© 2018 IJSRST | Volume 4 | Issue 5 | Print ISSN: 2395-6011 | Online ISSN: 2395-602X
Themed Section: Science and Technology

A Technical Review on Analysis of Centrifugal Pump

Parmar Piyushkumar*, Saurabh Valand, Rushabh Suthar
Mechanical Engineering Department, Vadodara Institute of Engineering, Vadodara, Gujarat, India

ABSTRACT

Centrifugal pumps are grossly used in chemical industries, oil refineries, sewage disposal systems, mines, food processing industries, power plants for the transportation of fluids. Hence it is required to find the geometric parameters and working conditions to achieve maximum head and efficiency without increasing the manufacturing cost. This paper evince the increasing use of computational fluid dynamics for the analysis of centrifugal pump, as there are various parameters which affect the performance of centrifugal pump it is necessary to analyze all the forces which are act on pump impeller such as drag and friction forces. CFD analysis is very useful to determine the amount of drag force acting on pump impeller when working with different fluids such as Newtonian and non- Newtonian fluids, as the amount of drag force increase than power requirement also increase results in poor performance. Therefore it is necessary to carry out CFD analysis on centrifugal pump maintain its performance in any condition.

Keywords: Centrifugal pump, Analysis, CFD, Impeller, Cavitation.

I. INTRODUCTION

A pump is mechanical device which is used to increase the head of fluid for further application. The centrifugal pump is rotodynamic device which increases the potential energy or head of liquids or slurries on the expense of its kinetic energy of fluid enters axially and leaves radially, due to the shape of spiral casing of pump kinetic energy possessed by fluid is converted into pressure energy. It is commonly used for pumping of chemicals, water, sewage, petroleum products etc. It consists of impeller, spiral casing, suction and discharge pipes, different valves, motor or engine, and other additional components.

It is not economical to manufacture a pump only to carry out analysis on it, when one parameter is change thane whole model or prototype has to be changed. Computational fluid dynamics (CFD) is very useful tool to carryout analysis on centrifugal pump and it is easy to change any parameter like blade angle, inlet or outlet conditions, type of fluid etc.

II. LITERATURE REVIEW

Wei li and all, by investigating the varying locus and pattern of transient radial force of centrifugal pump with time the results shows that the frequencies of pressure pulsations are mainly the blade passing frequency at the monitoring points and also at that point most intense pulsation appears. The magnitude of blade passing frequency which causes the pressure pulsation is 114.333 HZ, indicates the interference between the blades of impeller and collector. Under the non-designed flow rates, the radial force on impeller is large and the amplitude of fluctuations is largest. Under the designed small flow rates, the transient radial force on impeller is small with gentle fluctuations. [1]

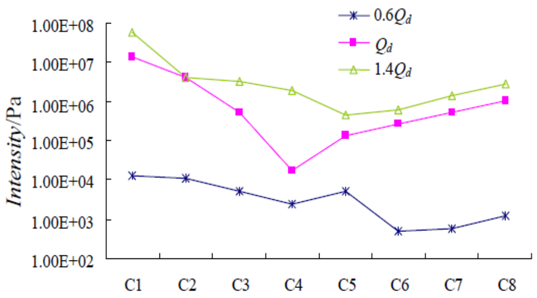

Figure 1. Intensity comparison of pressure pulsation

Dr.P.Muthu and Alex George described that among number of blade, Inlet and outlet pressure and characteristics of pump the number of blade has major influence on cavitation in the pump. With increase in the number of blade, the head of centrifugal pump increase all the time, the change regulations of efficiency and Net positive suction head required are complex. By numerical simulation they found that 5 to 7 numbers of blades are optimum value for best efficiency and capitation characteristics, with increase in blade number, capitation characteristic increases. So to get best efficiency of centrifugal pump and to avoid cavitation characteristics it is required to use minimum 5numbers of blades. [2]

Table 1

Blade number(Z)	Experimental Net positive suction head required(m)	Numerical Net positive suction head required (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

Deeptesh Singh and all studied the failure analysis of centrifugal pump from the perspective life of its component and frequency of occurrence of the failure. After doing questionnaires in the industry and analysis of data obtained from industry it clear that most critical component in a centrifugal pump are packing and bearing. Hence it is required to increase the reliability of these components. [3]

Tilahun Nigussie and Edessa Dribssa investigated the effect and distribution of velocity profile and pressure within a pump using 3D Navier stokes equation in ANSYS. From the simulation it is observed that the pressure increases gradually from impeller inlet to outlet and velocity increases from impeller inlet until it enters in the volute casing and drops to zero at outlet. It is found that the design and analysis methods lead to very good flow field and general performance predictions. In this way, design can be optimized to give reduced energy consumption, lower head loss, prolonged component life and better system flexibility without making prototype. [4]

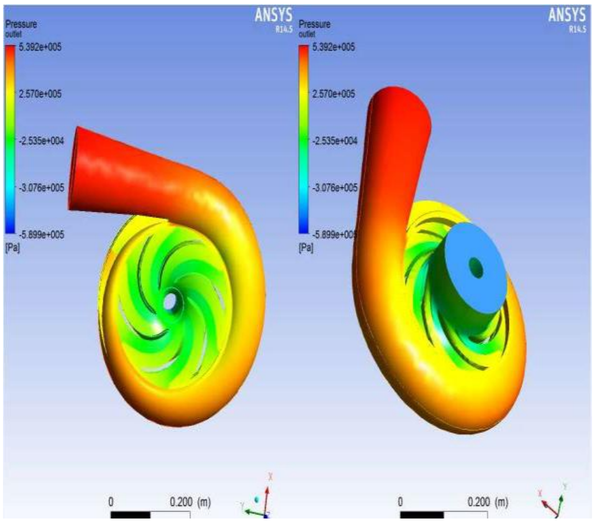

Figure 1. Pressure counters flow through the pump in the 3D view at design value of the flow rate

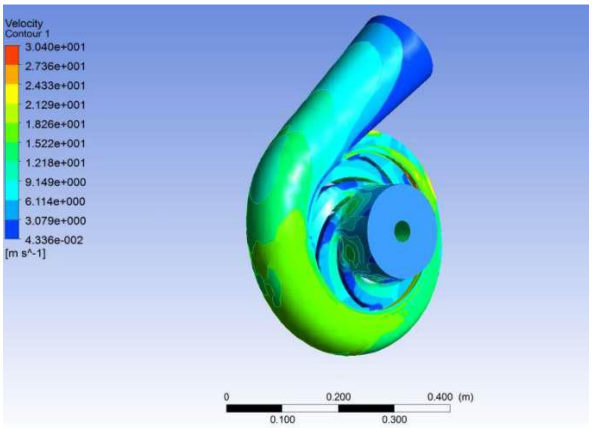

Figure 2. Velocity counters flow through the pump in the 3D view at design value of the flow rate.

P.Gurupranesh and all carried out a detailed CFD analysis on impeller of centrifugal pump to enhance its performance. Using solidworks 2012 modeling of impeller is done and CFD analysis is done using fluid flow simulation package. CFD analysis helps to predict pumps performance virtually. For the distribution of pressure, velocity and wall shear stress contours are plotted. The overall efficiency obtained is 61% and 50m of total head is predicted. [5]

Ajith M.S and Dr. Jeoju M. Issac suggested to use CFD approach to investigate the flow in centrifugal pump impeller using Ansys fluent. They used circular arc method and point to point method to generate impeller vane profile to perform CFD analysis. For both forward and backward curved vane analysis is done and simulation is on vane profile is solved by Navier stokes equation with modified k- w turbulence model. From result it is clear that backward curved vane has higher performance than forward curved vane, out of two methods to generate the vane profile equal head rise is obtained from both the methods.[6]

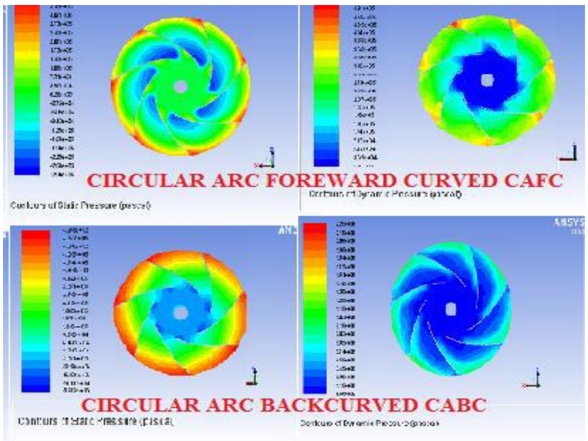

Figure 3. Pressure and velocity distributions in CAFC AND CABC.

Raghuvendra S. Muttali and all described the head, power, efficiency to evaluate the performance of pump using ANSIS 14. This is strongly related to cavitation flow phenomena which can occur either in rotating runner impeller or in stationary parts of the pump. To achieve safe operating for different flow rate and rotating speed, numerical simulation is used to detect cavitation in the pump. The simulation was performed using turbulent modeling k- Epsilon. The results shows that with increase in designed flow rate the total pump head reduces and also it is observed that formation of cavitation on the blade is increase with increase in rotating speed and mass flow rate. [7]

R. M. Pande and all discussed that about 90 to 95% of failure of pump is caused due to aeration, cavitation, contamination, excess pressure, excessive temperature or inadequate viscosity of the fluid. They used ANSYS 15.0 to solve the governing equations of the flow field to predict the velocity and pressure field of centrifugal pump. They used Finite Volume Method to examine the pump cavitation at the pressure drop region on the blade. As a result they found that there is a significant spike in residuals, in part due to the outlet pressure difference and absolute pressure is low enough to induce cavitation. [8]

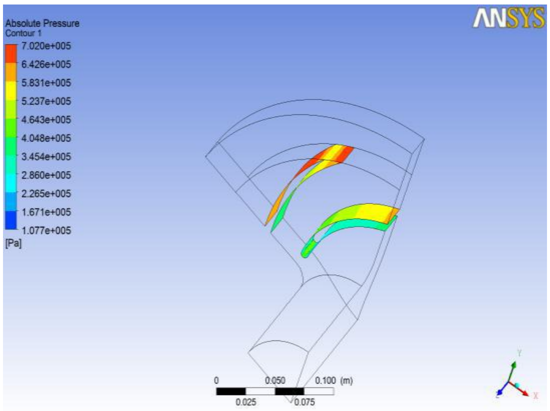

Figure 4. Pressure distribution at blade for without Cavitation model

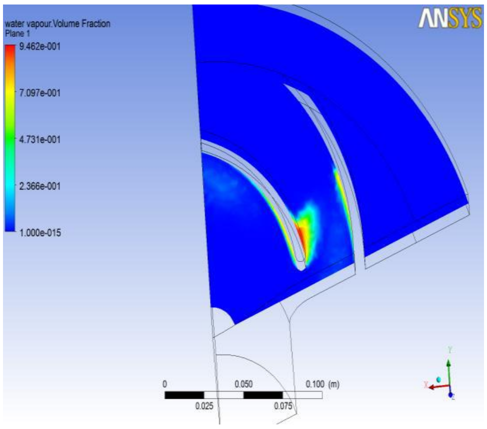

Figure 5. Cavitation induced at the pressure drop region on the blade of the centrifugal pump

T.Sivakumar and all described that impeller geometry has major impact on performance of pump and it is clear that by changing impeller blade angles overall performance of mixed flow pump is increases. As a result they found that in mixed flow pump impeller as the number of blade increases we get maximum head, efficiency and minimum power consumed. The analysis software predicated head of 19.65m, power consumed is 2524.49w, and efficiency is 79.83% when 42^0 inlet angle, 29^0 outlet angle and 6 numbers of blades are used. [9]

Table 2. Modified Impeller Data

Sr. no.	Parameters	New data
1.	Number of blade	6
2.	Blade inlet angle	38˚
3.	Blade outlet angle	27˚

Satish M. Rajmane and Dr. S.P. Kallurkar conducted a computational fluid dynamics analysis on discharge side of the pump to determine pressure losses for dividing and combining the fluid flow through a junction of discharge system. The mathematical model based on successive approximations is employed to estimate the pressure losses and also the effect of bend angle, pipe diameter and length, Reynold's number on the resistance coefficient is studied. As a result they found that vortices and cavitations causes inefficiency in the operation of centrifugal pump but it can be reduced by changing the blade angle of delivery pipe. Redesign of delivery side of pump facilitated the flow of water and improves the discharge and consequently the performance of centrifugal pump. [10]

C.Buratto and all in this study, the computational fluid dynamics analysis is carried out on centrifugal pump to investigate the difference and similarities in Newtonian and non- Newtonian fluids. As pump design is made according to the water as a working fluid, correlations are applied to predict the viscosity of non- Newtonian fluid. In particular, the comparison of the impeller curves of total pressure rise, disc friction loss and efficiency has highlighted a very close trend between tomato paste and Newtonian virtual fluid with dynamic viscosity of 1 Pa.s. A linear relationship between the Reynolds number and the power of impeller, for the laminar high viscous Newtonian flow inside the pump was found. A power law relationship for a set of virtual non Newtonian fluids with different viscosity index was also found. [11]

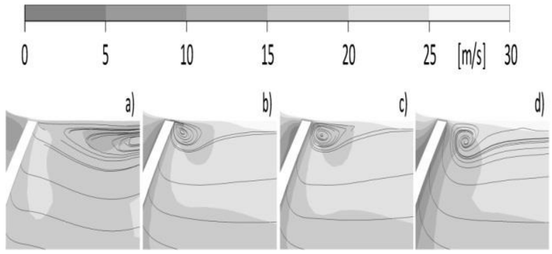

Figure 6. Comparison between the tip leakage flow, at flow rate, Q = 4,000 m³/h for: (a) water (H2O), (b) tomato paste (TOM), (c) Virtual Fluid (N1) and (d) Virtual Fluid (N11). The contour plots represent the magnitude of relative velocity (velocity in the rotating frame of reference).

Slawomir Dykas and Andrzej Wilk considered a single stage centrifugal pump having high rotational speed, high head and low discharge characteristic; it is designed and manufactured only for experimental work. The main purpose of the research is to determine the flow characteristic of pump by means of CFD analysis. The CFD flow simulation has allowed to make a detailed assessment of the flow field in the parts where measurement is very difficult or even impossible. The numerical data are verified in an experimental facility at Institute of power engineering and turbo machinery of the silesian university of technology. [12]

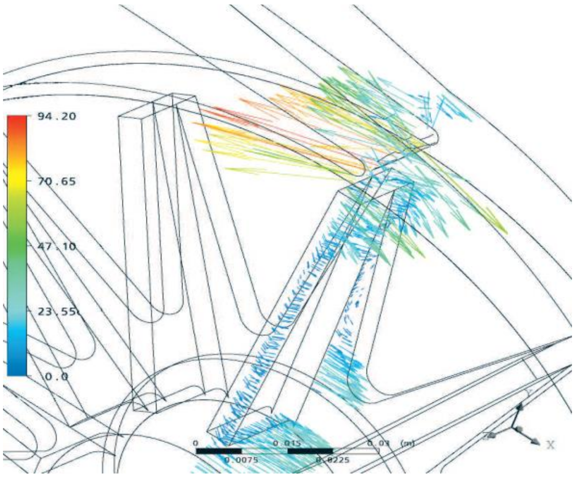

Figure 7. Velocity [m/s] vectors on the *YZ* plane along the main impeller blade And at the inlet to the outlet part of the diffuse

Y.Nagendra Babuand and K. Aparanaanalysed the effect varying vane angle on domestic centrifugal pump using CFD and structural analysis. Also static analysis is performed on impeller to determine the stresses by applying the rotational velocity using different material. The values of stresses and displacement are less when using Glass fiber reinforced plastic compared to structural steel, aluminium alloy and borosilicate glass. By observing CFD analysis result, the pressure, velocity and mass flow rates are decreased by increasing the vane angle. So it is concluded that the use of 160° vane angle is better for Glass fiber reinforced plastic material. [13]

Yu Zhang and all used fluid structure interaction (FSI) for the optimization of transient vibration performance of centrifugal pump. The results of FSI simulation are compared to kriging surrogate model which are nearly equal. The final optimized decision variables are obtained using multi island genetic algorithm, a prototype is manufactured using this optimized parameters of the pump. Experimental tests carried out on prototype well agreed with the results of FSI simulation and kriging surrogate model. The result showed that the radial force curve and moment curve exhibited cyclical fluctuations. [14]

Table 3. Results of Kriging, FSI simulation, and experiment

	KRIGIN G	FSI	EXPERIMEN T
RMS VALU E (mm)	0.334	0.329	0.3447

George Wilson discussed the effect of fine and coarse particle on performance of centrifugal slurry pump. It is clear that the particle size and roughness has a major impact on performance of pump. Consequently it is clear that the performance curves of pump for settling and non -settling slurries are different because they possess different characteristics. It is important to take caution when correcting for high concentration of non- settling slurries with extremely fine particulate such as kaolin clay. It is advisable to do pilot test on pump at manufacturers place if precdent has not been established already. [15]

III. CONCLUSION

The literatures above provide brief information about how the size of particles in fluid affects the power requirement and performance of pump. Also CFD is very useful tool to determine the pump performance without manufacturing prototype of pump. We can conclude that impeller geometry has major impact on the performance as the blade angles of impeller affects cavitation in pump which very crucial phenomenon in the pump which results in failure of the pump. CFD very helpful to determine the safe operation range to avoid cavitation phenomena. Also FSI simulation and kriging surrogate model are very helpful methods to analyze transient vibration in centrifugal pump.

IV. REFERENCES

[1]. Weili, Weidongshi, Zhongyongpan, Xiaopingjiang, Lingzhou, (2013), "Research on transient radial force of centrifugal pump",

Academic journals ISSN 1992-2248, PP 303-310, vol. 9

[2]. Dr.P.Muthu, Alex George, (2016), "CFD Analysis of performance characteristics of centrifugal pump impeller to.minimize cavitation" International Conference on Current Research in Engineering Science and Technology (ICCREST- 2016)

[3]. Deeptesh Singh, Dr.AmitSuhane, M.K.Thakur, (2013), "The study of failure analysis of centrifugal pump on the bases of survey" International Journal of Science and Research (IJSR),ISSN 2319-7064

[4]. Tilahun Nigussie, Edessa Dribssa, (2015), "Design and CFD Analysis of centrifugal pump", International Journal of Engineering Research and General Science, vol. 3, Issue 3, ISSN 2091-2730.

[5]. P.Gurupranesh, R.C.Radha, N.Karthikeyan, "CFD Analysis of Centrifugal Pump Impeller for Performance Enhancement", IOSR Journal of Mechanical and Civil Engineering, e- ISSN 2278-1684, p-ISSN 2320-334X, PP 33-41.

[6]. AjithM.S, Dr.JeojuM.Issac, (2015), "Design and Analysis of Centrifugal Pump Impeller Using Ansys Fluent ", International Journal of Science, Engineering and Technology Research, vol. 4, Issue 10.

[7]. Raghuvendra S.Muttali, Shweta Agrawal, Harshla Warudkar, (2014), "CFD Simulation of Centrifugal Pump Impeller Using ANSYS CFX", International Journal of Innovative Research in Science Engineering and Technology (IJIRSET), vol. 3, Issue 8, ISSN 2319-8753.

[8]. R.M.Pande,S.U.Kandharkar,R.B.Patthe,V.M.Nandedkar,V.B.Tungikar,(2015),"Computational Fluid Dynamics(CFD) of Centrifugal Pump to Study the Cavitation Effect", International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME),vol.4,Issue2,ISSN2319-3182.

[9]. T.Sivakumar,P.Aravind,U.AravindhKumar,R.Balamurugan,V.BharathiPriyan(2016)," PERFORMANCE ANALYSIS OF MIXED FLOW PUMP IMPELLER USING CFD", International Journal of Recent Trend In Engineering and Research (IJRTER),vol.2,Issue4,ISSN2455-1457.

[10]. SatishM.Rajmane, Dr.S.P.Kallurkar, (2015), "Flow Distribution Network Analysis Discharge Side of Centrifugal Pump", International Journal Innovations in Engineering Research and Technology (IJIERT), vol. 2, Issue 7, ISSN 2394-3696.

[11]. C.Buratto, M.Pinelli, P.R.Spina, A.Vaccuri, C.Verga (2015), "CFD Study on Special Duty Centrifugal Pump Operating With Viscous and Non- Newtonian Fluids",

[12]. SlawomirDykas, AndrzejWilk (2008), "Determination of the Flow Characteristics of A High Rotational Centrifugal Pump By Means of CFD Methods ",

[13]. Y.NagendraBabu, K.Aparana (2016), "Evaluation of Varying Vane Angle of Domestic Centrifugal Pump Using CFD and Structural Analysis ", International Journal Mechanical and Production Engineering, vol. 4, Issue 10, ISSN 2320-2092.

[14]. Yu Zhang, Sanbao Hu, Yunqing Zhang, Liping Chen (2014), "Optimization and Analysis of Centrifugal Pump Considering Fluid Structure Interaction", The Scientific World Journal, volume 2014, Article ID 131802.

[15]. George Wilson, "The effect of slurries on centrifugal pump performance".