International Journal of Multiphase Flow 63 (2014) 52–67

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

journal homepage: www.elsevier.com/locate/ijmulflow

Simulating cavitating liquid jets using a compressible and equilibrium two-phase flow solver

Yue Wang [a], Lu Qiu [a], Rolf D. Reitz [a,*], Ramachandra Diwakar [b]

[a] Engine Research Center, University of Wisconsin-Madison, Madison, WI 53706, United States
[b] Propulsion Systems Research Lab, General Motors Global R&D, Warren, MI 48093, United States

ARTICLE INFO

Article history:
Received 28 October 2013
Received in revised form 26 March 2014
Accepted 27 March 2014
Available online 5 April 2014

Keywords:
CFD
Two-phase flows
Compressible flows
Phase equilibrium
Cavitation

ABSTRACT

In internal combustion engines the injection of high-pressure liquid fuel into a low-pressure gas through a nozzle passage is an important process to atomize the liquid and achieve optimal fuel–air mixing. A Computational Fluid Dynamics (CFD) model is developed in the present work to simulate the internal- and external-nozzle flow fields in an integrated way. The model assumes that the flow within and near the nozzle is continuous, and an Eulerian flow solver is developed using the general conservation laws of fluid dynamics. Differences in the thermodynamic states of the liquid and gas phases are modeled with a Stiffened Gas Equation of State (EOS). A practical phase equilibrium solver is developed, and is implemented into the Eulerian flow solver to predict phase changes in the flows – in particular, cavitation of the liquid within the injector nozzle passage. The combined equilibrium solver is applied to single-component and two component flows with one component being non-condensable air. A number of test problems are simulated to verify the numerical methods and validate the proposed models. These include two-phase shock tube problems, a converging–diverging nozzle flow problem, a submerged liquid jet problem, and a cavitating liquid jet problem.

1. Introduction

The physics of interest in this work concerns high-pressure liquid injected through a nozzle passage into a low-pressure gas medium. Due to the momentum generated by the pressure gradient, the liquid jet interacts with the gas and is atomized into discrete droplets. One of its most important applications is found in automotive internal combustion engines, in which liquid fuel is directly injected into the combustion chamber during the intake or compression strokes (Heywood, 1988). In such engines, fuel atomization is a prerequisite for the efficient formation of the fuel–air mixture and its subsequent combustion. The amount and timing of the injected fuel are considered to be critical parameters to achieve the desired load conditions and combustion modes. Therefore, understanding the physical processes and controlling factors of liquid atomization is necessary for engine design.

The liquid atomization process consists of two steps: the near-nozzle primary breakup and the downstream secondary breakup (Reitz and Bracco, 1986, see Fig. 1). Liquid fuel is thought to be in the form of a continuous flow within a finite distance from the nozzle exit (Reitz, 1978; Liu et al., 2010), beyond which the primary breakup occurs due to flow instabilities generated by aerodynamics as well as nozzle disturbances (Sou et al., 2007). Secondary breakup refers to further breakup of the droplets into smaller ones.

Quantitative correlations based on fuel properties and the injection and chamber conditions have been established from experiments, and have been used as inputs to spray modeling (Naber and Siebers, 1996; Beale and Reitz, 1999). However, these correlations do not take into account the details of the nozzle geometry, such as the orifice inlet roundedness and length-to-diameter ratio, which are known to be important to the generation of cavitation and turbulence inside the nozzle. For high injection pressure cases, the local pressure can drop below the vapor pressure of the liquid fuel within the nozzle passage, which is thought to induce cavitation bubbles (Nurick, 1976). In addition to the thermodynamic pressure, the local viscous stress is also argued to contribute to cavitation inception (Joseph, 1995).

For nozzles with small length-to-diameter ratios, it is possible that the cavitated fuel reaches the nozzle exit and mixes with the chamber gas. This is called "super-cavitation" (Sou et al., 2007). In this case, the liquid fuel remains detached from the walls throughout the entire nozzle passage, and the liquid core is contracted at the nozzle exit compared to the nozzle hole size, so

* Corresponding author. Address: 1018A Engineering Research Building, 1500 Engineering Drive, Madison, WI 53706, United States. Tel.: +1 608 262 0145.
E-mail address: reitz@engr.wisc.edu (R.D. Reitz).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2014.03.006
0301-9322/

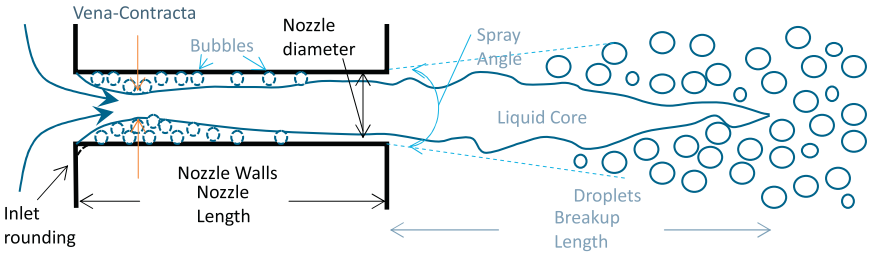

Fig. 1. Sketch of the atomization processes of liquid fuel injected from a nozzle.

the mass flow rate is reduced. The ratio of the actual mass flow rate over the theoretical one is defined as the discharge coefficient.

If the length of the nozzle passage is long enough, or if the injection pressure is not high, the liquid flow can re-attach to the walls downstream of the vena-contracta (Nurick, 1976). In this case, cavitation regions do not reach the nozzle exit and the discharge coefficient is higher compared to that of the super-cavitation case. However, cavitation still affects the downstream flow field by increasing the turbulent intensity within the nozzle (He and Ruiz, 1995). Finally, if the inlet corner is sufficiently rounded, the liquid fuel flows through the nozzle without detaching from the walls. Both the vena-contracta and the cavitation bubbles shown in Fig. 1 do not occur (Nurick, 1976), and the discharge coefficient is close to unity.

For automotive fuel injectors and practical high-speed injection conditions, it is difficult to visualize the internal flow and the near-nozzle dense liquid core with optical techniques unless high-intensity X-ray beams are used (see e.g., Liu et al., 2010). Such visualizations, however, do not provide detailed data about the local two-phase flow field. Further understanding of the physics and its application to engineering designs are aided by Computational Fluid Dynamics (CFD). CFD methods for fuel injection can be categorized into three classes. The first assumes a continuum for both the liquid and the gas phases, and the conservation laws are solved under Eulerian flow assumptions. Grid resolution is refined to the sub-micron level and is able to resolve droplets or bubbles without introducing any conceptual particles (see Fig. 2, left). A transport equation for an indicator function, e.g., a "level-set" function or the volume fraction, is used to track the liquid–gas interface. See the works of Menard et al. (2007), Desjardins and Pitsch (2010), Herrmann (2011) and Deshpande et al. (2012). Due to the very refined spatial resolution, this approach is termed quasi-DNS (Direct Numerical Simulation) by Gorokhovski and Herrmann (2008) and is considered to be a first-principles approach with high-fidelity. However, its application in engineering CFD is limited by its high computational cost. Also, the above-cited simulations apply only to the external nozzle flows, and do not account for internal nozzle flows or the physics of phase change. Other simulations focusing on internal nozzle flows either do not consider the effects of non-condensable gas within the spray chamber (see e.g., Schmidt et al., 1997; Habchi et al., 2008), or ignore the compressibility of the two-phase mixtures (Schmidt et al., 2010; Befrui et al., 2011).

A second method uses Lagrangian particles introduced by Dukowicz (1980) to represent one phase and a continuum Eulerian fluid to represent the other (see Fig. 2, right). The particles reside in the Eulerian grid cells and introduce source terms in the Eulerian conservation laws to account for interactions between the two phases. To simulate fuel injection, the continuous liquid core in the near-nozzle region is modeled with discrete "blobs" which typically are assumed to have the same characteristic size as the nozzle exit (Reitz, 1987), and phenomenological models are used to account for the blob breakup (Reitz, 1987; Beale and Reitz, 1999), as well as the collision and coalescence of the liquid droplets (Amsden, 1997). This significantly reduces the required mesh resolution and has been well-established in many open-source and commercial codes. Phenomenological models to consider the effects from nozzle flows are also proposed (Som and Aggarwal, 2010). However, a separate nozzle flow simulation which assumes an Eulerian liquid phase (e.g., Giannadakis et al., 2008) must be available to predict the flow conditions at the nozzle exit. The coupling between the internal and external nozzle simulations is inherently weak, due to potential inconsistencies of the two-phase models in the two separate simulations.

The third approach assumes both the liquid and gas to be a continuum in the internal- and near-nozzle regions. Therefore, no artificial boundary or coupling is required at the nozzle exit. The liquid phase is transitioned from Eulerian to Lagrangian particles only in far-downstream regions where coarse mesh is used. This methodology has been termed the "Eulerian–Lagrangian–Spray and

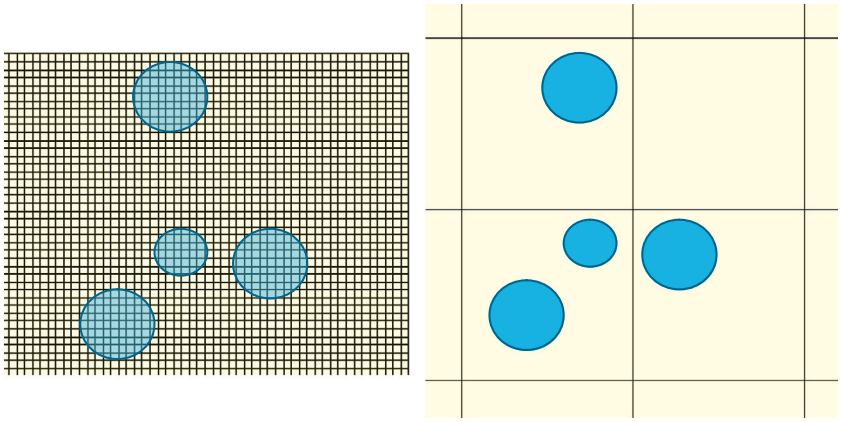

Fig. 2. Illustration of the Eulerian (left) and Lagrangian (right) CFD methods for two-phase flows.