

Disintegration Mechanisms of Intermittent Liquid Jets

Alexander Nygaard, Mireia Altimira, Lisa Prah Wittberg, and Laszlo Fuchs
Royal Institute of Technology

ABSTRACT

It has been observed that intermittent injection leads to improved spray characteristics in terms of mixing and gas entrainment. Although some experimental work has been carried out in the past, the disintegration mechanisms that govern the breakup of intermittent jets remain unknown. In this paper we have carried out a systematic numerical analysis of the breakup of pulsated jets under different injection conditions. More specifically, the duty cycle (share of active injection during one cycle) is varied, while the total cycle time is kept constant. The advection of the liquid phase is handled through the Volume of Fluid approach and, in order to provide an accurate, yet computationally acceptable, resolution of the turbulent structures, the implicit Large Eddy Simulation has been adopted. The results show that the primary disintegration results from a combination of stretching, collision and aerodynamic interaction effects. Moreover, there exists a strong coupling between stretching and collision as stretching makes the pulse thinner prior to the contact between pulses. In this work, the purpose is to study the collision contribution to breakup in terms of the near nozzle pulse disintegration rate. When approaching the low duty cycle limit, this effect is significant because of the lower liquid volume of the pulse. In contrast, for a high duty cycle, the stretching effect is limited and a wide tail region remains as an obstruction for following pulses. However, the integral momentum of the pulse is maintained to a larger degree that has an adverse effect on the outcome of the collision event.

CITATION: Nygaard, A., Altimira, M., Prah Wittberg, L., and Fuchs, L., "Disintegration Mechanisms of Intermittent Liquid Jets," *SAE Int. J. Fuels Lubr.* 9(1):2016, doi:10.4271/2016-01-0851.

INTRODUCTION

The field of atomization deals with the process of an initially coherent quantity of liquid, such as a jet or a sheet that undergoes breakup into smaller parts [1, 2]. The process of liquid jet disintegration and the subsequent mixing with the surrounding gas is used in a multitude of industrial applications, such as spray cooling, coating and fuel injection [3]. Depending on the application, the flow may be subject to restrictions since spatial confinement and the time available to generate the desired properties is generally limited. In order to obtain the desired spray quality while complying with such restrictions, different strategies are adopted, such as higher injection pressures, gas assisted atomization or intermittent injection. Many applied studies related to the automotive industry have investigated the concept of injection unsteadiness to modify the outcome of the process of fuel injection and combustion. Experimental works utilizing multiple injection strategies report substantial influence regarding emission levels and fuel efficiency. Payri et al. [4] investigated different multiple injection strategies in a low compression ratio Diesel engine, where split injection was found to improve the combustion. Using multiple pilot injections before injecting the main fuel charge resulted in higher energy conversion efficiency and total heat release. The gain was a result of the improved charge combustion, in turn due to the pilot injections providing favorable conditions. Similar improvements were observed in [5], reporting a significant reduction in heat release rate while maintaining the maximum combustion pressure and increasing the

operating stability using pilot injections. Experiments carried out in [6] on a heavy-duty Diesel engine also varied the injection timing and injection mass distribution to study its effect on the heat release and emission levels. The highest fuel efficiency was found at the most advanced main injection timing. Moreover, this injection timing was observed to decrease soot and particle concentration levels, but the NO_x levels were found to increase. In contrast, injection strategies utilizing a post injection after the main charge had the opposite effect. Injection strategy manipulation evidently has a great influence on the mixture formation process. Thus, understanding how the liquid breakup changes due to intermittence is key in order to meet the increasingly demanding emission requirements.

Extensive research on a more fundamental level has been carried out in the past, characterizing fully developed fuel sprays by the angle of the formed spray, penetration into the combustion chamber and droplet statistics at various locations downstream of the injector [7, 8]. At this stage, the liquid has already undergone a first disintegration, known as primary breakup, into a more dispersed state by detachment of larger fragments from the bulk liquid. Recent improvements in visualization techniques have shown that very important features develop in the region in close proximity of the nozzle. Experimental works like [9] show a clear influence of the near nozzle flow on the main spray characteristics further downstream of the injection. However, the requirements in terms of spatial and temporal resolution make detailed experimental characterization of the flow near the injector very difficult and

conditions found in actual internal combustion engines. Therefore, further investigation is needed to study the early development of intermittent fuel jets and its effect on the dominant breakup mechanisms as well as the outcome of the primary breakup process.

Because of the difficulties associated with the high-speed jet breakup process, it is experimentally a challenge to reveal the underlying mechanisms of these flows. Here, numerical simulations may provide further insight into the dominating features of the unsteady jet injection. Works focusing on the development of single-phase starting jets have found the generation of energetic flow structures near the jet core region to give rise to considerable increases in the local entrainment, already highly unsteady in nature. Bing et al. [10] numerically investigated the effect of sudden injection stop in single-phase jet flow, and confirmed a region of increased ambient fluid entrainment in the near-nozzle region. Their simulations revealed greater growth and separation of large vortical structures during the deceleration phase and pointed out changes in vorticity production and convection as the main cause. Furthermore, the dissipation rate of the mixture fraction was reported to decrease during deceleration, attributing entrainment increase to large scale vortical motion. For such flow phenomena, description by traditional parameters becomes difficult since transients in the flow become dominating.

Recently, highly resolved numerical simulations have contributed with valuable information on the fragmentation dynamics occurring on micro-scale [11, 12, 13, 14]. Numerical studies of liquid jet atomization by Umemura et al. [11, 12] considered the injection of a quasi-steady diesel jet into a stationary gaseous domain. In addition to the shearing effect between liquid and gas, disturbance pathways are pointed out to contribute to the disintegration of the jet. Vortices generated at the head of the jet carry energy to the jet region upstream. In addition, droplets that are created at the front of the jet follow the re-circulating gas flow causing them to collide with the jet stem. This in turn results in later breakup due to surface instability. Such features related to the flow unsteadiness play an important role in the breakup process during the initial liquid jet propagation.

The aim of this work is to investigate the dependence on the duty cycle of the liquid breakup characteristics close to an injector-like orifice by using numerical simulations. In this study, the injection is not continuous, rather it includes a series of consecutive injections, also taking the interaction between different pulses into account. The flow in this region is dense, in the sense that many unbroken liquid structures occupy the core and that the droplet concentration is high. In order to maintain accuracy and handle liquid irregularity associated to this region, we employ methods considering the large liquid structures with the purpose of quantifying the initial breakup performance. The dynamics of the two-phase flow is handled through the VOF (Volume of Fluid) model, while Large Eddy Simulations (LES) approach is adopted to resolve the unsteady flow fluctuations. Such modeling framework combining multiphase flows and high-fidelity resolution has been previously used in the field of atomization and cavitation [11, 12, 13, 14, 15, 16, 17, 18, 19, 20]. From those, the VOF method has been adopted and validated in [16, 17, 18, 19, 20].

By tracking individual liquid pulses, the deposition from each core into fragmented volume is assessed over time. However, this study does not consider the detached liquid fragments becoming unstable that eventually will lead to further disintegration by secondary breakup. Observed topology changes are supported by considering liquid transport within the core region of the pulses.

CASE SETUP

Liquid is introduced into a domain of stationary gas through a series of consecutive injections. The computational domain has the shape of a rectangular box of dimensions $8D \times 8D \times 18D$, where D is the diameter of the inlet orifice (see Fig. 1). A velocity boundary condition with no artificial turbulence is applied on the circular inlet, located at one of the short sides of the box. Adding slight turbulence to the inlet flow to account for turbulence or cavitation events inside the injector would obviously provide more realistic results, but at the same time it would hinder the identification of the effects of pulse collision. The outlet is located at the opposing side, at which a zero-gradient boundary condition is employed. All other surfaces are treated as no-slip walls. Since simulations represent the gas as incompressible, the numerical artifacts of pressure waves at the boundaries are not considered.

This configuration is representative of a highly confined jet, which is commonly found in engineering applications. Because of this setup, the entrainment of ambient gas will be affected. Typically, ambient gas entrainment is considered farther downstream in the dilute regions. However, the present work focuses on the flow in the near nozzle region, where inertial effects dominate the primary breakup, and the radial spreading is less affected by the distance to the side boundaries.

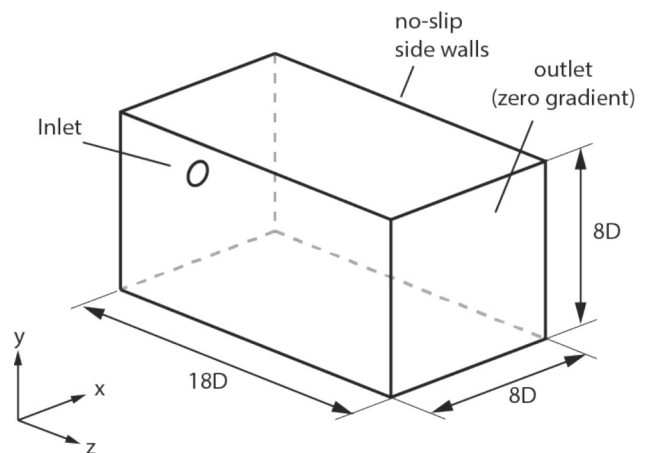


Figure 1. Schematic of the computational domain.

Five cases have been investigated varying the share of active injection to non-injection, the so called *duty cycle*. The total cycle duration, T_{cyc} , is kept constant. The injection velocity instantaneously alternates between U and zero (top hat profile), see Fig 2.

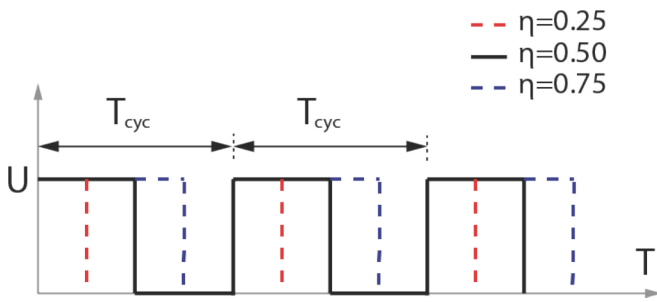


Figure 2. Schematic illustrating the injection strategy over time, with only three cases shown for clarity.

The injection scheme consists of the repetition of an injection period of duration T_{inj} and a dwell period (non-injection) of duration T_{dwell} that together form one injection cycle. The timing parameters of the different cases are presented in Tab.1. To define the injection condition, the Reynolds number (Re_{inj}) representing inertial to viscous forces, and Weber number (We_{inj}), representing inertial to surface tension forces, are used. These are defined as

$$Re_{inj} = \frac{\rho_l U D}{\mu_l} \quad (1)$$

$$We_{inj} = \frac{\rho_g U^2 D}{\sigma} \quad (2)$$

where ρ_g and ρ_l denote the gas and liquid densities, μ_l denotes the liquid dynamic viscosity and σ denotes the liquid-to-gas surface tension coefficient. All cases are run with $Re_{inj}=9300$, $We_{inj}=9000$ and liquid-to-gas density ratio of 20 and a viscosity ratio of 2.8. Such values were adopted as a reasonable compromise between computational capability and physical relevance. Similar values can be found in the literature [13, 17, 21].

Table 1. Injection and non-injection timing parameters.

Case	1	2	3	4	5
T_{inj} (D/U)	1.25	1.875	2.5	3.125	3.75
T_{dwell} (D/U)	3.75	3.125	2.5	1.875	1.25
η (-)	0.25	0.375	0.5	0.625	0.75

METHODOLOGY

The equations governing the flow are the isothermal Navier-Stokes equations with variable density.

$$\frac{\partial \rho}{\partial t} + u_i \frac{\partial \rho}{\partial x_i} = 0 \quad (3)$$

$$\frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re_{inj}} \left[\frac{\partial}{\partial x_j} \left(\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) \right] + \frac{\kappa \delta n_i}{We_{inj}} \quad (4)$$

The last term in (4) represents the contribution of surface tension, where κ is the curvature of the interface and δn represents the normal to the interface. The transport of liquid and gas is handled by the Volume of Fluid (VOF) method, representing the liquid distribution by solving the transport of the variable α , representing the liquid volume fraction.

$$\frac{\partial \alpha}{\partial t} + \frac{\partial u_i \alpha}{\partial x_i} = 0 \quad (5)$$

The two incompressible fluids are immiscible and no heat transport is taken into account. Density is uncoupled to the pressure and the variation of fluid properties is handled by weighting, using the liquid volume fraction according to

$$\phi = (1 - \alpha)\phi_g + \alpha\phi_l \quad (6)$$

At every time step, the phase distribution field is reconstructed and surface tension forces are computed using the first layer of neighboring cells. In order to avoid overestimations of the interface curvature, a smoothed version of the volume fraction field is used. Further details regarding the numerical framework can be found in Lörstad [12]. In the implicit Large Eddy Simulation framework, no explicit subgrid scale term is used to account for unresolved scales. Instead, the dissipation of energy at small scale is performed implicitly by utilizing a numerical scheme on a grid of finite resolution, which in combination acts as a filter. The approximation is justified by the argument that for an adequate grid resolution, the contribution of an explicit subgrid scale-term is comparable to the implicit dissipation associated to the numerics. The convective terms are discretized using a third order accurate upwind scheme. The pressure gradients and diffusive terms use a fourth order central difference scheme in the single-phase region and a second order accurate central difference scheme at the interface region. An implicit second order accurate scheme is used for discretization in time. The solver is based on the SIMPLE-algorithm and uses a multi-grid implementation and defect correction for efficiency. The numerical method maintains a sharp interface by employing a geometrical strategy to compute fluxes over each cell boundary [22].

We use a cell size of approximately 0.038D with an equidistant cartesian grid structure in the entire domain, producing a grid of close to 19 million cells. For resolution in time we use a second order accurate implicit scheme with a constant time-step $\Delta t \approx 0.014$ D/U, giving a maximum Courant number of 0.6 in the regions of highest velocity. It is true that the finite resolution does give rise to limitations in capturing interface dynamics on a very small scale. The same applies to the shapes of droplets and ligaments generated further downstream. A grid sensitivity study corresponding to a similar case can be found in [15], where it is proved that the adopted grid is fine enough to resolve the interaction between the larger scale pulse structures, which is the main focus of this work.