Large-eddy simulation of cavitating nozzle and jet flows

F Örley¹, T Trummler¹, S Hickel¹,², M S Mihatsch¹, 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 present implicit large-eddy simulations (LES) to study the primary break-up of cavitating liquid jets. The considered configuration, which consists of a rectangular nozzle geometry, adopts the setup of a reference experiment for validation. The setup is a generic reproduction of a scaled-up automotive fuel injector. Modelling of all components (i.e. gas, liquid, and vapor) is based on a barotropic two-fluid two-phase model and employs a homogenous mixture approach. The cavitating liquid model assumes thermodynamic-equilibrium. Compressibility of all phases is considered in order to capture pressure wave dynamics of collapse events. Since development of cavitation significantly affects jet break-up characteristics, we study three different operating points. We identify three main mechanisms which induce primary jet break-up: amplification of turbulent fluctuations, gas entrainment, and collapse events near the liquid-gas interface.

1. Introduction
Modern fuel injection systems play a key role for the optimization of the air-fuel mixing process in the combustion chamber. This is a key requirement for a more efficient combustion process in order to meet future emission standards. Recent developments aim towards increasing injection rail pressures, which enhances the jet break-up and mixing and hence improves the combustion. Higher flow acceleration goes hand in hand with thermo-hydrodynamic effects, such as cavitation, which occurs when the pressure locally drops below saturation conditions. The collapse of vapor pockets in regions of higher pressure causes strong shock-waves and high-velocity liquid jets directed towards the wall surface. This can generate high local stresses inside the surrounding structure. Loads induced by such phenomena are employed to clean nozzles from surface deposits, and can promote primary jet break-up. Also, cavitation can lead to choked conditions in a duct and hence maintains a mass flow rate independent of the pressure-drop. However, collapse events may cause material erosion and failure of the component.

A particular challenge in the context of injection of liquid jets into a combustion chamber is the mutual interaction of cavitating liquids and non-condensable gas. Sou et al [1, 2] assume that primary break-up of liquid jets is promoted by enhanced turbulent fluctuations caused by collapse events of cavitating structures near the nozzle outlet.

In this work we apply a simple, closed-form barotropic two-fluid cavitation model including a non-condensable gas component proposed by Örley et al [3]. The thermodynamic model is
an extension to our compressible framework for performing implicit large-eddy simulation of turbulent, cavitating flows of Egerer et al [4]. The model is used to simulate an experimental reference configuration of Sou et al [1, 2] of a cavitating water jet injected into air.

2. Thermodynamic model
The fluid consists of the three phases: liquid water (W), water-vapor-mixture (M) and non-condensable gas (G). The volume averaged density $\rho$ inside a computational cell is

$$\rho = \sum_\Phi \beta_\Phi \rho_\Phi,$$  \hspace{1cm} (1)

where $\beta_\Phi$ denotes the volume fraction and $\rho_\Phi$ the mean density of phase $\Phi$. For a detailed description of the thermodynamic model refer to Örley et al [3].

The actual vapor-liquid interface of cavitation structures is not reconstructed with sub-cell resolution. Surface tension thus is neglected. The cavitation model has been extensively validated for LES of turbulent wall-bounded flows by Egerer et al [4] and Hickel et al [5]. By using corresponding thermodynamic closure relations for each phase in Eqn. (1), the equation of state can be formulated in a suitable way to solve for the cell-averaged pressure $p = p(\rho)$.

Liquid water and liquid-vapor mixtures are modelled as barotropic mixture fluid. Integration of the isentropic speed of sound leads to

$$\rho = \rho_{s,liq} + (p - p_s)/c^2,$$  \hspace{1cm} (2)

where $p_s$ is the saturation pressure and $\rho_{s,liq}$ is the saturation density for liquid water. The formation of vapor is modelled by a homogenous mixture model. For purely liquid water, i.e. $p > p_s$, the speed of sound is $c = c_{liq} = 1482.35$ m/s at ambient conditions. Comparison to more accurate models, such as the Tait equation, shows negligible deviation for pressures up to 200 bar. For liquid-vapor mixtures, i.e. $p < p_s$, the same equation of state is used, but a different speed of sound is employed. Here, we consider an average of the speed of sound between a frozen and an equilibrium isentropic phase change in the two-phase region. We use a numerical value of $c = c_M = 1$ m/s, which corresponds to a conservative upper limit.

The non-condensible gas phase is modelled as an ideal, isothermal gas at reference temperature $T_{ref} = 293.15$ K

$$\rho_G = \frac{p}{R_G T_{ref}}.$$  \hspace{1cm} (3)

3. Numerical Method
We employ an implicit LES approach based on the Adaptive Local Deconvolution Method (ALDM) by Adams et al [6] and Hickel et al [7, 8]. In contrast to explicit SGS models, implicit LES merges turbulence modelling and numerical discretization. ALDM is a nonlinear finite volume method and incorporates free parameters that control the implicit SGS model. An SGS model that is consistent with turbulence theory is obtained through parameter calibration [7]. The compressible version of ALDM [8] 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 discussed by Egerer et al [4].

4. Setup
We adopt the setup presented by Sou et al [1, 2], see Fig. 1, who experimentally investigated different flow regimes in a nozzle flow. We conduct LES for a set of cavitation numbers that lead to different cavitation characteristics: Case 1: $\sigma = 1.27$ – no cavitation inside the nozzle; Case 2: $\sigma = 0.78$ – developing cavitation; and Case 3: $\sigma = 0.65$ – supercavitation. The
5. Results

Figure 2 shows transmitted light images of instantaneous vapor structures observed in the experiment in comparison to our LES data. Cases 1 and 2 compare well, whereas the numerical results for case 3 show vapor generation inside large vortical structures in the nozzle center. Turbulent structures inside the duct as iso-surfaces of the $\lambda_2$-criterion coloured by axial velocity, together with cavitation structures as iso-surfaces of the vapor volume fraction $\alpha$ and wall pressure are visualized in Fig. 3. Turbulence is damped in regions of high vapor content.
Stable, cavitating corner vortices are observed for low cavitation numbers. Finally, the effect of the cavitating nozzle flow on primary liquid jet break-up inside the gas domain is shown in Fig. 4. For cavitation numbers $\sigma = \{1.27, 0.78\}$, see Fig. 4(a/b), the jet structure in the experiment is similar. For $\sigma = 0.65$, Fig. 4(c), in contrast, spray formation is observed. Small droplets and ligaments of liquid detach from the surface and cause an increased jet angle. Our numerical results, see Fig. 4(d/e/f), show only little effect of the cavitation number on the jet angle in the $x$-$y$ plane. In contrast, significant differences between the higher cavitation numbers $\sigma = \{1.27, 0.78\}$, and the low cavitation number $\sigma = 0.65$ are found in the $x$-$z$ plane, see Fig. 4(g/h/i). Here, we clearly notice a widening of the jet and a detachment of liquid structures from its surface, which closely resembles the observations in the experiments.

From an analysis of the transient data we identified three main mechanisms that lead to distortions of the jet surface and, ultimately, to a widening and break-up of the jet. First, turbulent fluctuations, which are induced by collapse events in the proximity of the exit plane of the nozzle, add to the momentum in wall-normal direction. This observation confirms the hypothesis of Sou et al [2]. Second, low pressure vapor regions near the nozzle exit and the gas filled plenum form a pressure gradient, which enables entrainment of gas from the outlet region into the nozzle. When the gas is being ejected back out, the water is accelerated towards the side walls and creates large scale bulges of liquid. Third, collapse events of cavitation structures inside the jet near the liquid-gas interface induce high velocity liquid jets directed towards the interface. This effect resembles the findings of Kobel et al [9], and causes small, needle-like structures of liquid in our simulation. A detailed discussion is provided in Örley et al [3].

References

[1] Sou A, Tomiyama A, Hosokawa S, Nigorikawa S and Maeda T 2006 JSME International Journal. Series B, Fluids and Thermal Engineering 49 1253–1259
[2] Sou A, Hosokawa S and Tomiyama A 2007 International Journal of Heat and Mass Transfer 50 3575–3582
[3] Örley F, Trummel 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] Hickel S, Mihatsch M and Schmidt S J 2011 Implicit Large Eddy Simulation of Cavitation in Micro Channel Flows WIMRC 3rd International Cavitation Forum, University of Warwick, UK, 2011 ed Li S C
[6] Adams N A, Hickel S and Franz S 2004 Journal of Computational Physics 200 20–20
[7] Hickel S, Adams N A and Domaradzki J A 2006 Journal of Computational Physics 213 413–436
[8] Hickel S, Egerer C and Larsson J 2014 Physics of Fluids 26 106101
[9] Kobel P, Obreschkow D, Dorsaz N, de Bosset A, Nicollier C and Farhat M 2005 Phys Rev Lett 97