Available online at www.sciencedirect.com

ScienceDirect

Proceedings of the Combustion Institute 36 (2017) 2417–2424

Proceedings of the Combustion Institute

www.elsevier.com/locate/proci

Simulation of drop deformation and breakup in supersonic flow

F. Xiao*, Z.G. Wang, M.B. Sun*, N. Liu, X. Yang

Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha 410073, China

Received 4 December 2015; accepted 9 September 2016
Available online 13 October 2016

Abstract

The deformation and breakup process of a liquid drop in supersonic flow is numerically studied with a coupled Level Set and Volume of Fluid method as the interface tracking approach. The Navier–Stokes equations in the liquid phase are solved by an incompressible flow solver using a finite volume method, and the governing equations in the gas phase are solved by a compressible flow solver using a finite difference method. Proper boundary conditions are specified at the interface for both liquid and gas flow solvers in order to correctly capture the interaction between the liquid and gas flows. It is demonstrated that the simulation cost can be significantly reduced by reducing liquid/gas density ratio while keeping the same Weber number and Ohnesorge number. Drop breakup at different Weber numbers is simulated. Bag breakup, bag stamen breakup, and multimode breakup modes are reproduced by the present two-phase flow solver. The physical mechanism for drop breakup in supersonic flow is investigated, and Rayleigh–Taylor instability is found to determine the breakup morphology for the studied Weber number range.

Keywords: Drop deformation; Drop breakup; Supersonic flow; Interface tracking; Rayleigh–Taylor instability

1. Introduction

In order to achieve high combustion performance in a supersonic combustion ramjet (Scramjet) engine, the liquid fuel must mix well with the supersonic air flow. The understanding of the atomization process in a supersonic flow is required to design a superior fuel injection scheme. Computational Fluid Dynamics (CFD) in two-phase flow has made significant progress and furthered our understanding of the atomization mechanism [1–7]. As the liquid fuel is injected into the supersonic air flow, the liquid jet first undergoes primary breakup, producing both small and large drops, and then the large drops further undergo secondary breakup in the downstream, producing ever smaller drops. For primary breakup, the liquid jet disintegration process can be directly resolved using an interface tracking method [1]. For secondary breakup, a lot of work has been done to develop models for secondary atomization [8]. However, no single model can describe all aspects of secondary atomization accurately [8]. Interface tracking methods have been used in [9–15] to directly resolve the drop deformation and breakup process to elucidate the physical mechanism, which may help improve the models of secondary atomization.

* Corresponding authors.
E-mail addresses: xiaof03@aliyun.com (F. Xiao), wind_flowcfd@163.com (M.B. Sun).

http://dx.doi.org/10.1016/j.proci.2016.09.016
1540-7489

All the simulations in [9–15] are for a drop in the subsonic flow, and both liquid and gas phases are assumed to be incompressible. For a drop in the supersonic air flows as in a Scramjet engine, two approaches can be used: the first approach is to treat both liquid and gas phases as compressible flows [16–20]; the second is to treat the liquid phase as incompressible and the gas phase as compressible [21]. Since the sound speed is much higher in the liquid than in the gas, the constraint of the CFL condition requests the time step to be very small in the first approach, resulting in high computational cost. Simulations of drop breakup in [16–20] are carried out in two-dimensional or axisymmetric coordinates. Furthermore, the equation of state is significantly different between gas and liquid, which may induce oscillations in numerical solutions near the interface. In order to avoid these two problems, the second approach is applied here to simulate the drop breakup in the supersonic flow. Furthermore, the total pressure in the combustor of a SCRAMJET engine is several MPa, and the corresponding compressibility of the liquid fuel (e.g., kerosene) is in the order of 0.1%. Therefore, it is appropriate to assume that the liquid is incompressible. The method in [21] solves Euler equations by neglecting the effect of viscosity, and the tangential velocities are assumed to be discontinuous across the interface, resulting in considerable errors in the shear stress on the drop. In order to resolve well the shear forces acting on the interface, the effects of viscosity are taken into account, and the Navier–Stokes equations are solved in the current methodology.

In this study, the deformation and breakup morphology of liquid drop in supersonic flow at different Weber numbers is numerically investigated. The difference between the drop breakup in supersonic and subsonic flows is examined. The paper is to provide physical insight on the drop breakup in supersonic flow with some characteristic quantities, which can be useful in building secondary atomization models in supersonic flow.

2. Numerical methods

In the present formulation, the interface is tracked using the CLSVOF method where two functions are used to represent the interface. VOF function F is defined as the volume fraction of liquid in a cell. Level set (LS) function ϕ is defined as the signed distance from the interface, with the contour $\phi = 0$ representing the interface, $\phi > 0$ in the liquid, and $\phi < 0$ in the gas. The CLSVOF algorithm is detailed in [13]. The usual spatially filtered LES formulation is employed in the single-phase flow regions. The governing equations for the incompressible liquid phase and compressible gas phase are detailed in [13] and [22] respectively.

The governing equations are solved on a Cartesian grid, with the gas flow calculated by a finite difference method and the liquid flow by a finite volume method. For simplicity, the numerical methods are described on a two-dimensional grid for illustration; extension to 3D is straightforward. The arrangement of flow variables on the grid is demonstrated in Fig. 1. The variables solved by the liquid flow solver (liquid variables) are arranged in a staggered manner: pressure p is located at cell centres; velocity components u (u^L) and v (v^L) are located at the corresponding cell faces. The variables solved by the gas flow solver (gas variables: pressure P, velocity components U and V, temperature T, and density ρ) are all located at cell corners. LS function ϕ and VOF function F are located at cell centres.

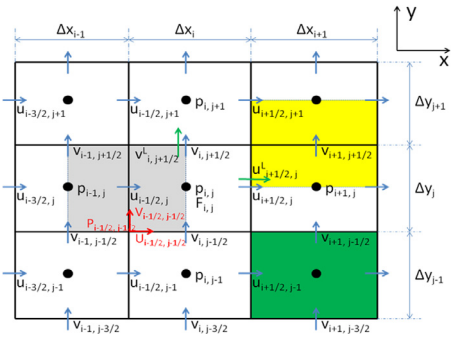

Fig. 1. Variable arrangement. Green-shaded region is pressure control volume (CV); grey-shaded region is x-momentum CV; yellow-shaded region is y-momentum CV.

2.1. Numerical methods for the gas flow solver

The supersonic gas flow is solved by the LES code developed by Sun et al. [22,23]. A second order TVD (total-variation-diminishing) Runge–Kutta method proposed by Shu [24] is used for temporal discretization of the compressible flow governing equation. A fifth-order WENO scheme developed by Jiang and Shu [25] is used here for spatial discretization of inviscid fluxes. A second-order central difference scheme is applied to discretize the viscous terms.

In order to solve the gas flow, boundary conditions should be specified at the interface. This can be achieved by specifying gas variables in the liquid region. The velocity in the liquid region ($\phi_{i+1/2,\,j-1/2} > 0$) is given by the liquid velocity field constructed in the liquid flow solver:

$$U_{i+1/2,\,j-1/2} = \frac{u^L_{i+1/2,\,j-1}\Delta y_j + u^L_{i+1/2,\,j}\Delta y_{j-1}}{\Delta y_j + \Delta y_{j-1}} \tag{1}$$

$$V_{i+1/2,\,j-1/2} = \frac{v^L_{i+1,\,j-1/2}\Delta x_i + v^L_{i,\,j-1/2}\Delta x_{i+1}}{\Delta x_i + \Delta x_{i+1}} \tag{2}$$

Eqs. (1) and (2) are interpolation of cell face values to cell corners, and the operation is done for all