure, temperature and mass.

REFERENCES

[1] The OpenFOAM Fundation: OpenFOAM, https://openfoam.org, accessed on 14/01/2022