International Journal of Heat and Fluid Flow 32 (2011) 1068–1075

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

# Numerical simulation of wall roughness effects in cavitating flow

F. Echouchene [a], H. Belmabrouk [a,*], L. Le Penven [b], M. Buffat [b]

[a] Laboratoire d'électronique et de microélectronique, Département de Physique, Faculté des Sciences de Monastir, 5000, Tunisia
[b] LMFA UMR CNRS 5509, Université de Claude Bernard Lyon 1, Ecole Centrale de Lyon, INSA de Lyon, France

ARTICLE INFO

*Article history:*
Received 24 April 2009
Received in revised form 25 February 2011
Accepted 20 May 2011

*Keywords:*
Cavitating flow
Wall roughness
Mixture model
Diesel injector

ABSTRACT

Hydrodynamic cavitation has an important effect on the performance of Diesel injectors. It influences the nature of the fuel spray and the efficiency of the combustion process. In the present study, we investigate numerically the effect of wall roughness in the cavitating and turbulent flow developing inside a Diesel injector. The mixture model based on a single fluid is adopted and the commercial Fluent software is used to solve the transport equations.

The discharge coefficient $C_d$ is computed for different cavitation numbers and wall roughness heights. Profiles of density mixture, vapor volume fraction, mean velocity and turbulent kinetic energy are reported. The effects of wall roughness and injection pressure are analyzed.



## 1. Introduction

The cavitation phenomenon occurs when the local pressure of the fluid is smaller than the vapor pressure at the temperature of the fluid. Vapor bubbles are then formed convected by the flow and may collapse. This may cause mechanical damage on turbomachinery and hydraulic systems. On the contrary, we can take advantage of cavitation in some industrial and medical applications. Indeed, cavitation may enhance spray breakup and it has an important effect on the performance of Diesel injector systems and efficiency of the combustion process (Dumont et al., 2000; Vortmann et al., 2003; Martynov, 2005).

Since the flow inside the injector nozzle has a high speed, the orifices are small, the pressure is very high and the injection duration is very short, it is very difficult to perform experiments in real-size devices. Numerical simulation of cavitating flow in high pressure Diesel injectors is therefore expected to provide valuable information. During the last decades, different experimental (Nurick (1976), Chaves et al. (1995), Schmidt et al. (1997), Payri et al. (2009)) and numerical works have addressed the subject of cavitating flow in Diesel injectors.

Von Dirke et al. (1999) simulated cavitating flows in Diesel injectors using two fluid models. Several geometries (sac-hole, one-hole nozzle and ball valve) have been investigated. The distributions of volume fraction show the existence of cavitation zones. Schmidt et al. (1997), Marcer et al. (2000), Dumont et al. (2001) presented a 2D and 3D numerical simulation of cavitating flow in Diesel injectors. They used an homogeneous equilibrium model based on equation of state (EOS). They studied the collapse of bubbles and the apparition of cavitation in a single and multi-hole-nozzle. Alajbegovic et al. (2002) used a multiphase model in a high-pressure swirl injector. They calculated the pressure and volume fraction fields. Vortmann et al. (2003) developed a new cavitation model based on non-equilibrium effects between the vapor phase and the liquid phase. It has been applied to converging–diverging nozzle.

Arcoumanis and Gavaises (1998), Giannadakis et al. (2004), Gavaises and Giannadakis (2004) developed a cavitation model based on the Eulerian–Lagrangian approach. It incorporates many fundamental physical processes inherent to cavitating flows such as bubble formation, growth and collapse, momentum exchange between vapor and liquid phases. A planar real-size single hole-nozzle was used and the effects of some operating conditions (pressure and geometrical parameters) have been reported. The vapor fraction distribution has been shown. The vapor fraction and axial velocity are calculated and compared with experimental data in Diesel injector nozzles.

Randy et al. (2005) developed a new multiphase mathematical model based on a mixture formulation for a multiphase flow. This model does not incorporate turbulence effects. The numerical simulation at high pressure, supersonic and three-phase cavitating flow within a nozzle injector is presented. The vapor volume fraction and velocity profile are calculated and compared with experimental results.

Dular et al. (2005) presented an experimental and numerical study of cavitating flow around hydrofoil. The commercial CFD

* Corresponding author. Fax: +216 73 500 280.
*E-mail addresses:* frchouchene@yahoo.fr, Hafedh.Belmabrouk@fsm.rnu.tn (H. Belmabrouk).

0142-727X/$ - see front matter © 2011 Elsevier Inc. All rights reserved.
doi:10.1016/j.ijheatfluidflow.2011.05.010

program-Fluent was used for the simulation of cavitation. The cavitation model is based on the mixture formulation and a $k$–$\varepsilon$ turbulence model was applied. Lei et al. (2006) used a two-phase mixture model to study unsteady cavitating flow. The flow is considered as compressible and a state equation between density and pressure is used. Martynov et al. (2006) developed a model of cavitation taking into account the bubbly nature of cavitation and assuming local homogeneity of the vapor–liquid flow. This model is built from correlations for evaporation and condensation and it is based on bubble dynamics theory. An equation describing the distribution of the number of bubbles is derived. The vapor volume fraction and the effect of liquid tension on this number along the nozzle are calculated.

Giannadakis et al. (2007), Andriotis et al. (2008) used two Eulerian models and a Lagrangian approach of cavitation to study a real-size six-hole mini-sac nozzle. Results have indicated that the two Eulerian models predict a large void area inside the injection-hole while the Lagrangian model predicts a more diffused and gradual vapor distribution. The collapse of the cavitation zone is not captured properly from the Eulerian models. This trend was better captured by the Lagrangian model. Befrui et al. (2007) have carried out a combined experimental and numerical study of the break-up structure of conical sprays of the high-pressure gasoline direct injection. The Volume-of-Fluid Large-Eddy-Simulation (VOF-LES) computational method for two-phase flow simulation is employed. The VOF-LES results were compared to experimental data.

Habchi et al. (2008) applied an homogeneous equilibrium modeling approach to simulate the cavitating flow inside a single-hole injector and six-hole injector. The needle displacement has been taken into account. The steady and unsteady flow features are discussed. Many valuable results concerning the extension of the cavitation region, fluid velocity and void fraction have been obtained.

Lee and Reitz (2009) used the KIVA code to simulate the transient cavitation in multi-hole injectors. The effect of needle motion on the flow structure has been highlighted.

It appears from this brief review that, in the last years, several models have been developed to simulate the cavitating flow inside Diesel injectors. They are based on a single-fluid or multi-fluid frame-work, Eulerian–Eulerian or Eulerian–Lagrangian approaches. The previous studies consider a smooth wall. However, wall roughness leads to higher shear stresses in the liquid near the wall and produces additional disturbance of the velocity and pressure. For this reason, we aim in the present paper to investigate numerically the effects of wall roughness in a cavitating flow inside a Diesel injector. The commercial CFD code Fluent 6.3 is used to simulate the cavitating flow.

## 2. Numerical simulation

### 2.1. Multiphase model

In this study, a ***single fluid*** approach is used i.e. the mixture can be considered as a *"single phase"* with its physical properties varying according to the local concentration of liquid and vapor. The flow is assumed to be steady, isothermal and incompressible and the fluid is Newtonian. In the configuration considered hereafter, the flow is assumed to be axisymmetric and the azimuthal component of the mean velocity is zero. The mass conservation equation of the mixture flow in $r$–$z$ cylindrical coordinates is:

$$\frac{1}{r}\frac{\partial r v}{\partial r}\frac{\partial u}{\partial z}=0 \tag{1}$$

where $v$ is the radial component and $u$ is the axial component of the Reynolds averaged velocity.

Conservation equations for the mean momentum read:

$$\rho\left(\frac{\partial}{\partial z}(uu)+\frac{1}{r}\frac{\partial}{\partial r}(ruv)\right)=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial z}\left[2(\mu+\mu_t)\frac{\partial u}{\partial z}\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r(\mu+\mu_t)\left(\frac{\partial u}{\partial r}+\frac{\partial v}{\partial z}\right)\right] \tag{2}$$

$$\rho\left(\frac{\partial}{\partial z}(vu)+\frac{1}{r}\frac{\partial}{\partial r}(rvv)\right)=-\frac{\partial p}{\partial r}+\frac{\partial}{\partial z}\left[r(\mu+\mu_t)\left(\frac{\partial v}{\partial z}+\frac{\partial u}{\partial r}\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[2r(\mu+\mu_t)\frac{\partial v}{\partial r}\right]-2(\mu+\mu_t)\frac{v}{r^2}+\rho\frac{u^2}{r} \tag{3}$$

where $p$ is the static pressure, $\mu$ is the mixture viscosity and $\mu_t$ is turbulent viscosity. The gravity force is assumed to be negligible. The relation between the mixture density $\rho$ and the volume fraction of the vapor phase $\alpha$ is:

$$\rho=\alpha\rho_v+(1-\alpha)\rho_1 \tag{4}$$

where $\rho_v = 0.026$ kg/m$^3$ and $\rho_l = 1000$ kg/m$^3$ are vapor density and liquid density respectively. The value of $\rho_v$ is obtained from thermodynamic tables for a saturation temperature $T = 296$ K.

Similarly, the mixture viscosity is expressed as:

$$\mu=\alpha\mu_v+(1-\alpha)\mu_1 \tag{5}$$

where $\mu_v = 1.2610^{-6}$ Pa s and $\mu_l = 10^{-3}$ Pa s are respectively vapor and liquid viscosity coefficients.

The relationship between the vapor mass fraction and the vapor volume fraction is:

$$\alpha=f\frac{\rho}{\rho v} \tag{6}$$

The previous set of equations should be completed by the boundary conditions, and the equations used to calculate the turbulent viscosity $\mu_t$ of the mixture and the vapor mass fraction $f$. These equations are given in the following sections.

### 2.2. Turbulence model

The flow is assumed to be turbulent, and the standard $k$–$\varepsilon$ model is employed. The turbulent viscosity $\mu_t$ of the mixture is related to the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$ according to:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{7}$$

It is worth noting that Dular et al. (2005) used a modified expression of $\mu_t$ in order to reduce the turbulent viscosity in the region containing mainly the vapor phase.

Transport equations of the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$ are expressed as:

$$\frac{\partial(\rho k u)}{\partial z}+\frac{1}{r}\frac{\partial(r\rho k v)}{\partial r}=\frac{\partial}{\partial z}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial z}\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial z}\right]+P-\rho\varepsilon \tag{8}$$

$$\frac{\partial(\rho\varepsilon u)}{\partial z}+\frac{1}{r}\frac{\partial(r\rho\varepsilon v)}{\partial r}=\frac{\partial}{\partial z}\left[\left(\mu+\frac{\partial_t}{\sigma_k}\right)\frac{\partial\varepsilon}{\partial z}\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial\varepsilon}{\partial r}\right]+C_{1\varepsilon}\frac{\epsilon}{k}P-C_{2\varepsilon}\frac{\epsilon^2}{k} \tag{9}$$

Here $P$ is the production term of turbulent kinetic energy. It is given by:

$$P=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_i} \tag{10}$$