LES of Single Droplet and Liquid Jet Primary Break-up Using a Combined Level Set/Volume of Fluid Method.

F. Xiao*, M. Dianat, and J. J. McGuirk
Dept. of Aero. & Auto. Eng., Loughborough University, UK.
F.Xiao2@lboro.ac.uk M.Dianat@lboro.ac.uk J.J.McGuirk@lboro.ac.uk

Abstract

An accurate method for prediction of liquid jet atomisation is of utmost significance in many industrial applications. The engineering driver of prime interest in the present work is the aeroengine gas turbine fuel injector. In this area, substantial experimental and computational research has been carried out to understand the first part of the atomisation process - primary break-up. However, numerical modeling of primary break-up of a liquid jet is very challenging, especially for high liquid/gas density ratio. Simulations can become unstable due to errors in dealing with discontinuous conditions across the interface. As a consequence, many published numerical simulations are limited to a liquid/gas density ratio less than O(100). Since the majority of liquid jet atomisation experiments are carried out at atmospheric pressure with high density liquids, quantitative comparison between numerical modeling and experiment is quite rare. The simulation reported by Li et al [1] applied the algorithm proposed by Sussman et al [2] to a liquid (Jet-A) jet in an air cross-flow and hence had a high (650) density ratio. Adaptive Mesh Refinement and the removal of under-resolved small liquid structures were necessary as the only experimental data available was far downstream; in spite of the advanced modelling, agreement with measurements was relatively poor. The current research is focussed specifically on the early stage of primary break-up and its objectives are: (i) to develop a Large Eddy Simulation (LES) methodology for liquid jet atomisation at high density ratio, and (ii) to validate the method against experimental data for: single droplet break-up, a liquid jet in a coaxial air flow, and a liquid jet in an air cross-flow.

A coupled Level Set and Volume of Fluid (CLSVOF) technique is adopted as the interface-tracking method. Benchmark test cases are used to demonstrate that the CLSVOF method conserves liquid mass and provides a superior interface representation, combining the advantages of LS and VOF methods. The pressure jump across the interface is treated using the ghost fluid approach. Since the density is discontinuous across the interface, momentum is also discontinuous. Thus, an appropriate numerical scheme is required to treat such discontinuities when discretising the momentum equations. A single set of momentum equations are solved to provide the local liquid/gas velocity depending on the local value of the Level Set. In the region of the interface, however, an extrapolated liquid velocity field is calculated, so that momentum errors are minimised; appropriate boundary conditions at the interface are imposed leading to stable solutions even for liquid/gas density ratio O(1000).

The current paper provides details on the developed method and uses three test cases for validation, all with water as liquid and air as gas. The break-up of a single liquid droplet in a uniform air flow has been studied extensively and experimental findings are well documented. Such a flow represents an appropriate first test case for validating the proposed two-phase modeling methodology. It is demonstrated that the numerical procedure leads to predictions that are consistent with experimental observations across the entire spectrum of Weber number. It is shown that the simulated droplet correctly undergoes oscillatory deformation, bag break-up, or sheet-thinning break-up depending on the Weber No. Characteristics such as break-up time, maximum cross-stream diameter and drag coefficient agree quantitatively with experimental data. The proposed method is then applied to simulate the break-up of a liquid jet in a coaxial air flow. Predicted break-up lengths at different air and liquid velocities agree closely with measured data [3], although for low Weber No. the turbulence conditions at injector nozzle exit require detailed modelling. Finally, the primary break-up of a liquid jet in a cross-flow has been investigated. The outer boundary of the liquid spray formed by the cross-flow is shown to agree closely with experimental results [4]. A detailed analysis of the mechanism involved in the primary break-up process is provided.

[1] Li X., Arienti M., Soteriou M., C., Sussman, M. M., AIAA-2010-0210, 48th AIAA Aerospace Sciences Meeting, Orlando, USA, 2010.
[2] Sussman M., Smith K. M., Hussaini M. Y., Ohta M., and Zhi-Wei R., Journal of Computational Physics 221-2: 469-505, 2007.
[3] Charalampous G., Hardalupas Y., Taylor A., Int. Jnl. of Spray & Combustion Dynamics 1-4: 389-415, 2009.
[4] El Shamy, O. M., Experimental Investigations of Steady and Dynamic Behaviour of Transverse Liquid Jets, PhD Thesis, Univ. of Cincinnati, 2007.

* Corresponding author: F.Xiao2@lboro.ac.uk