A Numerical Study on Cavitation in Diesel Fuel Injector

Dr. E.S. Prakash[1], Yankana Gouda T[2], Raghavendra H[3], Ravi K[4]
[1, 2, 3, 4] Department of Mechanical Engineering
[1] UBDT college of Engineering Davanagere
[2] Talla Padmavathi college of Engineering Somidi, Kazipet, Warangal
[3, 4] MITS Madanapalle, AP

Abstract- *In diesel engines, the internal flow characteristics in the fuel injection nozzles, such as the turbulence level and distribution, the cavitation pattern and the velocity profile affect significantly the air-fuel mixture in the spray and subsequently the combustion process. Since the possibility to observe experimentally and measure the flow inside real size diesel injectors is very limited, Computational Fluid Dynamics (CFD) calculations are generally used to obtain the relevant information.*

The present work is focused on the study of cavitation in real size automotive injectors by using a commercial CFD code. The first objective of the current work is to assess the ability of the cavitation model included in the CFD code ICEM CFD to predict cavitating flow conditions. For this, the model is validated for an injector-like study case defined in the literature, and for which experimental data is available in different operating conditions, pre-cavitation and after the pro-cavitation. Preliminary studies are performed to analyze the effects on the solution obtained for various numerical parameters of the cavitation model itself and of the solver ANSYS CFX, and to determine the adequate setup of the model. From this work concluded that overall the cavitation model is able to predict the onset and development of cavitation accurately. There is satisfactory agreement between the choked flow conditions and the corresponding numerical solution. This study serves as the basis for the physical and numerical understanding of the problem.

I. INTRODUCTION

1.1 DIESEL ENGINE INJECTOR

Combustion engines are used as an essential source of mechanical energy. In modern diesel engines, the combustion efficiency is strongly correlated to the spray behavior. In turns, as showed by many studies the main spray characteristics (atomization, penetration, and droplet diameter distribution) can be strongly correlated to injector operating conditions and injector geometric layout. A good understanding of the dynamics of internal nozzle flow has major importance due to its strong effects on spray and atomization characteristics.

Fuel spray atomization strongly affects diesel engine performance and emissions, which in turn is significantly influenced by transient cavitation behaviour inside an injector nozzle. The characteristics of a fuel spray depend on the fuel properties, geometry of the injector and the flow conditions upstream (inside) and downstream of the injection nozzle. Two main types of diesel injectors are used in direct injection systems:

Valve covered orifice (VCO) injectors and sac-type injectors (Figure 1.1
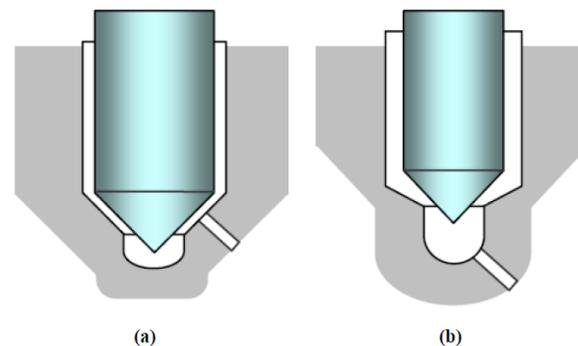

Figure 1.1 Two types of diesel injectors: valve-covered orifice (VCO) injector (a) and sac-type injector (b).

In direct injection diesel engines equipped with a common rail system high injection pressures are used to enhance the spray atomization and air-fuel formation in the combustion chamber. The more homogeneous the air-fuel mixture is, the shorter the ignition delay, the local temperatures are lower and CO, NOx and soot emissions are lower. As a result, the efficiency of performance of the engine is much affected by the fuel atomization [1]. However, observing the flow in diesel injectors is very difficult as they are characterized by high pressure injections through very small nozzles with diameters of around 100 microns or less. In addition, a decrease in pressure below a critical level leads to cavitation, which adds another level of complexity to the

problem. In most cases cavitation is undesirable since it causes performance loss, material damage, vibrations and noise. On the contrary, in engine fuel injection systems, and particularly in diesel injectors, it is expected that nozzle cavitation enhances jet turbulence, which in turn promotes fuel atomization. It is therefore of great importance to study the cavitation issues and to find out how it is correlated to the flow characteristics in the combustion chamber.
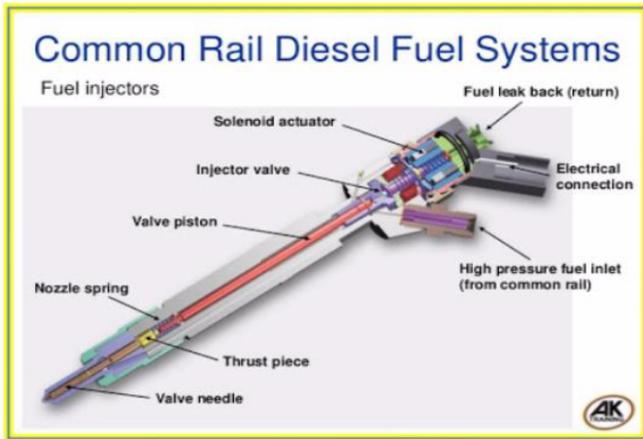

Figure 1.2 Common rail diesel fuel injection systems

1.2 CAVITATION

Cavitation is the formation of vapor bubbles of a flowing liquid in a region where the pressure of the liquid falls below its saturated vapor pressure. The formation of cavitation in diesel injector nozzles is primary ascribed of breakdown of non-equilibrium of liquid dynamics and the separation of boundary layers due to the geometric characteristics of nozzles. Cavitation is liable to occur when the fuel passes through the nozzle.

1.2.1 Theoretical background

Cavitation bubbles arise because of low static pressure that occurs near a sharp inlet corner in the nozzle flow. If the corner of the inlet is sufficiently sharp, the flow tends to separate and forms contraction (vena contract) inside the nozzle, which reduces the area through which the liquid flows. This reduced area is accompanied by an increase in velocity, as predicted by conservation of mass. Conservation of momentum predicts that the acceleration of the liquid through the vena contract causes a pressure depression in the throat of the nozzle. The low pressure inside the throat of nozzle may fall below the vapor pressure of the liquid, causing cavitation. A simple sketch of this flow is presented in figure 1.3. The formation of the bubbles is sensitive to the geometry of the corner and to any imperfection in the nozzle shape (i.e. roughness). For example in SAC and VCO nozzles is the

vapor mainly present in the holes with higher inclination angles. Cavitation is also very sensitive for the quality of the liquid (fuel).
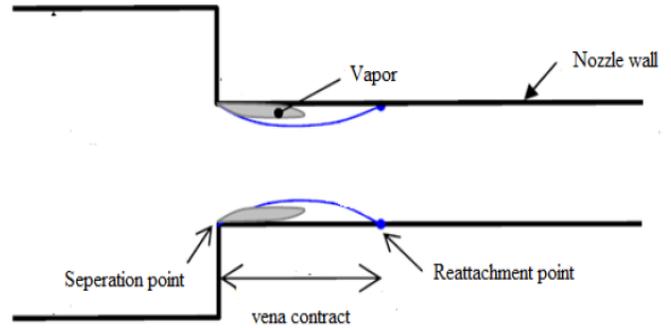

Figure 1.3 Sketch of the geometry-induced cavitation

Geometry-induced cavitation in nozzles occurs when the pressure drop at the nozzle entrance is accompanied by flow reattachment and formation of a recirculation region (Fig. 1.3). The presence of the recirculation region in the flow determines the distribution of tensions, and therefore has a crucial effect on the formation and development of cavitation. However, as will be shown later, flow reattachment may not happen in flows at low Reynolds numbers and in nozzles with a smooth entry configuration. Therefore, specific features of the liquid flow and its interaction with the growth and collapse of cavitation structures should be carefully achieved when developing a model of cavitation flow (see discussion of the bubble dynamics scale effects in reference [9]). Below graph represents that vapor-pressure graph of water as a fluid. At a particular temperature if the water pressure reduced then water turns to vapor. Diesel fuel also acts like water.
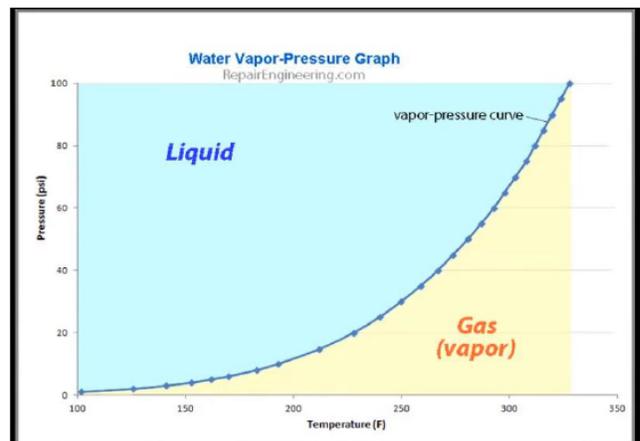

Figure 1.4 Water vapor-pressure graph [RepairEngg.com]

1.2.2 Types of cavitation

A method efficiently producing cavities can be taken as the main criterion in distinguishing among different types of cavitation.

The four principle types of cavitation and their causes can be summarised as follows
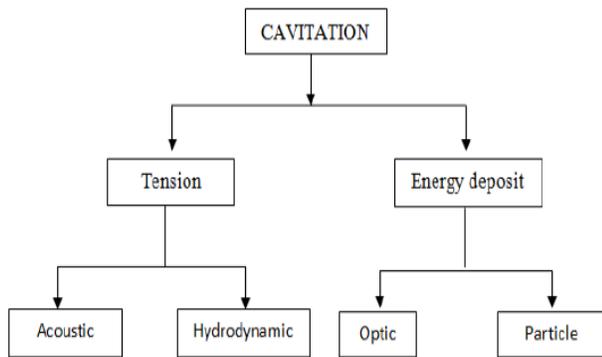

Fig.1.5 Schematic diagram of types of cavitation

a) Acoustic cavitation: In this case, the pressure variation in the liquid are effected using sound waves usually ultrasound (16 kHz to 100 kHz).the chemical changes taking place due to cavitation induced by passage of sound waves are commonly known as sonochemistry.

b) Hydrodynamic cavitation: Cavitation produced by pressure variations, which is obtained using geometry of the system creating velocity variation. For example based on geometry of the system, interchange of pressure and kinetic energy can be achieved resulting in the generation of cavities as in the case of flow through orifice, venturi and other cases.

c) Optic cavitation: It is produced by photons of high intensity light (laser) rupturing the liquid continuum.

d) Particle cavitation: It is produced by beam of elementary particles, example a photon rupturing a liquid, in the case of bubble chamber.

1.2.3 Stages of cavitation in nozzle

The types of cavitation observed in nozzle of three main stages: inception cavitation, super cavitation and hydraulic flip. The first one describes the state when the cavitation first appears, the second describes the situation where there is a strong cavitation zone close to the nozzle outlet and the last describes the situation where the liquid jet completely separates from nozzle wall.

1.2.4 Occurrence of cavitation

Cavitation phenomenon encountered mainly in liquid pumps and turbines, injection nozzles, throttles, pipes and channels with obstacles, ship propellers. It is also encountered in biology and even in the surrounding nature.

II. OBJECTIVE AND SCOPE OF THE PROJECT

2.1 OBJECTIVES

a.   The present study is to investigate the internal flow and cavitation phenomenon inside real size automotive injectors and to examine its effect on the nozzle exit flow.

b.   Computational Fluid Dynamics (CFD) calculations represent a useful tool to provide three dimensional analyses inside the nozzle.

c.   The current study also presents results of a computational fluid dynamics model to predict cavitation.

d.   Cavitation model implemented in commercial CFD code ANSYS ICEM CFD

e.   Numerical predictions were performed on a throttle channel at different operating conditions with cavitation.

f.   The present study intends to provide physical information, such as flow distribution, discharge coefficient, turbulence quantities, vapour fraction, and velocity distributions at the nozzle exit, which can subsequently be used in spray modelling.

2.2 SCOPE OF PRESENT STUDY

The present study is focused on the development of a model of cavitation, which is capable of predicting the hydrodynamic similarity of cavitation flows in nozzles. First, types of cavitation flows and factors responsible for the hydrodynamic similarity (scale effects) are identified from analysis of the experimental data. Then, an appropriate frame-work for cavitation modeling is chosen and additional assumptions and simplifications about the flow are made. Further, the analysis of cavitation flow models is performed and the problem in specification of the cavitation scale, which describes the liquid quality effect, is clarified. To develop a scalable model of cavitation, the governing equations are rearranged into dimensionless form and the cavitation length scale is shown to be linked to the number density of cavitation nuclei. This results in the development of a model for the number density of critical cavitation nuclei as a function of liquid tension in the flow.

COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and

gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. On going research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests. The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

III. METHODOLOGY

All established CFD software contain three elements
(i) Pre-processor
(ii) Main solver, and
(iii) Post-processor

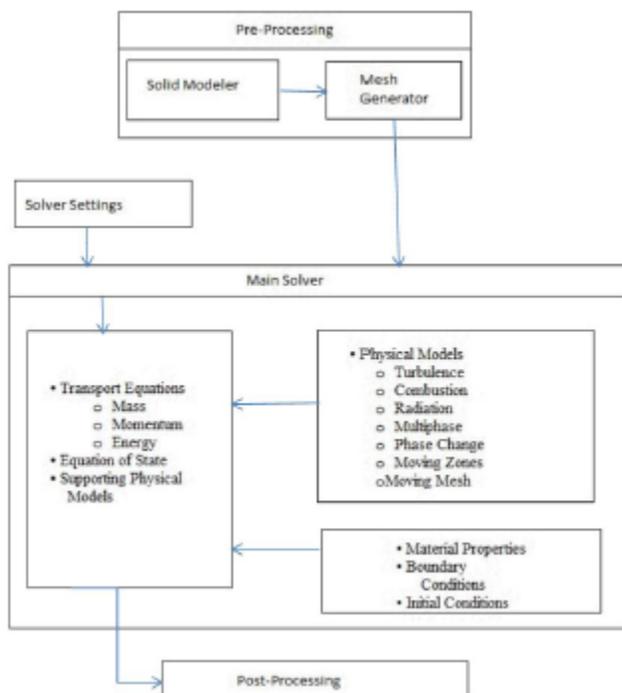These three elements can be explained with help of schematic diagram as following.

Figure 3.1 Schematic diagram of CFD procedure

IV. CAVITATION MODEL WITH AN ACADEMIC STUDY

This section discuss about the basic steps involved in studying the computational fluid dynamic software such as preliminary analysis, ANSYS ICEM CFD, ANSYS CFX, k-turbulence model etc. The detailed process of solving the FV-model is also studied i.e. using solver (ANSYS CFX). The detailed view the governing equations used.

4.1 ANSYS ICEM CFD

ANSYS ICEM CFD is a popularly developed software package used for creating the fluid transfer component as CAD geometry and to generate the mesh for it. The different types of cell geometries are developed for the structured, un-structured, multi-block and hybrid grids. This software is mainly meant for creating the mesh for the already dedicated CAD packages like CATIA, AUTOCAD etc. by directly importing the geometry. As a consequence, the geometry model is primarily meant for clean-up before importing it. Nevertheless, there exists some powerful geometry creation, repair and editing tools are available in the ANSYS ICEM CFD. The powerful tools available in the package assure at arriving the meshing stage very quickly. ICEM CFD treats the collection of surfaces as a whole volume of close body unlike the GAMBIT. Therefore, the topological issues encountered with GAMBIT (i.e. face cannot be deleted since it is referred by higher topology) don't show up here. ICEM CFD creates the body within water-tight geometry so that there shouldn't be any hole on the surfaces to leak. Apart from the simple tools like points, curves, surfaces etc. ICEM CFD has a topology checking tool in order to check the existence of any hole on the surfaces and also to provide the proper connectivity. It is relatively important to once check the connectivity for mesh generation on the area of interest. Meeting the requirement for integrated mesh generation and post processing tools for today's sophisticated analysis, ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools.

ANSYS ICEM CFD's mesh generation tools offer the capability to parametrically create meshes from geometry in numerous formats:
- Multiblock structured
- Unstructured hexahedral
- Unstructured tetrahedral
- Cartesian with H-grid refinement
- Hybrid Meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements
- Quadrilateral and triangular surface meshes

4.1.1 Computational domain

For the validation and parametric studies performed within this work, the injector-like throttle channel geometry of reference was considered. Since the geometry considered is symmetric

(Figure 4.1), only half of the geometry was calculated by imposing symmetry boundary conditions along the axis boundary. Constant pressure boundary conditions were set at the inlet and outlet and a no slip boundary condition was used at the wall.
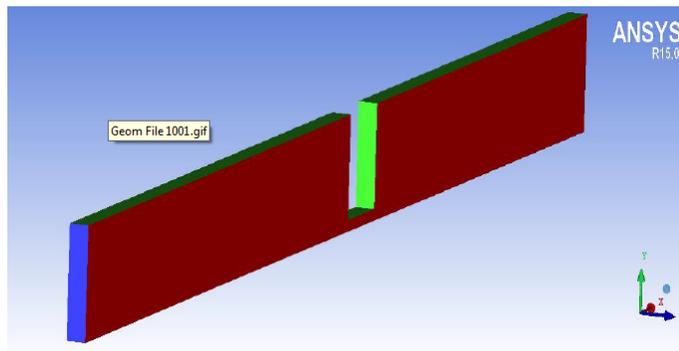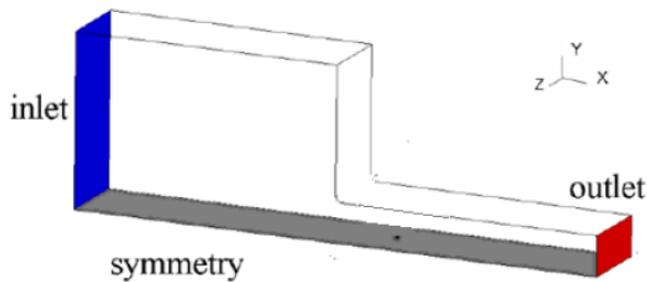

Figure 4.2 Finite volume model of rectangular nozzle configurations

Although the size of the channel is substantially larger than current Diesel injector orifices, this geometry is used as a reference in the literature [11] and can therefore be used to validate the model. The cross-section of the nozzle is almost square as shown in figure 5.1. The Geometrical values are shown below table 5.1. The results of a CFD calculation are known to be affected by the resolution of the computational mesh, especially in the regions of high gradients. In order to verify the grid independence of the solution, different adaptive refinements were performed.

Table 4.1 Geometry values of rectangular nozzle

Parameter	Values in mm
Total Length of Geometry (L)	20
Inlet Height (H)	1.5
inlet width (W)	0.3
Throat length (l)	1
Throat Height (h)	0.15
Length before throat	10
Length after throat	9

Physical properties fuel used

The equilibrium thermo-physical properties of a liquid depend on its content, temperature and pressure conditions. The working liquid also can make a solution with other liquids and gases, which affects the saturation pressure. Diesel fuel is a multi-component mixture of liquids, with different boiling points. This results is evaporation starting from the more volatile components present in the mixture. Density, viscosity, thermal conductivity, surface tension and latent heat of evaporation are varied for different type of diesel fuels. Most experiments on cavitation are performed using the light diesel fuel, which properties are listed in the Table.

Table 4.2 Physical properties of diesel fuel (200C, 1 bar)

Liquid properties	Liquid density, kg/m^3	Dynamic viscosity, Pa.sec	Thermal conductivity, W/m.K	Heat capacity, J/kg.K	Surface tension, N/m	Latent heat of evaporation, J/kg	Vapour pressure, bar
Diesel Fuel	840	0.003	0.11	1800	0.028	3.8x10^6	0.03

4.1.2 Meshing

Meshing of the model is done to improve the accuracy of the model. The accuracy of the model is checked by the histograms generated in the software. Aspect ratio is the one of the key parameter to be checked for good accuracy and it should be less than 1.

Types of meshing

Meshing mainly has the following types:
a) Tetra meshing

b) Hexagonal meshing
c) Prism meshing
d) Hybrid meshing

Tetra meshing is used for the surface mesh of the three dimensional model consisting of unstructured/structured cells. While, hexagonal meshing is mostly preferred for the two dimensional model because in this type there is need to generate blocking. In three dimensional models, it is difficult to create blocking because of complexity of the model. Prism mesh is used to generate the volume mesh at the boundary of the model.

Tetra Meshing Method

Initially the workspace is checked for topology errors of 0.06 mm of tolerance value and global scale factor of 1.0. A domain having the finned cylinder is meshed at its wall and the space available for the air flow in between the domain and a cylinder. The boundaries and edges of both domain and cylinder is subjected for fine surface mesh size of 2.0 with mesh scale factor of 0.1 and then complete mesh size of 16 with mesh scale factor of 1.0. The mesh size adopted should be 1/10th of the total length of the cylinder and should be in the factorial of 2. The mesh scale factor depends on the requirement of fine or rough surface mesh. Another important factor to be considered is the spacing between the two elements; it must be very small at the surface boundary.
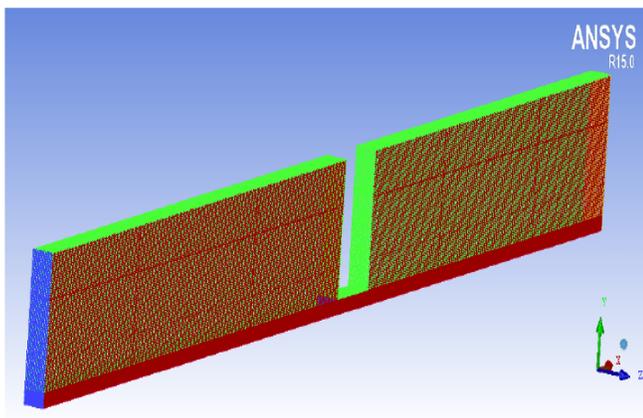

Figure 4.3 Visualization of mesh distribution

The size of the flow domain depends on the earlier studies and is in the multiple of 5 of the length of the rectangular nozzle. A rectangular square domain of 20 mm total length is created around the rectangular nozzle is shown in fig. 5.3. A finite volume and density mesh is generated using unstructured tetrahedral cells at the periphery of the geometry and in the area surrounding to it. Structured tetrahedral cells are used define the remaining fluid domain

.

Table 4.4 Domain physics report

Domain – Fluid domain	
Type	Fluid
Location	FLUID
Materials	
Diesel liquid	
Fluid definition	Material library
Morphology	Continuous fluid
Diesel vapor	
Fluid definition	Material library
Morphology	Continuous fluid
Settings	
Buoyancy model	Non buoyant
Domain motion	Stationary
Reference pressure	1.00 Pa
Heat transfer model	Isothermal
Fluid temperature	25^0C
Homogeneous model	True
Turbulence model	k-epsilon
Turbulent wall functions	Scalable

V. RESULTS AND DISCUSSIONS

A two-dimensional model is developed to study cavitation in diesel fuel injector for electronic packaging applications. A series of numerical calculations have been conducted by ANSYS CFX and the results are presented in order to show the effects of back pressure and discharge co-efficient, cavitation number. Analysis has been carried out at nozzle outlet pressure 30 bar, 40 bar, 50 bar and 60 bar.

ANSYS CFX

ANSYS CFX is used to simulate the fluid flow in range of applications and it is a conventional computational fluid dynamics program used as the post-processing tool in the analysis fluid flow techniques through the contours, plots and velocity vectors. The accessible program has been useful to the simulation of liquid or gas flowing past ship hulls, aircraft aerodynamics, gas turbine engines (including compressors, turbines, combustion chambers and afterburners), pumps, fans, HVAC systems, vessels, cylinders, vacuum cleaners and many more. Various general-purpose CFD packages have been published in the past decade. Important Among them are: PHOENICS, FLUENT, STAR-CD, CFX, CFD-ACE, ANSWER, CFD++, FLOW-3D and COMPACT. Generally all these packages are based on the finite volume method.But ANSYS CFX is just powerful computational fluid dynamics

program provides a superior bi-directional connection between the CAD geometry system, improved mesh generating tools in ANSYS meshing and powerful geometry creating and editing tools.

ANSYS CFX is a commercial Computational Fluid Dynamics (CFD) program, used to simulate fluid flow in a variety of applications. The ANSYS CFX product allows engineers to test systems in a virtual environment. The scalable program has been applied to the simulation of water flowing past ship hulls, gas turbine engines (including the compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydrocyclones, vacuum cleaners, and more. ANSYS CFX software has its roots in the programs CFX-TASC flow and CFX-4. CFX-4 was formerly Flow3D in the United Kingdom and originally developed in-house for use by the United Kingdom Atomic Energy Authority (UKAEA), and TASC flow which was developed by Advanced Scientific Computing (ASC), of Waterloo, Ontario, Canada. Measurements of the injection rate and momentum flux during the whole injection process for nozzle type injectors at various injection pressure conditions (see Table 3.2). These conditions presented in these tables were also simulated with the CFD code. The injection pressure (p1) is at 100 bar, while the back pressure ranges from 30 to 60 bars.

Table 5.1 Parameters obtaining at different outlet pressure by CFD simulation

Inlet pressure(p_1) in bar	Outlet Pressure(p_2) in bar	Diesel Vapour volume(V_a) in m^3	Discharge coefficient (C_d)	Reynolds number (Re)	cavitation Number (K)
100	30	1.12×10^{-07}	0.748	39908	1.47
100	40	2.77×10^{-13}	0.807	39167	1.719
100	50	1.72×10^{-13}	0.807	35742	2.064
100	60	1.61×10^{-15}	0.807	31945	2.58

Critical cavitation

From the numerical simulation values, as shown in the table, for p2 = 40 bar, the mass flow rate increases proportionally with the root of pressure differential until the point where it stabilizes. At this point, what is called ''choked flow'' occurs, and pressure conditions needed to arrive to this situation are named critical cavitation conditions.

The mass flow first follows the rising pressure drop as outlet pressure is increased from 30 bar to 60 bar. With the onset and growth of cavitation, however, the mass flow approaches a constant level and is insensitive to further reduction of outlet pressure (Fig. 2). This transition from pressure dependent mass flow to choked mass flow defines the

"critical cavitation" (CC) point. For practical purposes, the critical cavitation pressure (CCP) is defined as the pressure drop where the mass flow along the hydraulic line approaches 99% of the choked mass flow.

5.1 PLOTS

Variation Characteristics of Dimensionless Parameters:

Figures 3 shows the variations of discharge coefficient with the injection pressure of diesel fuel in nozzle. It can be seen from Figure 3 that discharge coefficient increases with increasing in back pressure before cavitation occurs. When the cavitation occurs, the discharge coefficients of diesel fuels increase continuously and reach its maximum value of 0.807, at the point of transition cavitation. After the transition cavitation, they both decrease with injection increase and have the similar tendency.
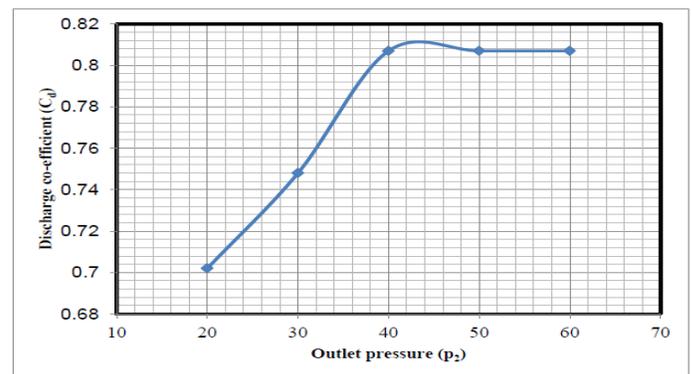

Figure 5.1 Relation between outlet pressure and discharge co-efficient

Cavitation in a diesel nozzle appears at the inlet of the nozzle hole. In this region, and due to the strong change in cross-section and flow direction, the boundary layer tends to separate from the hole wall and a so-called ''vena contracta'' is established. As a consequence, a recirculation zone appears between the ''vena contracta'' and the orifice wall. In this zone there is a pressure depression due to the acceleration of the fluid. If the static pressure falls below vapour pressure then the phenomenon of cavitation will arises. A simplified sketch of this situation can be observed in Fig. 4. Due to the complexity of theproblem, it is difficult to choose the proper parameters to characterize the flow. The usual measured pressures are the injection pressure p1 and the discharge backpressure p2. These two values will be used for calculation and analysis purposes. The pressure drop (p) across the orifice will be calculated by subtracting p2 from p1. This statement implies that the loss coefficients used throughout the analysis will be representative of all losses between the injector inlet and the discharge chamber.
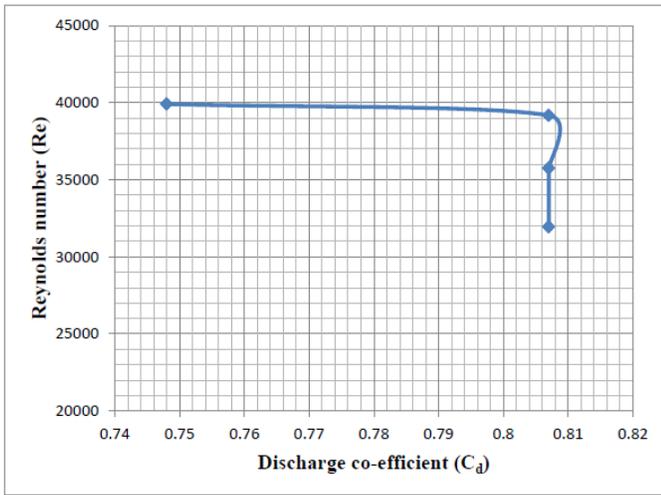
Figure 5.3 Relation between discharge co-efficient and Reynolds number

The Reynolds number strongly changes according to the temperature and pressure, given its strong dependency on the viscosity. In this case, the variations in temperature have been taken into account and have been controlled and registered in the interior of the test bench during the experiments.

The Reynolds number is related to the generation and evolution of the turbulence in the flow, and evaluates the importance of the inertia forces as opposed to those of the fuel viscosity. The variation law for the discharge coefficient with the Reynolds number is an indicator of the type of flow regime the fluid is experiencing. In the zone where the above-mentioned coefficient strongly depends on the Reynolds number the flow regime is considered laminar. It is considered turbulent when the coefficient passes the point of independence on the Reynolds number [7]. This figure 6.3 shows that the discharge coefficient generally increases with the increase of Reynolds numbers, and have a maximum value at a Reynolds number of around 30,000. Beyond this point, the discharge coefficient remains almost constant.
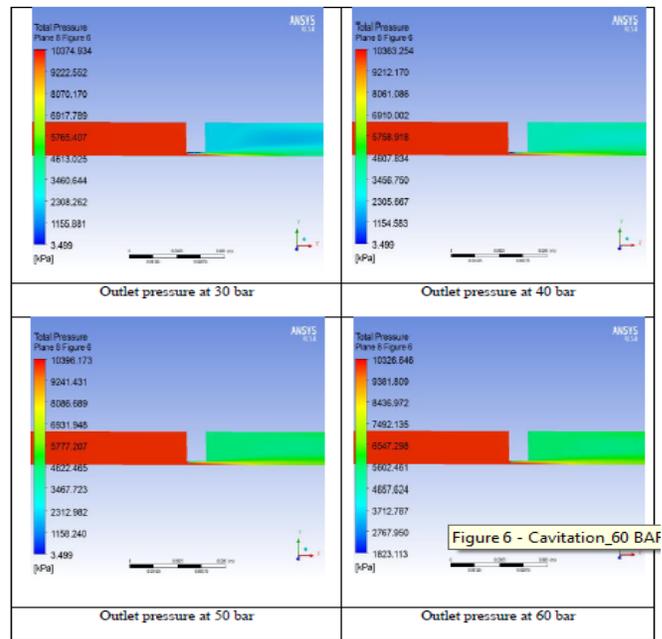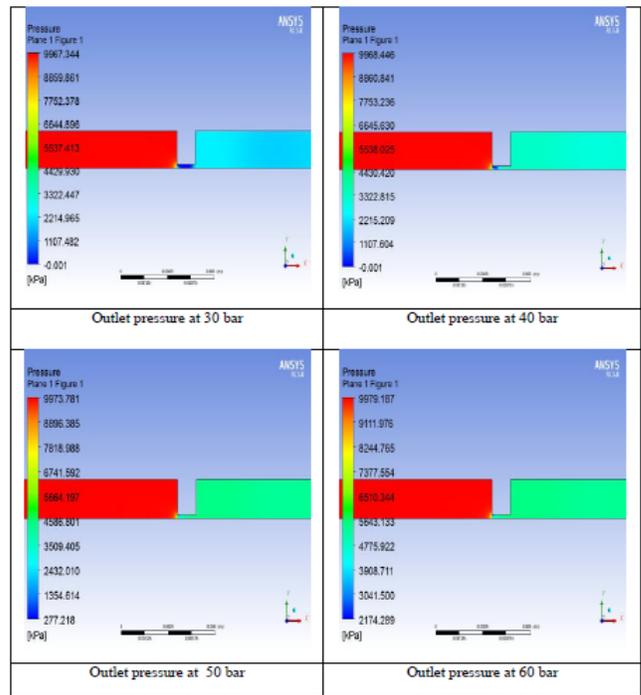

5.2.2 Liquid and vapour velocity contour

The velocity in the throttle entrance regime should obviously be affected by the flow separation and the presence of gas between the fluid and the channel wall. Velocity profiles are shown in Figure 6.6. The comparison of the velocity coefficients reveals that the model can capture the increase in mean velocity caused by the onset and development of cavitation. This velocity behaviour was observed previously. The cavitation develops mostly in the upper part of the nozzle and follows the vortical pattern

imposed by the flow velocity as evidenced in the density and velocity profiles. Low density peaks are observed at locations where there is maximum velocity, indicating that vapour bubbles are entrained by the swirling flow.
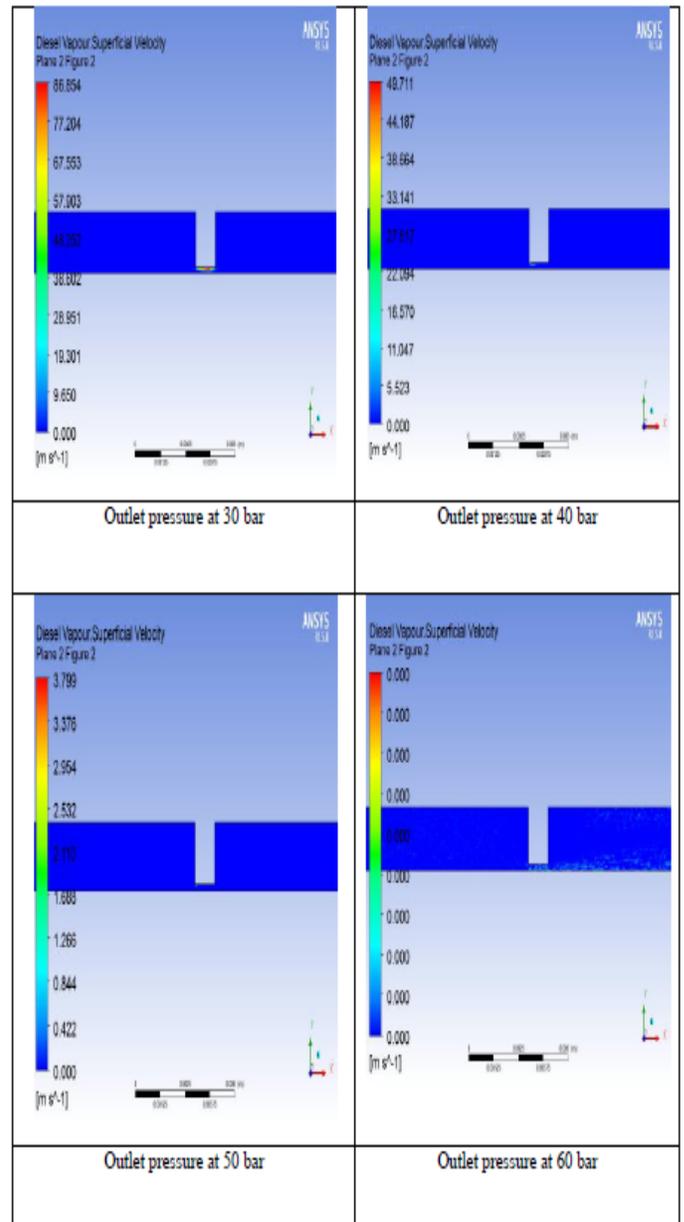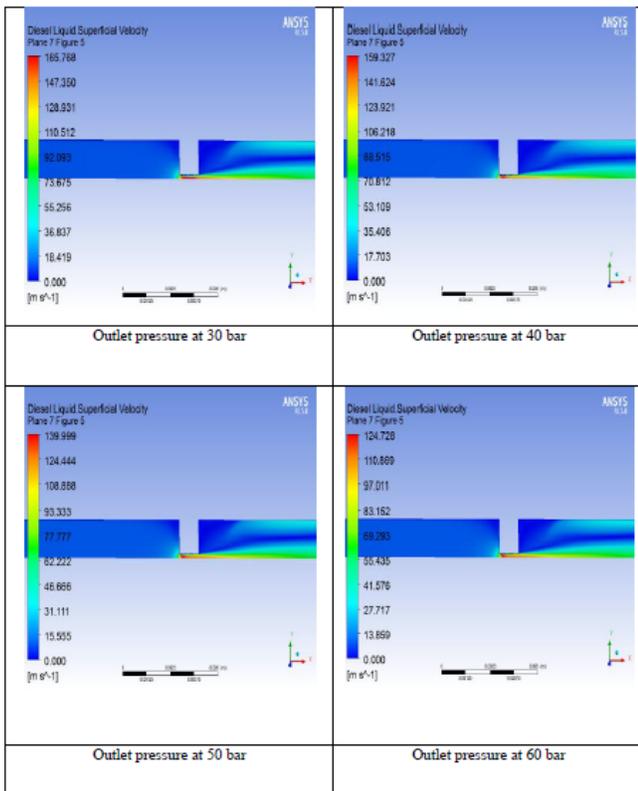
Liquid velocity

Because of vena contract region in nozzle which reduces the area through which liquid flows. This reduced area is accompanied by increase in velocity as predicted by conservation of mass. Figure 6.6 shows that diesel liquid velocity distribution along the length of the nozzle. The liquid velocity distribution increases with decreasing pressure difference at exit of throttle.

Vapour velocity

The diesel vapour velocity decreases with decrease cavitation number in throttle as shown in figure . At the entrance of the throat vapour formation decreases as increases in the outlet pressure from 30 to 50 bar. In between 40 to 50 bar attains choked flow condition.

Velocity vectors contour

Similarly the outlet pressure increases displacement diesel velocity also increases as shown in the figure.

Outlet pressure at 30 bar

Outlet pressure at 40 bar

Outlet pressure at 50 bar

Outlet pressure at 60 bar

Outlet pressure at 30 bar

Outlet pressure at 40 bar

Outlet pressure at 50 bar

Outlet pressure at 60 bar

5.2.3 Liquid and vapour volume fraction contour

Vapor and liquid volume fraction distribution in simplified nozzle is presented on figure. Considering the observation of the results with respect to the colors, the same explanation as at the model can be used. It can be stated that outlet pressure increase from 30 to 60 bar respectively diesel liquid volume also increases.The formation of cavitation in the recirculation region just downstream of the sharp edge corner. Then the cavitation remains attached at the corner, and develops downstream within the boundary layer.

Liquid volume fraction

Because of vena contract region in nozzle which reduces the area through which liquid flows. This reduced area is accompanied by increase in velocity as predicted by conservation of mass. Figure. shows that diesel liquid fraction distribution along the length of the nozzle. The liquid fraction distribution increases with decreasing pressure difference at exit of throttle

Vapour volume fraction

From figure concluded that vapour volume fraction reduces with increase in back pressure at the exit of throttle.
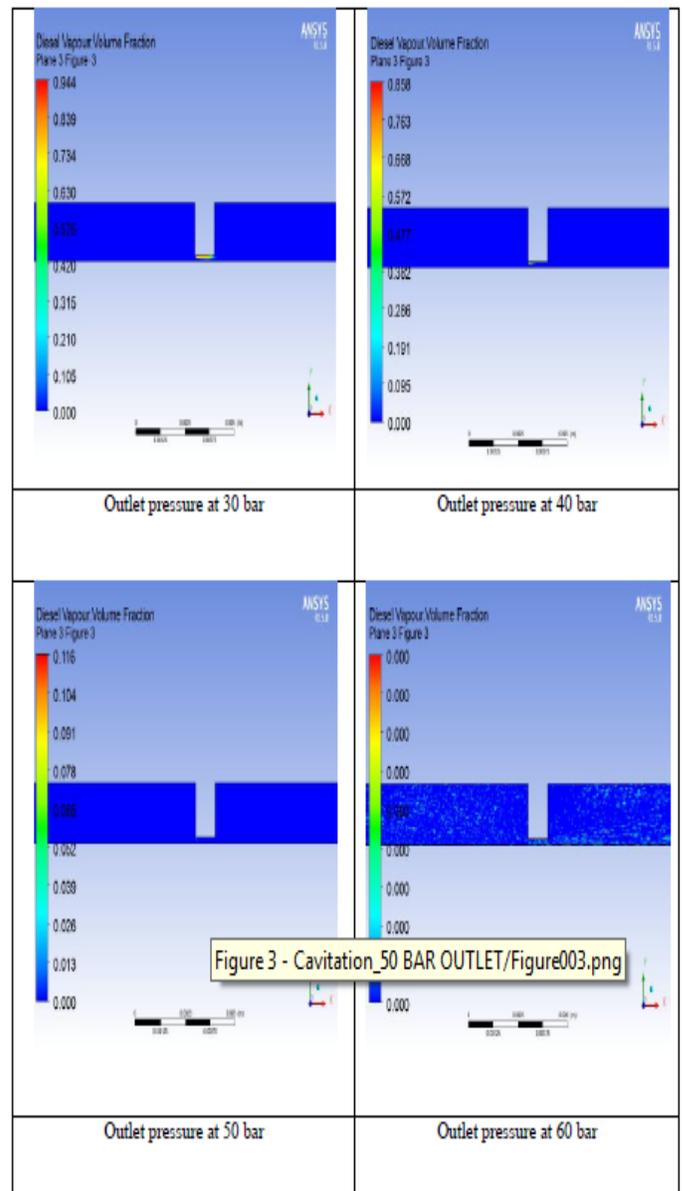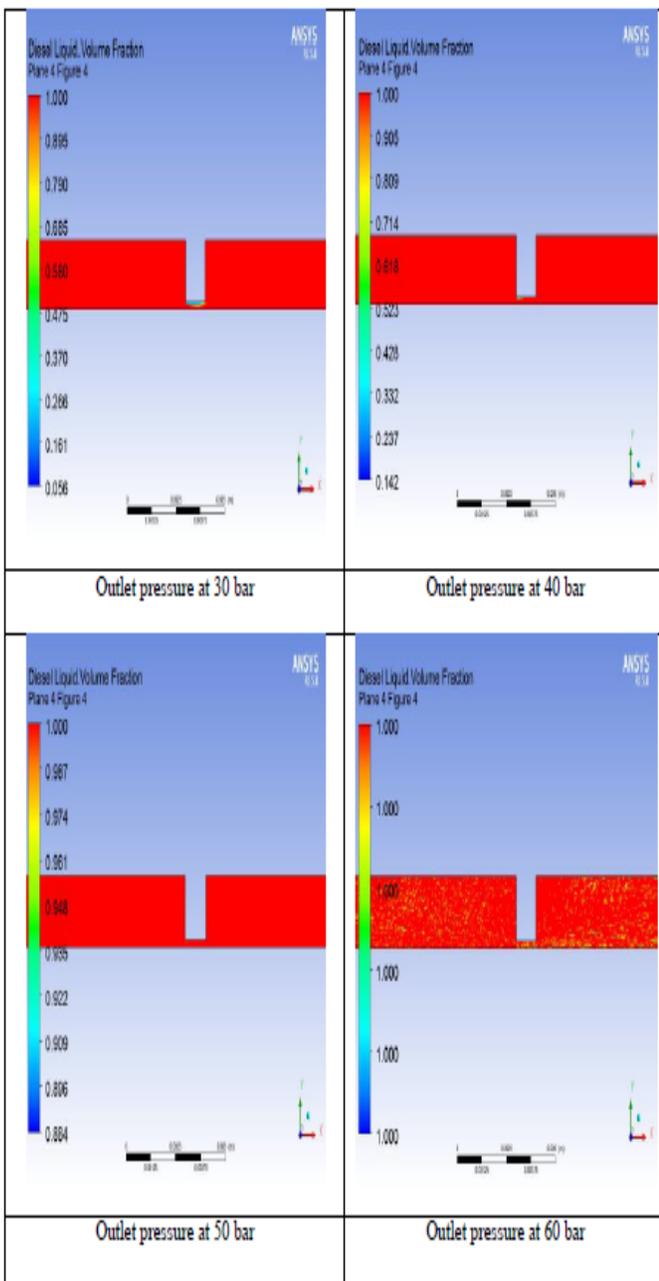

Figure 5.4 Diesel vapour volume fraction variations along the length

5.2.4 Liquid viscosity contour

The prediction may be linked to the uncertainties in the values of liquid viscosity. Indeed, it was found that the effect of liquid viscosity can have a significant influence on the amount of cavitation. Considering the lack of experimental values for viscosity, calculations were made with fuel properties found in the literature in order to check the effect of viscosity. It was observed that there was less generation of vapor with higher liquid viscosity. Higher viscosity leads to lower dynamic pressure and thus yields higher absolute pressure values and less cavitation in accordance with. From the figure 6.10 and 6.11 diesel liquid viscosity distribution parameter influence on cavitation can be predicted. As the outlet pressure increases vapor formation reduces and

increases the liquid viscosity. Studying and justifying the increase of injection velocity induced by cavitation. This velocity rise can be justified due to the reduction of viscosity as a consequence of the presence of vapor.
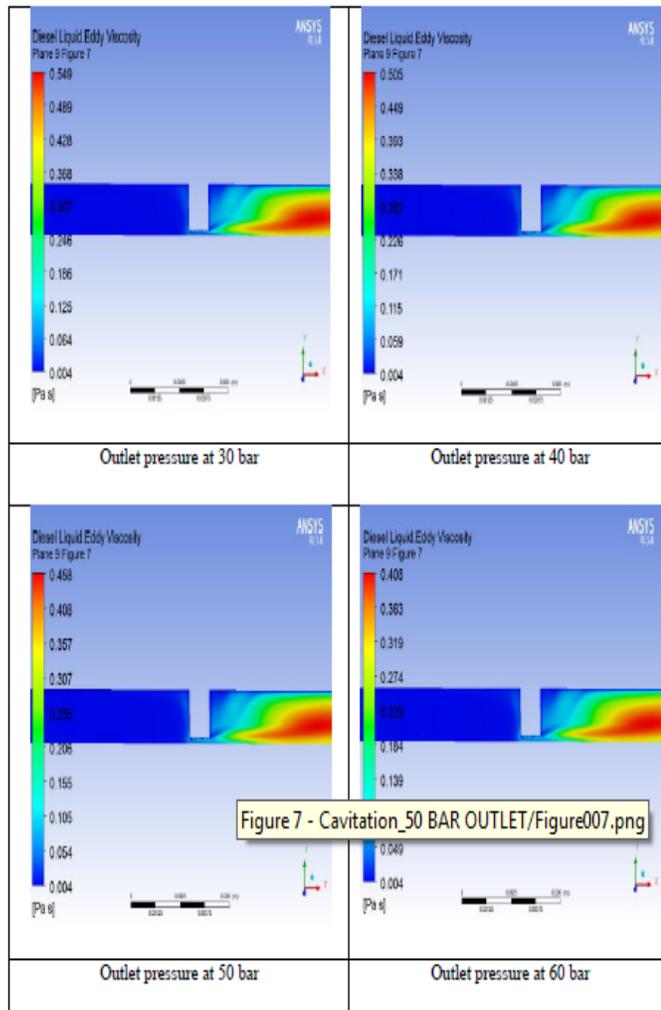

Figure 5.5 Diesel liquid eddy viscosity variations along the length

VI. CONCLUSIONS AND SCOPE OF FUTURE WORK

Conclusions

A commercial code ANSYS ICEM CFD has been used to study cavitation flow within an injector like geometry. Cavitation behavior predicted by the numerical simulations, showed a cyclic behavior. A cavity life cycle was provided by the model following clear phases: formation and growth, collapse and break-off. The mass flow choking induced by cavitation phenomena has been explored revealing that the reason of this collapse is a constant pressure at the throat. Till the 40 bar outlet pressure discharge coefficient increases with square root of cavitation number and after that discharge coefficient remains same. It was seen that numerically was independent of the pressure drop in the throttle nozzles. The

point at which discharge coefficient started remains same is called critical point or transition point. To left to that point all the cavitation pro region and right to that point is called cavitation free region. The phenomenon of cavitation only occurs in a narrow region where ambient pressure is very low. The cavitation within the nozzle is predominately the function of cavitation number, and nearly independent of the Reynolds number for those cases with a high Reynolds number. The amount of vapor generated in the cavitation region is underestimated by the model. This could be due to the overestimation of the liquid viscosity, the influence of the turbulence model or of the vapor-liquid phase interactions not taken into account. Cavitation number also increased due to increase in outlet pressure as Cavitation number linear function of outlet pressure. The predicted single-hole nozzle cavitation distribution was confined to the wall, showing major extension at enhanced cavitation conditions. The backpressure plays a more important role in the development of cavitation than the injection pressure. The vapor mass fraction is reduced due to saturation pressure at throat is increased which causes for reduction diesel vapor formation. The continuous changes in vapor distribution have a strong influence on the velocity profile at the hole exit, and therefore on the air–fuel mixing process in the combustion chamber.

Scope for the future work

a. Design variation for the diesel injector can be done to obtain further improvement in fuel spray formation and atomization.
b. Optimum nozzle geometry performance study can be done.
c. This model analysis and simulation can be improved also with nozzle movement, cylindrical nozzle, tapered nozzle.
d. Numerical simulation of multi hole injector also.
e. Numerical simulations of cavitation for different fuels are also performed.
f. To achieve more convergence solutions from other solvers like STAR-CD, ANSYS PHEONICS and ANSYS FLUENT.

REFERENCES

[1] J.B. Heywood, Internal combustion engine fundamentals, McGraw-Hill publications, London, 1988.

[2] C.E. Brennen, Cavitation and Bubble Dynamics, Oxford Engineering & Sciences Series 44, Oxford University Presentation, 1995.

[3] N. Dumont,O. Simonin,C. Habchi, Numerical simulations of cavitating flows in Diesel injectors by a

homogeneous equilibrium modelling approach, Fourth International Symposium on Cavitation, volume B6 (005), 2001.

[4] E. Giannadakis, M. Gavaises and C. Arcoumanis, Modelling of cavitation in diesel injector nozzles, Journal of Fluid Mechanics, volume 616 (153–193), 2008.

[5] H. Chaves, F. Obermeier, TU Bergakademie Freiberg, Lampadiusstr, Correlation between light absorption signals of cavitating nozzle flow within and outside of hole of a transparent diesel injection nozzle, Research Gate Journal, Publication 248392646, 1998.

[6] H. Roth, M. Gavaises, C. Arcoumanis, Cavitation initiation, its development and link with flow turbulence in Diesel injector nozzles, SAE paper 01-0214, 2002.

[7] L. C. Ganippa, G. Bark, S. Andersson and J. Chomiak, Comparison of cavitation phenomena in transparent scaled-up single-hole diesel nozzles, Fourth International Symposium on Cavitation,voume A9 (005), 2001.

[8] Daniel D. Joseph, Cavitation and the state of stress in a flowing liquid, Fourth International Symposium on Cavitation and journal of fluid mechanics, volume 366 (367-378), 2001.

[9] S.B. Martynov, D.J. Mason, and M. R. Heikal, Numerical simulation of cavitation flows based on their hydrodynamic similarity, International Journal of Engine Research, volume JER04105, 2005.

[10] F. Payri, V. Bermúdez, R. Payri, F.J. Salvador, The influence of cavitation on the internal flow and the spray characteristics in Diesel injection nozzles, Fuel, volume 83 (419-31), 2004.

[11] F. Payri, R. Payri, F.J. Salvador, J. Martinez-Lope, A contribution to the understanding of cavitation effects in Diesel injector nozzles through a combined experimental and computational investigation, Computers & Fluids, volume 58 (88–101), 2012