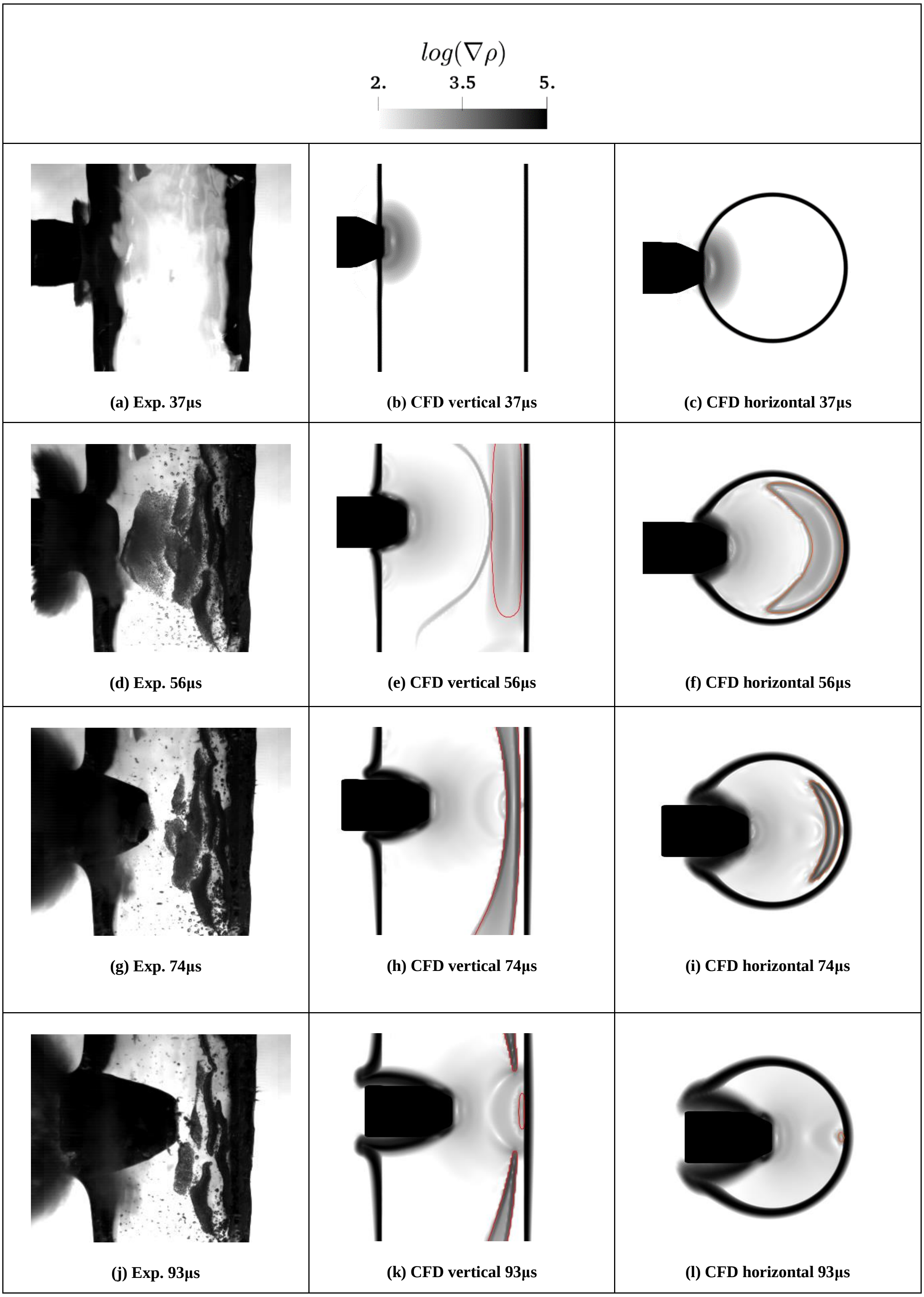
Summary of Deliverable D3.7: Validation of Immersed Boundary Method with Cavitation

Provided by ESR 10, Mr Evangelos Stavropoulos Vasilakis, under the supervision of Prof. Manolis Gavaises, CITY University of London, June 2018

Pressure wave interaction with material interfaces is known to lead to tension and consequently to formation of cavitation in liquids. The aforementioned effect is especially important in the interface of materials with a large acoustic impendence ratio and is relevant in many disciplines involving high velocity impacts, percussion and strong pressure waves (e.g. blast waves). High velocity liquid to liquid or solid impacts refer to physical problems of high engineering interest, as they are linked to cavitation inception, erosion development and eventual damage of mechanical structures and machines, including hydraulic machines, steam turbines, rain drops on airplanes. Moreover, shock wave interaction with liquid-gas interfaces may lead to cavitation induction as well and have important biomedical applications such as High Intensity Focused Ultrasound used for histotripsy or lithotripsy. Following the work of Field et al., who experimentally visualised shock propagation and cavity formation in impacted liquids at high velocities, the present study focuses on the simulation of the high velocity impact of a solid projectile on a water jet. The current study is the first attempt to quantify the qualitative experimental findings of the specific problem through computational fluid dynamics simulations, unveil the complex impact dynamics and provide detailed information about the shock wave structure, the cavitation formation, development and collapse, and the high-speed liquid jetting upon the impact. The aforementioned undeformable solid projectile is modelled through a direct forcing Immersed Boundary Method. An explicit density-based compressible flow solver suitable for flows with a large variation in Mach number is employed in combination with a two-phase flow model capturing the formation and collapse of cavitation. The proposed methodology adds a forcing term in the momentum equation to consider for the solid boundary in the domain and provides a reliable alternative to moving mesh strategies and a simpler formulation compared with the more mathematically accurate but computationally expensive cut-cell approach. During the projectile impact, high pressures are generated, and a shock wave is released that crosses the jet. As the initial shock travels across the jet and over the free surface, gets refracted on the water-air interface and rarefaction waves are generated, pressure drops under saturation values and large amounts of vapour are formed. As the projectile penetrates the jet and the waves propagate towards the opposite to the entry point free surface, the vapour cavity moves along, grows and collapses. The current study provides a better insight to the previous experimentally observed phenomena.



(a) Exp. 37µs
(b) CFD vertical 37µs
(c) CFD horizontal 37µs
(d) Exp. 56µs
(e) CFD vertical 56µs
(f) CFD horizontal 56µs
(g) Exp. 74µs
(h) CFD vertical 74µs
(i) CFD horizontal 74µs
(j) Exp. 93µs
(k) CFD vertical 93µs
(l) CFD horizontal 93µs

Consecutive frames from the experimental data from Field et al. (left) and numerical results (middle, right) for the impact of projectile on water jet, with U_{impact}=210m/s. For the CFD results, the logarithm of the magnitude of the gradient of density is plotted on the vertical (middle) and horizontal (right) middle plane on grayscale, along with the 0.1% contour for vapour volume fraction, represented by a red line