ABSTRACT  The formation of cavitation clusters and their influence on the flow structure inside high speed injector flows, discharging to a pressurized environment, is studied using a Homogeneous Equilibrium Model (HEM) type model. The model treats the liquid and vapor phases as continuum and uses an equation of state to link the phases and to obtain effective mixture properties. Two-dimensional compressible two-phase single-fluid Navier-Stokes equations are solved using pressure based methods, incorporating PISO schemes for pressure-velocity coupling. Within a computing cell the volume fraction of vapor is determined as a function of local density change, which allows the calculation of the speed of sound of the mixture. This value of the speed of sound is used to modify the compressibility term in the PISO pressure-correction equation during transient simulations. At the nozzle exit, a Navier-Stokes Characteristic Boundary Condition (NSCBC) is applied to allow acoustic waves to pass through the boundary. Application of NSCBC is equivalent to imposing a partially reflecting boundary condition, which helps in controlling the effect of downstream reflecting waves on the characteristics of the upstream cavitation and flow structure. In all simulations the nozzle inlet pressure was kept constant while the ambient pressure was changed to evaluate its influence on the cavitation characteristics. The simulations found that the internal nozzle flow portraits highly transient vapor formation and shedding cycles. For sufficiently low ambient pressure (i.e., high pressure drop along the injector), the cavitation clusters formed near the nozzle entrance, extend along the walls far downstream leading to events of super-cavitation. When the ambient pressure is increased a significant shifting between longer and shorter cavity lengths occurs, which generates strong recirculation regions within the nozzle. Mechanisms such as cavity stretching and cluster pinch-off were observed, exhibiting apparent non-periodic frequencies.

Keywords: Cavitation, Numerical simulation, HEM, Navier-Stokes, compressible flows, injector flows

1. INTRODUCTION

Cavitation inside nozzles has been extensively studied by researchers in order to assess its influence on spray atomization. Quantitative prediction of the vapor formation and their collapse using experimental techniques, are inhibited due to the small time and length scales involved in cavitation [1]. Bergwerk [2] observed that the jets emanating from cavitating nozzles had ruffled appearance and proposed that the creation of large amplitude turbulent disturbances due to cavitation mechanism enhanced jet atomization. Experiments of Wu et al. [3] indicated the influence of cavitation phenomena on atomized spray cone angle, while the studies conducted by Chaves et al. [4], showed development of elongated cavitation cluster cavities along the walls of orifice reaching the nozzle exit leading to events of supercavitation. Using the measurements on spray cone angle at different pressure drops along the injector, the combined influence of supercavitation and turbulence on jet atomization was assessed. Owing to the complexity associated with cavitating flow experiments, several authors turned to numerical methods for investigating cavitation phenomena.

Numerical simulations of multiphase flows entailing cavitation are very challenging due to the presence of steep local gradients of density in the liquid flow, which is predominantly incompressible. Interface tracking methods employ separate treatment of the phases for their hydrodynamic behavior with an added complexity of resolving the liquid-vapor interface with higher order surface reconstruction schemes. The continuum methods, however, consider the flow as a homogeneous mixture employing a void fraction variable to quantify the regions where the fluid has changed its properties, typically density. The smearing of liquid-vapor interface occurs at the sub-cell level and the mixture density reflects the fraction of vapor and liquid in a given cell. Since interface tracking is not employed, surface tension forces are often neglected. Continuum methods have been used in various forms: computations with vapor production terms [5], bubble dynamics [6] and barotropic equation of state [7, 8]. Vapor fraction methods encountered several stability problems at low void fractions which made it less attractive for cavitating flow computations, while the barotropic model computations were restricted by density ratios [7].

Recently, Schmidt et al. [9] performed numerical simulations on high speed cavitating injector flows using Homogeneous Equilibrium Model (HEM). A density based single-fluid compressible solver in conjunction with an explicit relation between pressure and density was used for numerical computations. This method takes less computational time, since no pressure equation treatment was performed. However, the density based solvers suffer from instability and slow convergence at low Mach number limits [10]. Turkel [11] identified that in the low Mach number limit, the discretized solution of the compressible
fluid flow fails to provide an accurate approximation to the incompressible equations. The stiffness of the time dependent equations associated with density based solver at low Mach number limit arises due to the mismatch of the acoustic speed and the fluid flow speed leading to slow convergence [12].

Numerical simulations of compressible cavitating flows using pressure based methods on continuum assumption are not frequent. In the current study, the compressible Navier-Stokes equations, decoupled from energy equation, in combination with HEM are solved using pressure based methods on arbitrary unstructured meshes. This method has the advantage of simulating wide gamut of Mach number flows without any preconditioning [11] or asymptotic schemes [10] together with continuum assumption reducing the computational time. In this framework, pressure variation remains finite irrespective of Mach number of the flow, offering computational capability throughout the entire spectrum of Mach numbers [13], an advantage over the density based methods. The pressure–coupling is achieved through PISO scheme [14], particularly suitable for simulating transient compressible flows. The requirement for a volume fraction equation to update the mixture density is eliminated by continuum assumption. Stability and boundedness of the pressure correction equation is ensured by proper numerical analysis of the terms involved. The treatment of the boundary conditions is dealt with special interest, since reflective nature of boundary conditions, typically encountered in compressible flows can lead to erroneous calculations [9]. The primary focus of this paper is to qualitatively characterize the inception of cavitation using HEM model in combination with pressure correction techniques. In the following section, brief description of the governing equations and HEM assumptions is presented.

2. MATHEMATICAL DESCRIPTION

2.1 Governing equations

In the homogenous method, with the hypothesis of thermodynamic equilibrium, Navier Stokes equations in the compressible form are solved for a single phase flow. By the nature of the barotropic state relations, the energy equation is decoupled from the system. This enables us to consider a reduced set of compressible flow equations involving only the mass and momentum conservation equations. The mass conservation equations, in tensorial form, can be expressed as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0,$$

while the momentum conservation equations are given by

$$\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U - T) = 0,$$

where $T$ is the total stress tensor defined as

$$T = -\left( P + \frac{2}{3} \mu \nabla U \right) I + 2\mu D.$$

$P$ in Eq. (3) is the pressure, $I$ the unit tensor and $D$ representing the rate of strain tensor. The strain rate tensor is expressed as

$$D = \frac{1}{2} \left( \nabla U + \nabla U^T \right).$$

The momentum equations are tightly coupled with density and viscosity relations. The density constitutive relations in the compressible scenario are given as functions of pressure:

$$\rho = \rho(P).$$

The effective viscosity of the mixture is given by

$$\mu = \alpha \mu_g + (1 - \alpha) \mu_l,$$

where $\mu_g$ and $\mu_l$ are viscosities of vapor and liquid phases respectively. In the current simulation, no turbulence model has been included to effect any change in the viscosity of the mixture. The additional viscosity, turbulent viscosity, due to the flow turbulence and dissipation has been neglected. This can be related to the fact that the order of magnitude of turbulence generated by cavitation itself is far higher than the flow turbulence [15]. Although several authors [1, 3] hinted at the influence of turbulence on the events of cavitation, implementation of turbulence models in the compressible framework would be carried out in our future studies. Here, we solve unsteady laminar compressible Navier Stokes equations in the flow domain using HEM assumptions for relating the mixture density to the variation in pressure fields.

2.2 Closure of hydrodynamic equations

The accuracy of numerical simulations on liquid flow with phase change rests in the modeling of the closure of hydrodynamic equations. The assumption of homogenous equilibrium of different phases of the liquid in the flow leads to the conservation of energy statement with viscous dissipation terms. The pressure and enthalpy of mixture terms are several order of magnitude higher than the conduction and viscous dissipation terms in the energy balance equations [9]. Using an isentropic assumption in the flow domain, and neglecting contributions of viscous dissipation and thermal conductivity, the energy statement can be reevaluated as

$$a^2 \frac{DP}{Dt} - \frac{\partial P}{\partial t},$$

Equation (8) can be considered as an isentropic model in a phase change process, where the speed of sound is given by the HEM [16] as

$$dP = a^2 \, d\rho,$$

where $a$, the speed of sound in the two phase mixture, is given by
a = \frac{1}{(1-a)\rho_g + a\rho_v} \left( \frac{a}{\rho_v \rho_g^2} + \frac{1-a}{\rho_v \rho_g^2} \right) \quad (10)

The values of \( a_g \) and \( a_v \) being the speed of sound in pure gas and pure liquid phases respectively. \( \rho \) can be identified as the mixture density related to pure liquid and vapor densities as

\[ \rho = a \rho_v + (1-a) \rho_g. \quad (11) \]

From Eq. (11), void fraction \( a \) is obtained as

\[ a = \frac{\rho - \rho_g}{\rho_v - \rho_g}. \quad (12) \]

The fluid in the current simulation is considered as single-phased, whose density varies from liquid to vapor according to the barotropic equation of state, based on the expression of the local sound speed depending on the local void fraction. This expression considers the fluid homogeneously mixed on the sub-grid scale and is called the homogenous equilibrium model. The relative velocity between the phases can be neglected based on the homogenous approximation leading to single phase equations with a mixture equation of state. When the pressure of the mixture is either well above or below the vapor pressure, the speed of sound is large but finite and the flow is considered weakly compressible. However in practical computations, the Mach number of the flow would vary from 0.001 to as high as 30. Hence, for efficient computation of the multiphase mixture with the homogenous assumption it is required that the time marching procedures are stable and accurate in the range of Mach numbers specified.

3. COMPUTATIONAL METHODOLOGY

In simulating cavitating injector flows, Schmidt et al.[9] used analytically integrated expressions to explicitly solve for the pressure values in the cell as a function of density. In performing an accurate simulation of the cavitating flows with incompressible zones and highly supersonic (vapor-liquid mixture), it is required to deal with the strong discontinuities with radical changes of physical quantities near saturation regions. The current simulations performed are based on finite volume discretization available in the openFoam framework [17], used for simulating continuum phenomena. Computations are performed over unstructured grids based collocated variable arrangement [18] and Rhie and Chow [19] interpolation for obtaining face values. This method works well on co-located meshes where pressure gradients are the major driving force which is typical in the case of injector domains. In the current methodology involving unsteady compressible flows, Pressure Implicit with Splitting of Operators (PISO) scheme has been utilized for pressure-velocity coupling. This pressure-velocity treatment proposed by Issa [14] was originally developed for non-iterative computation of unsteady compressible flows, involves a single predictor step and two corrector steps. This pressure implicit method uses a predictor-corrector approach to advance the momentum equation while enforcing the continuity equation. In the compressible PISO formulation [14], the pressure-density coupling is introduced only through the time-dependent term of the continuity equation, which helps enforce a strong coupling between the density variation and the flow field. In the following sub-section, a brief description of the formulation of pressure correction equation and relevant numerical analysis is presented.

3.1 Discretization procedure

In order to accurately solve the constitutive governing equations in resolving the transient behavior of the compressible flows, the development of pressure correction equation absorbing the effect of phase change is instrumental. In this section, we detail the discretization procedure of the governing equations, finally leading to the development of the pressure correction equation. Rewriting the momentum equation, we have

\[ \frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U - T) = 0. \quad (13) \]

The accurate prediction of pressure-velocity coupling is very crucial in resolving the non-linearity present in the momentum equation. In the finite volume framework, the non-linear convection terms can be linearized [18] and written as,

\[ \nabla \cdot (\rho U U) = \sum_f S \cdot (\rho U) f (U) f = \sum_f F (U) f, \]

\[ \sum_f F (U) f = a_p U_p + \sum_N a_N U_N \quad (14) \]

Where \( F \), the face flux and the coefficients \( a_p, a_N \) are functions of \( U \). After standard discretization procedure [18], the semi-discretized form of momentum equations Eq. (2) can be written as

\[ a_p U_p = H(U) - \nabla P. \quad (15) \]

Rewriting the discretization equation for velocity,

\[ U_p = \frac{H(U)}{a_p} - \frac{\nabla P}{a_p}, \quad (16) \]

where \( H(U) = \sum_N a_N U_N \).

3.2 Formulation of pressure equation

In evaluating compressible flows the fluid density can be related to the fluid compressibility factor as

\[ \rho = \psi P, \quad (18) \]

where \( \psi \) is the compressibility of the mixture. Using the isentropic phase change considerations, the compressibility can be calculated as,

\[ \psi = \frac{1}{a^2} \quad (19) \]

Equation (19) helps in the closure of hydrodynamic
equations by relating the density functional with the pressure variable using the compressibility factor. Combining Eqs. (1), (16) and (19), we have

$$\frac{\partial \psi \rho}{\partial t} + \nabla \cdot \left( \rho H \left( \frac{U}{a_p} \right) \right) - \nabla \cdot \left( \rho \nabla \rho \right) = 0. \quad (20)$$

Further, using Eqn. (18) to represent density as a function of pressure and speed of sound in the liquid-vapor mixture, we have

$$\frac{\partial \psi \rho}{\partial t} + \nabla \cdot \left( \psi \frac{H \left( \frac{U}{a_p} \right) \rho}{a_p} \right) - \nabla \cdot \left( \rho \nabla \rho \right) = 0. \quad (21)$$

Equation (21) represents the characteristic pressure equation for compressible flows derived from continuity and momentum equations. The equation has the compressibility factor associated with the pressure in the unsteady time derivative and with the velocity in the convection terms. From a preliminary analysis of the characteristic pressure equation, we can deduce that improper numerical treatment can result in an unbounded nature of the individual terms and can propagate into unphysical results in the domain. For improving accuracy and bounded solutions, blending scheme (combined upwinding and central differencing) are used in the treatment of the non-linear terms [18].

Treatment of the divergence terms in an explicit manner leads to an elliptic solving procedure which enhances convergence as compared to hyperbolic systems. Note that in the event that the flow regimes tend toward very low Mach numbers, the equations would typically reduce to the familiar solenoidality condition of incompressible flow [18]. The implicit implementation of the pressure equation helps in maintaining the computation convergence. Explicit first order time integration procedure with two corrector steps, within the PISO procedure, is carried out to increase time accuracy by an order [14]. The predictor corrector equations formulated using PISO are solved using incomplete-cholesky preconditioned conjugate gradient method (ICCG), while the momentum equations are solved using incomplete-cholesky preconditioned biconjugate gradient (BICCG) methods.

4. MODEL VALIDATION

The unsteady cavitating flow formulation presented in Section 3 is applied to a simple injector nozzle geometry used in the experiments of Roosen et al. [20]. In the present study, no interaction of cavitating flow with nozzle exterior is considered. The experimental set up of Roosen et al. consists of a rectangular shaped channel 0.2mm × 0.28mm × 1mm representing the width, height and length respectively. In the current study, only two-dimensional calculations are considered (width neglected). We use the same sac hole entrance curvature of $R_n = 0.1 \times$height as applied in the computations of Yuan et al. [21]. The simulation uses unstructured triangular meshes for finite volume discretization in the nozzle domain. Mesh clustering operation is performed near the contraction corners in the nozzle to capture high pressure and density gradients accompanying events of cavitation inception.

The left inlet boundary, representing a reservoir, is specified with a total inlet pressure. The outlet is designated as pressure outlet representing free stream conditions.

![Figure 1: Planar nozzle used in our numerical simulation.](image1)

The wall conditions with no-slip boundary conditions for velocity and zero gradient condition for pressure fields are imposed. For simulation and validation purposes, we use water, water-vapor mixture. All properties of water and water-vapor have been taken at STP.

![Figure 2: Speed of sound in a water-water vapor mixture estimated from Eqn. (10).](image2)

4.1 Treatment of Boundary Conditions

The conditions prevailing at pressure outlet are very complicated due to the interaction of the boundary with liquid-gas compressible mixture. In this context, it is essential to understand that the boundary conditions for subsonic outflows can be reflecting. Hence, the boundary condition must be able to accommodate two phase regimes and be able to allow acoustic waves generated by the two phase flows to pass through them. If the boundary conditions are of reflecting nature, they would back transmit perturbations affecting the upstream leading to totally non physical behavior of the flow structure. In highly non linear cavitating flows, exit boundaries are often supersonic when the neighbouring cells contain mixture of the phases although the exit conditions may be subsonic for the flow of pure liquid phase. This local supersonic regime due to the fact that the cell containing vapor phase has a low sonic speed. We use NSCBC (Navier Stokes Characteristic Boundary Conditions) approach, expressing Navier Stokes equations as characteristic equations at the boundary [22]. The conservative equations at the exit are written in terms of characteristic equations with characteristic wave amplitudes and propagation velocities of characteristic waves. Poinset et al. [23] suggested incoming characteristic wave of the form

$$l_i \propto \kappa \left( P_{exit} - P_2 \right), \quad (22)$$

where $l_i$ is the incoming characteristic wave that transmits downstream pressure back into domain with $P_{exit}$ being the current pressure value at the boundary between a cell and
exit. \( P_2 \) represents the specified exit pressure. An approach to estimate this constant has been presented by Poinso et al. [23]. With specified \( \kappa \) value, the boundary condition varies the downstream pressure from the specified downstream pressure to allow structures such as waves and vapor regions to pass through without reflection. The construction of this constant is critical, since it portrays the reflecting nature of the boundary condition. An increase in the \( \kappa \) value leads to a strong coupling with the specified downstream pressure which implies that the boundary becomes more reflecting. When the cavitation regions extend to the domain exit, no relaxation over the exit pressure should be made such that the regions exit the domain without any numerical collapse. Further, current study prohibits any occurrence of hydraulic flip phenomena [2], where gases present in the downstream of nozzle can move into the nozzle and act to stabilize the liquid flow, typically occurring during events of supercavitation.

5. TEST CASES

Experimental studies have confirmed that the cavitation number \( (\sigma) \) and Reynolds number \( (Re) \) are the most important criteria [1], which describe similarity of cavitating flows in nozzles:

\[
\sigma = \frac{P_1 - P_2}{P_2 - P_v},
\]

\[
Re = \frac{\rho U_l D_l}{\mu},
\]

where \( P_1 \) and \( P_2 \) are pressures upstream and downstream of the nozzle, \( P_v \) is the vapor pressure, \( U_l \) is the average velocity of the flow, and \( D_l \) is the hydraulic diameter of the nozzle. We would base our discussions on cavitation number, although \( Re \) is critical in providing insight on the turbulence intensity present in the system.

For validation purposes, the computations were performed with a constant inlet stagnation pressure of 80 bar and varying exit static pressures: (a) 16 bar (b) 19 bar and (c) 26 bar. These numerical experiments would help us analyze in detail, the influence of pressure drop, or equivalent cavitation number, on the transient nature of cavitating flows and related phenomena such as vapor shedding, collapse etc., A constant value of \( \kappa = 0.01 \) is used throughout the simulation unless stated otherwise.

5.1 Exit Pressure: 16 bar

The cavitation number computed from the given pressure drop corresponds to \( \sigma = 4.0 \). Our transient simulation revealed acceleration of the fluid near the sharp corners at the entrance of the bore hole resulting in flow separation resulting in vena contracta. Since, this separation zone serves the low pressure requirement for the inception of cavitation, first occurrence of vapor pockets were spotted near the entrance region. The flow separates from the outer wall as the fluid enters the sac, but then reattaches near the nozzle entrance. Formation of circulation regions and their stretching mechanism is illustrated in Fig. 3. The formation of vena contracta reduces the effective flow area resulting in higher velocities, due to mass conservation. Consequently, the conservation of momentum results in pressure depression due to accelerating liquid flow. This increase in liquid velocity supplies the vapor cavities with low pressure required for their growth [1].

As flow evolves, the reattachment zones restricting the cavity length, tend to elongate. The low density vapor pockets that are initiated near the nozzle corners extend downstream to form a stable film. The vapor cavities thrive on the low pressure content zones close the nozzle walls.

Fig. 4 depicts cavity cluster stretching and pinch off mechanism. Re-entrant regions at the initial stages of the simulation were observed. The formation of vapor cavity and subsequent events leading to a stable supercavitation regime is presented in Figs. 5(a) – 5(e).

Some collapse features near the exit section of the nozzles may vary due to the very fact that we allow partial transmission (partially reflecting condition) of pressure waves across the boundary. In reality, several other mechanisms such as free surface interaction of the issuing jet with atmosphere, hydraulic flip, might influence the collapse characteristics of the cavitation clusters. In the
present formulation, since the density change is tightly coupled with the fluid velocity, the formation of these low density regions, corresponding to cavitation clusters, are well identified based on the local flow conditions. In this case, the pressure drop across the sac is strong enough to carry portions of vapor cluster past the nozzle exit against the re-entrant jets leading to stable vapor film. It has been experimentally shown that, the cavitation number strongly influences the cavity length and events of supercavitation [4].

The clusters formed at the corners are swept downstream by the accelerating fluid. The local density modelled using transient compressible formulation in the current investigation reflects the pressure history encountered by the fluid in the regions prone to cavitation. The transient variation of vapor fraction and x-component of velocity at the nozzle exit is shown in Figs. 6(a) and 6(b) to analyze the constancy of the cavitating system.

![Figure 6: Plot of vapor volume fraction and x-velocity component at nozzle exit at different ascending time (T1 – T5).](image)

Our transient pressure based framework helps capture the temporal effects of the cavitation phenomenon in the entrance region and in the downstream region. In the current simulations, the initial cavity formation was highly transient with severely fluctuating vapor regions. However, as the simulation progressed the cavity length was seen stabilized and a quasi-steady behavior of the vapor regime was observed. Fig. 6(a) shows existence of a stable vapor film with a steady distribution at the nozzle exit. Pertaining to this vapor concentration, a steady exit velocity at the nozzle exit is achieved (Fig. 6(b)). A contour plot of the vorticity magnitude in log scale is shown in Fig. 7 indicates the stretching of vorticity layers close to expanding cavitation clusters.

![Figure 7: Vorticity layer stretching near cavitation clusters.](image)

5.2 Exit Pressure: 19 bar

Following our previous computation, we increase the counter pressure to 19 bar (σ = 3.2) to observe its effect on cavitation dynamics. Vapor pockets inception occurred at the nozzle entrance region. Severe unsteadiness in vapor pocket stretching and elongation leading to pinch off from the parent cluster, was observed. As flow evolved, events of supercavitation were detected. The vapor cluster fraction reaching the exit did not portray a steady nature. The decrease in cavitation number of the system, resulted in a regime consisting of prominent shifting between a longer and shorter cavity length influencing the flow structure and the exiting supercavitation behavior. Fig. (8) illustrates unsteady supercavitation and cavity length shifting phenomena.

![Figure 8: Development of vapor pockets leading to highly unsteady events of supercavitation (a) – (e). Experimental results (f)[20] resemble closely to our numerical simulation.](image)

Figs. 8(a) and 8(b) show highly transient vapor pockets formation and shedding, creating a highly perturbed flow field which intensifies the turbulent nature of the exiting liquid jet. Radial disturbances due to localized pressure variation in the cavitating regime potentially describes the wrinkles on jets surfaces [2] exiting from cavitating nozzles and enhanced atomization [24].

The cavities formed at the corners entering the sac would periodically stretched until exit and then collapsed entirely, generating strong pressure waves and vortices. This finite propagation of pressure waves can be satisfactorily accommodated in compressible formulations such as the present one. After collapse, bubble clusters would pinch off the main cavity and be swept downstream during the collapse process. Following this, a new cavity would form and this process repeated several times although no definite frequency was associated with this...
phenomenon. The flow structure looked highly distorted due to shifting cavity length mechanism. When the cavity gets shortened a strong re-entrant jets finds its way back to stabilize the cavity. However, the local flow conditions prevailing in the flow aid the growth of clusters which works its way downstream against the re-entrant jets. Analogous to the cavity formation, stretching and collapse, the re-entrant jets appear at intervals corresponding to the presence of short cavity lengths.

The stretching and collapse of the cavity induces very strong local recirculation regions depicted in Fig. 9. A plot of vapor fraction at the nozzle exit shown in Fig. 10(a), exhibits unsteady supercavitation behavior. Fig. 10(b) representing the \( x \)-component of velocity displays similar unsteady trends. Note that any fluctuating behavior in the \( x \)-component reflects on the \( y, z \) directional components due to continuity constraint.

The velocity of the liquid jet at the nozzle exit (not shown here) exhibits very steady nature with vapor pockets occasionally crossing the exit section.

### 5.3 Exit Pressure: 26 bar

With further increase in the cavitation number to \( \sigma = 2.1 \), cavitation events incepted at the corners of the nozzle sac are restricted to shedding phenomenon alone. The unfavourable pressure gradient oppose any stretching and supercavitation events phenomena.

Transient simulation results for the present case is shown in Fig. 11. Notice that the vapor pockets are restricted close to the nozzle entrance region. Clearly, shedding mechanisms and pinch-off are observed in the nozzle interior. This observation is well matched with the experimental results.

Figure 10: Plot of vapor volume fraction (a) and \( x \)-velocity component (b) at nozzle exit at different ascending time (T1 – T5).

The induced orthogonal components near the nozzle exit tends to amplify due to aerodynamic interaction leading to shorter breakup length of the jets [3, 24]. The distribution and intensity of exiting vapor pockets is highly transient and, as our simulation suggests, is a strong function of cavitation number.

### 6. CONCLUSIONS

We performed numerical simulations using compressible formulations with HEM assumption to capture cavitation dynamics in injector flows. Three different cases representing different modes of cavitation dynamics were tested and our results indicate events of supercavitation. Cluster stretching and pinch-off compared well with the experimental results. The formulation helps illustrate the effect of the cavitation number on the cavitation dynamics. Increasing the cavitation number leads to intensified liquid-vapor mixture flows. The current simulation is two-dimensional and hence some discrepancy in the cavitation sheet stabilization can be attributed to the true three dimensional nature of the cavitating flows. The effece of turbulence in a compressible framework with cavitation needs to be evaluated. Critically, the effect of reflecting boundary conditions needs to be investigated since a wide range of cavitation behavior can be achieved traversing from no-reflecting to fully-reflecting conditions. Also, the effect of grid size is very important since coarse gridding often results in smoothing the scalar function and hence inaccurate description of the related dynamics.

### 7. ACKNOWLEDGEMENTS

This study is supported by Toyota Motor Corporation,
8. NOMENCLATURE

- \( a \) mixture speed of sound \([\text{m/s}]\)
- \( D \) strain rate tensor \([1/\text{s}]\)
- \( H() \) operator
- \( F \) face flux \([\text{kg/m}^2/\text{s}]\)
- \( P \) pressure \([\text{N/m}^2]\)
- \( R \) radius \([\text{m}]\)
- \( t \) time \([\text{s}]\)
- \( T \) stress tensor \([\text{N/m}^2]\)
- \( U \) velocity \([\text{m/s}]\)
- \( \psi \) compressibility factor \([\text{s}^2/\text{m}^2]\)
- \( x, y, z \) coordinates \([\text{m}]\)
- \( Re \) Reynolds number (dimensionless)
- \( l \) incoming characteristic wave \([\text{m/s}^2]\)
- \( \mu \) viscosity \([\text{kg/m/s}]\)
- \( \rho \) density \([\text{kg/m}^3]\)
- \( \kappa \) Relaxation constant (dimensionless)
- \( \alpha \) vapor volume fraction (dimensionless)
- \( \sigma \) cavitation number (dimensionless)

Subscripts
- \( 1,2 \) upstream, downstream
- \( \text{exit} \) nozzle exit
- \( \text{vap} \) vapour
- \( x, y, z \) coordinate direction
- \( g \) vapor
- \( l \) liquid
- \( f \) face
- \( P \) computed cell
- \( N \) neighboring cells
- \( \text{in} \) inlet

Superscript
- \( T \) transpose

9. REFERENCES