Cavitation Modelling and Characteristic Study of a Centrifugal Pump Impeller

Arun mathew
M. Tech in Machine Design
S.N.G.C.E, Kolenchery, M.G. University, Kottayam

Abstract—The phenomenon of cavitation can be described as the vapour bubble formation of an originally liquid flow, this change of phase is carried through at constant temperature and local drop pressure, generated by flow conditions. Turbo machines like centrifugal pumps suffer with loss of performance, degradation of its useful life caused by the cavitation. The use of tools like CFD (Computational Fluid Dynamics) has been widely used in way to get better results in projects. The main objective of this work is to present the cavitation modelling in a centrifugal pump impeller without the use of experimental methods that in the most cases showing high coast. The blade number of impeller is an important design parameter of pumps, which affects the characteristics of pump heavily. In this study, to accurate predict cavitation of a centrifugal pump, numerical analysis was compared with experimental results to validating the analysis. Certain parameters are used to study the cavitational behaviour at each particular value and the values are taken from Ansys customer portal and experimental studies. The parameters are blade number, inlet and outlet pressure, inlet flow angle and trailing edge angle. From these parameters we get optimum values that regards to reduce the effect of presence of vapour content.

Keywords— *Cavitation, Centrifugal pumps, impeller, Number of blades, inlet flow angle, CFD*

I. INTRODUCTION

The centrifugal pump casing guides the liquid from the suction connection to the centre, or eye, of the impeller. The vanes of the rotating impeller impart a radial and rotary motion to the liquid, forcing it to the outer periphery of the pump casing where it is collected in the outer part of the pump casing called the volute. The volute is a region that expands in cross-sectional area as it wraps around the pump casing. The purpose of the volute is to collect the liquid discharged from the periphery of the impeller at high velocity and gradually cause a reduction in fluid velocity by increasing the flow area. This converts the velocity head to static pressure. The fluid is then discharged from the centrifugal pump through the discharge connection. Cavitation (from cavity) is defined as the rapid formation and collapse of vapour bubbles or pockets in a liquid, due to dynamic action, and resulting in the formation of cavities on the surfaces of solid boundaries. These solid boundaries can exist in any number of structures including hydrofoils, pipes, and fittings but, in our industry, the primary victims are impellers and propellers.

The goal of this thesis is to help to gain an intuitive understanding of the cavitation process and its causes. I will leave its prevention to the many articles that already exist. Since boiling is the precursor to cavitation, we will begin by taking a look at some of the properties of water and their role in the cavitation process. We will then put NPSH into perspective and review some of the dynamics of centrifugal pump operation that also directly affect the process. We will conclude with a discussion of the different types of centrifugal pump cavitation and the conditions that can cause them to occur.

II. AIM AND OBJECTIVES

• The aim of this thesis is to accurately predict the cavitation of the centrifugal pump with the help of Computational Fluid Dynamics (CFD).
• The main objective of this work is to present the cavitation modelling in a centrifugal pump impeller without the use of experimental works that in the most case showing high cost. Also how the operating conditions and input parameters affect the impeller geometry.

III. NUMERICAL ANALYSIS

Computational Fluid Dynamics (CFD) is a powerful tool used to model the real life behaviour of fluids. It allows the optimization of design parameters without the need for the costly testing of multiple prototypes. What is more, it is also a powerful graphical tool for visualizing flow patterns that can give insight into flow physics that otherwise would be very difficult and costly to discover experimentally, if possible at all. Governing equations exist to model fluid behaviour, but it is not always possible to apply them to many of the complex flow patterns we see in the real world directly as there would be too many unknown variables. However, CFD involves creating a computational mesh to divide up real world continuous fluids into more manageable discrete sections. The governing equations for fluid flow can then be applied to each section individually, but as the properties of each section are inevitably linked to its neighbouring sections, all the sections can be solved simultaneously until a full solution for the entire flow field can be found. This method obviously requires a huge amount of computational power, nevertheless with the advancement of modern computing, solutions that would take months to compute by hand can now be found in seconds using nothing more than an ordinary desktop or laptop computer.

As with everything, CFD is not without its limitations. Its accuracy or validity are dependent on a multitude of different factors: the quality and appropriateness of the mesh, the degree to which the chosen equations match the type of flow to be modelled, the interpretation of the results, the accuracy of the boundary conditions entered by the user or the level of convergence of the solution, to name but a few. Often it comes down to the skill of the user, as each flow problem will be slightly different and as a result, will require a slightly different modelling approach. However, experimental data can provide a valuable reference point with which to check the validity of CFD models.

IV. RESEARCH METHODOLOGY

The model pump has a design specific speed of 92.7 and an impeller with 5 blades. The blade number is varied to 4, 6, 7 with the casing and other geometric parameters keep constant. The inner flow fields and characteristics of the centrifugal pumps with different blade number are simulated and predicted in no cavitation and cavitation conditions by using commercial code FLUENT.

Here in this investigation we select various parameters for analysing the cavitation behaviour such as blade number, inlet and outlet pressures, blade angles and blade leading edge angle. The parameters are analysed by Ansys FLUENT.

A. CREATING IMPELLER USING BLADEGEN

ANSYS BladeGen is a geometry creation tool that is specialized for turbo machinery blades. BladeGen is a component of ANSYS Blade Modeller. The Blade Modeller software is a specialized, easy-to-use tool for the rapid 3-D design of rotating machinery components. With BladeGen, the user can re-design existing blades to achieve new design goals or create completely new blade designs from scratch. When either re-designing or evaluating an existing blade design, BladeGen facilitates the import of blade geometry interactively or through user supplied files. BladeGen allows sculpted or ruled element blades with linear or compound lean leading or trailing edges. Over/Under-Filing can be applied and leading and trailing edge shapes are easily specified as a full radius, an ellipse ratio, or a simple cut-off.

In analysing the performance of pump systems, the presence of cavitation can have a significant impact on the performance of the pump. Cavitation bubbles emerge when the local static pressure of the working fluid (a liquid) falls below the vapour pressure. Vigorous cavitation can lead to loss of head rise and, in many cases, structural damage due to erosion from collapsing vapour bubbles striking the pump blade passages. This involves the use of FLUENTs moving reference frame model and the multiphase mixture model (with the cavitation sub‑model enabled).Upon using blade gen tool we have to create an impeller and casing of a centrifugal pump and the certain operating conditions and geometry parameters are required to create the impeller geometry.
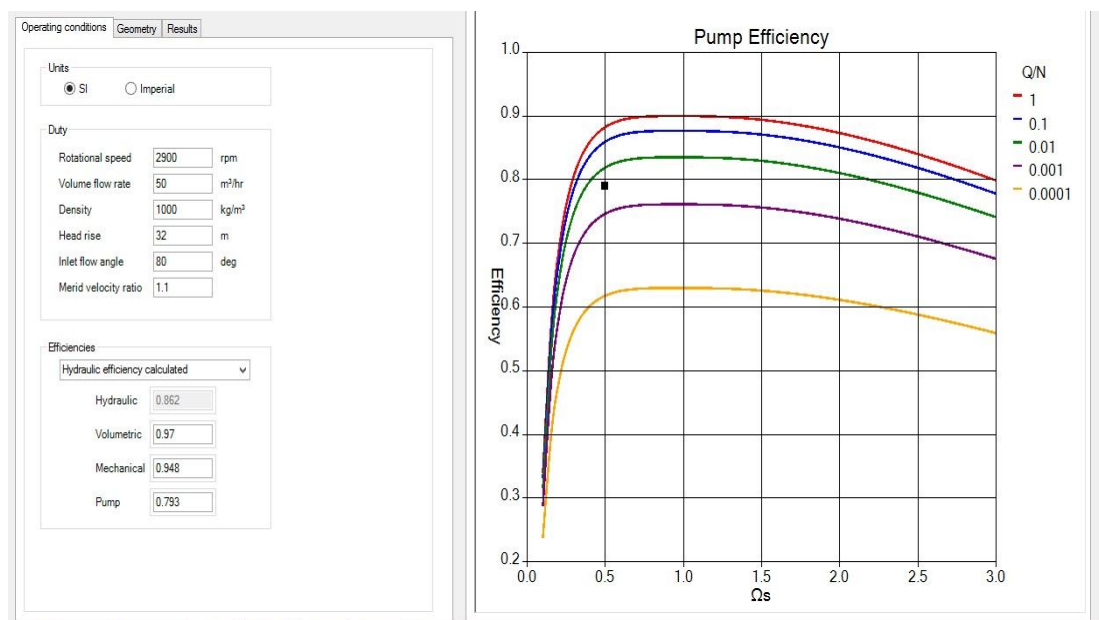

Fig: 01 Operating conditions

The pump under consideration is a five blade centrifugal pump operating at 2900 rpm. The flow enters through a circular inlet pipe, passes through the impeller, and is discharged radially through a uniform cylindrical exit boundary. The working fluid is water and the flow is assumed to be steady and incompressible.
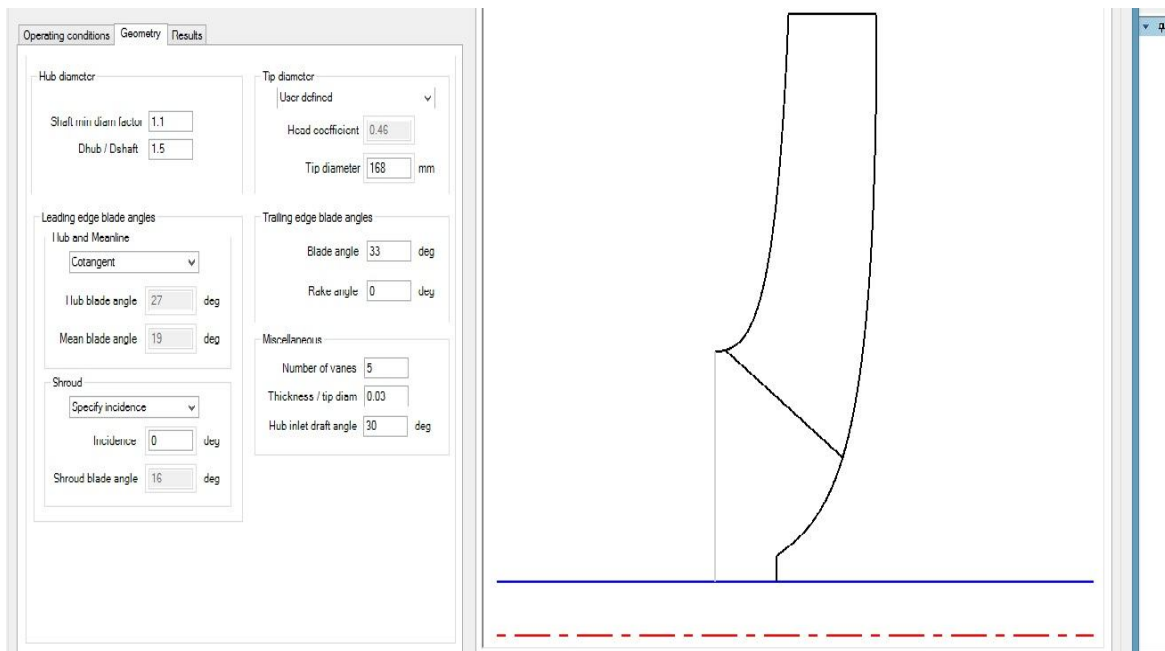

Fig. No.: 02 Geometric parameters

The fluid properties are as follows:
  Liquid density = 1000 kg/m3
  Liquid viscosity = 0.001 Pa‑s
  Vapour density = 0.01927 kg/m3
  Vapour viscosity = 8.8E‑6 Pa‑s
  Vapour pressure of liquid = 3540 Pa

    The simulation will assume a total fixed volume flow rate of 50 m3/hr at the inlet. This will be imposed using a fixed inlet velocity boundary condition. The flow will be assumed fully turbulent, with an entrance turbulence intensity of 5%. A uniform pressure outlet boundary condition is imposed at the outlet of the vane less space. The absolute inlet static pressure depends on the prescribed outlet absolute pressure. Hence, the outlet pressure will implicitly define the level of cavitation that occurs (relative to the minimum absolute pressure that occurs on the suction side of the impeller), based on the specified vapour pressure for the liquid. In the present case, the outlet pressure is set to 295000 Pa
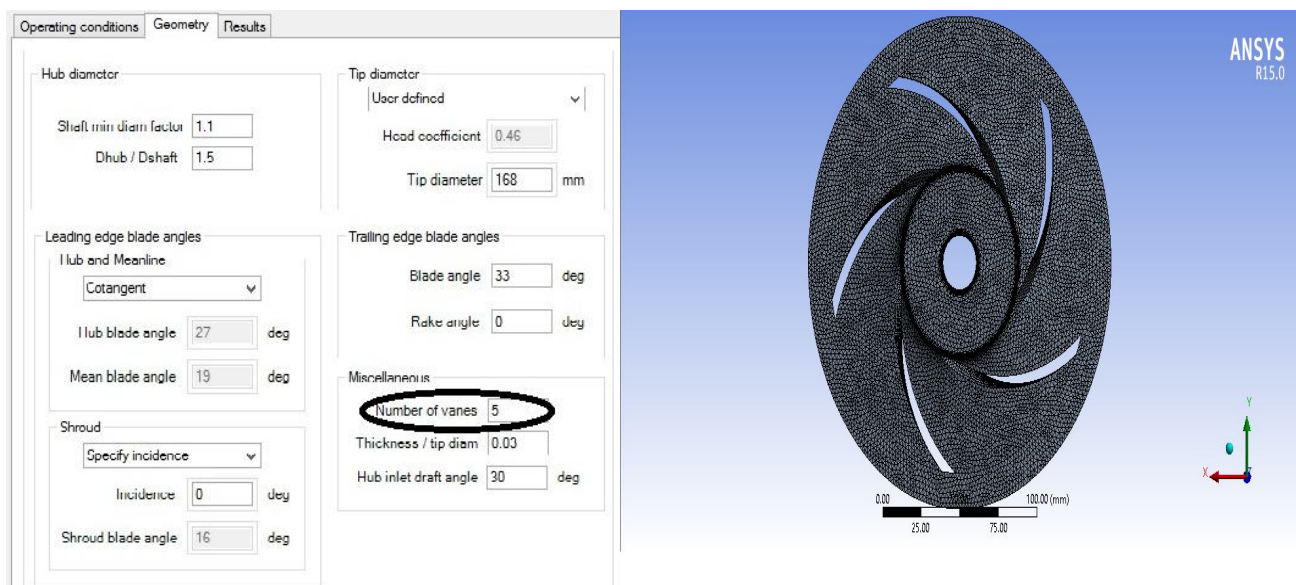

Fig No : 02 Number of blades 5   *Fig No: 03 Mesh model of impeller*

By putting the blade number as 5 and other parameters as like the experimental values we get the Net Positive Suction Head (NPSHR) as 3.63m. And this value is similar to the experimental analysis (NPSHR=3.68m). Thus from these observations we can say that our numerical analysis is validated.

V. RESULTS AND DISCUSSION

A. NUMBER OF BLADES

The blade number of impeller is an important design parameter of pumps, which affects the characteristics of pump heavily. At present, the investigation focuses mostly on the performance characteristics of axis flow pumps, the influence of blade number on inner flow filed and characteristics of centrifugal pump has not been understood completely. Therefore, the methods of numerical simulation and experimental verification are used to investigate the effects of blade number on flow field and characteristics of a centrifugal pump. The model pump has a design specific speed of 92.7 and an impeller with 5 blades. The blade number is varied to 4, 5, 6, 7.
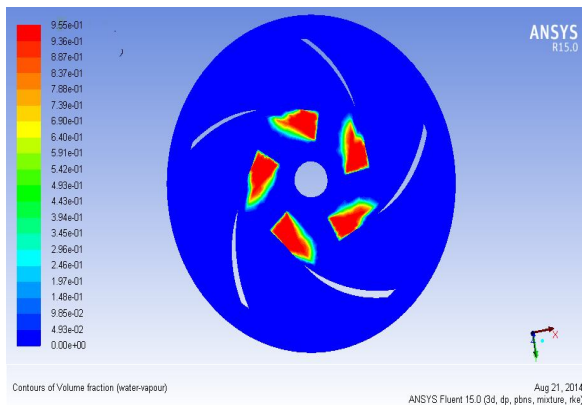

Fig No.: 04 Contours of volume fraction at Z=5

Blade number (Z)	Experimental NPSHR(m)	Numerical NPSHR (m)	% error
4	4.04	4.07	1.4
5	3.68	3.63	1.3
6	4.66	4.22	9.4
7	4.95	5.02	7

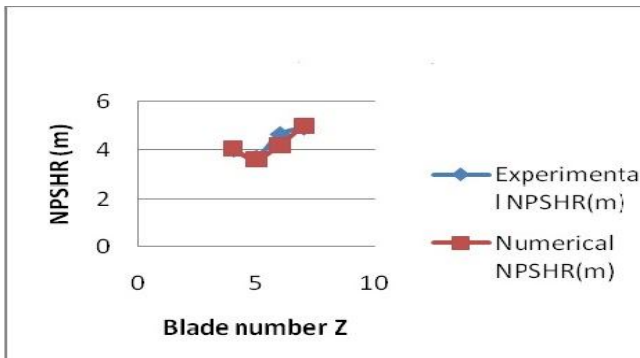*Table No. : 01 Experimental and numerical NPSHR*
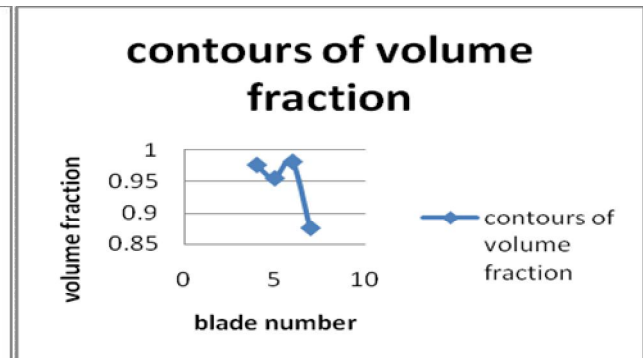

Fig No.: 05 NPSHR VS Blade number

Fig. No.: 06 Volume fraction VS Blade number

From the graph and table we conclude that the lower value of NPSHR is at when impeller number, Z=5 and the value of NPSHR is larger for the remaining values. So we conclude that the chance of cavitation parameter is less when the required net positive suction head (NPSHR) minimum and it is at blade number , Z=5. Thus the default value of number of blade is set as 5. Further analysis are done on this default value.

B. INLET AND OUTLET PRESSURES

The second parameter selected for the study is suction and delivery gauge pressures. From the published papers and Ansys customers port the normal pressure range is lies between 250000 Pa and 300000Pa. So in this tests fix the number of impeller is fix as 5 from the previous observations. In this study we set the input pressure value (283000 Pa) is setting as a default value and outlet pressure is varying. From the observations we can find out that while increasing the outlet gauge pressure with a constant value of inlet gauge pressure the amount of volume fraction is in a decreasing manner except in one case. That means the amount of vapour content is decreasing in the impeller blade region with increasing in pressure difference between the inlet and out let respectively. So we conclude that the amount of cavitation is decreasing with increasing the outlet pressure gauge reading.
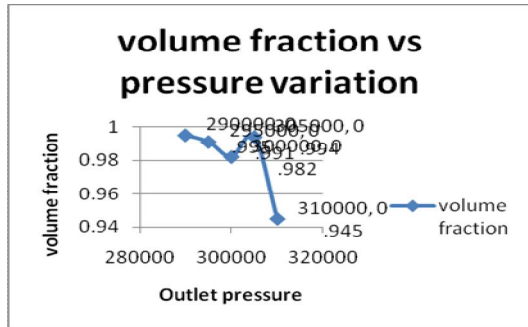
Fig. No. : 07 Volume fraction VS Outlet pressure

Inlet pressure (Pa)	Outlet pressure (Pa)	volume fraction
283000	290000	0.995
283000	295000	0.991
283000	300000	0.982
283000	305000	0.994
283000	310000	0.945

Table No.: 02 Volume fraction VS Outlet pressure

C. INLET FLOW ANGLE

Inlet flow angle is the angle between directions of fluid enters into the eye of the pump and the impeller. In normal working conditions the inlet flow angle is 90^0. In this study we have to change the angle as 80^0, 100^0 and 110^0 and check how these parameters affect the pump performances.
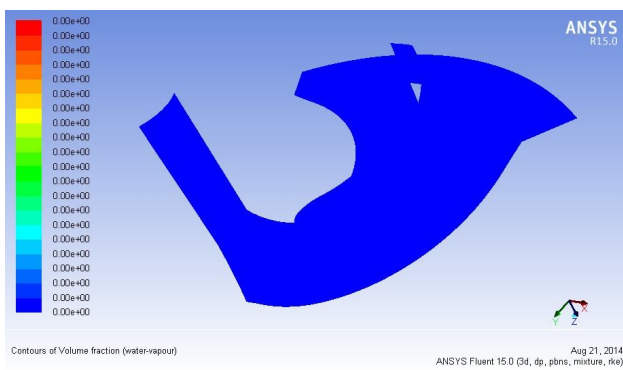
Fig. No. : 08 Contours of Volume fraction at 80^0
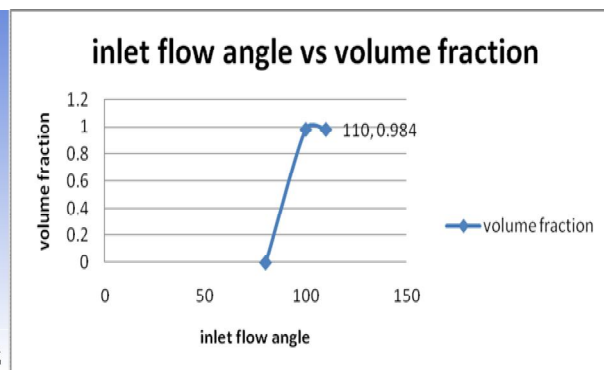

Fig. No. : 09 Volume fraction VS inlet flow angle

In normal working conditions the inlet flow angle is 90^0 that means radial impeller. In this study we have to change the inlet flow angle as above and below the default value.

From the results a graph is plotted as inlet flow angle vs. volume fraction. From the graph we can see that at 80^0 inlet flow angle the volume fraction is negligible while increasing the angle above the 90^0 the amount of volume fraction suddenly increases to a high value. Hence there is a large amount of cavitation is occurred in the impeller while increasing the inlet flow angle. But in the case of backward flow there is a chance of reverse flow of fluid. Hence in normal cases radial impellers are used.

D. TRAILING EDGE BLADE ANGLE

The present numerical study and passive flow control are concerned with the effect of the blade leading edge angle on the cavitation and performance of a centrifugal pump. There is a limited number of works regarding the effect of the blade leading edge angle on the pump cavitation.
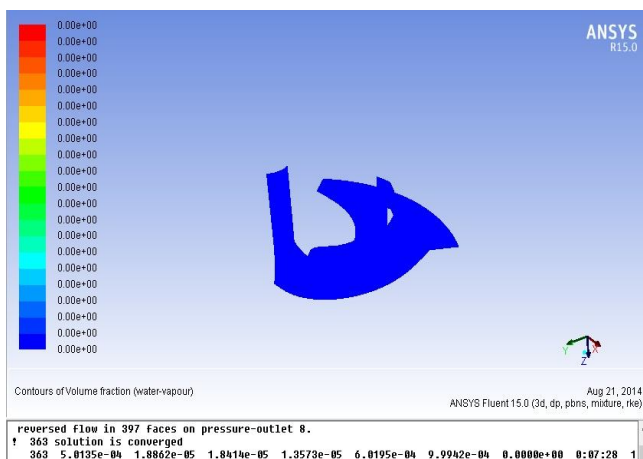

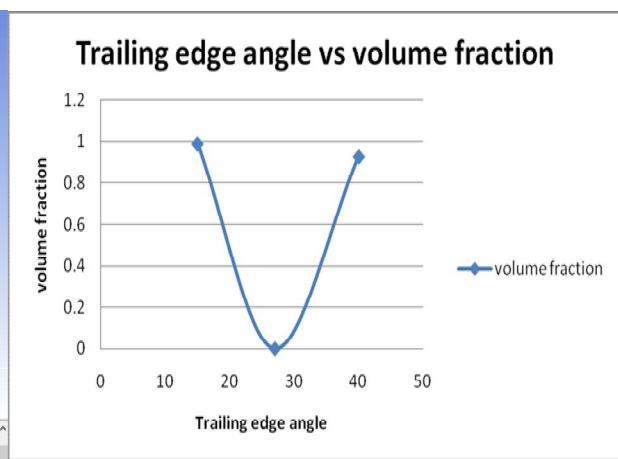*Fig. No. : 10 Contours of volume fraction at 27^0*

Fig. No. : 11 Volume fraction VS Trailing edge angle

Trailing edge angle is the outlet blade angle, by changing the corresponding angle there is a sudden change in cavitation phenomena. From fig 7.33we can see that at a particular value of trailing edge angle (27^0) value of volume fraction is zero. If any deviation from this value in upward or downward there is a sudden increment of volume fraction. So the mixture vapour bubble is large enough to cause the cavitation.

VI. CONCLUSIONS

The effects of blade number, inlet and outlet pressures, inlet flow angle and trailing edge angle are on inner flow and characteristics of centrifugal pump were researched by using the methods of numerical simulation. The main research conclusions are below.

With the increase of blade number, the head of centrifugal pump grows all the time, the change regulations of efficiency and NPSHR are complex, but there is an optimum value for the best efficiency and cavitation characteristics, the optimum blade number of the model pump in this paper for efficiency is 7, 5 blades secondly and for cavitation characteristic is 5, 4 blades secondly, so the model pump with the 5 blades has the best general characteristics.

With increasing the outlet gauge pressure with respect to the inlet pressure the volume fraction amount is in decreasing manner. But in the case of inlet flow angle the cavitation phenomenon is increases with blade angle. The considering values for blade angles are $80^0, 90^0$ and 110^0. And the negligible amount of vapour content present is at 80^0 blade angle. For trailing edge angle there is an optimum value (27^0) for minimum value of cavitation and any deviation will cause a large increment of vapour particles.

So in this paper we check different parameters how they affect or influence in the phenomenon of cavitation on impeller of a centrifugal pump and in each case there is an optimum value for minimum of that.

VII.   ACKNOWLEDGMENT

VIII.   REFERENCES

[1]   Shalin Marathe and Rishi Saxena," Numerical analysis on effect of exit blade angle on cavitation in centrifugal pump": June 2013.
[2]   LIU Houlin*, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai "Effects of Blade Number on Characteristics of Centrifugal Pumps" chinese journal of mechanical engineering Vol. 23,2010.
[3]   Myung Jin Kim, Hyun Bae Jin, and Wui Jun Chung: "A Study on Prediction of Cavitation for Centrifugal Pump", World Academy of Science, Engineering and Technology Vol: 6 2012-12-22.
[4]   Sujoy chakraborty, Kishan choudhary, Pransenjit datta, Bishop Debbarma; "Performance prediction of centrifugal pumps with variations of blade number"; Journal scientific & industrial research.
[5]   John's. Anagnostopoulos, "CFD Analysis and Design Effects in a Radial Pump Impeller",   Wear transactions on fluid mechanics Issue 7, Vol. 1, July 2006.
[6]   M. Hofmann, B. Stoffel, O. Coutter-delgosha, R. Fortes- patellia, JL. Rebound, "Experimental and numerical studies on a centrifugal pump with 2D-curved blades in cavitating motion", CAV 2001, Session B7.005
[7]   Sujoy Chakraborty a*, K.M. Pandey a, Bidesh Roya, "Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds" International Journal of Current Engineering and Technology, 2012.
[8]   Hou-lin LIU*, Dong-xi LIU, Yong WANG, Xian-fang WU, Jian WANG," Application of modified k-ω model to predicting cavitating flow in centrifugal pump" Water Science and Engineering, 2013, 6(3): 331-339
[9]   Walter Driedger "Controlling Centrifugal Pumps", Hydrocarbon Processing, July 1995,.
[10] Weidong Shi, Chuan Wang,Wei Wang, and Bing Pei." Numerical Calculation on Cavitation Pressure Pulsation in Centrifugal Pump" Hindawi Publishing Corporation Advances in Mechanical Engineering ,Volume 2014, Article ID 367631, 8 pages