LES of cavitating flow inside a Diesel injector including dynamic needle movement

F Örley¹, S Hickel¹,², S J Schmidt¹, and N A Adams¹

¹ Institute of Aerodynamics and Fluid Mechanics, Department of Mechanical Engineering, Technische Universität München
² Chair of Computational Aerodynamics, Faculty of Aerospace Engineering, TU Delft
E-mail: felix.oerley@aer.mw.tum.de

Abstract. We perform large-eddy simulations (LES) of the turbulent, cavitating flow inside a 9-hole solenoid common-rail injector including jet injection into gas during a full injection cycle. The liquid fuel, vapor, and gas phases are modelled by a homogeneous mixture approach. The cavitation model is based on a thermodynamic equilibrium assumption. The geometry of the injector is represented on a Cartesian grid by a conservative cut-element immersed boundary method. The strategy allows for the simulation of complex, moving geometries with sub-cell resolution. We evaluate the effects of needle movement on the cavitation characteristics in the needle seat and tip region during opening and closing of the injector. Moreover, we study the effect of cavitation inside the injector nozzles on primary jet break-up.

1. Introduction
Recent developments of fuel injection systems aim towards increasing rail pressures for improving the mixing and combustion to meet future emission standards. A detailed understanding of the flow is necessary to control hydrodynamic effects, such as cavitation, that go hand in hand with higher rail pressures, and their influence on jet and spray characteristics. Typical dimensions of fuel nozzles inside Diesel injectors are in the range of a few hundred micrometers. This makes the instrumentation with diagnostic equipment for an experimental flow characterization a challenging task. Moreover, short intrinsic timescales, imposed by inherent flow dynamics or by functional components such as the opening or closing of the injector needle, complicates time-accurate measurements. Experimental analysis of cavitation erosion can thus supply information on incubation time, position, and the progress of erosion damage, but does not necessarily reveal the relevant flow physics. Computational Fluid Dynamics, on the other hand, can provide time-resolved information on flow structures in arbitrary small geometries. Numerical simulations thus have become an important tool in the design process. Recent work was presented, e.g., by Andriotis et al [1] for stationary, and by Battistoni et al [2] for dynamic needle positions.

In this work, we investigate the flow inside a 9-hole solenoid common-rail (CR) injector in terms of fluid flow turbulence, cavitation characteristics, and primary jet break-up. Dynamic needle movement is handled by a conservative cut-element based immersed boundary method. We employ a simple, closed form barotropic single-fluid cavitation model for ISO4113 air-free test oil including non-condensable gas effects.
2. Thermodynamic model

We apply a homogenous-mixture single-fluid model for cavitation, which is extended by a component of free gas to a two-fluid mixture model. A validation of the model has recently been published in Örley et al [3]. The single-fluid cavitation model has been extensively validated in previous studies, e.g., for LES of cavitating, turbulent, wall-bounded fuel flows by Egerer et al [4], and closing control valves by Örley et al [5].

The fluid consists of the three phases: liquid Diesel fuel surrogate (ISO4113 air-free test oil, $D$), fuel-vapor-mixture ($M$), and non-condensable gas ($G$). The volume averaged density $\rho$ inside a computational cell is

$$\rho = \sum_\Phi \beta_\Phi \rho_\Phi,$$

where $\beta_\Phi$ denotes the volume fraction and $\rho_\Phi$ the mean density of phase $\Phi$. The actual vapor-liquid interface of discrete vapor structures inside a computational cell is not reconstructed, unlike with sharp-interface methods, see e.g., Lauer et al [6]. Surface tension is thus neglected.

Liquid fuel and liquid-vapor mixtures are modelled as a mixture fluid in thermodynamic equilibrium. For purely liquid fuel, i.e. if the pressure $p$ is higher than the saturation pressure $p_s$, we employ a Tait equation of state

$$\rho_D = \rho_{ref} \left( \frac{p + B}{p_{ref} + B} \right) \frac{1}{N},$$

where $B = 1.19 \times 10^8$ Pa and $N = 11.5$ are calibrated parameters. By choosing a set of $\{B, N\}$, the fuel can be modelled as either isothermal or isentropic. We use the saturation conditions for liquid Diesel fuel as reference conditions, i.e. $p_{ref} = p_s$ and $\rho_{ref} = \rho_{s,liq}$. The Tait equation is able to model the liquid phase in the full pressure range of interest.

For liquid-vapor mixtures, i.e. $p < p_s$, we use a linearized equation of state based on the integration of the isentropic speed of sound, which leads to

$$\rho_M = \rho_{s,liq} + (p - p_s)/c_M^2.$$

The mixture speed of sound is set to $c_M = \sqrt{p_s/\rho_s}$.

The non-condensable gas phase is modelled as an isothermal ideal gas

$$\rho_G = p/R_G T_{ref}$$

at reference temperature $T_{ref} = 293.15$ K.

Equations (1)-(4) are combined to a closed equation of state that employs the non-condensable gas mass fraction $\beta_G$, which is transported as a scalar. The equation of state is solved iteratively for the cell average pressure $p = p(\rho, \beta_G)$.

3. Numerical Method

We use an implicit LES approach based on the Adaptive Local Deconvolution (ALDM) proposed by Adams et al [7] and Hickel et al [8, 9]. ALDM merges turbulence modelling and numerical discretization of the conservation equations. A subgrid-scale (SGS) model that is consistent with turbulence theory was obtained through parameter calibration, see Hickel et al [8] for details. The compressible version of ALDM [9] can capture shock waves, while smooth pressure waves and turbulence are propagated without excessive numerical dissipation. More details on the validation of ALDM for cavitating flows are given in Egerer et al [4]. We employ Cartesian multi-block grids with hanging nodes. Complex moving geometries are represented with the conservative immersed boundary method recently proposed by Örley et al [5]. This so-called cut-element method is an extension of Ref. [10] and enables the highly accurate and oscillation free simulation of weakly compressible fluid flows in moving geometries.
4. Setup
We investigate the flow field during a full injection cycle at different rail pressures $p_R$, and include the full 360 degree injector geometry to capture circumferential asymmetries in the flow field and communication between the nozzles. The prescribed needle lift is obtained from a multi-domain simulation model by Huber and Ulbrich [11] for specific injection pressures, see Fig. 2. We initialize the domain with liquid fuel at $p_R = \{1500, 2000\}$ bar above the closed needle seat, and gas at chamber pressure $p_c = 10$ bar, otherwise.

Geometry data was provided by Huber and Ulbrich [11]. Multiple grid resolutions are tested. The medium grid resolution including the injector geometry and the needle is shown in Fig. 1. The nozzle holes have a mean diameter of $d_H \sim 200 \, \mu m$. The grid resolution in the proximity of the nozzle holes is on the order of $\Delta x = \{5, 10, 20\}$ $\mu m$. for the fine, medium, and coarse mesh with cubic cells. The fluid domain is resolved with approx. 20 cells over the nozzle diameter. Grid coarsening is employed in regions far away from the injector. The medium grid consists of approx. $50 \times 10^6$ cells. Owing to the small timestep size of 1.75 ns, the real time for one cycle is on the order of 5 days on 4000 CPUs, whereas the recomputation of geometrical parameters in each time step consumes 50% of the computational effort due to the large number of cut-cells. The computations are run on the Leibnitz Supercomputing Centre, Munich.

5. Results
Figure 3 shows coherent turbulent structures and cavitation structures during the initial phase of the injection cycle. While the interior flow field in the sac volume quickly undergoes transition to fully turbulent flow, larger vortical structures in the jet are generated in the shear layer of the liquid-gas interface inside the combustion chamber, see Fig. 3(a). Cavitation in the needle seat due to a strong expansion of the liquid, visualized in Fig. 3(b), is already found in the early stages of the injection. The liquid flow inside the nozzle holes shown in Fig. 4 does not yet cavitate, but already reveals characteristic vortical structures.
Figure 3. Initial phase of the injection process at \( t = 0.11 \) ms. Left: iso-surfaces of \( \lambda_2 = -1 \times 10^{10} \) 1/s colored by absolute velocity \( u_{\text{mag}} \); right: iso-surfaces of void fraction \( \alpha = 0.1 \) (blue) and gas volume fraction \( \beta_G = 0.9 \) (grey).

Figure 4. Vortical structures inside the injection holes at \( t = 0.11 \) ms visualized by iso-surfaces of \( \lambda_2 = -4 \times 10^{11} \) 1/s colored by absolute velocity \( u_{\text{mag}} \).

We will present a detailed analysis of the flow field during the injection using isothermal and isentropic flow, multiple rail pressures, and different injection times.

Acknowledgements
We would like to thank B. Huber and H. Ulbrich (Institute of Applied Mechanics, Technische Universität München) for providing the injector geometry data and curves for needle lift. Computing time was granted by the Leibnitz Supercomputing Centre (LRZ) of the Bavarian Academy of Sciences and Humanities.

References
[1] Andriotis A, Gavaises M and Arcoumanis C 2008 *Journal of Fluid Mechanics* **610** 195–215
[2] Battistoni M, Xue Q, Som S and Pomraning E 2014 Effect of Off-Axis Needle Motion on Internal Nozzle and Near Exit Flow in a Multi-Hole Diesel Injector Tech. rep. Warrendale, PA
[3] Örley F, Trummler T, Hickel S, Mihatsch M S, Schmidt S J and Adams N A 2015 *Physics of Fluids* **27** 086101
[4] Egerer C P, Hickel S, Schmidt S J and Adams N A 2014 *Physics of Fluids* **26** 085102
[5] Örley F, Pasquariello V, Hickel S and Adams N A 2015 *Journal of Computational Physics* **283** 1–22
[6] Lauer E, Hu X Y, Hickel S and Adams N A 2012 *Physics of Fluids* **24** 052104
[7] Adams N A, Hickel S and Franz S 2004 *Journal of Computational Physics* **200** 20–20
[8] Hickel S, Adams N A and Domaradzki J A 2006 *Journal of Computational Physics* **213** 413–436
[9] Hickel S, Egerer C and Larsson J 2014 *Physics of Fluids* **26** 106101
[10] Meyer M, Devesa A, Hickel S, Hu X Y and Adams N A 2010 *Journal of Computational Physics* **229** 6300–6317
[11] Huber B and Ulbrich H 2014 Simulation and Validation of a Common Rail Diesel Injector *Proceedings of the Ninth International Conference on Engineering Computational Technology*