COMPARISON OF MIXTURE AND VOLUME OF FLUID MULTIPHASE MODEL FOR CAVITATION INSIDE THE FUEL INJECTOR NOZZLE

[1]Ashish Malkan, [2]Mehul Bambhania,

[1] PG student, [2] Assistant Professor,

[1,2] Department of Mechanical Engineering, Faculty of Technology and Engineering, The M S University of Baroda, India.

Abstract: As energy is essential for human development, society faces a dual challenge: the rapid growth in energy demand and carbon emissions including the risks of climate change. One important method of reducing emissions in Diesel engines is to improve fuel injector spray breakup, producing smaller and more disperse droplets. The flow inside the fuel injector nozzle is known to have a significant effect on the spray. Recent investigations have suggested that the cavitation occurring within the fuel injector nozzle significantly affects spray breakup. However the hydrodynamic cavitation behavior of diesel flow inside the fuel injector nozzle is still need to explore. In this paper numerical simulations were performed with and extensive validation has been established with available experimental data. Geometry of two dimensional real size nozzles has been used to assess the effect of mixture and volume of fluid (VOF) multiphase model. Effect of back pressure (15 bar to 85 bar) with constant injection pressure (100 bar) has been performed with diesel fuel. The results obtained with mixture and VOF model is compared with experimental data. It is found that both model are equally capable to capture cavitation phenomena, however VOF will give closer results after cavitation inception.

***Index Terms* – Fuel injector nozzle, Cavitation, Mixture model, VOF model, CFD.**

I. INTRODUCTION

Hydrodynamic cavitation is the formation of bubbles and cavities in a liquid due to the decrease in static pressure below the vapour pressure, caused by the geometry through which the fluid flows. Usually, liquids cannot stand negative pressures, and if the vapour pressure is reached, the liquid evaporates. The inception of cavitation can be explained as follows. The liquid entering the injection hole is firmly accelerated due to the reduction of the cross-sectional area. Assuming a simplified one-dimensional, stationary, frictionless, incompressible, and isothermal flow, the Bernoulli equation as:

$$-p_1 + \frac{\rho u_1^2}{2} = p_2 + \frac{\rho u_2^2}{2} \qquad \text{Eq.(1)}$$

It can be used to explain the fact that, an increase in flow velocity u from a point 1 to a point 2 further downstream leads to a decrease in static pressure p. The lowest static pressure is reached at the inlet edges in the recirculation zones of the so-called vena-contracta. If the local pressure reaches below the vapour pressure of the liquid, than the recirculation zones fill with vapour bubble. An additional effect enhancing the onset of cavitation in this low-pressure zone is the high shear flow generated by the large velocity gradients in the region between recirculation zone and the main flow. This shear flow produces tiny turbulent vortices. The cavitation zones develop along the walls can separate from the walls, disintegrate finally into bubble clusters, and it will start to collapse in downstream when local liquid pressure increases above vapour pressure shown in Fig.1

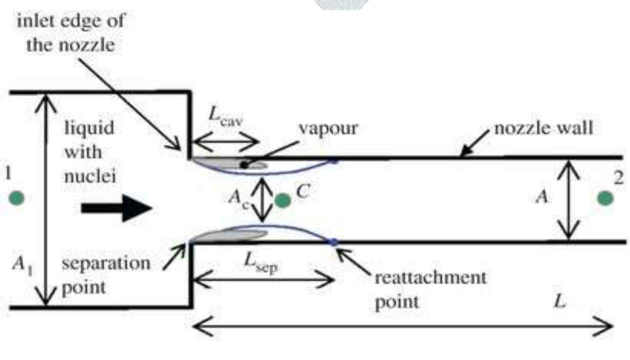

Fig. 1: Sketch of nozzle entrance that shows the cavitation inception [12]

Several computational and experimental studies have been reported & capture cavitating flow. However it is difficult to investigate the flow inside due to limitation of optical access, as a result many of the study reported global parameter such as total mass flow, empirical discharge coefficients, cavitation numbers and average velocities, to investigate the cavitation. The first real-size nozzle investigation was made by Arcoumanis et al. [2], conduct experiment with a hole of the real-size six-hole conical sac-type nozzle & the cavitation behavior observed indirectly. However Winklhofer et al. [3] reported experimental study with real size nozzle & developed optical method to capture cavitation inside fuel injector nozzle. An extensive study for measurement of velocity as well as mass flow was reported. In the addition, effect of inlet channel shape & shape of the target at high injection pressure were reported [1]. Due to experimental limitation, the use of CFD tool increases recently, which allows to capture inside detail of cavitation phenomena. Various cavitation model has been develop, which can be categorized as (a) single fluid/continuum model (b) two fluid model. In single fluid/continuum models, the average mixture properties, such as density and viscosity, are determined based on the vapour volume fraction. Schmidt et al. [6] developed model based on single fluid approach. In two-fluid models, the liquid and vapour phases are treated separately using two sets of conservation equations. Martynov et al. [4] studied two-fluid model with Eulerian–Eulerian approach. Giannadakis E. et al. [5] adopt Eulerian–Lagrangian based two-fluid model. The use of commercial CFD code allows understanding hydrodynamic behavior of flow in detail. S.Som et al. [7], Kaushik Saha et al. [8], Michele Battistoni et al. [9], Salvador et al.[10] carried out extensive computational study on cavitation and carried out parametric study based on nozzle geometry, fuel properties, pressure difference & needle movement. Following section illustrate summary of literature referred to understand cavitation in fuel nozzle.

II. NUMERICAL SETUP AND MESHING

Present work is proposed to carry out with the use of commercial CFD platform ANSYS-Fluent with the use of mixture multiphase model. The mixture and VOF multiphase model will be implemented in the framework of cavitation models by Schnerr-Sauer (SS) in ANSYS-Fluent. In addition to these conservation equations (mass & moment) and turbulence momentum equation are solve in ANSYS-Fluent. The problem considered in this work is a cavitating two-phase flow in a nozzle. Real size fuel injector nozzle geometry with its boundary conditions has been shown in Fig 2. Model was generated in design modeler tool of ANSYS package as per the dimensions used by Kaushik shaha et al. [8]. Discretization of model geometry was done with the help of meshing tool in ANSYS. Further adaption of boundaries of nozzle was done in order to get finer mesh elements in critical regions shown in Fig.3. The fuel properties of diesel are listed in Table-1.

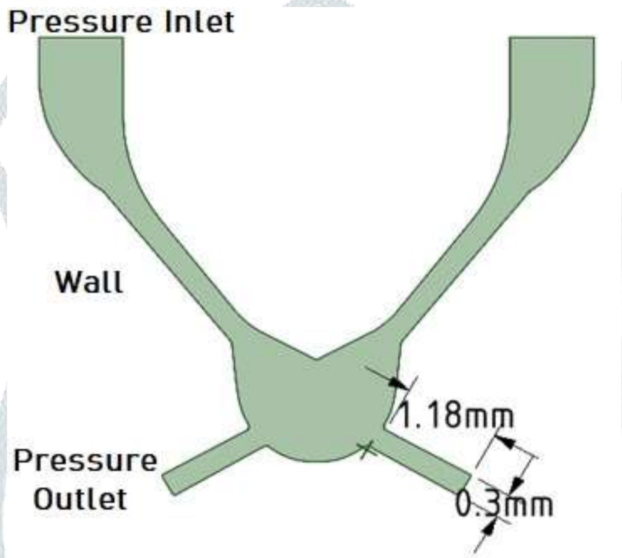

Fig. 2 Nozzle Geometry with its Boundary condition

Table 1 Fuel Properties

Properties	**Diesel**
Liquid Phase Density(kg/m^3)	822.7
Vapour phase Density (kg/m^3)	0.1361
Liquid Phase Viscosity(Pa*s)	0.0025
Vapour Phase Viscosity (Pa*s)	0.00004
Saturation Pressure(Pa)	1000
Bulk Modulus(GPa)	1.5
Molecular Weight(g/mole)	198
Surface Tension (N/m)	0.02

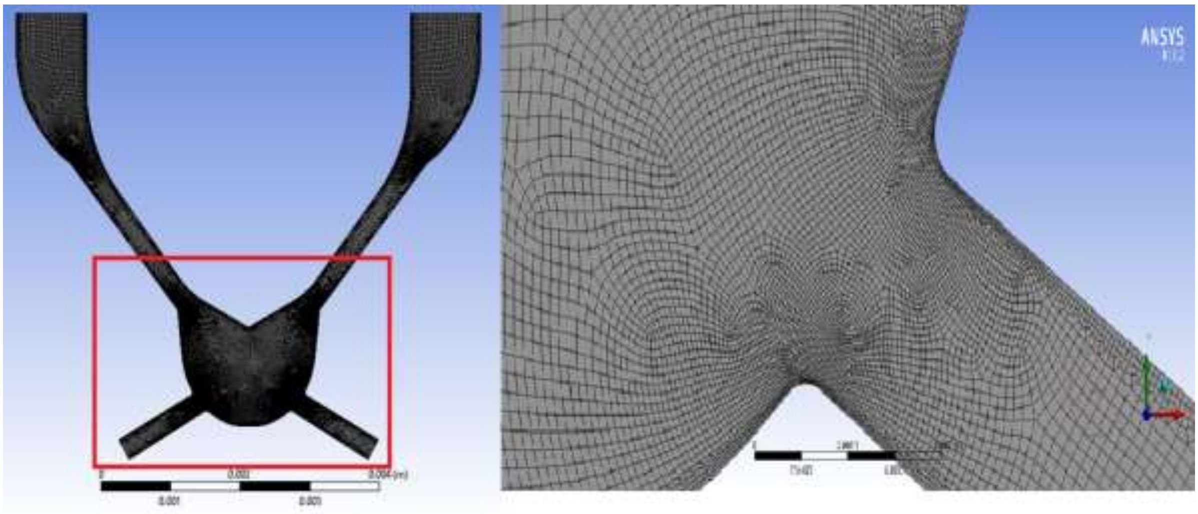

Fig. 3 Meshing with detail view

2.1 Multiphase models and governing equations

As discussed earlier there are two most effective models for analysis of two phase flow namely Mixture model and VOF (Volume of fluid) model. Both these models work on different set of governing equations. Here both methods are used to study two phase flow but basic difference between both these methods is that Mixture model assumes two phase flow with only one liquid whereas VOF model assumes two different fluids, one in liquid phase and one in liquid phase for single liquid itself. After giving its basic difference both models with their governing equations are explained below.

2.1.1 Mixture model approach

The governing equations for a two-phase flow when considered as only single fluid then this is called Mixture Model in ANSYS Fluent [8]. The flow field is then solved for only single fluid i.e. the mixture for continuity and momentum equations,

$$\frac{\partial \rho_m}{\partial t} + \nabla(\rho_m \vec{v}) = 0 \qquad \text{Eq.(2)}$$

$$\frac{\partial \rho_m}{\partial t} + \nabla(\rho_m \, \vec{v}) = -\nabla P + \nabla[\mu_m(\nabla\vec{v} + \nabla\vec{v}^T)] + \rho\vec{g} \qquad \text{Eq.(3)}$$

$$\rho_m = \alpha_v\rho_v + (1 - \alpha_v - \alpha_g)\rho_l + \alpha_g\rho_g$$

$$\mu_m = \alpha_v\mu_v + (1 - \alpha_v - \alpha_g)\mu_l + \alpha_g\mu_g$$

$$\alpha = \frac{n_b * \frac{4}{3} * \pi R_b^3}{1 + n_b * \frac{4}{3} * \pi R_b^3}$$

Where ρ_m is mixture density and μ_m is mixture viscosity and the subscripts *v*, *l* and *g* represent the vapour, liquid, and gas, respectively. Also α is vapour volume fraction, n_b is number of bubble per volume of liquid and $R_b$is bubble's radius.
The vapour mass fraction is the dependent variable in the transport equation. This formulation is given as follow:

$$\frac{\partial (f_v\rho)}{\partial t} + \nabla(f_v\rho\vec{v}_v) = \nabla(\nabla\Gamma f_v) + R_e - R_c \qquad \text{Eq.(4)}$$

Where, v_f is the vapour mass fraction, f_g is the non-condensable gases and Γ is the diffusion coefficient. The rates of mass exchange are given by the following equations:

$$R_e = F_{max}\frac{\max\left(\frac{1}{\sqrt{k}}\right)(1 - f_v - f_g)}{\sigma}\rho_l\rho_v\sqrt{\frac{2(P_v - P)}{g\rho_l}}$$

$$R_c = F_{con}\frac{\max(1/\sqrt{k})f_v}{\sigma}\rho_l\rho_v\sqrt{\frac{2(P_v - P)}{g\rho_l}}$$

2.1.2 VOF model approach

It is an Euler-Lagrangian approach and its accuracy depends upon how well the interface is captured by VOF model and also on the accuracy of the droplet identification algorithm. Algorithm of this model separates out the liquid droplets based on its size and shape, which is the measure of how close, is the shape of the lump to a perfect sphere. This algorithm helps in transferring lump of fluid from Eulerian to Lagrangian field [11]. Main difference of governing equations of VOF model from Mixture model is that here equations of conservation of mass and momentum both are solved for both liquid as well as vapour phase of fluid.
Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0 \qquad \text{Eq.(5)}$$

Reynolds Averaged Navier Stokes Equation:

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \mu\left(\frac{\partial^2 (U_i)}{\partial x_j^2}\right) + \frac{\partial (\rho U_i^{'} U_j^{'})}{\partial x_j} \qquad \text{Eq.(6)}$$

Turbulence equations:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k U_j)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + \rho\varepsilon - Y_M + S_k \qquad \text{Eq.(7)}$$

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial \rho k U_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_e}\right)\frac{\partial k}{\partial x_j}\right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + (\varepsilon\nu)^{0.5}} + S_\varepsilon \qquad \text{Eq.(8)}$$

In above equations [7,8] the terms on right side S_k &S_ε are called source and sink term which indicates the source or sink of mass flow rate of any liquid phase from any source defined by the user.

III. RESULTS AND DISCUSSIONS

3.1 Effect of Mesh Size

To determine optimum numbers of cells in this model of nozzle mesh sensitivity test is performed. For different numbers of elements mass flow rates are calculated. Fig. 4 shows that number of elements can be more then 20000. Further increasing these numbers of elements may improve accuracy but consumes more time.

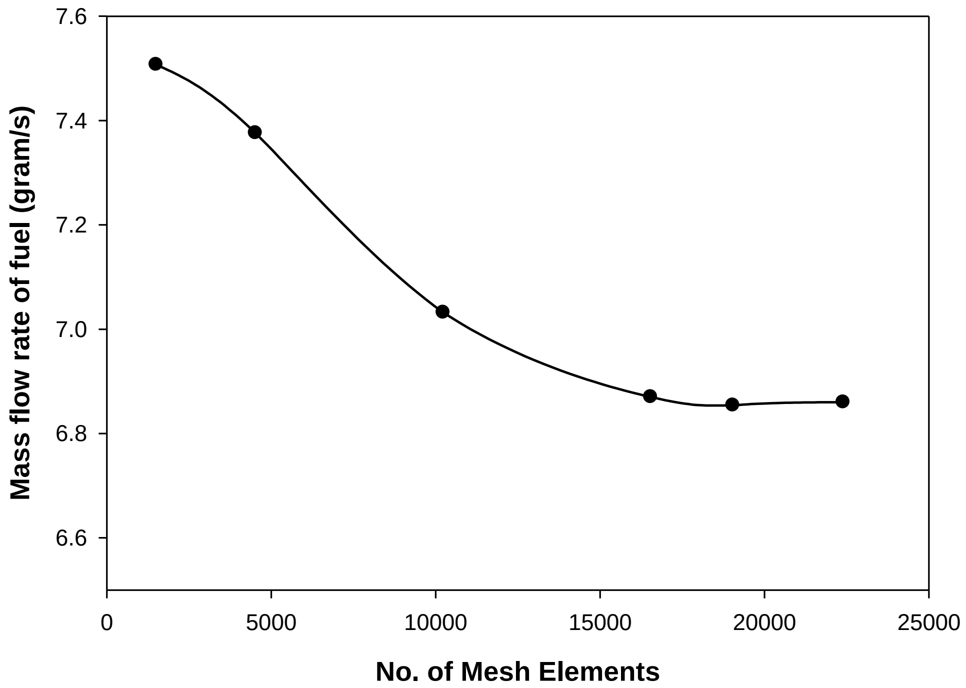

Fig. 4 Mesh sensitivity study

3.2 Model Validation

The reliability of the developed model is assessed by comparing its predictions with the experimental observations of Winklhofer et al. [3]. The "U" nozzle used for comparison has a rectangular cross section with the depth of 0.3 mm, an inlet width of 0.301 mm, an outlet width of 0.284 mm and an inlet rounding radius of 0.02 mm. The nozzle length is 1 mm. Figure 3.3 displays the schematic illustration of the U nozzle. In the set-up of the experimental study [3], the inlet pressure was maintained at 10 MPa (100 bar), and the outlet pressure was varied from 1.5 to 8 MPa. The temperature was fixed at 300 K. Mass flow rate of fuel has been calculated with mixture and VOF multiphase model and compared with experimental results of Winklhofer et al. [3] shown in Fig. 5.

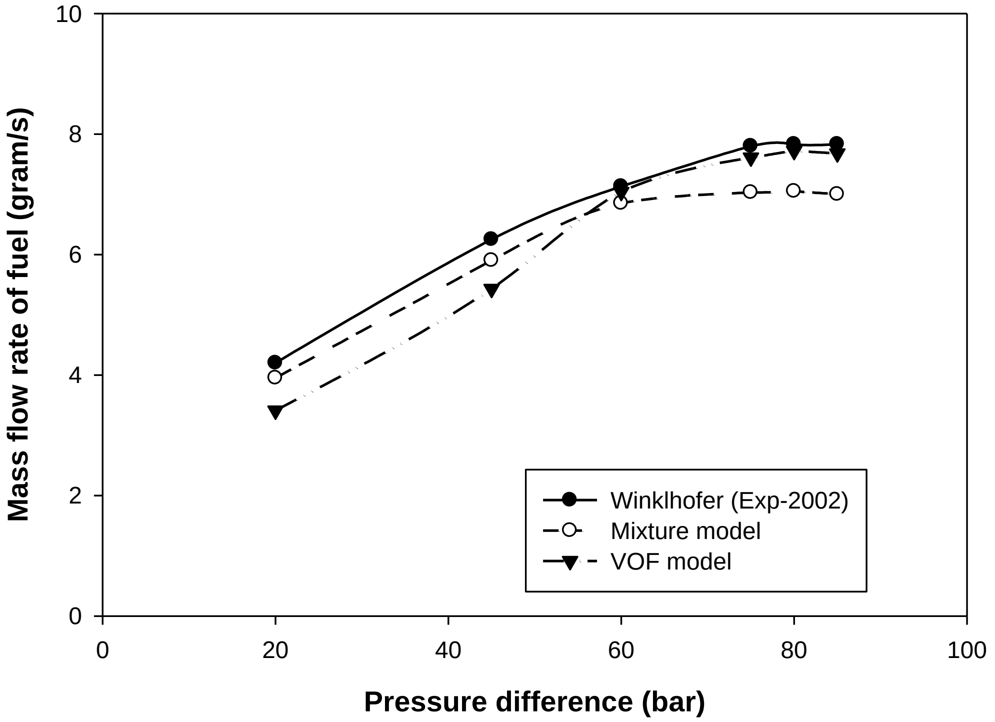

Fig.5 Mass flow rate of fuel with mixture and VOF model at different pressure difference

3.3 Effect of Pressure difference

Velocity distribution

Simulation has been carried out with real size nozzle and mixture and VOF multiphase model is compared on the basis of velocity contour and vapour fraction contour shown in Fig. 6 and Fig.7. It is found that VOF predict early cavitation inception. The magnitude of vapour fraction is observed larger in case of VOF model compared to mixture model.

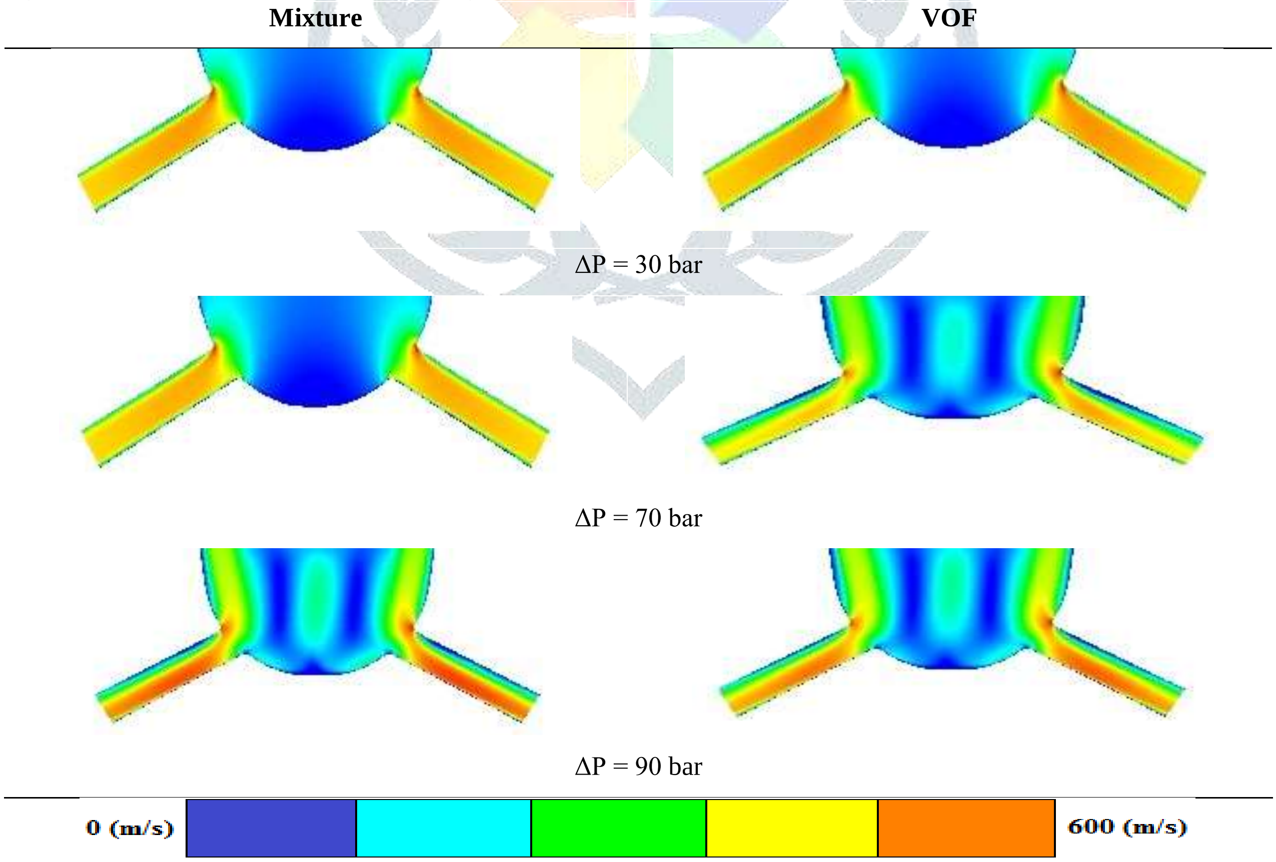

Fig. 6 Velocity contour for mixture and VOF model at different pressure difference

Vapour fraction

Mixture **VOF**

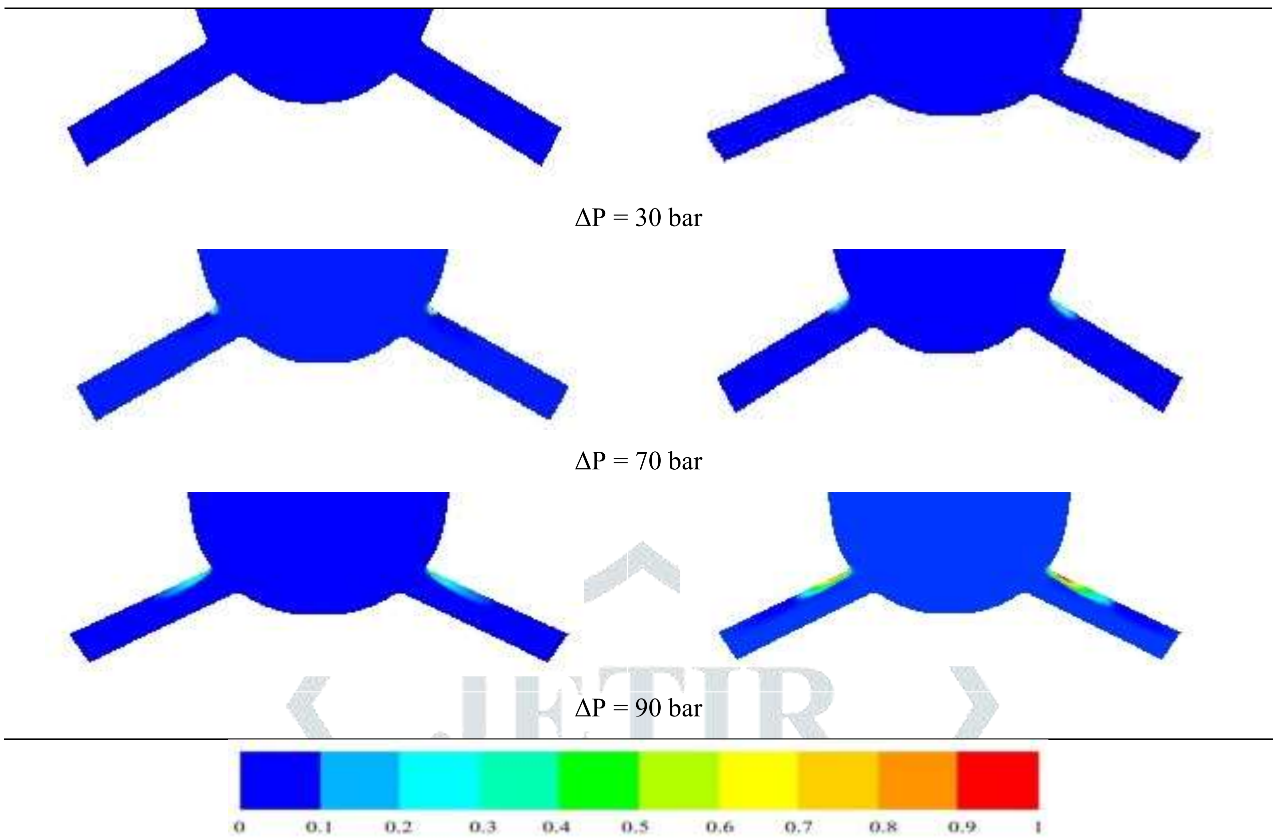

Fig. 7 Vapour fraction contour of mixture and VOF model at different pressure difference

CONCLUSION

- It is observed that VOF as well as mixture multiphase model are equally capable of capturing cavitation phenomena in nozzle.
- The results obtained from simulation are in good agreement with experimental data. It is found that at lower pressure difference mass flow rate calculated with mixture multiphase model have closer values as compared to VOF model. In case of high pressure difference mass flow rate value closely match with VOF model.
- Both model are indicating chocking condition as it was found by Winklhoffer er al [3]
- The inception of cavitation observed at the pressure difference of 60 kPa with VOF model, while mixture model shows inception of cavitation at the pressure difference of 70 kPa.
- However time taken to converge the solution with the use of VOF multiphase model is more than that of mixture model.

IV. ACKNOWLEDGMENT

I had privilege to work under the esteemed guidance of **Assistant Prof. Mehul Bambhania sir**, mechanical engineering department, The M.S. University, Baroda, who visualized this concept and gave invaluable suggestions throughout the process that acted as the weapons against all the obstacles in the successful completion of this paper.

V. REFERENCES

[1] H. Chaves, M. Knapp, A. Kubitzek, F. Obermeier, and T. Schneider, "Experimental Study of Cavitation in the Nozzle Hole of Diesel Injectors Using Transparent Nozzles," SAE Paper No. 950290, 1995.
[2] C. Arcoumanis, M. Gavaises, B. French, Effect of fuel injection process on the structure of diesel sprays, SAE Paper, 1997.
[3] Winklhofer E, Kull E, Kelz E, Morozov A., "Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions", ILASS-EUROPE, Zurich, Switzerland (2001).
[4] S Martynov, "Numerical Simulation of the Cavitation Process in Diesel Fuel Injectors", Ph.D. thesis, The University of Brighton, 2005.
[5] E. Giannadakis, M. Gavaises, H. Roth, and C. Arcoumanis, "Cavitation Modeling in Single-Hole Diesel Injector Based on Eulerian-Lagrangian Approach," Proceedings of the THIESEL International Conference on Thermo- and Fluid Dynamic Process in Diesel Engines, Valencia, Spain, 2004.
[6] D Schmidt, P. Rutland, C. J. Corrandini, P. Roosen, P., and O. Genge, "Cavitation in 2-D Asymmetric Nozzles," SAE Paper No. 1999-01- 0518, 1999.
[7] Som S., Aggarwal S.K., El-Hannouny E. M., Longman D.E., "Investigation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector", article in journal of engineering for gas turbines and power, 2010.

[8] Saha Kaushik, Ehab Abu-Ramadan, Xianguo Li1, “Modified Single-Fluid Cavitation Model for Pure Diesel and Biodiesel Fuels in Direct Injection Fuel Injectors”, University of Waterloo, Waterloo, Canada, 2013.
[9] Michele Battistoni, Sibendu Som, and Douglas E Longman. Comparison of mixture and multifluid models for in-nozzle cavitation prediction. Journal of Engineering for Gas Turbines and Power, 136(6):061506, 2014.
[10] Salvador, F.J, Martínez-Lopez, J, Caballer, M, De Alfonso, “Study of the influence of the needle lift on the internal flow and cavitation phenomenon in diesel injector nozzles by CFD using RANS methods”, Energy Conversion and Management, 246–256, 2013.
[11] Mehul Bambhania, Nikul Patel, “Numerical simulation to predict cavitation inside diesel engine fuel injector nozzle”, International conference on Multidisciplinary Research Approach for the accomplishment of academic excellence in Higher technical education through industrial practices, June 2016,Pattaya-Bangkok.