Unstructured cut-cell method for flow problems with moving surfaces

W. Schröder[1,2], T. Wegmann[1] and M. Meinke[1]

[1]Institute of Aerodynamics and Chair of Fluid Mechanics, RWTH Aachen University,
Wüllnerstr. 5a, 52062 Aachen, Germany

[2]JARA Center for Simulation and Data Science, RWTH Aachen University,
Seffenter Weg 23, 52074 Aachen, Germany
E-Mail: office@aia.rwth-aachen.de

Numerical analyses of fluid-structure interaction problems are discussed in the context of internal combustion engines. Their flow field is characterized by the formation and break-up of large-scale vortical flow structures, in particular the tumble vortex. The production and convection of turbulence is responsible for the fuel-air mixing process and thus, these structures are major influencing factors for a stable combustion. The evolution of the tumble vortex is linked with the fluid jets past the engine valves during the intake stroke and the forced piston motion. Large-eddy simulations with high mesh resolution are necessary to accurately predict all relevant turbulent scales. A Cartesian mesh based numerical solution method for these fluid-structure interaction problems is presented. A semi-Lagrange level-set solver is used to track the location of the immersed moving boundaries individually. To handle the valve opening and closing, an extended multiple-level-set approach is introduced. The compressible Navier-Stokes equations are solved using a finite-volume method, where boundary surfaces are represented by a conservative sharp multiple cut-cell and split-cell method.