International Journal of Advance Engineering and Research Development (IJAERD)
Volume 1,Issue 6,June 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406

A Critical review on CFD Analysis of centrifugal pump impeller

Kapil Pandya[1], Chetankumar M.Patel[2]

[1]M. Tech. Scholar, Mechanical Engineering Department, RK University, Rajkot, Gujarat, India, kapild1984@gmail.com

[2]Assistant Professor, Mechanical Engineering Department, SOE RK University, Rajkot, Gujarat, India, chetan.patel@rku.ac.in

Abstract: The main objective of this work is to go through various approaches used in CFD analysis of centrifugal pump and highlight the advantages and application of CFD analysis in turbo industries. The CFD analysis is the advanced tool to overcome the limitation of conventional method to design the pump. Now a days CFD analysis is very familiar approach to improve the design of centrifugal pump and optimize it's operational parameters like Head, Power, Discharge and Speed. The performance prediction of centrifugal pump using experimental approach cannot fulfill the desired outcome for the researcher. The CFD analysis provides more options towards the evaluation and synthesis of pump.

Key Words: Mixed flow pumps, Computational Fluid dynamics, Impeller design, Performance prediction.

I. INTRODUCTION

Computational fluid dynamics investigation is very useful tool in designing of machinery in turbo commercial enterprise. With the aid of the CFD proceed towards, the compounded internal flows in water pump impellers, which are not fully comprehend yet, can be well augur, to speed up the pump design procedure. Computational fluid dynamics (CFD) analysis is being progressively applied in the design of centrifugal pumps. [1]

Recent advances in computing power, in junction with powerful graphics and interactive 3D influence of models have made the process of creating a CFD model and analyzing results much less labour intensive, lesser time and, hence, cost. Thus, CFD is any dominant tool for pump designers. The application of CFD tools in industry which uses turbo machinery is quite common today. Many tasks can arithmetically be solved much faster and cheaper than by means of experiments. Nevertheless the highly wobbly flow in turbo machinery raises the question of the most appropriate method for modeling the rotation of the impeller. [1]

II. LITERATURE Review:

Jekim J. Damor,et.al., conducted experimental investigation, on centrifugal water pump with a 111 mm outlet impeller diameter, backward curved blades, formal discharge of 4.00 lps and 12 m of head to assess the consequence of various operating state like Head, Discharge, Power and Speed on the performance of the pump. Further the impeller is modeled using Solid works software and Computational Fluid Dynamics (CFD) analysis is carried out using ANSYS CFX software on the developed model of impeller to predict the performance virtually and to verify with the experimental result of the pump. [1]

Sujoy Chakraborty, et.al.Carried out two dimensional study of steady, static pressure given out and incompressible flow characteristics inside the passage with different numbers of blades of centrifugal pump impeller. The investigation focuses mainly on the effective efficiency of the pump. Centrifugal pump with impeller blades 5, 6 and 7 have been prepared and its efficiency at 3000 rpm is assess by FLUENT 6.3 software. The numerical analysis displays that with the increase of the head, blade number, the head and static pressure of the model become greater, but the efficiency of centrifugal pump varies with number of blades. [2]

S. C. Chaudhary,et.al, describes an enhance the head of mixed flow pump impeller, Computational Fluid Dynamics (CFD) analysis is one of the advanced CAE tools used in the manufacturing of pump . By performing CFD analysis, the velocity and pressure in the outlet of the impeller is augur. The optimum inlet and outlet vane angles are find out for the existing impeller by using the empirical correlation. The CAD models of the mixed flow impeller with optimum inlet and outlet angles are modeled using CAD modeling software Solid Works 2009. By alteration the outlet blade angle and the Numeral of blade of impeller the head of the impeller is improved .From this analysis it is understood that the changes in the inlet angle of blade and Numeral of blade change the head of the impeller. From the CFD investigation the head of the impeller with optimum blade angles is calculated. Thus, head of the mixed flow impeller is improved by changing the inlet and outlet angels of blade and Numeral of blade. [3]

A. Manivannan,et.al**,** performed detailed CFD analysis to predict the flow pattern inside the impeller which is an active pump component. From the results of CFD investigation, the velocity and pressure in the outlet of the impeller is anticipated. These outlet flow states are used to calculate the efficiency of the impeller. The most appropriate inlet and outlet vane angles are calculated for the existing impeller by using the empirical correlations. The CAD models of the mixed flow impeller with optimum inlet and outlet angles are modeled using CAD modeling software. To find the relationship between the vane angles and the impeller performance the optimum vane angle is achieved in various stages. Three CAD models of centrifugal pump are modeled with the vane angles between existing and most appropriate values. These models are analyzed individually to find the performance of the impeller. [4]

Lamloumi Hedi et al., studied Numerical simulation of the three-dimensional fluid flow inside a centrifugal pump The numerical simulation is used for solving governing equations of incompressible viscous/turbulent flows through the pump. The k-ε turbulence model is choosing to describe the turbulent flow action. In this study, the viscous Navier-Stokes equations are used to simulate the flow inside the vane less impeller and volute Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD proceed towards, the complicated internal flows in water pump impellers, which are not fully comprehend yet, can be well estimated, to speed up the pump design procedure. A solution method is developed to gain three-dimensional velocity and pressure distribution within a centrifugal pump. The method is based on solving completely elliptic partial differential equations for the conservation of mass and momentum. [5]

Rakesh Joshi, performed approach to analyze the pressure and velocity distribution inside the pump passage and evaluate the pump performance with the help of Fluent, a computational fluid dynamics simulation tool. A numerical model of an impeller and casing has been generated and the complex internal pressure and velocity distribution are by applying fluent computational code. Pressure and velocity distribution inside of the centrifugal pump has direct influence due to change of stream wise location. [6]

Mehta Mehul P, performed the CFD analysis, the velocity and pressure in the exits of the impeller is forecast. These outlet flow conditions are used to determine the efficiency of the impeller. The incomparable inlet and outlet vane angles are calculate for the existing impeller by using the empirical correlations. In the first case outlet angle is become greater, and second case inlet angle is become smaller obtain from the CFD analysis, it is lesser outlet recirculation or it is increase outlet recirculation flow cause to improve efficiency. By changing the outlet angle the head of the impeller is improved. Finally, from CFD analysis the calculated efficiency of the impeller with optimum vane angle can be improved by changing the inlet and outlet angle. The Head turn out by this analysis would be higher.

A. V.S. Kadam, carried numerical analysis out with multiple frames of reference to predict the flow field inside the entire pump casing and impeller. [7]

III. METHODOLOGY OF CFD ANALYSIS

Following figure 3.1 describes methodology of CFD analysis.

International Journal of Advance Engineering and Research Development (IJAERD)
Volume 1,Issue 6,June 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406

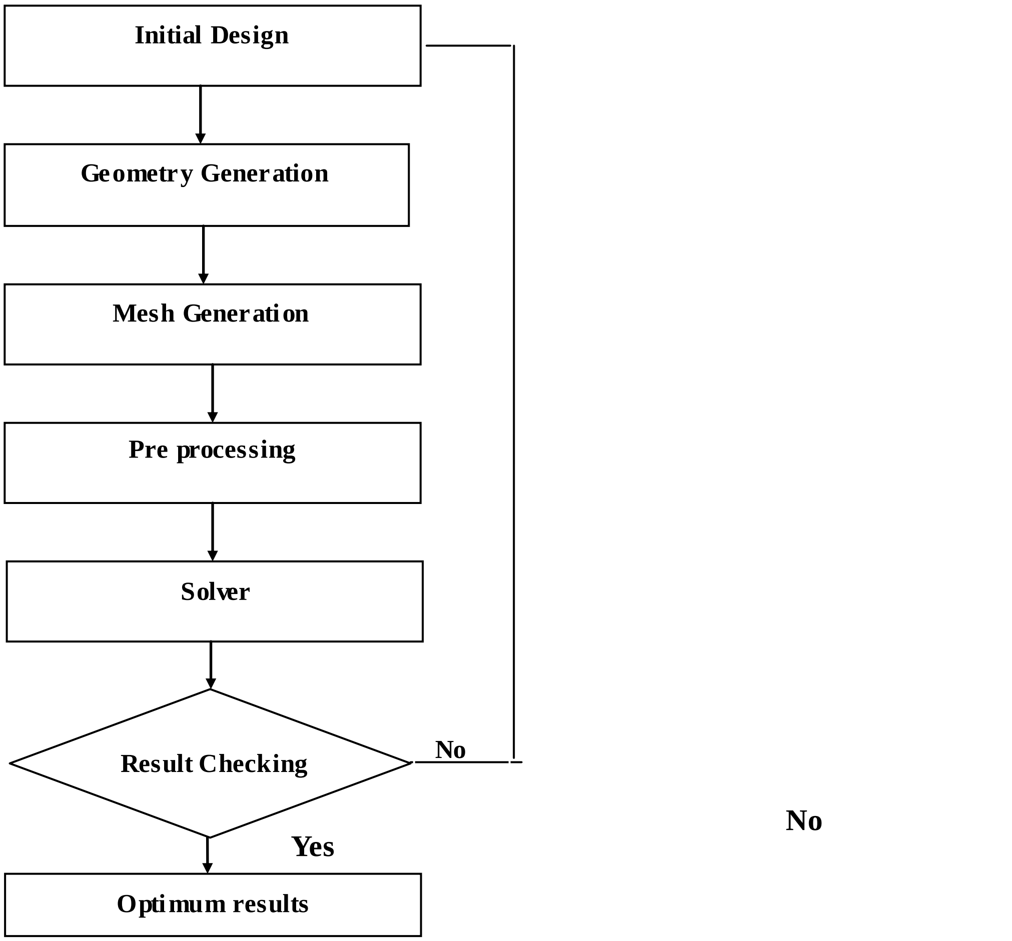

Fig. 3.1 Flow chart of CFD[7]

3.1 Initial Design:

Initial design of model is the planning decision and the geometry is generated depending on this initial design considerations, using CFD model tool or other design, which is used to input the problem geometry, generates the grid; define the flow perameter and the boudary condition to code. [6]

3.2 Geometry generation:

The geometry of design needs to be created from the initial design. Any modeling software can be used for modeling and then shifted to some other simulation software for the analysis purposes.[6]

3.3 Mesh Design Generation:

Mesh generation (Gridding) is the process for subdividing a region to be modeled in to set of small control volume. Associated with each control volume there will be one or more value of the dependent flow variable (e.g. velocity, pressure and temperature etc.) usually they represents some local type of averaged values. Numerical algorithms constituting representing approximations to the conservations of law of mass, momentum and energy are then used to compute these variables in each control volume. [6]

3.4 Pre processor

Pre processor is used to input the problem geometry, generates the grid; define the flow parameter and the boundary conditions to code. [6]

3.5 Solver

Solver is used to get governing equations of the flow subject to the conditions provided there are three different methods to use as given below:

- **Finite difference method:** Finite difference method utilized Taylor series expansion to write the derivatives of the variables as the differences between the values of variable at various points in space or time. [6]

- **Finite elements method:** In the finite element method, the fluid domain under the consideration is divided in to small number of sub domains, known as element. The simple function is assumed for the variation of each variable in each element. The summation of the variations of variable in each element is used to describe the whole fluid flow. [6]

- **Finite volume method:** Finite volume method is currently the most popular in CFD. The main reason is that it can resolve all difficulties that the other two have. Generally the finite volume method is the special case of finite element method. [6]

3.6 Post processor

Post processor is used to message the data and show the results graphically and easy to read format. [6]

IV. CFD ANALYSIS APPROACH

Following approaches are used for CFD analysis.

4.1 performance prediction approach

Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps, where the performance of pump deteriorates [8]. With the aid of the CFD approach, the complex internal flows through the different components of pump can be studied at different operating conditions which help in improvement in the performance at off-design conditions.

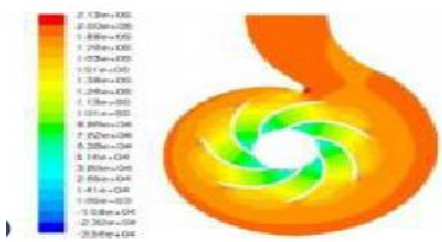

Fig. : 4.1 : static pressure contours in the pump[9]

Mentzos et al. [9] simulated the flow through the impeller of centrifugal pump using finite-volume method along with astructured grid system for the solution of the discretized governing equations. The CFD technique was applied to predict the flow patterns, pressure distribution and head-capacity curve. It was reported that, although the grid size was not adequate to investigate the local boundary layer variables, global ones were well captured. The proposed approach was advocated forthe basic understanding of the flow at various operating points.

Shah et al. [10] carried out steady state simulation of 200 m^3/hr capacity centrifugal pump using using RANS equations. The non-uniformities were observed in different parts of the pump at off-design conditions which resulted in the decrease in efficiency. The k-ω SST turbulence model provided better results compared to RNG k-e model. The operating characteristic curves predicted by the numerical simulation were compared with the results of model testing and were found in good agreement. The static pressure contours in the pump at rated discharge are shown in Fig.4.1

4.2 Parametric Approach

CFD helps in prediction of flow behavior in different parts of the hydraulic machines before actually manufacturing them. In case of modification of existing systems, the modifications can be incorporated in numerical model and their effects can be predicted before implementing them. CFD analysis helps in studying the effects of various parameters, independently as well as by forming the non dimensional groups, on pump performance.

Bacharoudis et al. [11] analyzed the performance of pump by varying the outlet blade angles by keeping the same outlet diameter. The numerical simulation of 3-D, incompressible Navier-Stokes equations was carried out with a commercial CFD finite-volume code. At nominal capacity, when the outlet blade angle was increased from 20° to 50°, the head was increased by more than 6% but the hydraulic efficiency was reduced by 4.5%. However, at high flow rates, the increase of the outlet blade angle caused a significant improvement of the hydraulic efficiency.

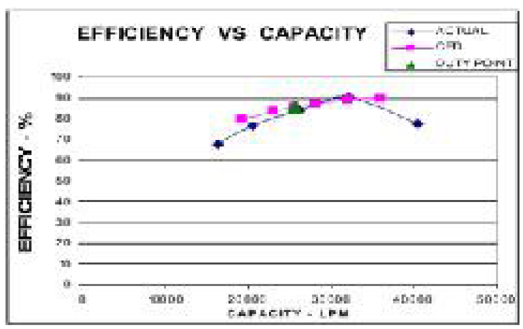

Fig. : 4.2 : Efficiency Vs Capacity curve[12]

Patel and Rama Krishnan [12] numerically studied the effects of changing hub curve profile and stator angle in mixed flow pump at duty point and at part load. The analysis concluded that: (i) the nature of head & power versus capacity curves obtained was similar to that of standard mixed flow pump (ii) pump efficiency was predicted within + 5% range at duty point. However, more variation was observed at off-design conditions and (iii) efficiency was improved by 1% after matching stator angle and changing hub curve profile. The efficiency versus capacity curves, actual and predicted by CFD analysis, are shown in Fig.4.2.

4.3 Cavitations approach

Cavitation may occur in different regions of the pump when local pressure goes below the vapour pressure correspond to fluid temperature. The mechanism of cavitation erosion has been studied for more than a hundred years, but until now there has been no general theory of cavitation erosion damage to analytically calculate cavitation erosion rate in impellers of centrifugal pumps or to evaluate erosion intensity at the pump design stage.

Medvitz et al. [13] used multi-phase CFD method to analyze centrifugal pump performance under cavitating conditions. The homogeneous two phase RANS equations were used wherein mixture momentum and volume continuity equations were solved along with vapor volume fraction. Performance trends of partial discharge and blade cavitation, including breakdown, were observed and compared qualitatively with experimental measurements.

Nohmiet al. [14] studied the cavitation flow in a low specific speed centrifugal pump with compressible air-vapor-liquid two-phase medium (TE model) and constant enthalpy vaporization (CEV) model. The study revealed that, at the high flowrate cavitation bubbles appear at the leading edge on pressure side and the head drops gradually. The TE model was able to predict the gradual head drop but the computations were found to be unstable; whereas, CEV model was unable to predict the gradual head drop. In both the codes, further modification was recommended to achieve stable and accurate results.

Caridad et al. [15] carried out numerical analysis in a centrifugal pump impeller of submersible pump conveying an air– water mixture, which was similar to cavitating flow. A sensibility analysis with regard to the gas-void fraction and the bubble diameter was performed. The variations in impeller head and relative flow angle at the outlet were presented as a function of liquid flow rate and phase distribution within the impeller. It was found that, larger bubble diameter lead to larger head experimented by the impeller. The numerical results and diffuser losses showed excellent agreement with the experimental results.

4.4: Mini/micro pump analysis approach

For the turbo-pump being the key machine for liquid transportation, its further development is always desirable. Impeller diameter between 5 mm and 50 mm is defined as mini pump.
Liu et al. [16] carried out experimental and numerical studies on impeller-geometry of mini turbo-pump. The law of similitude was observed for the pump characteristics in the range of Reynolds number larger than 1.0×10^5. The effect of tip clearance was found to be attenuated by the impeller geometry such as larger outlet blade angle. It was concluded that, numerical 3-D flow analysis based on RANS equations with k-ω turbulence model may be reasonably applicable to study the hydraulic performance of mini impellers.

Tsui and Lu [17] analyzed the unsteady flow field prevailing in the valve less micro pump by using both the CFD and the lumped-system method. The moving membrane was modeled by imposing a reciprocating velocity boundary condition. In the multidimensional simulation, the Navier–Stokes equations were solved using a finite volume method suitable for the use of unstructured grids. It was reported that the variation of the flow rate ratio is quite different in the lumped-system analysis compared with the multidimensional calculations, due to negligence of inertial effect

CONCLUSION

From the CFD analysis, the velocity and pressure in the outlet of the impeller can be used to calculate the efficiency of the existing impeller by using the empirical relations.
CFD analysis is one of the very important approaches to enhance the design of mechanical system such as centrifugal pump.
By studying CFD analysis in case of centrifugal pump, improvement of design can be done by comparing it with experimental analysis.

As the experimental analysis and Conventional Methods for pump designing in turbo industry consumed more time and less accurate result with expenses.
In case of CFD Analysis the model of pump can be generated using modeling software and can be analyzed in ANSY .These approach of analysis gives better results using finite element method compare to Experimental performance Test.

REFERENCES

[1] Jekim J. Damor, Dilip S. Patel, Kamlesh H.Thakkar, Pragnesh K. Brahmbhatt; Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study; IJERT

[2] Sujoy chakraborty, Kishan choudhary, Pransenjit datta, Bishop Debbarma; Performance prediction of centrifugal pumps with variations of blade number; Journal scientific & industrial research.

[3] S. C. Chaudhari, C. O. Yadav & A. B. Damor; a comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump; IMPACT

[4] A. Manivannan; Computational fluid dynamics analysis of a mixed flow pump impeller; International Journal of Engineering, Science and Technology.

[5] Lamloumi Hedi, Kanfoudi Hatem, Zgolli Ridha; Simulation study and Three-Dimensional Numerical Flow in a Centrifugal Pump; International Journal of Thermal Technologies.

[6] Rakesh joshi; Computation investigation of flow field in a centrifugal slurry pump; Mechanical engineering department; Thapar University; Patiala

[7] V.s. kadam, S.s. gawade, H.h. mohite, N.k.chapkhane; Design and development of split case pump using computational fluid dynamics; Institute of technology, Nirma university, Ahmedabad

[8] Hedi, L., Hatem, K.,Ridha, Z., 2010."Numerical flow simulation in a centrifugal pump," International Renewable Energy Congress. Sousse, Tunisia. pp. 300-304

[9] Mentzos, M., Filios, A., Margaris, P., Papanikas, D., 2005. "CFD predictions of flow through a centrifugal pump impeller," Proceedings of International Conf. Experiments/Process/System Modelling/ Simulation/Optimization. Athens, pp. 1-8.

[10] Shah, S., Jain, S.,Lakhera, V., 2010. "CFD based flow analysis of centrifugal pump," proceedings ofInternational Conferenceon Fluid Mechanics and Fluid Power. Chennai, India, paper#TM08.

[11] Bacharoudis, E.,Filios, A.,Mentzos, M.,Margaris, D., 2008.Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle. The Open Mechanical Engineering Journal 2, p. 75.

[12] Patel K.,Ramakrishnan, N., "CFD analysis of mixed flow pump."

[13] Medvitz, R., Kunz, R., Boger, D., Adam, J.,Yocum, A., Pauley, L., 2002. Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD. Journal of Fluid Engineering 124, p. 377.

[14] Nohmi, M.,Goto, A.,Iga, Y.,Ikohagi, T., 2003."Cavitation CFD in a centrifugal pump," Proceedings of International Symposium on Cavitation. Osaka, Japan, pp. 1-7.

[15] Caridad, J., Asuaje, M.,Kenyery, F.,Tremante, A.,Aguillon, O., 2008. Characterization of a Centrifugal Pump Impeller under Two-Phase Flow Conditions. Journal of Petroleum Science and Engineering 63,p. 18.

[16] Liu, S.,Michihiro, N., Yoshida, K., 2001. Impeller Geometry Suitable for Mini Turbo-Pump.Journal of Fluids Engineering 123,p. 500.

[17] Tsui, Y., Lu, S., 2008.Evaluation of Performance of a ValvelessMicropump by CFD and Lumped System Analyses. Sensors and Actuators 148, p. 138.