Proceedings of IMECE'05
2005 ASME International Mechanical Engineering Congress Expo
Orlando, Florida, USA, November 5-11, 2005

IMECE2005-82948

NUMERICAL SIMULATION OF A SUPERSONIC THREE-PHASE CAVITATING JET FLOW THROUGH A GASEOUS MEDIUM IN INJECTION NOZZLE

Randy S. Lagumbay
Department of Mechanical Engineering
University of Colorado at Boulder, 427 UCB
Boulder, CO 80309, USA
Email: Randy.Lagumbay@colorado.edu

Oleg V. Vasilyev[*]
Department of Mechanical Engineering
University of Colorado at Boulder, 427 UCB
Boulder, CO 80309, USA
Email: Oleg.Vasilyev@colorado.edu

Andreas Haselbacher
Center for Simulation of Advanced Rockets
University of Illinois at Urbana-Champaign
2253 Digital Computer Laboratory, MC 278
1304 West Springfield Ave.
Urbana, IL 61801, USA
Email: haselbac@uiuc.edu

Jin Wang
Argonne National Laboratory
IL 60439, USA
Email address: wangj@aps.anl.gov

ABSTRACT

A new multiphase mathematical model based on a mixture formulation of the laws of conservation for a multiphase flow is used to simulate a supersonic three-phase cavitating jet flow through a gaseous medium. The model does not require an ad-hoc closure for the variation of mixture density with regards to the attendant pressure and yields a thermodynamically accurate value for the acoustical propagation generated by the process. A source term for cavitation is added into the equations of the mixture formulation and the resultant cavitation is mathematically modeled accordingly. The new numerical formulation has been incorporated into a multi-physics unstructured code "RocfluMP" that solves the modified three-dimensional time-dependent Euler/Navier-Stokes equations for a multiphase framework in integral form. A modified form of the Harten, Lax and van Leer approximate Riemann equations are used to resolve the isolated shock and contact waves. The newly developed multiphase flow equations provide a general framework for analyzing coupled incompressible-compressible multiphase flows that can be applied to a variety of supersonic multiphase jet flow problems such as fuel injection systems and liquid-jet machining. Preliminary results for three-phase cavitating jet flow through a gaseous medium in injection nozzle are presented and discussed.

INTRODUCTION

The injector nozzle plays a very significant role in various engineering applications from the fuel injection systems used in combustion engineering to the liquid-jet machining of materials in processing engineering. The internal flow through an injection nozzle is a very important aspect of spray formation and atomization generated by the nozzle, and is directly correlated to the nozzle's combustion efficiency and the resultant formation of pollutants. A thorough understanding of the complex nature of flow intra nozzle and flow contiguous to the nozzle is necessary for predicting spray development. The internal flow process is very complex. The injection pressure is high and the internal flow is accelerated toward the small nozzle orifice, which can cause a pressure drop within a particular internal region. When the pressure drops within this region below the saturation pressure, a cavitation will occur. Cavitation and gas entrapment inside the injector nozzle orifice can greatly affect the local and global be-

---

[*] Address all correspondence to this author.

havior of flow. Experimental evidence demonstrates that cavitation within the nozzle influences the characteristics of the nozzle exit spray [1, 2].

Experimental investigation in this area has been rare and detailed quantitative data is very limited due to the difficulties involved resulting from the small scale size of the injector nozzle that ranges from 100 μm - 1 mm, high injection pressure that may reach upwards to 2000 bar in a very short time period, the high speed flow, which may reach supersonic velocity [3], and the cost and difficulty of manufacturing experimental devices. The aforementioned time frame, on the order of 1 - 10 μs [3], presents the greatest challenge to modeling the internal flow of a nozzle and to capturing the process of cavitation formation for investigation. Hence, most experimental studies have been performed on large scale transparent models in order to visualize the cavitation structure within the nozzles [1, 2, 4–13]. However, there are still several authors [14–21] who have performed experiments on actual scale and have observed the instability of cavitations that is generated by the alternating growth and collapse of the cavity. Recent experimental studies involve the investigation of the spray cone angles for a cavitating nozzle flow [17, 22], investigation of cavitation oscillation frequencies [20] and the determination of the length of the cavity [21]. Moreover, Payri *et al.* [23] examined the influence of cavitation on the internal flow and the macroscopic behavior of spray in diesel injection nozzles, and concluded that cavitation leads to an incremental changes of the spray cone angle and increases the outlet velocity.

Numerical and theoretical investigations have complimented each other and provide detailed quantitative explanations of the complex structure of the jet from single phase to more complex multiphase flows. Numerical modeling and simulation has become the best of the alternative tools to supplement experimental study, and provide very promising results in improving our understanding of the complex nature of flow in the injection nozzle. The majority of current research projects have focused on modeling and simulation. Bunnell *et al.* [24] have performed a three-dimensional simulation of unstable cavitating flows in injector passages and have demonstrated the effect of cavitation on both the mean and unsteady components of the orifice discharge coefficient. Similarly, Dirke *et al.* [25] have carried out three-dimensional simulations of cavitating flows in diesel injectors and have shown the distribution of cavitation zones. Yuan *et al.* demonstrated the strong interaction of a cavitating nozzle flow with the external jet formation [3] and the effect of injection pressure fluctuations on the cavitation processes in injection nozzles [26]. A numerical study of cavitating flow through various nozzle geometries was performed by Schmidt *et al.* [27] and the results show that the upstream geometry has a small influence on the nozzle flow.

The flow inside the nozzle is composed of a mixture of liquid, vapor, and gas, and exits the nozzle through a gaseous medium. The numerical modeling and simulation of a multiphase flow is a very daunting task as compared to a single phase flow. In a multiphase flow the phases will assume a large number of complicated configurations, in which small-scale interactions between the phases can have profound effects on the macroscopic properties of the flow [28]. The most common problem encountered in multiphase flow modeling is the fluid interface where a large density variation exists. This causes flow phenomena to become more complex because of additional non-linearity introduced by the indeterminacy of such surfaces. Also, the fluid interface may be unstable, changing the flow configuration of the problem [29]. There are two possible methods to model multiphase flow problems. The first is to solve the mass, momentum, and energy equations for each phase, supplemented with the conditions that account for the constitutive relationships of the interfacial balance between phases and the interfacial boundary condition [30]. The second method treats the multiphase flow regime as a mixture and considers the interface as part of the flow solution, supplemented with additional equations for the void or mass fraction of the mixture constituents. The advantage for this approach is that it only solves one set of mass, momentum and energy equations for the mixture composition [31].

Several numerical simulations of multiphase flow have been conducted over the past few years with different modeling approach and numerical techniques. It has been shown that each model and numerical method has limits to their respective capabilities. To this date, there has been a great need of improving the technique of modeling multiphase flow and of a numerical method to perfectly capture the complex nature of the flow. Dumont *et al.* [32] and Shin [33] have used the homogeneous equilibrium model to solve two-phase (liquid and vapor) cavitating flows. Their models differ on the equation of state of the mixture and the numerical scheme. Taking a different approach, Alajbegovic *et al.* [34] and Tatschl *et al.* [35] have used the two-fluid formulation for multiphase flows [36] to simulate three phase cavitating flows. The conservative form of the Favre-averaged Navier-Stokes equations have been utilized by Senocak *et al.* [37] to study cavitating flows through convergent-divergent nozzles. A direct calculation of cavitating flows by the Space-Time Conservation Element and Solution Element (CE/SE) method is reported by Qin *et al.* [38]. The CE/SE method is applicable for flows at wide range of Mach numbers and suitable for time accurate simulations. Vortman *et al.* [39] have proposed a new approach based on postulating Gibbs free energy for the phase mixture. The two-phase flow is treated numerically by combining the rate equation with a volume of fluid approach.

Moreover, Singhal *et al.* [40] have developed a full cavitation model that accounts the formation and transport of vapor bubble, turbulent fluctuations of pressure and velocity, and the magnitude of noncondensible gases that are dissolved in the operating liquid. A reduced form of Rayleigh-Plesset equation for bubble dynamics has been used to derived the phase change rate equations. The model has assumed an isothermal flow process, and decoupling the cavitation module from heat transfer and radiation modules. The model has been implemented in an advanced, commercial, general-purpose CFD code called CFD-ACE+ by CFD Research Corporation (CFDRC). Taking a similar approach, Delale *et al.* [41] have used a continuum bubbly liquid flow model with bubble nucleation, and nonlinear bubble dynamics described by the classical Rayleigh-Plesset equation in a quasi one dimensional steady state cavitating nozzle flows. On the other hand, Ahuja *et al.* [42] have formulated a multiphase model for low speed gas/liquid mixtures by reducing the compressible system of equations to an acoustically accurate form for multi-fluid mixtures. In addition, Hosangadi *et al.* [43] have developed a generalized numerical framework for transient and multiphase problems that involve a combination of gas, bulk liquid, and a dense dispersed phase. The model has been improved by allowing flexibility of specifying variables for thermodynamic properties and specifying physical equations of state for mixture constituents [44].

Our previous work [45] has focused on a numerical simulation of the structure and dynamics of a high pressure, high speed jet of gas/liquid mixtures through a gaseous medium close to the nozzle region. A new multiphase model based on a mixture formulation of the conservation laws for a multiphase flow has been developed. The new numerical formulation has been incorporated into a multi-physics unstructured code called “RocfluMP” [46] and [47]. RocfluMP solves the three-dimensional time-dependent Euler and Navier-Stokes equations in integral form. A finite volume and unstructured mesh method has been used for the spatial discretization, and a fourth-order accurate Runge-Kutta method for the time integration. The modified Harten, Lax and van Leer approximate Riemann equations (HLLC) have been used to capture and resolve the isolated shock and contact waves [48].

In this paper, we have extended and improved our new multiphase model from two phase (liquid and gas) into three phase (liquid, vapor, and gas), with the addition of cavitation model. A source term for cavitation is added into the equations of the new mixture formulation and the resultant cavitation is mathematically modeled accordingly. The main objectives of this paper are to capture the complex structure of the flow in the injector nozzle and to investigate the effects of cavitation inside the nozzle injector to the external jet formation. Previous experiment by MacPhee *et al.* [49] have demonstrated a generation of shock wave in the gaseous medium, and cavitation inside the nozzle have been observed by [14–21]. In particular, the specific objective is to develop and provide a general framework for analyzing coupled incompressible-compressible multiphase flows that can be applied to a variety of supersonic multiphase jet flow problems such as fuel injection systems [50], thermal and plasma spray coating [51] and liquid-jet machining [52].

MATHEMATICAL FORMULATION

A high-pressure, high-speed three-phase (liquid, vapor, and gas) cavitating nozzle jet flow through a gaseous medium is modeled using the multiphase mixture formulation, supplemented with equations for the volume or mass fraction of the mixture constituents. Cavitation and gas entrapment that occur inside the nozzle affect the behavior of the flow. The structure of the jet is composed of a bubbly fluid traveling at a speed higher than the speed of sound of the gas through a gaseous medium. An oblique shock wave is generated in a surrounding gas and a compression wave is moving through the jet with a relative velocity opposite to the direction of the jet flow. High pressure occurs at the front region of the jet due to the impact of the momentum of the heavy fluid to the stationary gas. The external jet formation is greatly influenced by the presence of cavitation inside the nozzle. Background gas near the external jet is drawn along with the flow of the jet.

The mathematical model used for the simulation is based on a mixture formulation of the conservation laws for a multiphase flow. The multiphase flow regime including the background gaseous medium is treated as a mixture and considering the interface as part of the solution domain. The advantage for this model is that it only solves one set of mass, momentum and energy equations for the mixture composition [31]. However, there are challenges associated with the use of this method such as the mathematical closure of the system in a consistent manner with the thermodynamics of the system and the resulting acoustic speed of the multiphase system. The governing equations for the mixture system that are solved include the continuity, momentum for the three components, the energy equation, and the evolution equation for volume fraction of the mixture constituents. An equilibrium formulation is assumed, so that the temperatures, pressures and velocities for the mixture (liquid, vapor, and gas) are the same. The governing Euler equations for a multiphase mixture in conservative vector form can be written:

$$\frac{\partial \mathbf{Q}}{\partial t}+\frac{\partial \mathbf{E}}{\partial x}+\frac{\partial \mathbf{F}}{\partial y}+\frac{\partial \mathbf{G}}{\partial z}=\mathbf{S}, \tag{1}$$

where $\mathbf{Q}$ is the vector of the conserved variables, and $\mathbf{E}$, $\mathbf{F}$, $\mathbf{G}$ are the flux vectors given by

$$\mathbf{Q}=\begin{bmatrix}\rho_m\\ \rho_m u\\ \rho_m v\\ \rho_m w\\ \rho_m e_{m_T}\\ \rho_g\phi_g\\ \rho_v\phi_v\end{bmatrix}, \quad \mathbf{E}=\begin{bmatrix}\rho_m u\\ \rho_m u^2+P\\ \rho_m uv\\ \rho_m uw\\ (\rho_m e_{m_T}+P)u\\ \rho_g\phi_g u\\ \rho_v\phi_v u\end{bmatrix} \tag{2}$$

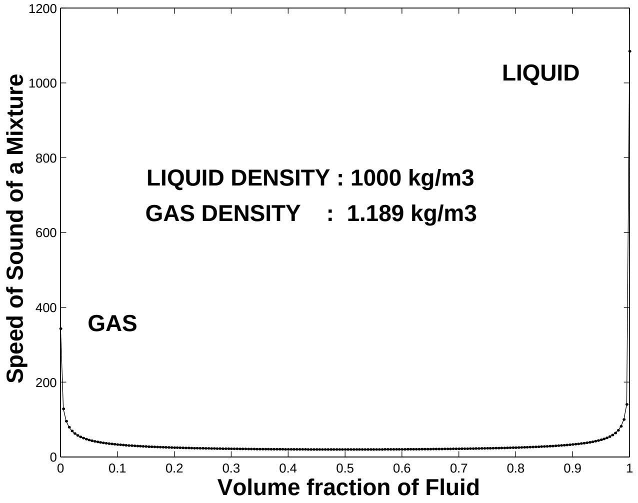

Figure 1. PLOT OF THE SPEED OF SOUND OF THE TWO-PHASE MIXTURE (LIQUID AND GAS).

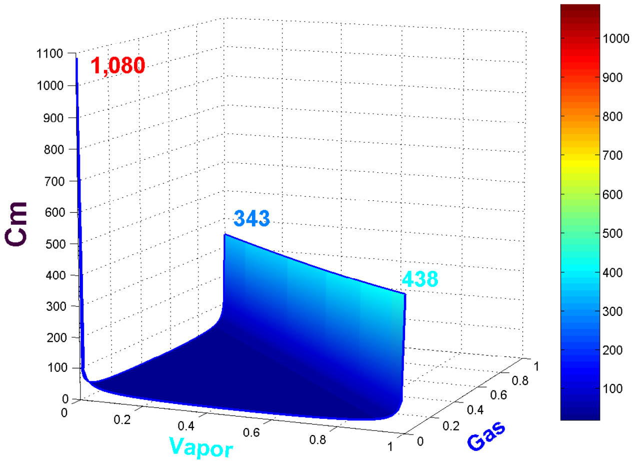

Figure 2. PLOT OF THE SPEED OF SOUND OF THE THREE-PHASE MIXTURE (LIQUID, VAPOR AND GAS).

$$\mathbf{F}=\begin{bmatrix}\rho_m v\\ \rho_m vu\\ \rho_m v^2+P\\ \rho_m vw\\ (\rho_m e_{m_T}+P)v\\ \rho_g\phi_g v\\ \rho_v\phi_v v\end{bmatrix},\quad \mathbf{G}=\begin{bmatrix}\rho_m w\\ \rho_m wu\\ \rho_m wv\\ \rho_m w^2+P\\ (\rho_m e_{m_T}+P)w\\ \rho_g\phi_g w\\ \rho_v\phi_v w\end{bmatrix}. \tag{3}$$

On the right hand side of the equation (1) is the source terms and is given by $[0\ 0\ 0\ 0\ 0\ 0\ S_v]^T$. Presently, the cavitation source term, S_v is based on the model by Hosangadi *et al.* [43].

In the above formulation, ρ_m is the mixture effective density, ϕ_g is the gas volume fraction, and ϕ_v is the vapor volume fraction. P, u, v, w are respectively the equilibrium pressure, x-, y-, and z-velocity components of the mixture. The mixture effective density, volume fractions for liquid, gas, and vapor, total energy of the mixture, and specific heat constant of the mixture are coupled as follows:

$$\rho_m=\rho_l\phi_l+\rho_g\phi_g+\rho_v\phi_v, \tag{4}$$

$$\phi_l+\phi_g+\phi_v=1, \tag{5}$$

$$\rho_m e_{m_T}=\rho_m c_{\mathrm{vm}}T+\frac{1}{2}\rho_m V^2, \tag{6}$$

$$c_{vm}=\frac{\rho_l\phi_l c_{\mathrm{vl}}+\rho_g\phi_g c_{\mathrm{vg}}+\rho_v\phi_v c_{\mathrm{vv}}}{\rho_m}, \tag{7}$$

where ρ_l, ρ_g, and ρ_v are respectively the densities of the liquid, gas, and vapor phases. Similarly, c_{vl}, c_{vg} and c_{vv} are respectively the specific heat constants of the liquid, gas and vapor phase, while T and V are respectively the equilibrium temperature of the mixture and velocity magnitude.

The constitutive equation for the liquid is mathematically modeled as a linear function of pressure and temperature. For the gas and vapor, the constitutive equations are currently based on the ideal gas law. These are given by

$$\rho_l=\rho_o+\beta_P(P-P_o)+\beta_T(T-T_o), \tag{8}$$

$$\rho_g=\frac{P}{R_gT}. \tag{9}$$

$$\rho_v=\frac{P}{R_vT}. \tag{10}$$

The corresponding acoustic form of density differential for the liquid, gas, and vapor phase is shown below:

$$d\rho_l=\frac{1}{C_l^2}dP-\frac{\beta_l^2}{C_l^2}dT, \tag{11}$$

$$d\rho_g=\frac{1}{C_g^2}dP-\frac{\beta_g^2}{C_g^2}dT, \tag{12}$$

$$d\rho_v=\frac{1}{C_v^2}dP-\frac{\beta_v^2}{C_v^2}dT, \tag{13}$$

where C_l, C_g, and C_v are respectively the isothermal speeds of sound in the liquid, gas, and vapor, while, β_l, β_g, and β_v are respectively the compressibility constants in the liquid, gas, and vapor. Combining (4), (11), (12), and (13) the differential form of the mixture density can be written as follows:

$$d\rho_m=(\rho_v-\rho_l)d\phi_v+(\rho_g-\rho_l)d\phi_g+\frac{1}{C_\phi^2}dP-\frac{\beta_\phi^2}{C_\phi^2}dP, \tag{14}$$

where

$$\frac{1}{C_\phi^2} = \frac{\phi_l}{C_l^2} + \frac{\phi_g}{C_g^2} + \frac{\phi_v}{C_v^2}. \tag{15}$$

and

$$\frac{\beta_\phi^2}{C_\phi^2} = \frac{\phi_l}{C_l^2}\beta_l^2 + \frac{\phi_g}{C_g^2}\beta_g^2 + \frac{\phi_v}{C_v^2}\beta_v^2. \tag{16}$$

The speed of sound of the mixture can be obtained easily by transforming the conservative variables form into a primitive variables form. Using equation (14), the Euler equations can be rewritten in the following conservative form

$$\Gamma\frac{\partial \mathbf{Q}_v}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} = \mathbf{S}, \tag{17}$$

Here, $\mathbf{Q}_v$ is the vector of the primitive variables

$$\mathbf{Q}_v = [P,\ u,\ v,\ w,\ T,\ \phi_g\ \phi_v]^T \tag{18}$$

and Γ is the transformation matrix from which the eigenvalues and eigenvectors of the system can be obtained. The resulting eigenvalues, λ_i of the system are given as

$$\boldsymbol{\lambda} = (u,\ u,\ u,\ u,\ u,\ u - C_m,\ u + C_m), \tag{19}$$

where C_m is the speed of sound of the multiphase mixture of liquid, gas and vapor and is given by

$$\frac{1}{C_m^2} = \frac{\rho_m^2 c_{\mathrm{vm}}}{\rho_m c_{\mathrm{vm}} + \frac{\phi_l \beta_l^2 P}{\rho_l C_l^2} + \frac{\phi_v \beta_v^2 P}{\rho_v C_v^2} + \frac{\phi_g \beta_g^2 P}{\rho_g C_g^2}} \left[\frac{\phi_l}{\rho_l C_l^2} + \frac{\phi_v}{\rho_v C_v^2} + \frac{\phi_g}{\rho_g C_g^2}\right]. \tag{20}$$

The resulting speed of sound of the mixture is in good agreement with the result presented in [44]. For the case of a two purely incompressible fluids, the energy equation decouples and the acoustic speed reduces to a harmonic expression for the speed of sound in a two-phase isothermal mixture [53]. Figures 1 and 2 show the plot of the speed of sound for the two-phase (liquid and gas) and the three-phase (liquid, vapor and gas) mixture. As can be seen, the acoustic speed of the mixture drops off rapidly as the variation of density increases.

CAVITATION MODEL

The cavitation source term provided on the right hand side of the equation (1) is similar to [53] and is given by

$$S_v = \kappa_l \rho_l \phi_l + \kappa_v \rho_v \phi_v \tag{21}$$

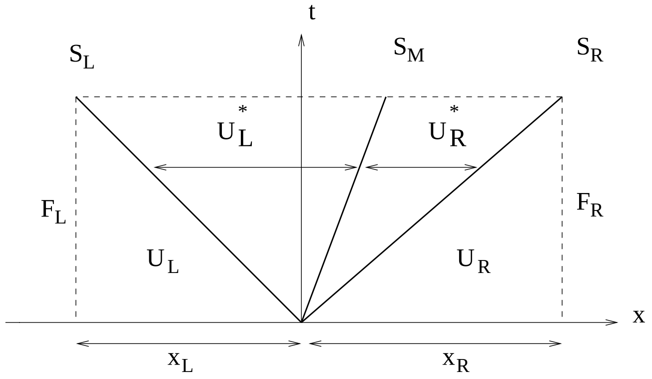

Figure 3. THE SIMPLIFIED RIEMANN WAVE DIAGRAM WITH TWO INTERMEDIATE STATES USED IN THE HLLC SCHEME.

where, κ_l is the constant rate of liquid vaporization in the region where the local pressure drops below the vapor saturation pressure at the corresponding temperature, while, κ_v is the constant rate of vapor condensation in the region where the local pressure exceeds the vapor saturation pressure at the corresponding temperature. The constants κ_l and κ_v are specified in a similar form given by [31] except that the vapor saturation pressure is determined from the empirical equation. The constants κ_l and κ_v are given by

$$\kappa_l = 0.5(\mathrm{SIGN}(1.0, P - Ps) - 1.0)\,[Cav.No.]\left[\frac{1}{\tau_f}\right]\left[\frac{Q_\infty}{L_\infty}\right], \tag{22}$$

$$\kappa_v = 0.5(\mathrm{SIGN}(-1.0, P - Ps) - 1.0)\,[Cav.No.]\left[\frac{1}{\tau_v}\right]\left[\frac{Q_\infty}{L_\infty}\right] \tag{23}$$

where $Cav.No.$ is the cavitation number and the vapor saturation pressure Ps is calculated from the empirical equation derived by [54]

$$Ps = 10.^{10.79574*(1.-\frac{Ts}{T})} - 5.02800 * \mathrm{LOG}_{10}(\tfrac{T}{Ts}) + 1.50475\mathrm{e}^{-04} * (1. - 10.^{(} - 8.2969 * (\tfrac{T}{Ts} - 1.))) + 0.42873\mathrm{e}^{-03} * (10.^{(4.76955*(1.-\frac{Ts}{T}))-1.}) + 0.78614)$$

where, $Ts = 273.16\mathrm{K}$ is the steam point temperature, and T is the local temperature in a region.

It is noted that equations (22) and (23) are reasonably sufficient for steady attached cavitation since the cavitation time scales does not interact with the fluid time scales particularly for faster rate of cavitation process. The time constant for vapor formation,τ_f and the time constant for vapor condensation,τ_v are determined from [53].

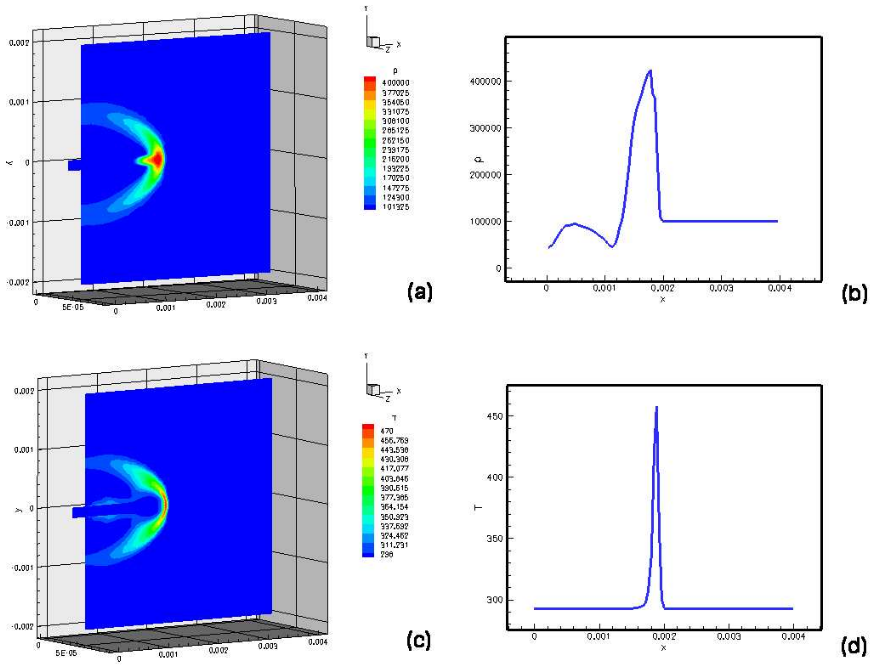

Figure 4. PLOT OF (a) MIXTURE PRESSURE SHOWING THE OBLIQUE SHOCK IN A GASEOUS MEDIUM, (b) MIXTURE PRESSURE ALONG THE CENTERLINE OF THE JET AXIS, (c) MIXTURE TEMPERATURE SHOWING THE OBLIQUE SHOCK IN A GASEOUS MEDIUM, AND (d) MIXTURE TEMPERATURE ALONG THE CENTERLINE OF THE JET AXIS. HEAVY FLUID DENSITY, ρ_l = 1000.0 kg/m^3; GAS DENSITY, ρ_g = 1.189 kg/m^3; INFLOW CONDITION, INFLOW JET VELOCITY IS TWICE THE SPEED OF SOUND IN THE SURROUNDING GAS (MACH = 2.0 WITH RESPECT TO THE GAS).

NUMERICAL METHOD

The numerical scheme employed in the formulation in both spatial discretization and time integration is adopted from [46] and [47], except that the system of equations solved are mixture of liquid, vapor and gas. The upwind finite differencing scheme is used to approximate the fluxes to allow for capturing of shock waves and contact discontinuities. A classical 4-stage Runge-Kutta method in low-storage formulation is used for the temporal discretization [55]. The modified Harten, Lax and van Leer approximate Riemann equations (HLLC) are used to capture and resolve the isolated shock and contact waves [48].

The HLLC Scheme

The HLLC scheme yields the exact resolution of isolated shock and contact waves by choosing the right acoustic and contact wave speeds [48]. The simplified Riemann wave diagram with two intermediate states is illustrated in Fig.3. Harten, Lax, and van Leer [56] defined the two-state approximate Riemann solution as

$$\mathbf{U}_{HLLC} = \begin{cases} \mathbf{U}_L \text{ if } S_L > 0, \\ \mathbf{U}_L^* \text{ if } S_L \leq 0 < S_M, \\ \mathbf{U}_R^* \text{ if } S_M \leq 0 \leq S_R, \\ \mathbf{U}_R \text{ if } S_R < 0, \end{cases} \tag{24}$$

where $\mathbf{U}$ is the vector of the conserved variables defined similar to $\mathbf{Q}$ in Ea. (2). The corresponding interface flux, $\mathbf{F}_{HLLC}$, is defined as

$$\mathbf{F}_{HLLC} = \begin{cases} \mathbf{F}_L \text{ if } S_L > 0, \\ \mathbf{F}_L^* \text{ if } S_L \leq 0 < S_M, \\ \mathbf{F}_R^* \text{ if } S_M \leq 0 \leq S_R, \\ \mathbf{F}_R \text{ if } S_R < 0. \end{cases} \tag{25}$$

Applying the Rankine-Hugoniot conditions across the S_L waves and simplifying yields

$$\mathbf{F}_L^* = \mathbf{F}_L + S_L(\mathbf{U}_L^* - \mathbf{U}_L), \tag{26}$$

$$S_L\mathbf{U}_L^* - \mathbf{F}_L^* = S_L\mathbf{U}_L - \mathbf{F}_L. \tag{27}$$

Similarly, for the S_R waves yields

$$\mathbf{F}_R^* = \mathbf{F}_R + S_R(\mathbf{U}_R^* - \mathbf{U}_R), \tag{28}$$

$$S_R\mathbf{U}_R^* - \mathbf{F}_R^* = S_R\mathbf{U}_R - \mathbf{F}_R. \tag{29}$$

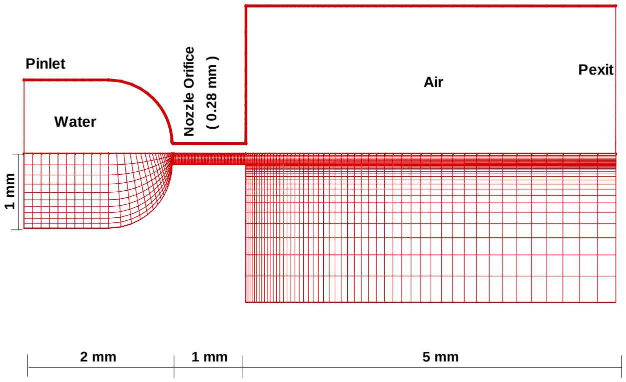

Figure 5. PLOT OF THE COMPUTATIONAL MESH AND BOUNDARY CONDITIONS OF THE TWO-DIMENSIONAL PLANE MODEL NOZZLE INJECTOR.

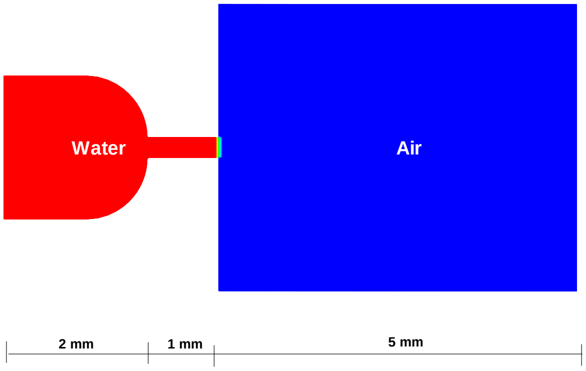

Figure 6. PLOT OF THE INITIAL CONDITION OF THE SYSTEM. THE NOZZLE REGION IS FILLED WITH LIQUID AND THE CHAMBER DOWNSTREAM FROM THE EXIT OF THE NOZZLE IS FILLED WITH GAS.

To determine $\mathbf{U}^*_L$ and $\mathbf{F}^*_L$, assumption is made such that

$$S_M = q^*_L = q^*_R = q^*, \tag{30}$$

where q^* is the average directed velocity between the two acoustic waves, and

$$S_M = \frac{\rho_R q_R (S_R - q_R) - \rho_L q_L (S_L - q_L) + P_L - P_R}{\rho_R (S_R - q_R) - \rho_L (S_L - q_L)}. \tag{31}$$

Also,

$$S_L = \min[q_L - C_{mL}, \overline{q} - \overline{C}_m], \tag{32}$$

$$S_R = \max[q_R + C_{mR}, \overline{q} + \overline{C}_m], \tag{33}$$

where $\overline{q}$ is the sum of the average of the x-, y-, z-velocity components and $\overline{C}_m$ is the mixture average speed of sound.

RESULTS AND DISCUSSIONS

In our previous work [45], we have demonstrated the capabilities of the new multiphase model equations, and the new numerical scheme to capture the complex structure of the jet flow close to the nozzle region. The results in our simulation have shown an oblique shock wave in the gaseous medium, which confirmed the previous experiment conducted by MacPhee *et al.* [49]. We have extended and improved our new multiphase model from two phase (liquid and gas) into three phase (liquid, vapor, and gas), with the addition of cavitation model. A source term for cavitation is added into the equations of the new mixture formulation and the resultant cavitation is mathematically modeled accordingly. The results of the simulation for a cavitating nozzle jet flow, and the effect of fluctuating inflow conditions to the external jet formation are presented, and compared with other numerical simulation and experimental observation.

In the simulation, we have used the two-dimensional plane experimental test case of Roosen *et al.* [57], which has been used also by Yuan *et al.* [3] to validate the numerical simulation. The dimension of the nozzle hole is $1mm$ x $0.28mm$ x $0.2mm$ (length x height x width). Figure 5 illustrates the computational domain of the nozzle injector and the treatment of the boundary conditions. To reduce the computational time, a symmetric boundary condition is imposed along the nozzle axial axis, and solves only the lower half of the computational domain. A total of 7140 computational cells are used in the numerical computation. We have initialized the system with liquid in the nozzle region and gas in the exit region. Figure 6 shows the initial condition of the system. Note the sharp interface between the liquid and gas just after the nozzle exit.

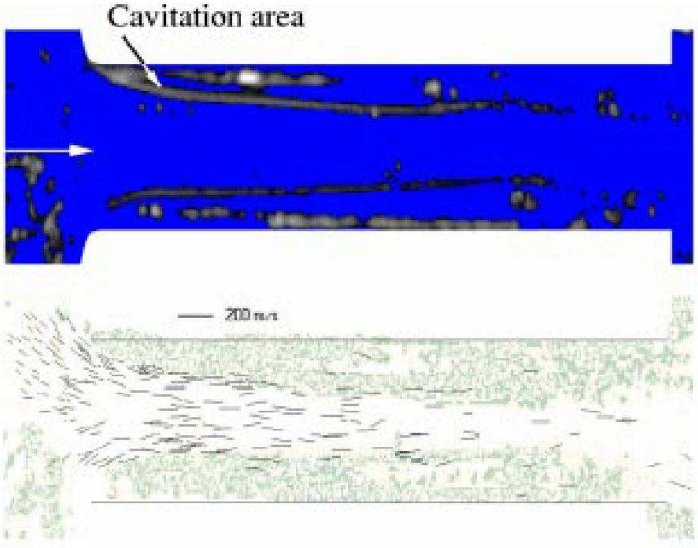

Figure 7. CAVITATION INSIDE THE NOZZLE INJECTOR. EXPERIMENTAL RESULTS TAKEN FROM ROOSEN *et al.* [57]. INJECTION PRESSURE, $P_{inlet} = 80$ bar; EXIT PRESSURE, $P_{exit} = 11$ bar.

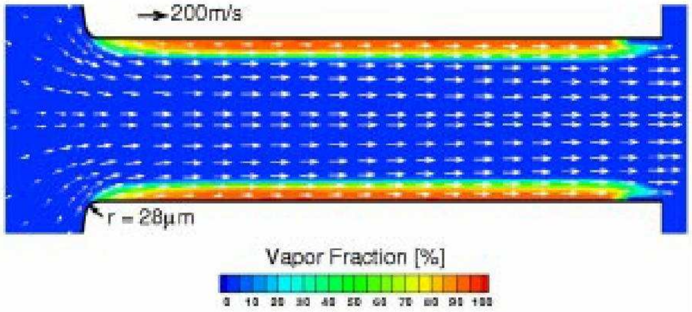

Figure 8. CAVITATION INSIDE THE NOZZLE INJECTOR. NUMERICAL SIMULATION BY YUAN *et al.* [3]. INJECTION PRESSURE, $P_{inlet} = 80$ bar; EXIT PRESSURE, $P_{exit} = 11$ bar.

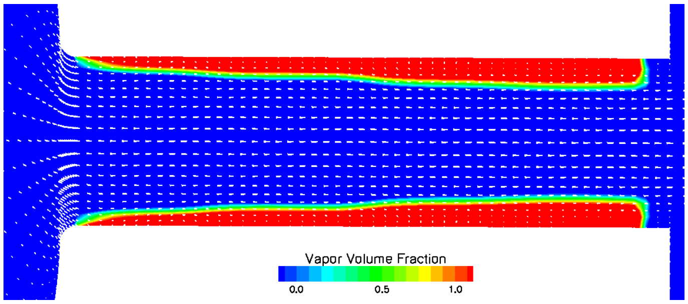

Figure 9. RESULTS OF THE NUMERICAL SIMULATION FOR A CAVITATING NOZZLE INJECTOR USING THE MULTIPHASE MIXTURE FORMULATION WITH A CAVITATION MODEL. INJECTION PRESSURE, $P_{inlet} = 80$ bar; EXIT PRESSURE, $P_{exit} = 11$ bar.

Cavitation in Nozzle Injector

Cavitation is a physical phenomenon that occurs when the local pressure drops below the vapor saturation pressure. Cavitation will normally occur inside the nozzle injector in many modern fuel injection systems primarily due to high injection pressure. Other important parameters that additionally affect cavitation are the orifice inlet curvature, injection angle, and nozzle aspect ratio. A numerical simulation is carried out for a high injection pressure to confirm cavitation inside a nozzle injector. There are two scenarios performed in the simulation to compare with the experimental results of Roosen *et al.* [57], and the numerical simulation results of Yuan *et al.* [3]. The first scenario considers an injection pressure of 80 bar and an ambient condition of 21 bar, while the second scenario considers an injection pressure of 80 bar and an ambient condition of 11 bar. The injection pressure is assigned a value of the inlet pressure for simplicity and is assumed to be a constant steady inlet pressure throughout the calculation. In the next section, the injection pressure is fluctuated and the effect to the external jet formation is observed.

In both scenarios, the results of the simulation, see Fig. 9, were in substantial agreement with the experimental results of Roosen *et al.* [57], see Fig. 7, and the numerical simulation results of Yuan *et al.* [3], see Fig. 8, respectively. Figure 10 shows the contour plot of the vapor and gas volume fraction, inside and close to the exit of the nozzle injector at different time levels. It was observed that the time scale of the internal cavitation dynamics is on the order of 10 μs, which confirms the assertion of Yuan *et al.* [3]. It was also noticed that the bubble cavities start to develop near the inlet corner of the nozzle orifice, and extend further downstream to the exit of the nozzle. The maximum length of cavity is observed after steady flow is achieved. Maximum length of cavity is reached after 5.0e-05 seconds. As can be seen on Fig. 10, for the gas volume fraction, a presence of gas inside and near the exit of the nozzle is observed. This could be a reentrant jet of gas that starts to penetrate the cavity and causes it to collapse. However, the re-entrant jet is not strong enough to penetrate inside the nozzle further upstream. This might be due to the steadiness of the flow and constant steady injection pressure.

Figure 11 shows the contour plot of the liquid volume fraction and velocity, inside and close to the exit of the nozzle injector at different time levels. It is clear that cavitation causes a reduction of the cross sectional area of the liquid jet flow as can be seen on the left of Fig. 11. This means that cavitation chokes the flow of liquid and reduces the discharge significantly. The coefficient of discharge within the orifice for a 1.2 cavitation number was computed to be equal to a value of 0.75, which is about 7 % higher as compared to Nurick's [6] correlation and Singhal *et al.* [40] simulation. Also, shown to the right of Fig.

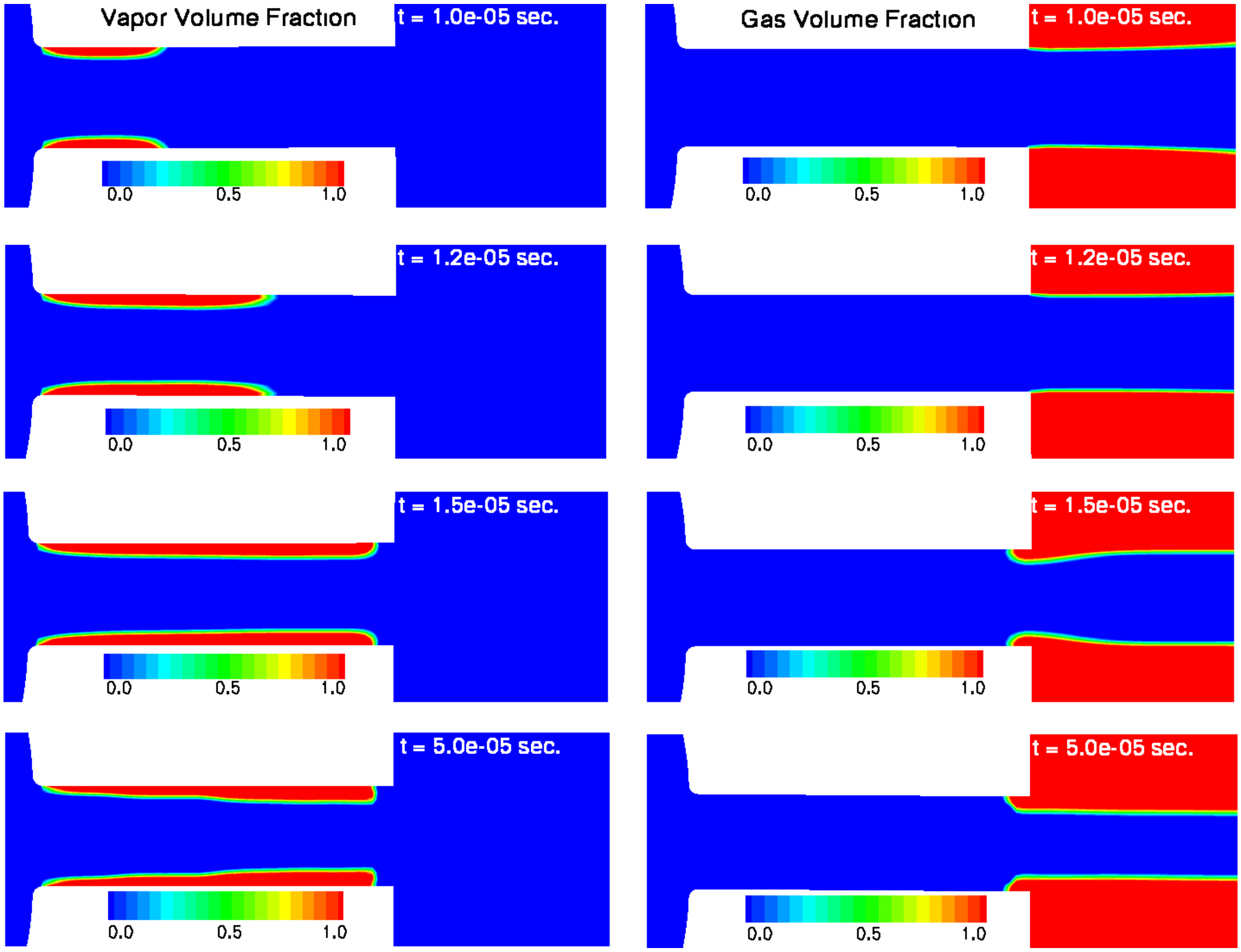

Figure 10. CONTOUR PLOT OF THE VAPOR (LEFT) AND GAS (RIGHT) VOLUME FRACTION INSIDE AND CLOSE TO THE EXIT OF THE NOZZLE INJECTOR. CONSTANT STEADY INJECTION PRESSURE, $P_{inlet} = 80$ bar; EXIT PRESSURE, $P_{exit} = 11$ bar.

11 is how cavitation separates the flow inside the nozzle injector and affects the velocity profile of the jet flow.

Periodic Inflow Condition and External Jet Formation

It has been illustrated in the previous section that cavitation occurs inside the nozzle injector and has significant effect on the liquid flow discharge and velocity profile of the flow. In the actual injection process, the injection pressure and exit pressure fluctuate at high frequencies. To understand the effect of injection pressure fluctuation to the cavitation process and the external jet formation, a periodic rectangular inflow pressure condition is implemented. Figure 12 shows the results of the simulation at different time levels for a fluctuating inflow pressure condition. The volume fraction of liquid that relates to the amount of liquid discharge and the formation of the external jet is shown on the left of Fig. 12, while the volume fraction of vapor inside and close to the exit of the nozzle injector is depicted on the right of Fig. 12.

The cavitation process in the actual injection system becomes more complex due to fluctuating inflow boundary condition. There is significant interaction of the bubble cavities and gas from the nozzle exit. It can be seen on the right of Fig. 12 that the cavitation process is more complicated. A re-entrant jet of gases from the downstream chamber causes the cavities to collapse and separate. The separation of the bubble cavities creates a bubble cloud downstream of the cavity. The bubble clouds will interact with a large rotating vortex of gas external to the nozzle as it exits from the nozzle injector. Qin *et al.* [38] further postulated that the external pressure propagates into the orifice once the re-entrant jet reaches the nozzle orifice inlet and occupies the entire upper part of the orifice, causing the flow in the nozzle to revert to a non-cavitating mode. This phenomenon is called hydraulic flip, which was observed by Bergwerk *et al.* [4].

The discharge of the nozzle is strongly dependent on the cavitation process and the magnitude of the bubble cavities. As can

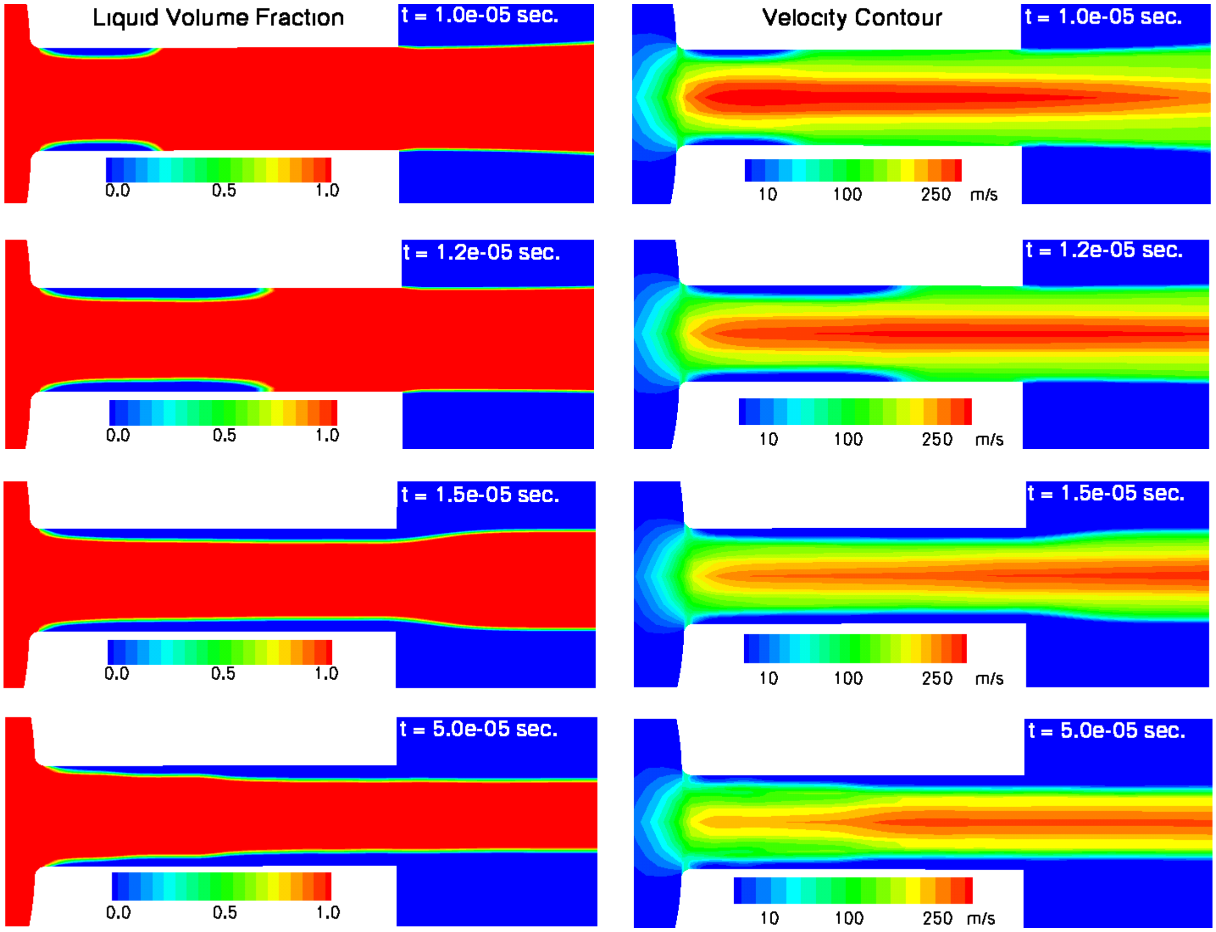

Figure 11. CONTOUR PLOT OF THE LIQUID VOLUME FRACTION (LEFT) AND VELOCITY PROFILE (RIGHT) INSIDE AND CLOSE TO THE EXIT OF THE NOZZLE INJECTOR. CONSTANT STEADY INJECTION PRESSURE, $P_{inlet} = 80$ bar; EXIT PRESSURE, $P_{exit} = 11$ bar.

be seen on the left of Fig. 12 , the magnitude of discharge fluctuations is related to the fluctuation of the inflow boundary condition. It should be noticed that the cavitation process for a fluctuating inflow condition intensifies the unsteadiness of the external jet formation. The instability of the jet formation and the fluctuating amount of discharge greatly affect the jet break-up and the atomization process further downstream .

CONCLUSIONS

The new multiphase mathematical model based on a mixture formulation of the laws of conservation for a multiphase flow was successfully applied to numerically simulate a high pressure, supersonic three-phase cavitating jet flow within a nozzle injector through a gaseous medium. The influence of cavitation process inside the nozzle injector to the external jet formation was successfully demonstrated. The cavitation model captures well the cavitation process inside the nozzle injector. The time scale of the internal cavitation dynamics was observed to be on the order of 10 μs, which confirms the assertion of Yuan *et al.* [3]. The flow inside and close to the nozzle injector was observed and found to be complex. The discharge of the nozzle is strongly dependent on the cavitation process and the magnitude of the bubble cavities. Cavitation separates the flow inside the nozzle injector and intensifies the unsteadiness of the external jet formation for a fluctuating inflow condition.

The objective of capturing the complex structure of the flow inside and close to the nozzle region by modeling and simulating multiphase cavitating jet flow in a nozzle injector was successfully achieved. Furthermore, the modified numerical scheme by Harten, Lax and van Leer (HLLC) was successfully implemented into the multiphase mixture formulation. The HLLC scheme is suitable for resolving isolated shock and contact waves. With the HLLC scheme, multiphase flow problem with strong contact discontinuities of the mass and momentum in the flow field can easily be solved.

The Reynolds number for a typical injection system process is very high. Due to a high Reynolds number, turbulent flow is

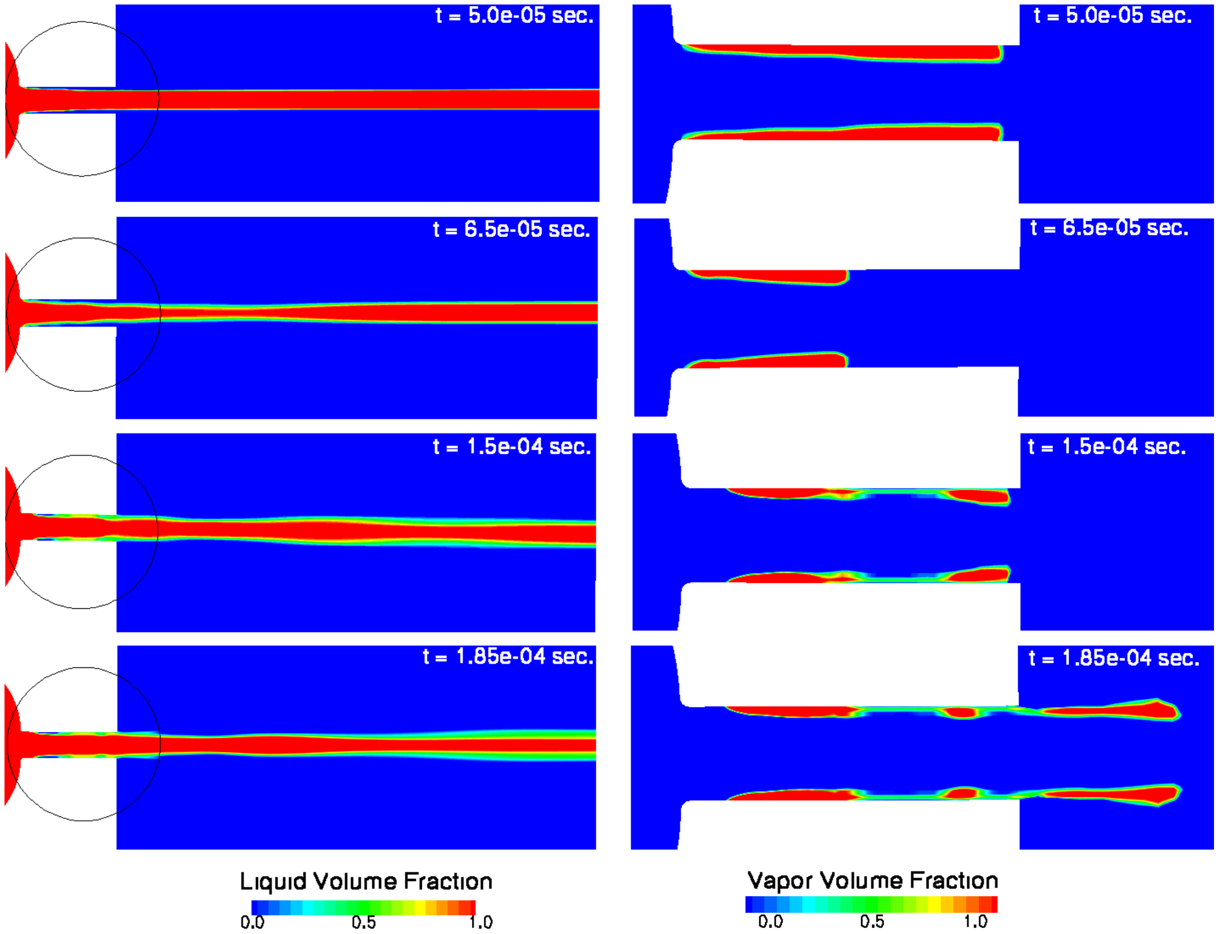

Figure 12. PLOT OF THE UNSTEADY EXTERNAL JET FORMATION (LEFT) AND DISTRIBUTION OF VAPOR VOLUME FRACTION (RIGHT) INSIDE AND CLOSE TO THE EXIT OF THE NOZZLE INJECTOR. PERIODIC UNSTEADY INJECTION PRESSURE, $P_{inlet} = 80 \pm 10$ bar; FREQUENCY, $f = 3.725$ Mhz; EXIT PRESSURE, $P_{exiit} = 11$ bar.

developed inside the nozzle injector. The multiphase model in its present state of development does not incorporate turbulence model. The multiphase Large Eddy Simulation (LES) turbulence model is currently under development, and will be implemented in the next work. However, turbulence does not influence the cavitation process, particularly in the region where pressure decreases dramatically like near the edge of the entrance of the nozzle orifice [32]. On the other hand, experimental observations show that downstream the nozzle and the reattachment region, turbulence affect the breakup and coalescence of the collapsing bubble cavities.

ACKNOWLEDGMENT

This work was sponsored by Argonne National Laboratory. This support is gratefully acknowledged. The third author is supported by the Department of Energy through the University of California under subcontract number B341494.

REFERENCES

[1] He, L., and Ruiz, F., 1995. “Effect of cavitation on flow and turbulence in plain orifices for high-speed atomization”. *Atom Sprays*, **5** , pp. 569–84.

[2] Soteriou, C., Andrews, R., and Smith, M., 1995. “Direct injection diesel sprays and the effect of cavitation and hydraulic flip on atomization”. *SAE Paper 950080* .

[3] Yuan, W., and Schnerr, G., 2003. “Numerical simulation of two-phase flow in injection nozzles: Interaction of cavitation and external jet formation”. *Journal of Fluids Engineering*, **125** , pp. 963–969.

[4] Bergwerk, W., 1959. “Flow pattern in diesel nozzle spray holes”. *Proc. Inst. Mech. Eng.*, **173** , pp. 655–660.

[5] Lichtarowicz, A., Duggins, R., and Markland, E., 1965. “Dsicharge coefficients for incompressible non-cavitating flow through long orifices”. *J. Mech. Eng. Sci.*, **Vol. 7, No. 2** , pp. 210–219.

[6] Nurick, W. H., 1976. “Orifice cavitation and its effect on spray mixing”. *ASME J. Fluids Eng.* , pp. 681–687.

[7] Ruiz, F., and He, L., 1999. “Turbulence under quasi cavitating conditions: a new species”. *Atom Sprays,* **9** , pp. 419–29.

[8] Arcoumanis, C., Nouri, J., and Andrews, R., April 14-15, 1992. “Application of refractive index matching to a diesel nozzle internal flow.”. *Proceedings of the ImechE Seminar on Diesel Fuel Injection System* .

[9] Arcoumanis, C., Flora, H., Gavaises, M., Kampanis, N., and Horrocks, R., 1999. “Investigation of cavitation in a vertical multi-hole diesel injector”. *SAE Paper 1999-01-0524* .

[10] Hiroyasu, H., Arai, M., and Shimizu, M., Gaithersburg, Maryland , July, 1991. “Break-up length of a liquid spray and internal flow in a nozzle”. *ICLASS-91* .

[11] Knox-Kelecy, A., and Farrell, P., 1997. “Spectral characteristics of turbulent flow in a scale model of a diesel fuel injector nozzle.”. *SAE Paper 972942* .

[12] Soteriou, C., Smith, M., and Andrews, R., 1998. “Diesel injector laser light sheet illumination of the development of cavitation in orifices”. *Proc ImechE C529/018/98* .

[13] Ganippa, L., Bark, G., Anderson, S., and Chomiak, J., June 20-23, 2001. “Comparison of cavitation phenomenon in transparent scaled-up single-hole diesel nozzles”. *Proceedings of the 4th International Symposium on Cavitation* .

[14] Arcoumanis, C., Badami, M., Flora, H., and Gavaises, M., 2000. “Cavitation in real size multi-hole diesel injector nozzles”. *SAE Paper 2000-01-1249* .

[15] Badock, C., Wirth, R., and Tropea, C., Toulouse, July 5-7, 1999. “The influence of hydro-grinding on cavitation inside a diesel injection nozzle and primary break-up under unsteady pressure conditions”. *Proceedings of the 15th ILASS-Europe 99* , pp. 5–7.

[16] Tamaki, N., Shimizu, M., Nishida, K., and Hiroyasu, H., 1998. “Effects of cavitation and internal flow on atomization of a liquid jet”. *Atom. Sprays,* **8** , pp. 179–197.

[17] Chavez, H., Knapp, M., and Kubitzek, A., 1995. “Experimental study of cavitation in the nozzle hole of diesel injectors using transparent nozzles”. *SAE Paper No. 950290* .

[18] Li, Haiyun, and Collicott, S., Indianapolis, IN, 1999. “Initial images of cavitation inside high-pressure atomizers”. *ILASS-99* , pp. 409–413.

[19] Karasawa, T., Tanaka, M., Abe, K., and Kurabayashi, T., 1992. “Effect of nozzle configuration on the atomization of a steady spray”. *Atom. Sprays,* **2** , pp. 411–426.

[20] Chandra, B., and Collicott, S., Indianapolis, IN, 1999. “Experimental investigation of cavitation frequency in a slot orifice”. *ILASS-99* , pp. 379–384.

[21] Sanchez, and Collicott, Indianapolis, IN, 1999. “Statistics of cavitation length in a 2-d slot orifice”. *ILASS-99* , pp. 403–408.

[22] Wu, K., Su, C., Steinberg, R., Santavicca, D., and Bracco, F., 1983. “Measurements of the spray angle of atomizing jets”. *ASME J. Fluids Eng.,* **105** , pp. 406–413.

[23] Payri, F., Bermudez, R., Payri, R., and Salvador, F., 2004. “The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzles”. *Fuel,* **83** , pp. 419–431.

[24] Bunnell, R., and Heister, S., 2000. “Three-dimensional unsteady simulation of cavitating flows in injector passages”. *Journal of Fluids Engineering,* **122** , pp. 791–797.

[25] Dirke, v. M., Krautter, A., Ostertag, J., Mennicken, M., and Badock, C., 1999. “Simulation of cavitating flows in diesel injectors”. *Oil and Gas Science and Technology,* **Vol. 54, No. 2** , pp. 223–226.

[26] Yuan, W., and Schnerr, G., 2001. “Cavitation in injection nozzles: effect of injection pressure fluctuations”. *CAV2001:session A7.001* .

[27] Schmidt, D. P., Rutland, C. J., and Coarradini, M. L., 1997. “A numerical study of cavitating flow through various nozzle shapes”. Transactions of the SAE, **106** (3) , pp. 1644–1673.

[28] Hanratty, T. J., Theofanous, T., Delhaye, J.-M., Eaton, J., McLaughlin, J., Prosperetti, A., Sundaresan, S., and Tryggvason, G., 2003. “Workshop findings”. *International Journal of Multiphase Flow,* **29** , pp. 1047–1059.

[29] Power, H., and Brebbia, C. A., 2001. *Computational Methods in Multiphase Flow*. WITPRESS Southampton, Boston.

[30] Ishii, M., 1975. “Thermo-fluid dynamic theory of two-phase flow”. *Eyrolles* .

[31] Merkle, C. L., Feng, J. Z., and Buelow, P. E. O., 1998. “Computational modeling of the dynamics of sheet cavitation”. In *Proceedings of the 3rd International Symposium on Cavitation*. Grenoble.

[32] Dumont, N., Simonin, O., and Habchi, C., 2001. “Numerical simulation of cavitating flows in diesel injectors by a homogeneous equilibrium modeling approach”. *CAV2001:sessionB6.005* .

[33] Shin, B., June 11-14, Anaheim, CA, 2001. “Numerical analysis of unsteady cavitating flow by a homogeneous equilibrium model”. *31st AIAA Fluid Dynamics Conference and Exhibit* .

[34] Alajbegovic, A., Meister, G., Greif, D., and Basara, B., 2001. “Three phase cavitating flows in high pressure swirl injectors”. In *Fourth International Conference on Multiphase Flow*. ICMF.

[35] Tatschl, R., Sarre, C., Alajbegovic, A., and Winklhofer, E., September 11-13, Darmstadt, 2000. “Diesel spray break-up modeling including multidimensional cavitating nozzle flow effects”. *ILASS-Europe* .

[36] Drew, D., and Passman, S., 1998. “Theory of multicompo-

nent fluids". *Springer, New York* .

[37] Senocak, I., and Shyy, W., July 6-10, Honolulu, Hawaii, USA, 2003. "Computations of unsteady cavitation with a pressure-based method". *Proceedings of ASME FEDSM2003-450009* .

[38] Qin, J., Yu, S., and Lai, M., October 25-28, Toronto, Ontario, Canada, 1999. "Direct calculations of cavitating flows by the method of space-time conservation element and solution element". *SAE 1999-01-3554* .

[39] Vortman, C., Schnerr, G. H., and Seelecke, S., 2003. "Thermodynamic modeling and simulation of cavitating nozzle flow". *International Journal of Heat and Fluid Flow,* **24** , pp. 774–783.

[40] Singhal, A., Athavale, M., Li, H., and Jiang, Y., 2002. "Mathematical basis and validation of the full cavitation model". *Journal of Fluids Engineering,* **124** , pp. 617–624.

[41] Delale, C., Okita, K., and Matsumoto, Y., November 1-4, Osaka, Japan, 2003. "Steady-state cavitating nozzle flows with nucleation". *Fifth International Symposium on Cavitation* .

[42] Ahuja, V., Hosangadi, A., Ungewitter, R., and Dash, S. M., 1999. "A hybrid unstructured mesh solver for multi-fluid mixtures". *AIAA-99-3330* , pp. 1–9.

[43] Hosangadi, A., Sinha, N., and Dash, S., 1997. "A unified hyperbolic interface capturing scheme for gas-liquid flows". *AIAA-97-2081* , pp. 1227–1237.

[44] Ahuja, V., and Hosangadi, A., 2001. "Unstructured simulation of multi-phase mixtures with compressibility effects". *AIAA-2001-2671* , pp. 1–15.

[45] Lagumbay, R., Vasilyev, O., Haselbacher, A., and Wang, J., Nov. 13-19, Anaheim, CA., USA, 2004. "Numerical simulation of a high pressure supersonic multiphase jet flow through a gaseous medium". *Proceedings of IMECE2004-61008* .

[46] Haselbacher, A., 2003. "Rocflu manual". In *CSAR*. University of Illinois at Urbana-Champaign.

[47] Haselbacher, A., 2005. "A weno reconstruction method for unstructured grids based on explicit stencil construction". *AIAA-2005-0879* .

[48] Batten, P., Clarke, N., Lambert, C., and Causon, D. M., 1997. "On the choice of wavespeeds for the HLLC riemann solver". SIAM J. Sci. Comput., **18** (6) , pp. 1553–1570.

[49] MacPhee, A. G., Tate, M. W., Powell, C. F., Yue, Y., Renzi, M. J., Ercan, A., Narayanan, S., Fontes, E., Walther, J., Schaller, J., Gruner, S., and Wang, J., 2002. "X-ray imaging of shock waves generated by high pressure fuel sprays". *SCIENCE,* **295** , pp. 1261–1263.

[50] Kuo, K., 1996. "Recent advances in spray combustion: Spray atomization and drop burning phenomena". *AIAA,* **1-2** .

[51] Herman, H., 1988. "Plasma spray deposition processes". *Materials Research Society bulletin,* **13** , p. 60.

[52] Tikhomirov, R., Babanin, V., Petukhov, E., Starikov, I., and Kovalev, V., 1992. *High-Pressure Jetcutting*. ASME Press.

[53] Hosangadi, A., Ahuja, V., and Arunajatesan, S., 2001. "A generalized compressible cavitation model". *CAV2001:sessionB4.0043* .

[54] Goff, J., 1957. "Saturation pressure of water on the new kelvin temperature scale, transactions of the american society of heating and ventilating engineers". *American society of heating and ventilating engineers* , pp. 347–354.

[55] Chapra, S. C., and Canale, R. P., 1988. *Numerical Methods for Engineers*, 2nd ed. McGraw-Hill, Inc, New York.

[56] Harten, A., Lax, P., and Van Leer, B., 1983. "On upstream differencing and godunov-type schemes for hyperbolic conservation laws". *SIAM Rev.,* **25** , pp. 35–61.

[57] Roosen, P., Unruh, O., and Behmann, M., 1996. "Untersuchung und modellierung des transienten verhaltens von kavitationserscheinungen bei ein- undmehrkomponentigen kraftstoffen in schnell durchstromten dusen". *Report of the Institute for Technical Thermodynamics* .