International Journal of Heat and Mass Transfer 112 (2017) 97–112

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

A large eddy simulation of the breakup and atomization of a liquid jet into a cross turbulent flow at various spray conditions

Young-Lin Yoo [a], Doo-Hee Han [a], Ji-Seok Hong [a], Hong-Gye Sung [b,*]

[a] Department of Aerospace and Mechanical Engineering, Korea Aerospace University, South Korea
[b] School of Aerospace and Mechanical Engineering, Korea Aerospace University, South Korea

ARTICLE INFO

Article history:
Received 5 October 2016
Received in revised form 6 March 2017
Accepted 13 April 2017

Keywords:
LES
Two-phase flow
Eulerian-Lagrangian
KH-RT breakup model
Penetration depth
Sauter mean diameter
SMD

ABSTRACT

A three-dimensional large eddy simulation (LES) is conducted to investigate the breakup and atomization of a liquid jet into a cross turbulent flow for several variants of a liquid-gas momentum flux ratio by varying the liquid injection velocity and cross flow temperature. The spray-field dynamics are treated using a combined Eulerian-Lagrangian approach in which the gas phase is discretized using a density-based, finite-volume approach. A Kelvin-Helmholtz and Rayleigh-Taylor (KH-RT) hybrid wave breakup model is implemented to simulate the liquid column and droplet breakup process. While the KH model is applied to the liquid column breakup (primary breakup), the RT model is implemented to the breakup of the small droplets (secondary breakup). The detail flow structures of the counter-rotating vortex pair (CVP) and vortex interaction behind the injector are observed. The spray penetration depth in the cross-flow compares well with the experimental data and similar to empirical equations. The Sauter mean diameter (SMD) distribution is analyzed along the flow downstream representing somewhat different, and an analytical correlation model is proposed to pre-evaluate the SMD in the flowfield.

1. Introduction

The combustion efficiency of air-breathing propulsion systems using liquid fuel is sensitive to fuel spray characteristics such as liquid fuel atomization and fuel-air mixing. Using the jet in a cross-flow is one method of improving the atomization and air-fuel mixing characteristics in air-breathing and liquid rocket engine systems. The atomization process is strongly correlated with various complicated parameters such as turbulent intensity, spray condition, mixing condition, and even geometry characteristics.

With gas fuel injection, combustion performance is determined almost entirely by mixing the performance of the inlet airflow and gas fuel. However, liquid fuel is much more complicated because of the liquid column breakup and atomization processes, as shown in Fig. 1. Therefore, it is essential to understand these processes to design and analyze liquid fuel engines. In general, experimental investigations of atomization and air-fuel mixing processes in high-speed engines require precise experimental techniques (which are very expensive) to measure unsteady liquid-gas interactions and liquid fuel spray characteristics. A numerical dynamic investigation of the air-liquid mixing process is a potential alternative. A numerical analysis of this process in a high-speed engine combustor needs to take into account both the turbulent effects and liquid fuel spray characteristics simultaneously. Since the 1980s, several liquid breakup models have been developed to determine liquid spray characteristics based on experiments.

O'Rourke [2] studied a numerical analogy using the Taylor analogy breakup (TAB) model, which was based on an analogy between a forced oscillating spring-mass system and its correspondence to conditions at a low Weber number (We < 12). Ibrahim [3] proposed a droplet deformation and breakup (DDB) model that assumed that the liquid drop had deformed to an oblate spheroid due to a pure extensional flow from an initial spherical drop. This corresponded to conditions at a high Weber number (We > 40). However, the breakup criterion for DDB model presents difficulties at large Weber number because it is linearly proportional to it [4]. Also, Beale and Reitz [5] implemented a Kelvin-Helmholtz and Rayleigh-Taylor (KH-RT) breakup model, which was associated with two modes of wave instability on the liquid surface. The first was the Kelvin-Helmholtz instability growing on the surface of a cylindrical liquid jet. The second was the Rayleigh-Taylor instability of the interface between two fluids of different densities, which occurred with the acceleration (or deceleration) normal to this interface. The KH-RT model is the most popular of all hybrid models used today [6,7]. It successfully predicts the disintegration

* Corresponding author.
E-mail address: hgsung@kau.ac.kr (H.-G. Sung).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.04.064
0017-9310/

Nomenclature

A_1	characteristic coefficient of injector nozzle shape
a	parent droplet radius
a_i	acceleration vector
B_0, B_1	breakup model constants
C_D	drag coefficient
C_{RT}	radius constant of RT breakup model
C_τ	time constant of RT breakup model
D_{32}	Sauter mean diameter (SMD)
D_{ij}	nonlinearity of viscous stress term
d	diameter
E	specific total energy
e	specific internal energy
F_i	force vector
f	vortex frequency
f_{birth}	child droplet formation constant
g	gravity acceleration
H_i	energy flux
h	enthalpy
L_b	Levich length
MW	molecular weight
m	mass
N	number of droplets
p	pressure
Q	energy transfer
q	momentum flux ratio
q_i	heat flux vector
R	gas constant
R_u	universal gas constant
Re	Reynolds number
r	droplet radius
$\mathbf{S}_f$	source term of resolved-scale
S_k	source flux of species k
St	Strouhal number
T	temperature
Ta	Taylor number
t	time
$U_{k,i}$	reaction velocity of species k
u	velocity or x-velocity
V	volume or cell volume
v	y-velocity
We	Weber number
w	z-velocity
x	spatial coordinate
x_b	breakup length
Y_k	mass fraction of species k
Z	Ohnesorge number

Greek

δ_{ij}	Kronecker delta
θ	spray angle in blob model
λ	thermal conductivity
μ	viscous coefficient
ρ	density
σ	surface tension
σ_i	viscous work
τ_{ij}	viscous stress tensor
τ_{ij}^{sgs}	subgrid stress
τ_{KH}	breakup time constant of KH model
τ_{RT}	breakup time constant of RT model
ω_k	mass production of species k
$\Theta_{k,i}$	species diffusive flux of species k
$\Phi_{k,i}$	species flux of species k
Λ	wave length
Ω	maximum growth rate

Superscripts

$\cdot$	time difference
$\bar{\cdot}$	time average
$\tilde{\cdot}$	Favre average
sgs	subgrid-scale

Subscripts

b	breakup
child	child droplet
d	droplet
fs	latent heat
g	gas phase
H_2O	water
i, *j*, *ij*	spatial coordinate index
KH	Kelvin-Helmholtz
k	species index
l	liquid phase
m	mixture
o	injection or initial
parent	parent droplet
RT	Rayleigh-Taylor
rel	relative
s	source term
v	viscous term

process of high-pressure diesel sprays [8]. Kim et al. [9] studied TAB and KH breakup model for both non-evaporating and evaporating conditions. The TAB model showed rapid breakup near the nozzle exit causing under-predict breakup length, whereas KH model is better predicted for dense spray dynamics. KH-RT model showed better predictions of SMD distribution and the transient spray tip penetration than KH-DDB model [6]. Wang et al. [10] carried out a numerical analogy of a liquid jet injected on a plate using a sub-grid-scale (SGS) algebraic model of a large eddy simulation (LES) with KH breakup model. Yang et al. [11] carried out a numerical analysis of a jet in the crossflow of a square duct and only room temperature using an SGS algebraic model of an LES in a KH-RT breakup model. However, the penetration depth and SMD distributions of their numerical results were somewhat different from the experimental data.

The objective of the present study is the numerical investigation of the penetration depth and the SMD distributions at various conditions; momentum ratio (q) 9, 12, 18, 27 and crossflow temperature (T_g) 300, 473, 573 K. The Eulerian approach is used to analyze gas phase physics, and the Lagrangian approach is used to analyze liquid phase physics for calculation efficiency. The KH-RT breakup model is implemented to investigate the liquid breakup characteristics via an in-house code. An SGS dynamic model of an LES is implemented to simulate the unsteady turbulent flow fields.

2. Numerical method

A numerical analysis of the spray breakup is carried out using a compressible, two-phase, multi-block in-house code. The Eulerian-Lagrangian solver handles the coupled gas-liquid flow. The Eulerian approach is suitable for the gas phase under the assumption of the continuum. It is solved in three-dimensional space, while the Lagrangian solver locates and tracks droplets in a domain with interpolated Eulerian fluid properties.