© June 2019 | IJIRT | Volume 6 Issue 1 | ISSN: 2349-6002

CFD Analysis for Cavitation of a Centrifugal Pump Impeller

Vikash Siddhu[1], Pooja Kaushal[2], Swati Sharma[3]
[1, 2, 3] *Government Polytechnic College, Lecturer, ITARSI, MP, INDIA*

***Abstract*- Centrifugal Pumps are the most common appliances used in various industries, agriculture and domestic application& thus its impeller design thus required a very precise understanding of the internal flow at rated and part load operating conditions. Design and development of turbo machines like centrifugal pump is highly complex due to turbulence flow structure, unsteadiness and cavitation inside the pump. The pump suffers with loss of efficiency, erosion of material, degradation of its useful life caused by cavitation. The Phenomenon of cavitation can be described as the vapor bubbles formation in 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. Under the analytical point of view the cavitation phenomenon shows very complex, bringing great physical and numerical modeling challenges. The use of tools like CFD (Computational Fluid Dynamics) has been widely used in way to get better results in projects and developments on the dynamics of fluids. With the use of CFD tools it is possible to have a forecast about the cavitation places looking for the pressure field, since the cavitation has a direct relation with the vapor pressure at the flow fluid temperature, becoming possible to add improvements in the project of the equipment in order to prevent or to minimize the phenomenon, without the use of experimental methods that in the most cases showing high cost. The main objective of this work is to present the cavitation modeling in a centrifugal pump impeller using CFD tool.**

Index Terms- Cavitation, CFD, turbo machine, centrifugal pumps impeller.

I. INTRODUCTION

Centrifugal pump is a type of a turbo machine in which mechanical energy is converted into pressure energy by means of centrifugal force acting on the fluid. It is classified as rotor dynamic type of pump in which dynamic pressure is developed which enables the lifting of liquids from lower level to higher level. Since lifting of liquid is due to centrifugal action, it is called as centrifugal pump. Centrifugal pump has high output and high efficiency compared to other types of pumps. To develop a consistent machine for high demand operations, before they are put in actual use the performance of the flow in the entire pump has to be predicted.

1.1 WORKING PRINCIPLE

A centrifugal pump consists of a set of rotating vanes enclosed within a housing or casing that is utilized to impart energy to a fluid through centrifugal force. The vanes are usually slope backwards, away from the direction of rotation. The blades of the rotating impeller transfer energy to the fluid & thus increase velocity. The fluid is sucked into the impeller through impeller eye and flows through the impeller channels formed by the curved blades between the shroud and hub. The fluid is accelerated by pulse transmission while following through the curvature of the impeller vanes from the impeller Centre (eye) outwards. It reaches its maximum velocity at impeller’s outer diameter and leaves the impeller into a diffuser or volute chamber.

1.2 CAVITATION IN PUMP

Cavitation is not a new phenomenon that can impact a pump system, but it is an issue that is growing. While no official figures exist, it is not misleading to say that in the last five years, cases of pump cavitation have increased markedly. Often the pump itself is unfairly blamed. Pumping system problems, including cavitation, often manifest themselves at the pump but are rarely caused by it. In fact, nine out of 10 pump problems are not caused by the pump itself but by issues such as cavitation, poor system design and lack of maintenance. Additional issues caused by cavitation, such as vibration, can be severe and may lead to mechanical damage to the pump. Cavitation related problems also have the potential to reduce pump life from 10-15 years, down to just two years in extreme cases. Why is cavitation on the increase when 20 years ago it was a more isolated occurrence? One of the causes may lie in the fact that design engineers in the water industry are now expected to understand a wide range of different technologies. It is impractical for these professionals to be experts in several areas. Cavitation is primarily due to poor pump system design and a lack of awareness about how cavitation is caused.

1.2.1 CAVITATION

Cavitation can have a serious negative impact on pump operation and lifespan. It can affect many aspects of a pump, but it is often the pump impeller that is most severely impacted. A relatively new impeller that has suffered from cavitation typically looks like it has been in use for many years; the impeller material may be eroded and it can be damaged beyond repair.

Cavitation occurs when the liquid in a pump turns to a vapor at low pressure. It occurs because there is not enough pressure at the suction end of the pump, or insufficient Net Positive Suction Head available (NPSHa). When cavitation takes place, air bubbles are created at low pressure. As the liquid passes from the suction side of the impeller to the delivery side, the bubbles implode. This creates a shockwave that hits the impeller and creates pump vibration and mechanical damage, possibly leading to complete failure of the pump at some stage.

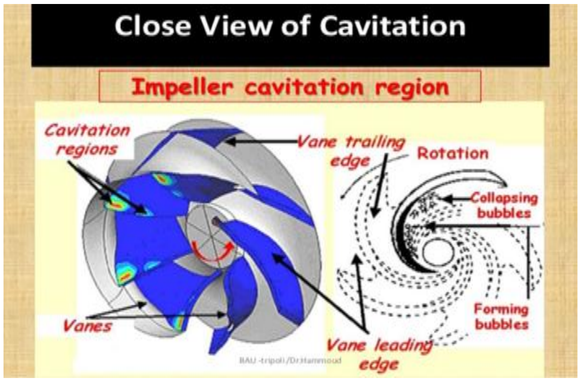


Figure 1 Impeller cavitation Regions

1.2.2 VAPOUR PRESSUE

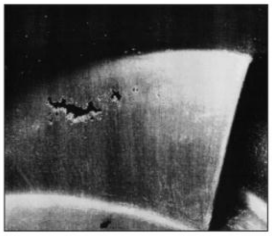
At a specific combination of pressure and temperature, which is different for different liquids, the liquid molecules turn to vapor. An everyday example is a pot of water on the kitchen stove. When boiled to 100o Celsius, atmospheric pressure bubbles form on the bottom of the pan and steam rises. This indicates vapor pressure and temperature have been reached and the water will begin boiling. Vapor pressure is defined as the pressure at which liquid molecules will turn into vapor (see Figure 1.3). It should be noted that the vapor pressure for all liquids varies with temperature. It is also important to understand that vapor pressure and temperature are linked. A half full bottle of water subjected to a partial vacuum will begin to boil without the addition of any heat whatsoever.

1.2.2.1 VARIABILITY OF VAPOR PRESSURE

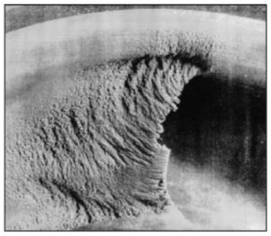
At 5ºC, water vapor pressure is 872 in round figures, but at 100ºC it is 101,000 which is a major change in pressure. This means that for any combination of pressure and temperature in the table, the water will begin boiling and turn into .For example, a glass of water subjected to a pressure of 2337 N/m2 at 20ºC will begin boiling. At 100ºC the vapor pressure is 101325 N/m2, which is atmospheric pressure. With cavitation one must deal with absolute pressures, not gauge pressures. By regulating the temperature and pressure, it is possible to boil water at different points. Other liquids have similar charts, but the values will be different.

1.2.3 IMPACT OF CAVITATION ON A PUMP

Cavitation causes pump performance deterioration, mechanical damage, noise and vibration which can ultimately lead to pump failure. Vibration is a common symptom of cavitation, and many times the first sign of an issue. Vibration causes problems for many pump components, including the shaft, bearings and seals.



Hole in pump impeller vane.



Impeller damage.

Figure 2 photographs show cavitation damage to an impeller with segments chipped away and severe mechanical damage.

1.2.4 HOW TO AVOID CAVITATION

Assuming no changes to the suction conditions or liquid properties during operation, cavitation can be avoided most easily during the design stage. The key is to understand Net Positive Suction Head (NPSH) and take it into account throughout the design process. In order to understand this term more easily it is helpful to break it down:

- Net refers to that which is remaining after all deductions have been made
- Positive is obvious
- Suction Head refers to the pressure at the pump inlet flange.

NPSH is defined as the difference between the pressure available at the pump inlet and the vapor pressure of the liquid. Vapor pressure is different for different liquids and varies with pressure and temperature. The pressure available at the pump inlet is what remains after friction loss, velocity head loss and inlet and outlet losses have been taken into account within the suction pipework of the pumping system. Because of this, during the design phase, it is necessary to calculate these losses and process unit losses in the suction pipework and then deduct those losses from the suction head available to the pump. By doing this, at the point where the pump is installed, one is left with net pressure remaining and available for the pump.

Net Positive Suction Head available (NPSHa) has nothing to do with the pump; it is a system value specific to the system design being considered. NPSHa is the head available at the pump suction flange pipework connection for the pumping system in question, and is completely independent of the pump to be installed there. It is the actual difference between the pressure at the pump inlet flange and the vapor pressure of the liquid for the installation and is determined by the design, configuration and relative levels for the suction side of a particular system.

NPSHa = Pump inlet -vapor pressure (m)

Pressure available at the pump inlet is that which remains after allowances have been made for all the losses as described above. Net Positive Suction Head Required (NPSHr) Net positive suction head required (NPSHr) is a pump characteristic that is not related to the system. All pump NPSHr's are different and the values can be obtained from the pump manufacturer.

1.2.5 WHAT CAUSES CAVITATION

Cavitation occurs in a pump when the temperature and pressure of the liquid at the suction of the impeller equals the vapor pressure. It can happen at low pressures and normal operating temperatures. Locally, it results in the liquid turning to a vapor and creating very high temperatures and pressures, which can reach circa 10,000K and 1GN/m2. Bubbles form during cavitation. As the pressure in the pump increases, those bubbles collapse in the form of an implosion – equally as violent as an explosion. The implosion causes shockwaves to travel through the liquid and hit the impeller causing mechanical damage.

II. LITERATURE REVIEW

This section describes the research work done by the investigators, with CFD as a numerical simulation tool to carry out effects of cavitation, in the performance of centrifugal pump for performance enhancement.

A. Performance Prediction Analysis

Salem (2013) presented information about the experimental work which has been done to monitor cavitation in centrifugal pump using acoustic signals. Analysis of head, efficiency, flow rate are shown graphically with the amplitude of the signals. Standard deviation methods are implemented for analyses the influence of flow rate in pump.

Weidong Shi et al. (2014) investigated the flow feature in the design condition in a turbulence model. Head flow and back flows were analyzed. It was found that when the flow rate decreased below a certain value of the design flow rate, backflow occurred near the pressure surface of the pump impeller and cavitation is occurred take place at this stage. Three different pump models were investigated using the turbulence model.

Shalin Marathe et al. (2013) presented the effect of outlet blade angle on cavitation in centrifugal pump. Modeling of the centrifugal pump along with the different configuration of the impeller having different exit blade angles was carried out using Creo Parametric. Numerical simulation was carried out using ANSYS CFX and standard k-ε turbulence model is implemented for the analysis purpose. From the results, the author found that the pump having low value of the blade exit angle will have less chances of getting affected by the cavitation phenomenon.

B. Different Turbulence Model

Pande et al (2015) used a general three dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. CFD was used to solve the governing equations of the flow field. This study described a finite volume method to examine the pump cavitation at the pressure drop region on the blade. It easy found that there was a significant spike in residuals, in the outlet pressure difference, and absolute pressure is low enough to induce cavitation.

Mouhammed khuladhar abbas et al. (2010) numerically studied the flow through the blade passage with CFD code, ANSYS CFX in order to detect formation of cavitation in centrifugal pump. Head drop curve has a knee shape that head remain constant while NPSH decreased and head will be rapidly decreased at critical point. The beginning of cavitation in the blade passage can be detected and shown in quality and quantity with numerical simulation. The inception of cavitation is take place on the suction surface where the leading edges meet the tip. In pressure distribution plot shows that the cavitation zone expanding to the trailing edge especially in super cavitation case. Author concluded that the available NPSH of the system must be equal to or greater than the NPSH required by the centrifugal pump in order to avoid cavitation difficulties.

C. Various Flow Conditions

Myung Jin Kim et al. (2012) accurately predicted cavitation of a centrifugal pump in various flow conditions. In this study, numerical analysis was compared with experimental results modeled on a small industrial centrifugal pump for reliable prediction on cavitation of a centrifugal pump. To improve validity of the numerical analysis, transient analysis was conducted on the calculated domain of full type geometry. The numerical analysis from the results was considered to be a reliable prediction of cavitation.

Ragavendra muttali et al. (2014) analyzed the flow through a centrifugal pump using commercial CFD package. CFD analysis was carried out at design and off design condition and was reported. The simulation results were obtained at different operating speed with different mass flow rates for transportation of fluid. The Simulation was performed by using turbulent modeling k-ε and cavitation analysis, pressure contours and velocity vector contour were predicted. The performance results were satisfactorily matching with test data.

SUMMARY OF LITARATURE REVIEW

Various authors proved that below listed parameters have influence on the performance of centrifugal pump but the research not has been done on the area of cavitation. As the number of impellers changes the performance of the hydraulic machine also changes. In order to find its influence on cavitation blade number variations is taken as a parameter. The cavitation inside the volute and between the impeller occurs mainly due to alteration in the pressure. So the inlet and outlet pressure is allowed to vary over a range and find its effect on the cavitation. The fluid is sucked up in to the pump due to the difference between pressure inside the pump and the atmospheric pressure. As the fluid enters to the system and starts move along with vanes. So the edge angle and inlet flow angle being contact with the fluid. These parameters like flow angle and edge angle are also affect the performance of the pump. In order to find the effect of these parameters in the performance of the pump further investigations are necessary. Based on the literature review, the following objectives have been found for the present study.

- To accurately predict the cavitation behavior of the centrifugal pump with the help of Computational Fluid Dynamics.
- To present the cavitation model in a centrifugal pump impeller without the use of experimental works.
- To shows how the operating conditions and input parameters affect the impeller geometry.
- To modify the blade flow passage this affects the development of cavitation. To design centrifugal impellers that can operate more efficiently and quietly.

III. METHODOLOGY

The geometry dimensions for the work are given as below and made in Ansys 14.5:

Impeller Blade Dimensions:
Inlet Dia. = 61mm
Impeller thickness at inlet = 25mm
Impeller thickness at tip = 5mm
No. of Blades = 8
Blade angle = 45 Degree

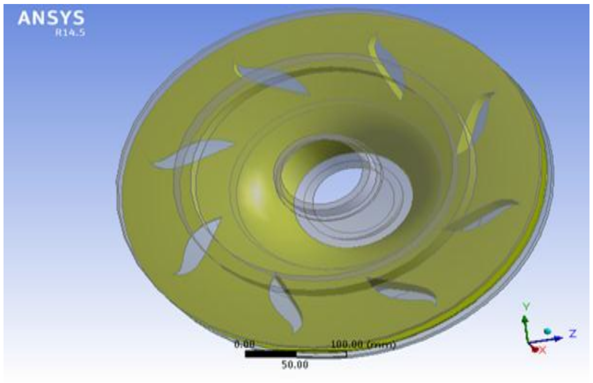


Figure 3 Geometry of the Model

Ansys Blade Gen was used to simulate the inner flow field under non cavitation condition. The standard k - ε turbulence Model applied to solve the RANS equations. The simulation is steady and moving reference frame is applied to take into account the impeller volute interaction. ANSYS BladeGen is a geometry creation tool that is specialized for turbo machinery blades. BladeGen is a component of ANSYS Blade Modeler.

The Blade Modeler software is a specialized, easy to use tool for the rapid 3-D design of rotating machinery components. Incorporating extensive turbo machinery expertise into a user-friendly graphical environment, the software can be used to design axial, mixed-flow and radial blade components in applications such as pumps, compressors, fans, blowers, turbines, expanders, turbochargers, inducers and others.

In order to develop the blade model following parameters are taken in to the consideration:

Flow rate, Q= 80 m^3 /hr
Head, H = 40 m
Rotation speed, N = 4500 r.p.m.
Specific speed Ns =42.1755
Blade number, Z= 8
Inlet flow angle, =90º
Trailing edge angle, = 45º
Inlet pressure, =300000Pa
Outlet pressure, =320000 Pa
Velocity inlet, = 3 m/s

The model created in the Blade Gen software. After the generation of the blade it is meshed in to fine number of nodes for effective analysis.

Meshing Type: Mapped face Meshing (Tetrahedral)
No. of Node: 3508
No. of Element: 11155

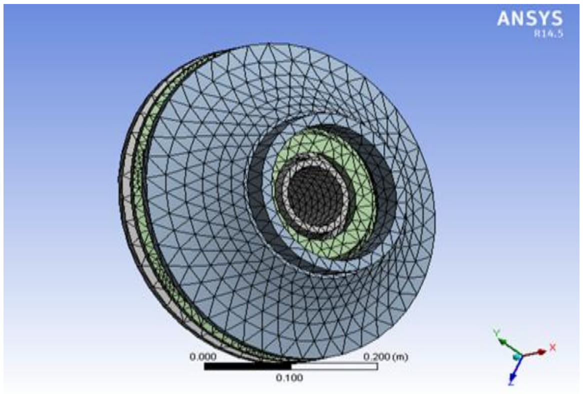


Figure 4 Mesh view

For the effective analysis of the impeller geometry should me meshed to fine number of contours. The above figures show the meshed geometry of an eight blade impeller.

IV. RESULTS AND DISCUSSIONS

The number of impeller is an important design parameter among pumps, which affects the characteristics of pump heavily. 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 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 and an impeller with 8 blades. When blade number is increased to 8, it’s clear that pressure is also rising according to the increment. There is an accumulation of pressure at the edge of the impeller. This is the static pressure developed inside the pump. The pressure contours for the number of blade is shown in the fig5.

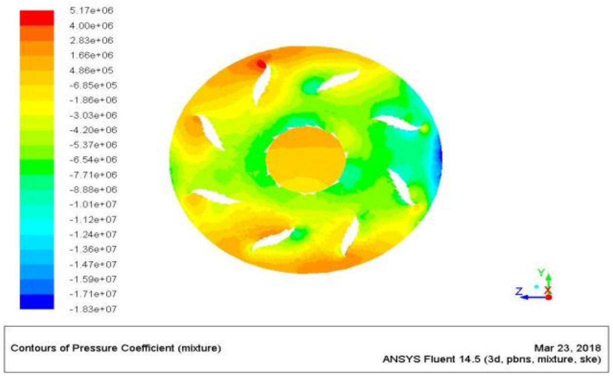


Figure 5 Contours of Pressure Coefficient

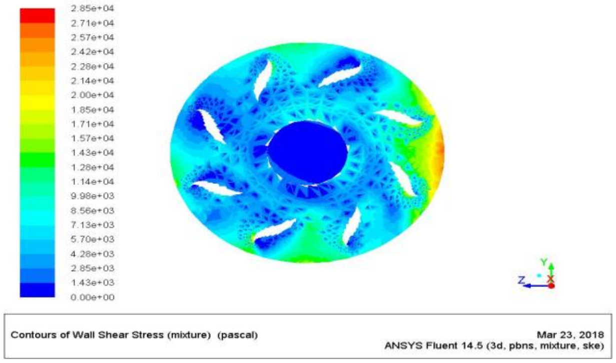


Figure 6 Contours of Wall shear stress

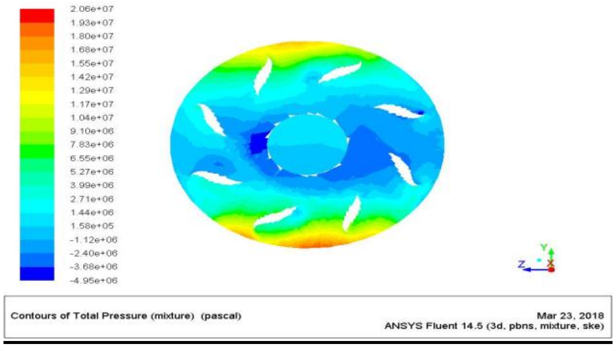


Figure 7 Contours of Total Pressure

As the number of blade is increases, there is a decrement in the development of vapor.

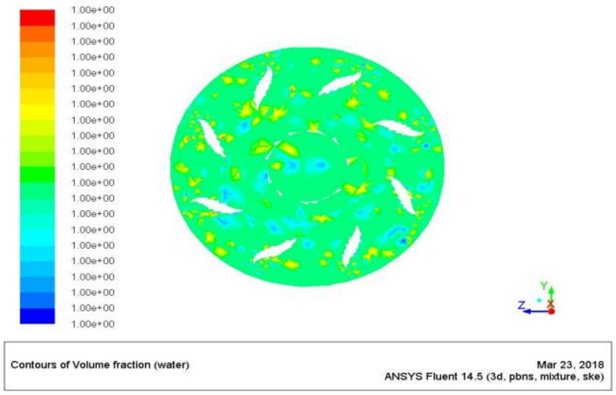


Figure 8 Contours of Volume Fraction

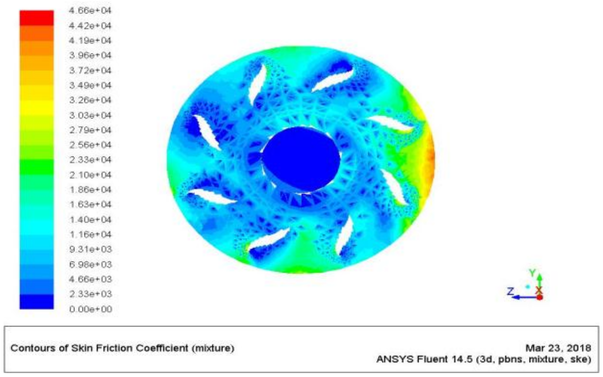


Figure 9 Contours of Skin Friction Coefficient

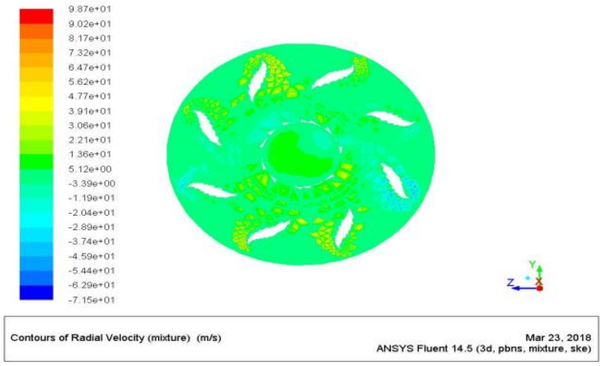


Figure 10 Contours of Radial Velocity

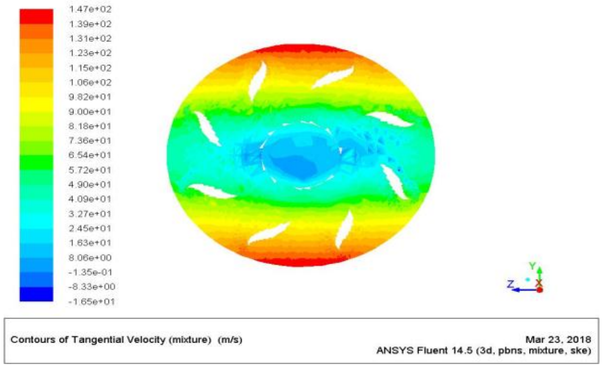


Figure 11 Contours of Tangential Velocity

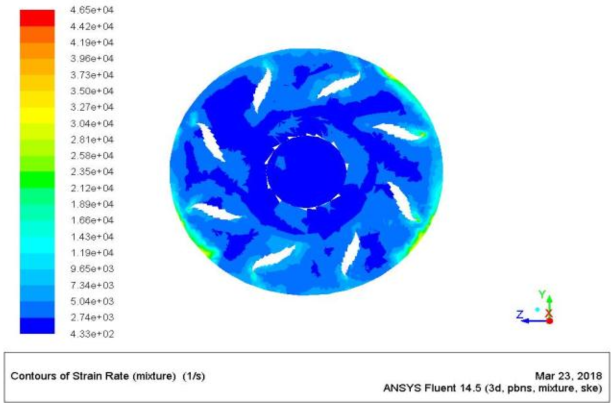


Figure 12 Contours of Strain Rate

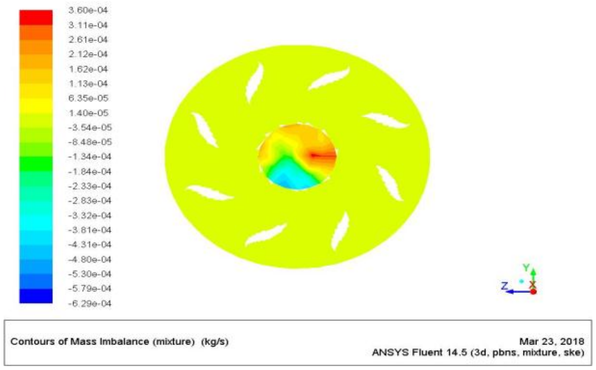


Figure 13 Contours of Mass imbalance

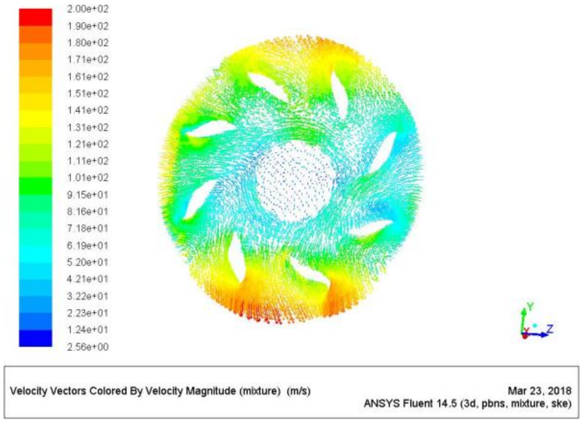
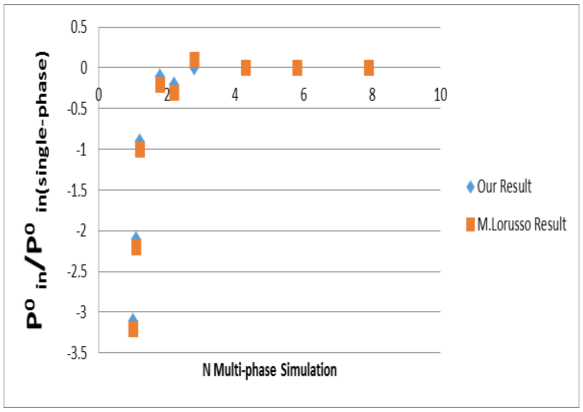
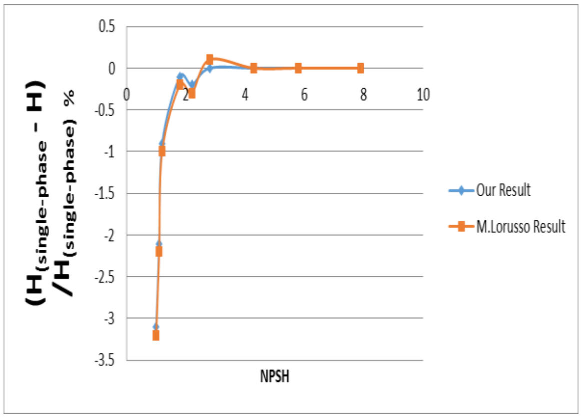


Figure 14 Contours of Velocity Magnitude

We conclude that the chance of cavitation parameter is less at blade number, Z<8. This is because net positive suction head required is increasing as the number of blade increases. In the 8 number of blade, cavitation is found the minimum value compared to all other number of blades. But the NPSHR is quite high for this number of blade. Thus the default value of number of blade is set as 8. Further analyses are done on this default value. The contours of volume fraction shows 8 number of blade impeller having minimum amount of cavitation. But for Z<8 of blade impeller, having quite large value in head.



Graph 1 Values of the total pressure imposed at the inlet boundary during the construction of the pressure drop curve



Graph 2 Pressure drops and head drop lines (with respect to single-phase)

V. CONCLUSIONS

The effects of blade number, inlet and outlet pressures, 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 is 8 blades.

The head required for the pump is quite large for 8 impeller blade. With increasing the outlet gauge pressure with respect to the inlet pressure the volume fraction amount is in decreasing manner. The optimum pressure value for the minimum occurrence of cavitation is 320000Pa. Effective operation of centrifugal pump with minimum cavitation effect can be achieved by operating the pump in above mentioned conditions. The model is free from cavitation at design point and it is also noted that, the formation of cavitation on the blade is increasing with the increase of mass flow rate and rotating speed.

REFERENCES

[1] S.Rajendran and Dr.K.Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS-CFX,” International Journal of

Engineering Research & Technology, Vol. 1, Issue 3, 2012.

[2] S R Shah, S V Jain and V J Lakhera, “CFD based flow analysis of centrifugal pump,” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[3] P.Usha Shri ans C.Syamsundar, “computational analysis on performance of a centrifugal pump impeller,” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[4] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, “Parametric Study of a Centrifugal Pump Impeller by Varying th e Outlet Blade Angle,” The Open Mechanical Engineering Journal, no 2, 75-83, 2008.

[5] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova, “Cavitation modelling of a centrifugal pump impeller,” 22nd International Congress of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil, 2013.

[6] Mohammed Khudhair Abbas, “cavitation in centrifugal pumps,” Diyala Journal of Engineering Sciences, pp. 170-180, 2010.