

Large Eddy Simulation and Optical Studies of the Primary Break-up of a Thin Planar-Sheet Liquid Jet

2010-01-0622

Published
04/12/2010Giovanni Corbinelli
Delphi LuxembourgBizhan Befrui
Delphi LuxembourgWolfgang Reckers
Delphi Luxembourg

Copyright © 2010 SAE International

ABSTRACT

Volume-of-fluid large-eddy-simulations (VOF-LES) and optical imaging results of the primary breakup of a pulsed planar-sheet liquid jet are presented and compared. The planar-sheet thickness pertains to the GDI outward-opening conical-sheet spray. The investigations include injection conditions of 0.5 and 1 MPa fuel pressure and 0.3 MPa ambient pressure. The objective of the study is to assess the predictive accuracy of the VOF-LES method for analysis of the Kelvin-Helmholtz (KH) instability and the primary breakup of a transient, pulsed, liquid sheet jet, through inclusion of the injector nozzle flow domain into the simulations. Thus, the simulations do not resort to prescription of the issuing liquid jet velocity boundary condition. The results show good qualitative and quantitative accuracy for prediction of the KH interface instability waves and the liquid-sheet breakup process for the conditions studied.

INTRODUCTION

The design of the GDI injector valve-group (c.f. the conduit directing fuel flow to the nozzle, the pintle / seat designs, the nozzle geometry) for a specific spray geometry and atomization characteristics, through hardware design iteration, is a time-consuming and expensive exercise. This is due to the large number of (mostly) inter-dependent design parameters, coupled with precise dimensions and manufacturing tolerances, that render execution of a hardware-based DOE complicated and tainted with

uncertainty. Therefore, the development of a computational method that can assist the injector valve-group design analysis and optimization, through computer simulation, is highly desirable.

Over the past decade, there has been significant scientific progress in development of numerical methods for interface capturing/tracking for Volume-of-Fluid (VOF) two-phase flow simulations [1,2] and efforts for demonstration of the capability of the Volume-of-Fluid Large-Eddy-Simulation (VOF-LES) method for analysis of the liquid jet primary breakup process [3,4,5,6,7]. A review of the literature - for various liquid jet geometries and jet breakup regimes - indicates a remarkable potential of the method. However, in majority of the studies, the simulation domain excludes any significant portion of the injector nozzle domain (i.e. is limited to the vicinity of the nozzle exit). Consequently, the simulations of the liquid jet interface instability growth and primary breakup - and comparison with measurement data, when available - have exhibited strong dependence on the “imposed” liquid jet upstream boundary conditions (e.g. mean velocity profile, ‘turbulent’ velocity perturbations scale, etc.). Therefore, it can be argued that for a proper assessment of the VOF-LES predictive capability - especially to demonstrate the capability of the method to account for influence of valve-group design on breakup characteristics - it is important to incorporate the valve-group into the VOF-LES simulation domain.

In order to evaluate the VOF-LES method - especially to assess its capability to account for the influence of valve-

group design on the breakup characteristics - a combined experimental-computational research activity has been conducted at Delphi Customer Technology Centre Luxembourg. The objective is to perform VOF-LES simulations of the injector (valve-group) internal flow and near-field liquid-jet primary breakup and compare the results with the imaging data. The ultimate goals are two-folds: (1) eliminate the uncertainty in the boundary conditions in VOF-LES simulation of the emerging liquid jet, and (2) enable an evaluation of the method to reproduce the influence of the valve-group design features on the liquid jet primary breakup characteristics [8].

This paper describes the investigations of the primary breakup of a pulsed planar-sheet liquid jet, for one nozzle design, for different injection conditions. The objective is to assess the VOF-LES simulation capability for prediction of the liquid jet breakup process, without resort to ad-hoc perturbations of the liquid jet velocity boundary condition. Section 1 provides a description of the planar-sheet injector device. Section 2 presents the experimental test facility and setup for optical studies of the liquid-jet primary breakup. Section 3 presents an over-view of the mathematical and numerical features of the VOF-LES method used for analysis of the injector internal flow and near-field primary breakup process. Section 4 presents the VOF-LES simulation results and comparisons with the imaging data.

THE PLANAR-SHEET INJECTOR DEVICE

Figure 1 depicts the design of the planar-sheet injector hardware: it consists of the “body” part - with fixture accessories for its installation into the experimental test rig - and the movable “pintle” part. A planar-slit nozzle of cross-sectional dimension ($50\ \mu\text{m} \times 1.5\ \text{mm}$) and (vertical) length of 2.0 mm is formed between the “body” and the “pintle” parts. The large nozzle aspect ratio provides a two-dimensional planar-sheet spray, amenable to analysis with a simple experimental setup and light source.

The “body” provides a perfectly-sized cavity for installation of a GDI injector, that enables generation and controlled pulse-width duration / schedule of the pulsed liquid jet. A large channel connects the GDI injector cavity to the slit nozzle, designed to ensure uniform - with largely diminished turbulence pressure / velocity disturbances - supply of the injected liquid to the top of the planar-slit nozzle.

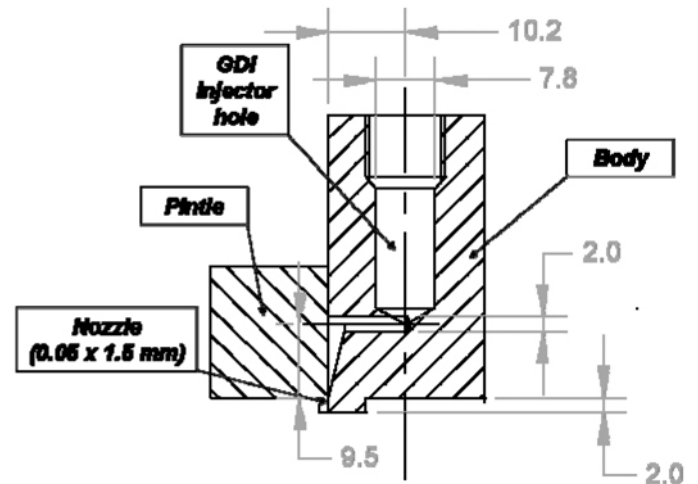


Figure 1. The planar-sheet injector device

The “pintle” does not control the injector (static) flow rate or injection pulse-width (these functions are done by GDI injector), rather its function is to direct and influence the planar-sheet spray. The different position settings of the “pintle” part - with respect to the “body” part - provide alternative nozzle configurations (e.g. protruding or flush); in addition, non-parallel nozzle wall configurations - including surface discontinuities that invoke flow separation - could be investigated.

THE EXPERIMENTAL INVESTIGATION

The objective of the optical investigation is to provide close-up imaging of the liquid jet primary breakup process, specially the features of the liquid jet - ambient air interface instability waves, with sufficient resolution for quantitative assessment of the VOF-LES simulation results. A second objective is to characterize the dependence of the spray microstructure (e.g. breakup length) and macrostructure (e.g. spread angle, skew angle) on the small geometric features of the nozzle design. According to the type of imaging (micro - or macro structure features) different setups of the optical system is employed.

EXPERIMENTAL APPARATUS

Figure 2 shows the experimental rig, which is comprised of a compressed air supply, a fuel supply, the continuous-flow optically-accessible pressure chamber, the planar-sheet injector device and the optical components for image acquisition. The fuel is supplied from a 200 bar fuel reservoir (with controllable fuel supply pressure in the range 0.3 - 20 MPa) and its delivery is controlled with the aid of the DIG injector driver-box electronics. The pressure chamber enables variation of the ambient thermodynamic conditions for pressure (0.3 to 2 MPa) and temperature (23° - 300° C).

The image acquisition system comprises of the light sources (flash light, short pulsed laser light) and CCD camera devices.

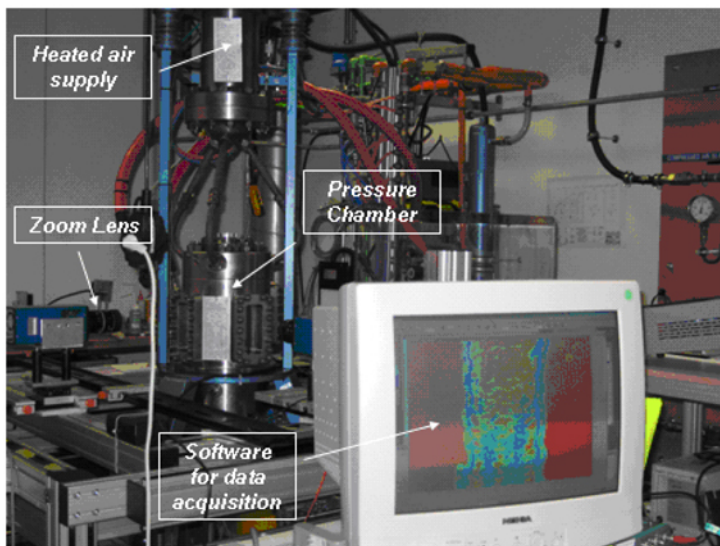


Figure 2. The experimental apparatus

The experimental arrangement for the spray imaging is illustrated in [Figures 3](#). In this setup, the planar-sheet liquid jet is oriented perpendicular to the axis camera/ conversion optics, providing a “frontal” spray image. The liquid-sheet interface instability waves are visualized in a back-illumination arrangement, using a ND YAG Laser (532 nm). The Laser illuminates the spray through a set of conversion optics in order to prohibit coherent-light interferences. The images are acquired by a CCD camera (1280×1024 Pixel) equipped with a far-distance microscope - providing a field-of-view of 2.1mm × 2.8mm.

<[figure 3](#) here>

EXPERIMENTAL DATA

Liquid Sheet Break-up data

[Figure 4](#) presents the close-up images of the spray development, obtained with the optical setup in [Figure 3](#). The field of view is 2.1 × 2.8 mm. The injection and ambient test conditions are $P_{\text{fuel}} = 0.5 \text{ MPa}$, $P_{\text{ambient}} = 0.3 \text{ MPa}$, and $T_{\text{ambient}} = 23^\circ\text{C}$. These injection conditions were deliberately selected to slow-down the breakup process and permit its optical investigation.

The sequence of images in [Figure 4](#) illustrate the initiation and growth of the Kelvin-Helmholtz instabilities through a combination of influences, namely the turbulence-induced liquid jet perturbations, surface-tension induced oscillations and - most notably - the interfacial aerodynamic shear force. The images illustrate development of largely two-

dimensional interface instabilities - with growing three-dimensional irregularities. The images indicate major growth of the three-dimensional irregularities, accompanied with evident interaction of the waves on the front and back sides of the liquid sheet at a distance of $Z < 1 \text{ mm}$ downstream of the nozzle (680 μsec . after start of injection), however it is difficult to ascertain if this represents a concurrent occurrence of the sheet primary breakup. (In the rest of the paper, for brevity, we will refer to this distance as the “breakup” length, in spite of a clear evidence of the liquid-sheet breakup). The results provide evidence that the three-dimensional flow effects associated with the nozzle-ends are confined to a narrow region, and do not influence the structure and growth of the instabilities; hence exclusion of these influences in the computational analysis of the liquid jet breakup is acceptable.

The experimental data show a reduction of the apparent break-up distance (as indicated by the change in the light refraction index) after the onset of the breakup. It must be noted that the injection velocity increases during the initial phase of the injection ($t = 0 - 700 \mu\text{s}$) due to the acceleration of the liquid volume within the planar-sheet injector device. Therefore, the observed reduction of the breakup distance is a consequence of the increased injection velocity and associated amplification of the liquid- and the gas-phase velocity and pressure disturbances. Also, the spray-front profile shows the dominant effects of gravity and the surface-tension during the initial stages of the injection ($t = 0 - 600 \mu\text{s}$) that diminish with the increase of the injection velocity.

<[figure 4](#) here>

Influence of fuel system pressure

The increase of the injection pressure (i.e. injection velocity) or the ambient pressure (i.e. the air density) are experimental parameters that can cause enhancement of the interfacial aerodynamic force and precipitation of the break-up process. [Figure 5](#) illustrates the liquid jet propagation and development of the KH interface waves for the injection conditions of fuel pressure = 1MPa, ambient pressure = 0.3MPa. The selection of imaging data is to allow comparison with the results of [Figure 4](#), at comparable liquid jet penetration distances.

<[figure 5](#) here>

The most notable features of the increased liquid jet velocity, associated with increase of the fuel pressure, as indicated by [Figure 5](#) images, are (1) a reduction of the KH wave-length, (2) a reduction of the breakup length, (3) a reduction of the surface tension effects (evident through visible contraction of planar-sheet jet, at the nozzle ends), and (4) an enhancement/ amplification of the three dimensional effects engendered by the nozzle ends. However, notably the KH waves are predominantly two-dimensional outside the region of the

three-dimensional nozzle-end effects, thus allowing comparison of the VOF-LES simulations (that do not account for the nozzle-end geometry) with the data.

THE COMPUTATIONAL INVESTIGATIONS

THE MATHEMATICAL METHOD

Conservation Equations of Multi-Phase System

The mathematical model comprises of the transport equations for conservation of mass and momentum of a two-phase flow system, comprised of two immiscible, compressible or incompressible Newtonian fluids, including surface tension. Accordingly, the single set of conservation equations that simultaneously describe the flow the two-phase mixture are [5,9]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (1)$$

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla P + \nabla \cdot \tau + \int_{S(t)} \sigma \kappa' n' \delta(x - x') dS \quad (2)$$

Where, U is the velocity, ρ is the density, σ is the surface tension coefficient, τ is the stress tensor, κ is the curvature of the liquid surface, n represents a unit vector normal to the liquid surface, and the operators $\nabla ()$ and $\nabla \cdot ()$ signify the gradient and the divergence operations, respectively. The integral term on the right-hand-side of equation (2) represents the momentum source due to surface tension: it acts only at the interface (indicated by the Dirac function $\delta(x)$) over the entire liquid surface $S(t)$.

The VOF-Based Interface Tracking Method

The principle idea of the “Volume of Fluid” approach is that the two- (or indeed multi-) phases system can be represented as a mixture of phases in which the phase-fraction distribution includes sharp, yet resolvable, transitions between the phases. Accordingly, the interface in a two-phase flow system is computed with the aid of the transport equation for the liquid volume fraction as the indicator function to locate the interface. The transport equation for the phase fraction α is:

$$\frac{\partial \rho \alpha}{\partial t} + \nabla \cdot (\rho U \alpha) = 0 \quad (3)$$

According to the definition of α , the mixture thermo-physical properties are calculated as

$$\rho = \alpha \rho_f + (1 - \alpha) \rho_g \quad (4)$$

$$\mu = \alpha \eta_f + (1 - \alpha) \mu_g \quad (5)$$

The “Volume of Fluid” (VOF) interface capturing/tracking method is a simple and flexible approach to simulation of multi-phase systems with free surfaces, especially for circumstances that the surface tension effects are not dominant. The main problems of VOF method relate to the accuracy of the numerical approaches used to ensure that the interface remains sharp, yet not unduly affected by numerical dispersion, conservation and mesh alignment bias [1].

Large-Eddy-Simulation Method

The LES-VOF equations are derived from equations (1),(2), (3), through a process of local volume averaging of phase-weighted properties, that entails decomposition of the dependent variables into resolvable and sub-grid scales of turbulent fluctuations and application of a filter that removes the sub-grid scales fluctuations from direct simulation. The filtering process, in conjunction with the non-linear terms in equation (2), produce additional terms, involving correlation of the variable fluctuations at sub-grid scales that require closure through mathematical models. The most notable of these terms is the sub-grid-scale (SGS) stress that represents the effect of unresolved scales of turbulence on the momentum transport process and its viscose dissipation. The Sub-Grid-Scale stress is defined as:

$$\tau_{sgs} = \overline{UU} - \overline{U}\overline{U} \quad (6)$$

The closure of the sub-grid-scale stresses is affected through a sub-grid-scale eddy-viscosity model that adopts a one-equation transport model for the sub-grid turbulent kinetic energy (κ), in conjunction with the Smagorinski's dissipation model [5, 9]. The sub-grid correlation terms associated with the fluctuations of density, phase-fraction, surface tension, etc. are neglected in the absence of suitable closure models [5, 9]. The present calculations are performed on a computational mesh size smaller than the Taylor (micro) turbulence length scale; thus the influence of the sub-grid-scale terms on the solution is expected to be negligible.

The Numerical Solution Method

The OpenFOAM finite volume CFD code [10] is used for solution of the LES-VOF system of conservation equations. OpenFOAM incorporates special treatment of the phase-fraction transport equation, in order to preserve a sharp