iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: VI Month of publication: June 2017

DOI:

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Mathematical Modelling and CFD Simulation of Cavity Formation and its Elimination in Water Jet Pump Using Fluent

Balkrishna Patel[1] Suresh Badholiya[2,] Rohit Choudhary[3]
[1]*M.Tech Scholar Mechanical Engg Dept. BIST BHOPAL*
[2]*Asst. Prof. Mechanical Engg Dept. BIST BHOPAL*
[3]*HOD Mechanical Engg. Dept. BIST BHOPAL*

Abstract: when we change inlet pressure the pressure drop is decreased and cavity is reduced or vapor pressure is reduced in efficient manner. Again when we simulate the problem by increasing both inlet and outlet pressure same phenomenon we found. Beyond the inlet pressure (7e5 pascal) no change will appear and in whole simulation system is stable in nature so fluid flow system is stable in nature which shows good results and excellent accuracy that maintain. When we change inlet pressure the pressure drop is decreased and cavity is reduced or vapor pressure is reduced in efficient manner. Again when we simulate the problem by increasing both inlet and outlet pressure same phenomenon we found. Beyond the inlet pressure (7e5 pascal) no change will appear and in whole simulation system is stable in nature so fluid flow system is stable in nature which shows good results and excellent accuracy that maintain. With this research we eliminate the cavity by 25 %
Keywords: - water jet, cfd, cavity, cavitation etc.

I. INTRODUCTION

A. General

Cavitations has always been the area of comprehensive experimental and theoretical analysis since there has been understood a numbers of undesirable technological development and progress. Cavitation causes certainly prominent destructive effects like noise, vibration and erosion produced by the expansion, growth and collapse of small vapour bubbles. The influence of model cavitating flows has presented the study in the cavitational fluid dynamics (CFD) community. It encompasses a various industrial applications, like water turbines, fuel pump, pump inducers and nozzles etc. Hydraulic cavitation in sharp areas such as in draft tube as normally appear in mechanical organization. In fluid machinery a very low pressures measure consistently generated by the action of machine, e.g. on top of blade surfaces, with an instant peril of cavitation .The continued existence of cavitations is frequently undesirable; as a result, performance of the mechanism will be humiliated, which become redundant phenomena of explosion , consequence in physical destruction to the apparatus and have a certain effect on the reliability of structure. Personal property furtherance, range and information of cavitation will be of severe make easy, correct way throughout the impression are given the organism of many stages in the fluid machinery, in an challenge to satisfy cavitation or to explanation for those effects and optimize the look. There have seen appreciable analysis on cavitation and intensive reviews measure on the market within the literature from past many decades [1], [8], [11]. Totally different aspects of this advanced development are explored, including, e.g., cavitation bubble collapse [24] and harm due to erosion, cavitation acoustics, cloud cavitation [23], [24], and rotating cavitation [3], [4]. Supported the belief that the flow is in sticky in nature, numerous numerical strategies are up to now projected to simulate cavitating flows; the conformal mapping methodology, the singularity methodology, and the panel methodology. The flow inside a centrifugal blade and [9], [10], [21] , [6] around hydrofoil may well be calculated persecution these in sticky flow models. Experimental observations have unconcealed that the cavitation leak relates closely to the sticky phenomena of the liquid-phase, like the physical phenomenon and therefore the vortex motion . Viscous flow models that regard the cavitating flow because the bubbly flow containing spherical bubbles, were introduced to supply extremely exact calculations. Within the viscous flow models, the Navier-Stokes equation as well as cavitation bubble is resolved in conjunction with Rayleigh's equation governing the amendment within the bubble radius. Kubota et al. [10] Analyzed the flows around a hydrofoil by the Finite methodology, and Bunnell et al. [5] Calculated the flow in a very mechanical system pump for diesel engines by the management volume methodology. These strategies agree well with the cavitation regions to obtained the predominating regions of the high volumetric fraction of bubbles discovered by experimentation.

www.ijraset.com
IC Value: 45.98
Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

To account for the cavitation dynamics in a very additional versatile manner, recently, a transport equation model has been developed. During this approach volume or mass fraction of liquid (and vapour) section is accountable. Singhal et al. [20], Merkle et al. [14] and Kunz et al. [13] have used similar models supported this idea with variations within the supply terms. Merkle et al. [14] and Kunz et al. [13] have used the substitute sponginess methodology. Kunz et al. [13] have adopted a non-conservative type of the continuity equation and applied the model to totally different geometries. Their solutions measure in smart concurrence with experimental measurements of pressure distributions.

The present work study addresses the process analysis of sheet hydrofoil cavitation. Different forms of cavitation that occur with sheet cavitation embrace cloud and bubble cavitation. Sheet cavitation is incredibly common on hydraulic machinery and therefore this study was intended by learning the literature regarding the experimental observations and theoretical aspects. A sheet cavity is characterized by a definite undernourished vapour bubble connected to the blade surface. Over the years, many models are developed that describe finite cavities. These measures characterized by the way, during which the cavity is terminated. Two-phase cavitating flow models supported consistent mixture approach, with a transport equation for the vapour volume fraction are enclosed in knowledgeable business codes like FLUENT [25]. We lean to initially measure this model for the benchmark drawback of a blunt cavitator, and compare the numerical results with experimental knowledge of Rouse & McNown [17]. We've got performed such associate analysis for the subfigure cavitator in [15]. Second, we tends to study the process analysis of sheet hydrofoil cavitation. The legal action corresponds to a NACA 0009 isolated hydrofoil, wherever experimental knowledge measure on the market in [7]. We tend to conclude that for steady cavitating flow, the model evaluated during this paper properly captures the pressure distribution on the hydrofoil. The present effort relies on the application of the recently developed a full cavitation model that utilizes the changed Rayleigh-Plesset equations for bubble dynamics and include the consequences of turbulent pressure fluctuations and non-condensable gases (ventilated cavitation) with rotating cavitation in numerous forms of fluid turbo machines.

B. Cavitation

Cavitation is one of the most important searched themes in fluid machinery, which is justified, because cavitation is unhappily appeared in the pumps to failure the surface on the mobile mechanism. Cavitation is initiated by the autumn of absolute pressure to level wherever gas bubbles are a unit made in an exceedingly liquid matrix. Ideally, the gas would be composed entirely of vapor of the liquid matrix, but most real systems have dissolved gas gift within the liquid. These substance gases promptly precipitate out of resolution (usually at a press beyond the pressure of the liquid) to create micro-bubbles, that act as nucleation sites for vaporization, so additional sanctioning the cavitation growth path. It's typically assumed that this growth path consists of associate adiabatic enlarging of the substance gas to the saturated pressure of the liquid [14]. The cavity, then continues to grow with the liquid turning to vapor, for as long because the bubble is in an exceedingly region of air mass. It ought to be noted that entire cavitation cycles typically occur at intervals a couple of milliseconds. The collapse of cavities creates "white" noise, that occupy a large information measure of up to one megacycle per second, with smaller bubbles leading to higher frequencies [20]. In general, peak noise amplitudes tend to correlate to peak erosion rates.

Figure 1.1: General Cavitation

These regions of air mass area unit usually the merchandise of fast liquid in hydraulic systems. In fact, any machinery that moves liquid could expertise cavitation. Another to making a district of air mass is raising the saturated pressure regionally. It's this principle exploited by some cavitation experimentalists wherever an electrical spark or optical device pulse is employed to make a cavity. Primarily, the optical device or spark provides adequate energy to a tiny, low region to lift the pressure higher than that of the close pressure, so boiling an especially localized region of liquid.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

The presence of nuclei also can have an important impact on the cavitating nature of a system. Within the absence of nuclei, it's doable for a liquid to resist tensions kind of like solids. On paper, a pure liquid should have tensions applied thereto adequate to beat building block forces and "crack" or "tear" the liquid (Trevena 1987). In most cases this calculates to negative pressures at tens of atmospheres, instead of its pressure. In reality, it's just about not possible to get rid of all of those nucleation sites; thus it's additional vital to leave the scale and distribution of those sites. In most cases, this can be additionally terribly tough to try and do; associated as a result the entire gas content of a liquid is often used as an indicator. Sadly, the entire gas content wills nothing on the far side indicating the relative propensity for a liquid to cavitated, as size and distribution don't seem to be expected.

C. Cavitation Types

Depending upon the flow configurations, there are many types of cavitation can be examined. There are two major categories addicted to which the manykinds of cavitation categories can be separated; one is the attached cavitation and other is the convicted cavitation. Attached cavitation is a part of the cavity which is associated in the surfaces of pump such as, sheet cavitation and tip vortex cavitation. Convected cavitation communicates with the vaccume in a pipe, which is completely approved by the convecting flow such as, bubble cavitation and cloud cavitation.

1) *Bubble Cavitation:* Individual bubbles are travelling with the convecting flow as they expand and collapse. This phenomenon occurs for low pressure gradient flows, corresponding to a flow around a hydrofoil at a low incidence angle (Figure 1.1).

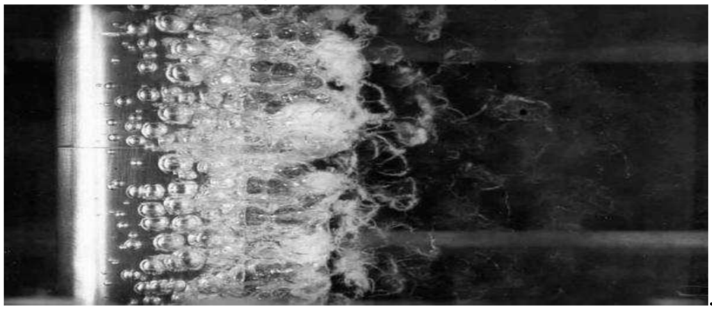


Figure 1.2: Bubble Cavitation

2) *Sheet Cavitation:* As the incidence angle of a hydrofoil increases a separated zone of vapour starts to take form. This cavity can be of different size. A sheet cavity that closes on the suction surface of the foil is called "partial Cavitation" (Figure 1.2) whereas "supercavitation" covers the entire foil. Where the cavity closers and meets the surface a stagnation point appear and a part of the flow, the so called re-entrant jet, is turned back towards the leading edge of the hydrofoil. As the re-entrant jet propagates along the foil surface the sheet cavity is shedding away from the hydrofoil.

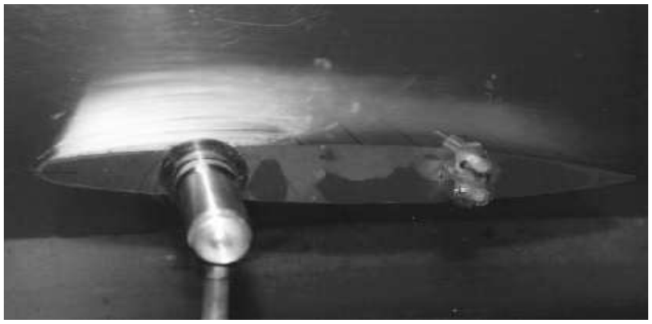


Figure 1.3: Sheet Cavitation on the upper surface

3) *Cloud Cavitation:* After the collapse of the sheet cavity the bubble density increases and a cavitation cloud move downstream with a rotating motion (Figure 1.3). Cloud cavitation may be seen in other flows with temporal periodicity. In a pump this periodicity may occur due to fluctuations caused by the rotor-stator-interaction.
4) *Vortex Cavitation:* Flows of high Reynolds number often contain regions of concentrated vorticity. In the vortex core the pressure is much smaller than in the rest of the flow, therefore this is a critical zone for cavitation. In pump flows this phenomenon can occur at the tip vortices and be seen as a helix travelling downstream (Figure 1.4).

D. Cavitating Modelling

Cavitation modelling is a sort of computational fluid dynamic (CFD) that represents the flow of fluid during cavitations. It covers a broad series of applications, such as pumps, water turbines, pump inducers, and fuel cavitation in orifices as commonly encountered in fuel injection systems.

Cavitational Modelling efforts will be alienated into two broad categories: vapour transport models and separate bubble models.

1) *Vapour transport model:* Vapour transport models are best apposite to large-scale cavitation, like sheet cavitation that usually struck on rudders and propellers. These models embody two-way interactions between the phases.
2) *Separate bubble model:* The separate bubble model includes the results of the encompassing fluid on the bubbles. Separate bubble models, e.g. [1] The Rayleigh-Plesset [2] Gilmore [3] Keller-Miksis, portray the relation between the external pressure, bubble radius and the velocity and acceleration of the bubble wall.

E. Two-phase modelling

Two-phase modeling is the modelling, as in a free surface code. The common types of two phase models are homogeneous mixture models and sharp interface models. The distinction between both the models is in the behaviour of the contents of cells contains both phases.

1) *Homogenous mixture models:* Primarily cavitation modelling have used homogenous mixture models, during which the contents of individual cells are assumed to be uniform. This advance is best suited to modelling substantial numbers of bubbles that are prolific smaller than one cell. The drawback of this approach is that once the cavities are larger than one cell, the vapour fraction is faint across neighbouring cells by the vapour transport model.This is entirely different from the sharp interface models there in the vapour and liquid or profile of distinct phases separated by an interface.
2) *Sharp interface models:* In a sharp interface models, the interface isn't subtle by temperature change. The model maintain a point interface. Naturally, this is often solely acceptable once the bubble size is a least of an order of a number of cells.
3) *Action Models:* Action model represent the mass transfer between the phases. In cavitation, pressure is liable for the mass transfer between the liquid and vapour phase. This is often in peculiarity to boiling, during which the temperature cause the action. There are three general classes of action models used for cavitation: the barotropic models and equilibrium models. This section can be discuss the benefits and drawbacks of every sort.

II. LITERATURE

Xinping Long, Maosen Xu, Qiao Lyu, Jialin Zou. The present paper describes a jet fish pump designed to safely convey fish in the aquatic industry due to its simple structure and reduced tendency to cause mechanical injury. High-speed imaging, physiology investigations and CFD simulations are used to demonstrate the impacts of the flow on the grass carp during transport in the jet fish pump for various operating conditions. The results show that none of the tested fish were dead, had organ injuries or had swimming problems after passing through the jet fish pump. The respiratory rates and most of the blood indexes of the tested fish were affected by the flow field in the pump, but they were able to recover to normal levels after just 24 h. The surface injuries such as descaling and operculum injuries were mainly caused by recirculation flows, shear flows and pressure gradients. Further analyses indicated that bruising was caused by the flow direction changes, with cavitation potentially causing eye injuries. A deflection number, C_d, was proposed to describe the amount of flow direction changes for various operating conditions in the jet fish pump. The results show that, even though shear flows and pressure gradients are inevitable, the jet fish pump operating conditions can be optimized to reduce the risk of fish injuries. Recirculation regions and intensive cavitation should be avoided since they may cause serious injuries in the fish, reduce the fish transport rate and reduce the flow efficiency. These results provide a basis for improved jet fish pump designs. 2 STERN, Zhaoyuan WANG, Jianming YANG-An overview is provided of CFDShip-Iowa modeling, numerical methods and high performance computing (HPC), including both current V4.5 and V5.5 and next generation V6. Examples for naval architecture highlight capability and needs. High fidelity V6 simulations for ocean engineering and fundamental physics describe increased resolution for analysis of physics of fluids. Uncertainty quantification research is overviewed as the first step towards development stochastic optimization. 3. H. Nouraei, K. Kowsari, B. Samareh, J.K. Spelt, M. Papini-Abrasive slurry jet micro-machining (ASJM) uses a relatively high-speed jet of fine abrasive slurry to precisely machine controlled-depth micro-features such as channels. Existing surface evolution models, developed for air-driven erosion processes, cannot account for the effect of slurry flow on the channel sidewall erosion that leads to the progressive channel widening observed in the ASJM of ductile materials. This paper presents a novel numerical–empirical model to predict the profiles of micro-channels in ductile materials (i.e. polymethylmethacrylate (PMMA), 6061-T6 aluminum alloy, 316L stainless steel and Ti–6Al–4V titanium alloy) using ASJM. The specific erosion rates of these materials were measured as a function of jet angle using a 10 μm nominal diameter aluminum oxide slurry. The erosion rate-impact angle relations were corrected using computational fluid dynamic (CFD) models to account for the local impact angles and velocities of the particles. The erosion rate-impact angle relations were then used in three-dimensional CFD models to obtain the particle trajectories, impact angles, and velocities on a shallow eroded profile, and thus predict the erosion for deeper profiles. The model was verified by comparison with experiments which showed the previously-observed widening of the machined channels as the depth increased due to secondary erosion by particles impacting the channel sidewalls. The widening effect was found to be substantial in the PMMA but less important in Ti–6Al–4V titanium alloy due to its greater erosion resistance. The numerical–empirical approach could accurately estimate the widening and predicted the channel cross-sections up to an aspect ratio of approximately 1 wita maximum error of less than 5%. 4.Jinya Zhang, Cong Xu, Yongxue Zhang, Xin Zhou-Liquefied natural gas (LNG) cryogenic submerged pumps are important transmission devices in LNG terminals and filling stations. In this study, the impeller of a two-stage LNG submerged pump was designed by the quasi-3D hydraulic design method based on the S_1 and S_2 relative stream surfaces theory. In the design procedure, the finite element method (FEM) with a quadrilateral nine-node element was adopted for the S_1 stream surfaces calculation, and the quasi-orthogonal method was used for the average S_2 stream surface calculation. The flow field was obtained by the iterative computations of S_1 and S_2 stream surfaces. Given a reasonable velocity moment distribution along streamlines considered cavitation, the blade drawing was realized by iterating the camber lines and circulation equations on an average S_2 stream surface. Moreover, a steady numerical simulation of the designed pump was conducted. The simulation result showed that the head of the designed pump was 260.15 m and the efficiency was 62.82% at the designed flow rate condition. The net positive suction head required (NPSHr) at the conditions with 0.9 Q_0, 1.0 Q_0 and 1.1 Q_0 were, respectively, 1.69 m, 2.54 m and 3.12 m, which met the industrial needs. Furthermore, both the cavitation and hydraulic performance of an impeller designed by the method presented in this study were better than those of an impeller which was designed by the two-dimensional method. 5.Desheng Zhang, Lei Shi-Tip leakage vortex (TLV) in an axial flow pump was simulated by using the shear-stress transport (SST) k–ω turbulence model with a refined high-quality structured grid at different flow rate conditions. The TLV trajectories were obtained by using the swirling strength method corresponding to the cross-sections of streamlines of the TLV. High-speed photography experiments were conducted to observe the TLV trajectory based on cavitation tracing bubbles in an axial flow pump with a transparent casing. The TLV trajectories predicted by the SST k–ω turbulence model agreed well with the visualization results. The numerical and experimental results show that the starting point of the TLV is near the leading edge at part-load flow rate condition.

III. METHODOLOGY AND RESULTS

A. Basic Steps to perform CFD Analysis:

1) Pre-processing:

a) *CAD Modeling:* Creation of CAD Model by using CAD modeling tools for creating the geometry of the part/assembly of which we want to perform FEA.CAD model may be 2D or 3d.

b) *Meshing:* Meshing is a critical operation in CFD. In this operation, the CAD geometry is discretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in a proper manner is called mesh. The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed decrease but the accuracy increase.

c) *Type of Solver:* Choose the solver for the problem from Pressure Based and density based solver. Physical model: Choose the required physical model for the problem i.e. laminar, turbulent, energy, multiphase, etc.

d) *Material Property:* Choose the Material property of flowing fluid.

e) *Boundary Condition:* Define the desired boundary condition for the problem i.e. velocity, mass flow rate, temperature, heat flux etc.

2) Solution

a) *Solution Method:* Choose the Solution method to solve the problem i.e. First order, second order

b) *Solution Initialization:* Initialized the solution to get the initial solution for the problem.

c) *Run Solution:* Run the solution by giving no of iteration for solution to converge**.**

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

3) *Post processing.*

a) *Post Processing:* For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

B. *CFD Analysis of hydrogen combustion using Ansys Fluent*

1) Pre-processing:

a) CAD Model: Generation of 2d axisymmetric geometry influent.

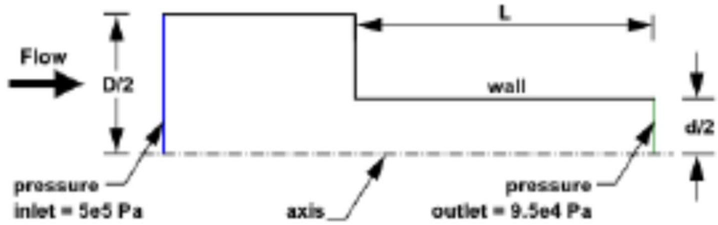


Fig-3. 2d axisymmetric geometry

b) *Mesh:* Generate the mesh in the Ansys Mesh software .

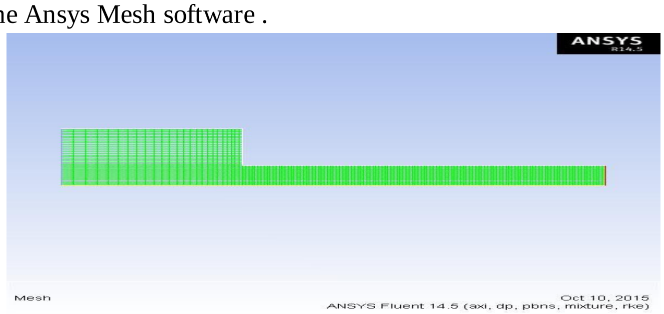


Figure 4 Mesh model

Mesh Type: grid meshing

Element Edge Length =2.95e-004 m
No. of Nodes = 10051
No. of Element = 50042

c) Fluent setup: After mesh generation define the following setup in the Ansys fluent.
d) Problem Type : 2D axisymmetric
e) Type of Solver: Pressure-based solver.
f) Physical model: Viscous: K-epsilon two equation turbulence model.
g) Finite rate/ Eddy dissipation model
h) Material Property: Flowing fluid is water
i) Density of water = 1000 kg/m^3
j) Viscosity = 0.001kg/m-s
k) Second fluid is water vapor
l) Density of water = .2558 kg/m^3
m) Viscosity = 1.26e-6 kg/m-s
n) Phases- Water is select for primary phase Water vapour is secondary phase
o) Cavitation model is enabled –No of mass transfer mechanism is 1and ensure that mass transfer is occurred from liquid phase to water vapor
p) In cavitation model- Bubble density is 1e13kg/m^3 and vaporization pressure is 3540 pa

For the multiphase mixture model, we will specify conditions for the mixture (i.e., conditions that apply to all phases) and the

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

conditions that are specific to the primary and secondary phases. In this tutorial, boundary conditions are required only for the mixture and secondary phase of two boundaries: the pressure inlet (consisting of two boundary zones) and the pressure outlet. The pressure outlet is the downstream boundary, opposite the pressure inlets.

2) *Boundary Condition:*

a) Pressure inlet for the mixture- Gauge Total Pressure- 500000 Pascal

b) Supersonic/Initial Gauge Pressure-449000 Pascal

c) Pressure inlet for the phase that is vapour –volume fraction is zero.

d) Pressure outlet for the mixture- Gauge Total Pressure- 95000 Pascal

e) Pressure outlet for the vapour-Backflow vapour fraction is set to zero

3) *Solution:*

a) *Solution Method :*

Pressure- velocity coupling – Scheme -SIMPLE
Pressure – Standard
Momentum – Second order
Turbulent Kinetic Energy (k) Second order
Turbulent Dissipation Rate (e) Second order

b) *Solution Initialization:* Initialized the solution to get the initial solution for the problem.

d) *Run Solution:* Run the solution by giving 500 no of iteration for solution to converge.

e) *Post Processing:* For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

IV. RESULTS AND DISCUSSION

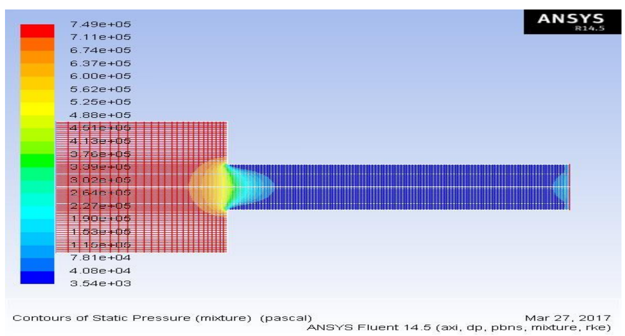


Figure 1: Contours of Static Pressure

It is noticed that dramatic pressure drop at the flow restriction in Figure 1: Contours of Static Pressure. Low static pressure is the major factor causing cavitations. In figure we see that pressure drop is increased so cavity will be decreased.

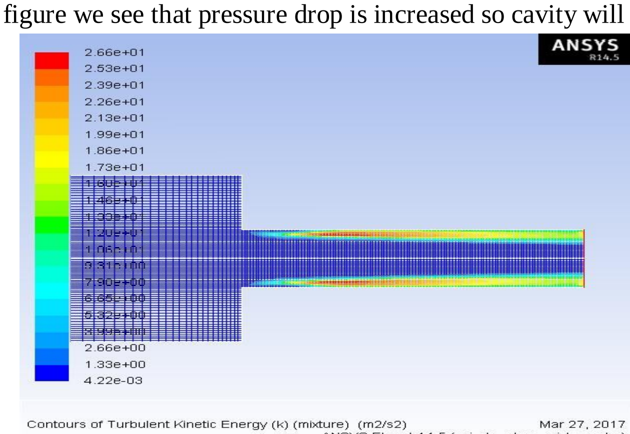


Figure 2: Contours of Turbulent Kinetic Energy

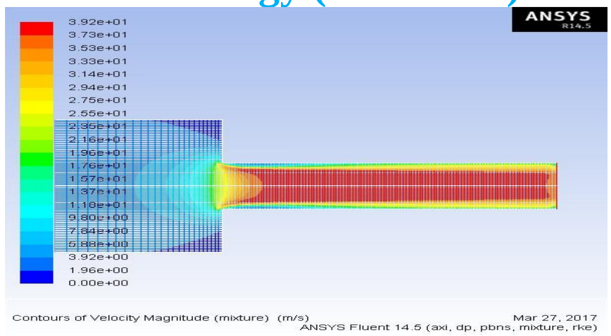


Figure 3: Contours of Velocity stream

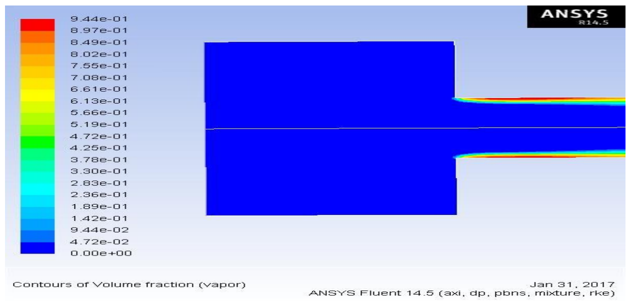


Figure 4: Contours of Vapor volume fraction at inlet pressure is 6e5 Pascal

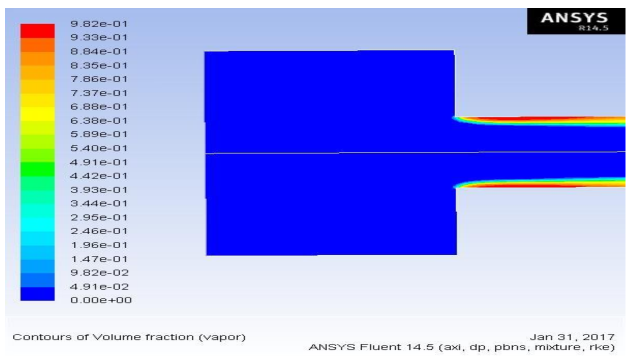


Figure 5: Contours of Vapor volume fraction at both inlet and outlet pressure is 6e5 Pascal

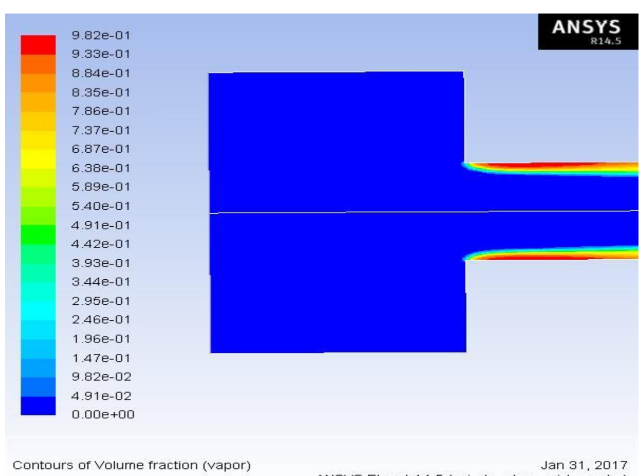


Figure 6: Contours of Vapour volume fraction at inlet pressure is 7e5 Pascal

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

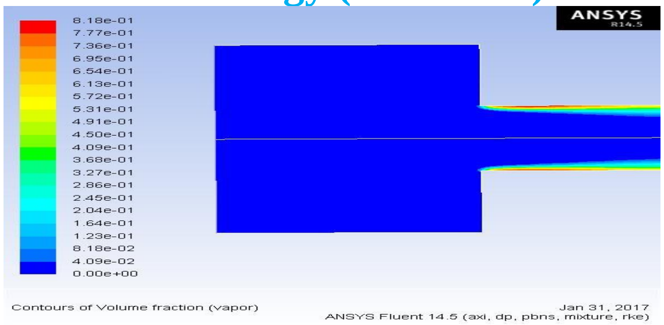


Figure 7: Contours of Vapour volume fraction at both inlet and outlet pressure is 7e5 Pascal

The high turbulent kinetic energy region near the neck of the orifice in Figure 5: Contours of Turbulent Kinetic Energy coincides with the highest volume fraction of vapor in Figure 6: Contours of Vapor Volume Fraction .This indicates the correct prediction of a localized high phase change rate. The vapor then gets convected downstream by the main flow. When we change inlet pressure the pressure drop is decreased and cavity is reduced or vapor pressure is reduced in efficient manner. Again when we simulate the problem by increasing both inlet and outlet pressure same phenomenon we found. Beyond the inlet pressure (7e5 Pascal) no change will appear and in whole simulation system is stable in nature so fluid flow system is stable in nature which shows good results and excellent accuracy that maintain. The high turbulent kinetic energy region near the neck of the orifice in Figure 5: Contours of Turbulent Kinetic Energy coincides with the highest volume fraction of vapor in Figure 6: Contours of Vapor Volume Fraction .This indicates the correct prediction of a localized high phase change rate. The vapor then gets convected downstream by the main flow.

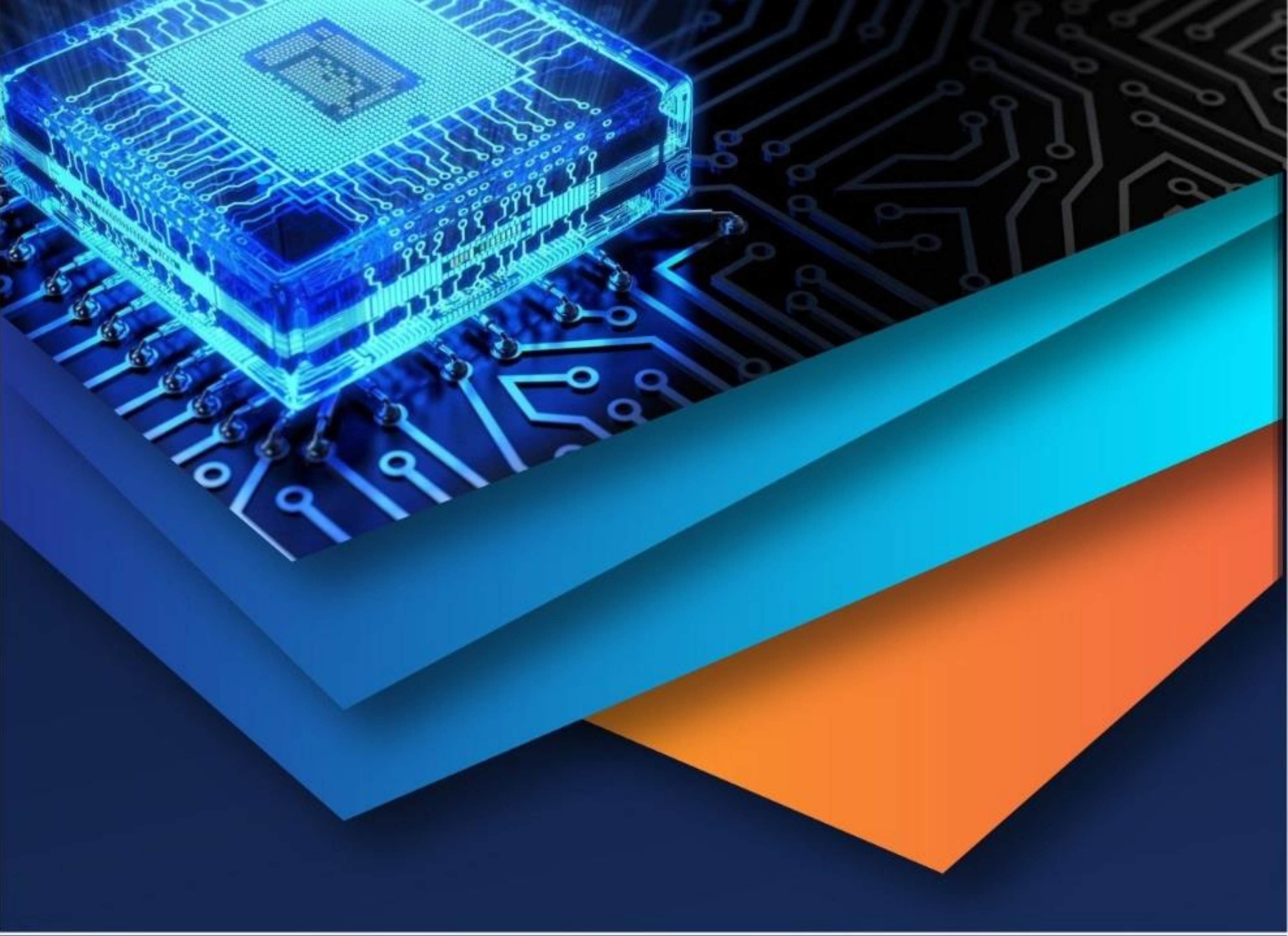
V. CONCLUSION

In present study demonstrated how to set up and resolve a strongly cavitations pressure-driven flow through an orifice, using multiphase mixture model of ANSYS FLUENT with cavitations effects. With this study we learned how to set the boundary conditions for an internal flow. A steady-state solution was calculated to simulate the formation of vapor in the neck of the flow after the section restriction at the orifice. A more computationally intensive transient calculation is necessary to accurately simulate the irregular cyclic process of bubble formation, growth, filling by water jet re-entry, and break-off. Practically it is very difficult to predict the exact location because of vibration and noise sound. When we change inlet pressure the pressure drop is decreased and cavity is reduced or vapor pressure is reduced in efficient manner. Again when we simulate the problem by increasing both inlet and outlet pressure same phenomenon we found. Beyond the inlet pressure (7e5 Pascal) no change will appear and in whole simulation system is stable in nature so fluid flow system is stable in nature which shows good results and excellent accuracy that maintain. With this research we eliminate the cavity by 25 %.

REFERENCES

[1] Arai M, Shimizu M, Hiroyasu H (1985). Breakup length and spray angle of high speed jet. Proceedings of the 3rd International Conference on Liquid Atomization and Spray Systems (ICLASS), London, IB/4/1-10.
[2] Arcoumanis C, Badami M, Flora H, Gavaises M (2000). Cavitation in real-size multi-hole diesel injector nozzles. Society of Automotive Engineers International (SAE) 2000-01-1249.
[3] Balasubramanyam MS, Bazarov VG, Chen CP (2010). Numerical design investigation of a hydromechanical pulsator for rocket motor injector dynamics research. Engineering Applications of Computational Fluid Mechanics 4:314-325
[4] Berwerk W (1959). Flow pattern in diesel nozzle spray holes. Proceedings of the Institute of Mechanical Engineers173(25):655-660.
[5] Bierbrauer F, Zhu SP (2008). A numerical model for multiphase flow based on the GMPPS formulation, part II: Dynamics. Engineering Applications of Computational Fluid Mechanics 2:284-298
[6] 7. Chaves H, Knapp M, Kubitzek A (1995). Experimental study of cavitation in the nozzle hole of diesel injectors using transparent nozzles. Society of Automotive EngineersInternational (SAE) 950290.
[7] Christafakis A, Tsangaris S (2008). Twophase flows of droplets in contractions and double bends. Engineering Applications of Computational Fluid Mechanics 2:299-308
[8] Gavaises M, Andriotis A (2006). Cavitation inside multi-hole injectors for large Diesel engines and its effect on the near-nozzle spray structure. Society of Automotive Enginners International (SAE) 2006-01-1114
[9] He L, Ruiz F (1995). Effect of cavitation on flow and turbulence in plain orifices for highspeed atomization. Atomization and Sprays 5(6):569-584
[10] Schilling R. et al. (2003) Three-dimensional Unsteady Cavitation Effects on a Single Hydrofoil and in a Radial Pump - Measurements and numerical Simulations, Part Two: Numerical Simulations, Fifth International Symposium on Cavitation, Osaka, Japa
[11] Singhal et al. (2002) Mathematical Basis and Validation of the Full Cav- itation Model, Journal of Fluids Engineering, vol.124, p.617-624

[12] Susan-Resiga R.F. et al. (2004) Numerical Simulation of Two-phase Cavitating Flow in Turbomachines, The 6th International Conference on Hydraulic Machinery and Hydrodynamics, Timisoara, Romani
[13] Szantyr J.A. (2008) The crucial contemporary problems of the computa- tional methods for ship propulsor hydrodynamics, Archives of Civil and Mechanical Engineering, vol.8, p.69-95
[14] van Terwisga T. et al. (2007) Achievements and challenges in cavitation research on ship propellers, International Shipbuilding Progress, vol.54, p.165-18
[15] Tritton D.J. (1988) Physical Fluid Dynamics, Second edition, Oxford University Press Inc., New Yor
[16] Versteeg H.K., Malalasekera W. (2007) An Introduction to Computa- tional Fluid Dynamics: The Finite Volume Method, Second edition, Bell & Bain Limited, Glasgo
[17] Yakhot V., Orszag S.A (1986) Renormalization group analysis of turbu- lence, Journal of Scientific Computing, vol.1, p.3-51



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY