Towards Industrial-Scale Simulation of Internal Combustion Engines Using OpenFOAM

Bulut Tekgul*[1], Karri Keskinen[1], Sakari Leponiemi[1], Mahmoud Gadalla[1]
*lead presenter: bulut.tekgul@wartsila.com

1 Thermofluids and Simulation Team, Wärsilä Finland

The recent developments made to OpenFOAM by CFD Direct [1] have significantly enhanced its capabilities in handling complex engineering simulations. These developments include robust dynamic mesh handling for moving geometries, improvements in conjugate heat transfer (CHT) modeling, and improvements in zone handling in OpenFOAM. These advancements address the long standing challenges in performing internal combustion engine (ICE) simulations in OpenFOAM, such as maintaining an acceptable mesh quality during valve and piston motion, accurately resolving heat flux across fluid-solid interfaces, and mesh generation for engine geometries. We present a selection of examples, built upon these developments, designed to perform robust and repeatable multi-cycle engine simulations.

References

[1] CFD Direct, “OpenFOAM: Open Source Computational Dynamics” [Online]. Available: https://cfd.direct. [Accessed: May 15, 2025]