Integrated CFD on Atomization Process of Lateral Flow in Injector Nozzle

Jun ISHIMOTO*

*Institute of Fluid Science, Tohoku University, Japan
Tel: +81-22-217-5271; E-mail: ishimotojun@ieee.org

The governing equations for high-speed lateral atomizing injector nozzle flow based on the LES-VOF model in conjunction with the CSF model are presented, and then an integrated parallel computation are performed to clarify the detailed atomization process of a high speed nozzle flow and to acquire data which is difficult to confirm by experiment such as atomization length, liquid core shapes, droplets size distributions, spray angle and droplets velocity profiles. According to the present analysis, it is found that the atomization rate and the droplets-gas two-phase flow characteristics are controlled by the turbulence perturbation upstream of the injector nozzle, hydrodynamic instabilities at the gas-liquid interface, shear stresses between liquid core and periphery of the jet. Furthermore, stable and a high-resolution computation can be attained in the high density ratio ($\rho_l/\rho_g = 554$) conditions conditions by using our numerical method.

Keywords: Atomization, Injecor, Multiphase flow, Nozzle flow, Spray, Integrated CFD

1 INTRODUCTION

Fuel injection is essential for the operation and performance of internal combustion (IC) engines. High pressure fuel injectors are used in both Diesel and Direct Gasoline Injection (DGI) engines. The CFD combined with experiments prove to be a useful tool to test many configurations in a short time, to test different injectors, as well as to vary the operational parameters, such as injector positions, injection timing, duration, etc. For IC engines (both fuel and diesel), control of exhaust emission (such as unburned hydrocarbons and NO_x) and engine efficiency depends directly on the atomization of the liquid jet inside the combustion chamber (direct injection) or inside the admission pipe (indirect injection). Especially, the fundamental research on atomization and multi-phase processes in injector nozzle is multi-disciplinary in the sense that it involves the disciplines of fluid mechanics, multi-phase systems, measuring techniques and modeling. Therefore, a precise investigation of the mechanism of primary breakup process including upstream region is required because it controls either the length or the evolution of the potential core region (liquid core region where the magnitude of velocity is not damping toward the central axis) of the spray as well as it generates all the characteristics of the dispersed region (drops size, spray angle). We mainly are focusing on the breakup process of liquid column, formation of liquid film, and the formation of small droplets in a high-speed turbulent injector nozzle flow. The computational domain and rectangular structed mesh is created in reference to the actual injector nozzle which is used in the gasoline engines.

2 Governing equations

The numerical model represents the simultaneous unsteady flow of two immiscible, incompressible fluids, each having a constant viscosity and including surface tension. The flow is considered to be a laminar incompressible Newtonian and isothermal flow and governed by the Navier-Stokes equations and continuity equations. The surface tension is taken into account through the Continuum Surface Force (CSF) model [1], where the surface force is transformed to a body force which is only non-zero in the limited thickness interface region. The scalar F is used to denote the volume fraction field also called VOF field. Therefore, the governing equations for the one-fluid VOF-CSF model include the Navier-Stokes equations, continuity equation and VOF advection equation. It comprises a single set of conservation equations for the whole flow field even though fluid properties are discontinuous across the fluid boundaries. These equations are written as follows. The mass conservation equation:

$$\nabla \cdot \boldsymbol{v} = 0, \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \cdot (\rho \boldsymbol{v}\boldsymbol{v}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \int_{S(t)} \sigma \kappa' \boldsymbol{n}' \delta(\boldsymbol{x} - \boldsymbol{x}')\, dS, \tag{2}$$

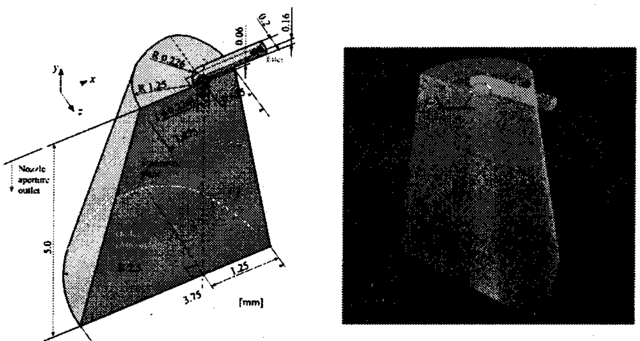
When the interface is advected by the flow, the evolution of the VOF advection function is given by

$$\frac{\partial F}{\partial t} + \nabla \cdot (F\boldsymbol{v}) = 0, \tag{3}$$

where $\boldsymbol{v}$ is the velocity, ρ the density, σ is the surface tension coefficient, κ the curvature of the liquid surface and $\boldsymbol{\tau}$ is the viscous stress tensor. Also, $\boldsymbol{n}$ represents a unit vector normal to the liquid surface. The last term on the right hand side of Eq. (2) represents the source of momentum due to surface tension. It acts only at the interface (represented by the Dirac function $\delta(\boldsymbol{x})$) over the entire surface described by $S(t)$. The interface between the phases is simultaneously computed using a surface capturing methodology which employs the volume fraction of one of the phases (here taken to be the liquid) as an indicator function to mark the different fluids. The interface is not defined as a sharp boundary and a transition region exists where the fluid is treated as a mixture of the two fluids on each side of the interface, which would in reality be a discontinuous step. The indicator function, which is equivalent to the liquid-phase volume fraction F, is defined as:

$$F = \begin{cases} 0 & \text{for a cell inside the gas} \\ 0 < F < 1 & \text{for a cell in the transitional region} \\ 1 & \text{for a cell completely in the liquid} \end{cases}$$

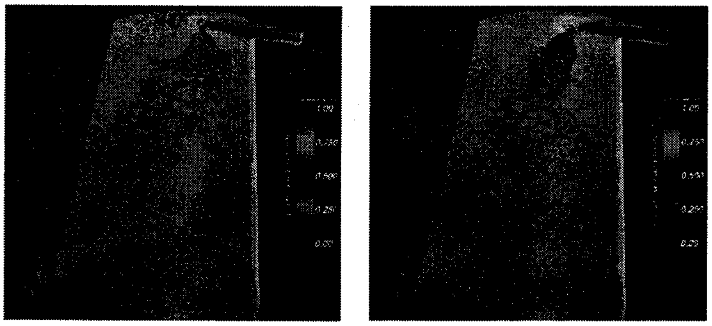
The VOF advection in Eq. (3) appears in the conservative form adopted from Puckett et al. According to the



(a) Geometry of the computational domain

(b) Used computational mesh

Fig. 1 Computational system for lateral entry flow model of injector nozzle



(a) $t = 5.3\text{e-}5$ s

(b) $t = 1.01\text{e-}4$ s

Fig. 2 Instantaneous iso-surface of liquid-phase volume fraction

definition of the indicator function F, the local density ρ and the local viscosity μ of the fluid are typically interpolated across the interface as follows:

$$\begin{cases} \rho = F\rho_l + (1-F)\,\rho_g \\ \mu = F\mu_l + (1-F)\,\mu_g \end{cases} \tag{4}$$

where the subscripts l and g denote the liquid and gas respectively. Since the interface is treated as a transitional zone, its exact shape and location are not explicitly known. During the numerical solution process, we apply the free-surface boundary conditions. There are three hydrodynamic boundary conditions at free surfaces: normal stress balance, tangential stress balance, and the kinematic equation. The kinematic condition is implied by the VOF advection. The surface integral in Eq. (2) that represents the surface tension therefore cannot be calculated directly. Brackbill et al. [1] overcame this problem with their continuum surface force (CSF) model, which represents the surface tension effects as a continuous volumetric force acting within the transition region. The body force term of $\boldsymbol{F}_{sv}$ in r.h.s. in momentum Eq. (2) effectively removes the explicit boundary condition at the interface in the governing equations. The LES-VOF equations are derived from Eq. (2) through a localized volume averaging of the phase weighted properties. This process is more commonly known as filtering, because it removes the very small scales of motion from direct calculation. This averaging in conjunction with the non-linear convection term in Eq. (2) produces an additional quantity into the momentum equation that cannot be directly calculated.

3 RESULTS AND DISCUSSIONS

The computational domain and rectangular structed mesh is created in reference to the actual injector nozzle which is used in the gasoline engines. The geometry of the computational domain and the generated mesh are shown in Fig. 1. The generated mesh is enough fine to apply the LES-VOF model.

Figure 2 shows the instantaneous iso-surface of liquid-phase volume fraction. The liquid-phase volume fraction is perturbed in the process of liquid-phase flow passing until the aperture inlet portion of injector nozzle. The lateral inflow of liquid column is transform to a small wavy liquid film at downstream of the aperture exit due to the small vortex induced by the wake passing through the nozzle throat, which is due to the effect of the nozzle atomized turbulence and negative pressure gradient. When the magnitude of the perturbation amplitude for liquid film is above a certain value, due to turbulent generation resulting from the boundary layer separation at the nozzle throat, the liquid film in the nozzle downstream is stretched to form ligaments. After that, the ligaments are broken up and subsequently forms the liquid droplets.

References

[1] Brackbill, J. U., Kothe D. B. and Zemach C. A, *J. Comput. Phys.*, **100** (1992), pp. 335–354.