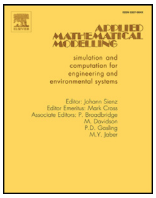
Applied Mathematical Modelling 40 (2016) 4712–4726

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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



Application of the improved cavitation model to turbulent cavitating flow in fuel injector nozzle

Barış Biçer[a,b,*], Akira Sou[a,1]

[a] Graduate School of Maritime Sciences, Kobe University, 5-1-1, Fukaeminami, Higashinada, 658-0022 Kobe, Hyogo, Japan
[b] Softflow Co., Ltd., Tama Center Tosei Bldg. 4F, 1-15-2 Ochiai, Tama-shi, Tokyo, Japan

ARTICLE INFO

Article history:
Received 7 May 2015
Revised 3 November 2015
Accepted 26 November 2015
Available online 12 December 2015

Keywords:
Nozzle cavitation
Fuel injector
Critical pressure
Bubble dynamics
Numerical simulation
OpenFOAM

ABSTRACT

Cavitation phenomenon occurring inside diesel injector nozzles plays a key role in atomization of fuel spray. The most common approach to numerically model the cavitating flow is Volume-Of-Fluid (VOF) method, which employs the governing equations for a perfect gas–liquid mixture often in combination with a transport equation for liquid or gas volume fraction. A mass transfer model is required to evaluate the phase change between liquid and vapor. Most of the mass transfer models use the simplest bubble dynamics model, Rayleigh (R) equation which is sometimes called simplified Rayleigh–Plesset (RP) equation, to simulate the growth and collapse of bubbles based on the vapor saturation pressure P_v. We have found that R equation over-predicts cavitation when local pressure is slightly below P_v. We have proposed the Modified Rayleigh (MR) equation taking into account the critical pressure P_c, and showed its validity in some simple test cases with uniform pressure. In this study, the applicability of the MR equation to turbulent cavitating flows in a fuel injector nozzle is examined. OpenFOAM is used for the numerical simulation of turbulent cavitating flows in an one-side rectangular nozzle whose images have been captured by a high-speed camera and the turbulent velocity has been measured by a Laser Doppler Velocimetry (LDV). Turbulent effect is taken into account using RNG k-ε model. The numerical results are compared with the experimental data and the turbulent recirculation flow, re-entrant jet and cloud cavitation shedding are well simulated by the MR model.

1. Introduction

Turbulent cavitating flow in fuel injector nozzles for diesel engines has been widely investigated, since it plays a key role in atomization characteristics of fuel spray which strongly affects diesel engine performance and emissions. A large variety of experimental works have been performed to clarify the influence of cavitation in injector nozzles. An early experimental work was performed by Bergwerk [1], who showed that cavitation results in large amplitude of disturbance which enhances jet atomization. Nurick [2] carried out lots of cavitation experiments using various types of nozzles with different sizes. Hiroyasu et al. [3] showed that spray atomization is improved by the extension of cavitation to the exit of injector nozzle. Soteriou and Andrews [4] investigated the internal cavitating flow structure in a scaled-up nozzle and classified the cavitation into three distinct regions, i.e., a separated boundary layer inner region, a main stream flow, and an attached boundary layer inner region.

---

[*] Corresponding author at: Softflow Co., Ltd., Tama Center Tosei Bldg. 4F, 1-15-2 Ochiai, Tama-shi, Tokyo, Japan. Tel.: +81 80 3854 0340; fax: 81 78 431 6294.
E-mail addresses: bicerbaris@hotmail.com, 103w204w@stu.kobe-u.ac.jp (B. Biçer), sou@maritime.kobe-u.ac.jp (A. Sou).

[1] Tel.: +81 78 431 6294; fax: +81 78 431 6294.

http://dx.doi.org/10.1016/j.apm.2015.11.049

Nomenclature

C_c	condensation rate constant
C_v	vaporization rate constant
D_0	nuclei diameter (μm)
$\mathbf{U}$	mixture velocity (m/s)
K	cavitation number
k	turbulent kinetic energy (m^2/s^2)
l	characteristic length (mm)
L_n	nozzle length (mm)
n_0	bubble number density ($1/m^3$)
P	pressure (Pa)
t_n	nozzle thickness (mm)
V_n	mean velocity (m/s)
V_{in}	inlet velocity (m/s)
R	radius (μm)
R_c	condensation source term
R_e	evaporation source term
W_n	nozzle width (mm)

Greek symbols

υ	kinematic viscosity (m^2/s)
Δt	time step (s)
Δx	cell size in direction of velocity (mm)
α	volume fraction
ρ	fluid density (kg/m^3)
μ	dynamic viscosity (Pa.s)
σ	surface tension (N/m)
ε	turbulent dissipation rate (m^2/s^3)

Subscripts

0	initial
a	atmospheric
b	bubble
c	critical
eff	effective
in	injector
G	gas
L	liquid
m	mixture
out	outlet
t	turbulence
v	vapor
y	y-direction

Sou et al. [5] observed that cavitation inception takes place as bubble clouds in the recirculation zone near the inlet of a nozzle. The generation of a long cavitation film forms the development of cavitation zone almost to the exit (supercavitation), and shedding of cavitation clouds accompanied in vortices finally induces a large deformation of the liquid jet.

A large number of experimental studies have been conducted using large-scale transparent nozzles, which enable to facilitate clear visualization of cavitation structure and LDV measurement [5–9]. Although important and useful knowledge has been obtained from the experimental attempts, the real effects of the turbulent cavitating flows in injector nozzles remain still unknown due to the refraction of light at the cylindrical side wall of the nozzles, tiny scale of the nozzle of about 0.1 mm in diameter and 1 mm in length, operating at high injection pressure and high velocity up to hundreds meters per second in the nozzles, and complexity of the turbulent cavitating flow. These difficulties also make the experimental visualizations and measurements extremely difficult. Therefore, lots of numerical models have been developed for many years in the literature to simulate cavitation inside injector nozzles [9–26].

In the numerical modeling of the turbulent cavitating flows inside injector nozzles, it is necessary to precisely describe gas or liquid fraction and the resulting change in the mixture fluid properties. Homogeneous Equilibrium Model (HEM) is one of the well-known and widely used multiphase approach due to its simplicity [10,19–24]. This model treats the cavitating flow as a