International Journal of Advance Engineering and Research Development (IJAERD)
Volume 1,Issue 5,May 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406

CFD ANALYSIS AND MODIFICATION OF OPEN WELL CENTRIFUGAL PUMP : A REVIEW

ASIF F. Hathiyari[1], Vikram H. Modi[2]

[1]*M. Tech. Scholar, Mechanical Engineering Department, RK University, Rajkot-* 360020, *Gujarat, India, asifhathiyari@gmail.com*

[2]*Assistant Professor, Mechanical Engineering Department, RK University, Rajkot* -360020, *Gujarat, India, vikram.modi@rku.ac.in*

Abstract: The centrifugal pump is the most used pump type in the world. The flow analysis inside the centrifugal pump is highly complex mainly due to 3D flow structure involving turbulence, secondary flow, cavitation and unsteadiness. In recent years, a growing availability of computational resources and progress in the accuracy of numerical methods brought turbo machinery Computational Fluid Dynamics (CFD) methods from pure research work into the competitive industrial markets. The critical review of CFD analysis of centrifugal pumps along with the future scopes for further improvement is presented in this paper. CFD technique has been applied by the researchers to carry out different investigations on centrifugal pumps viz. performance prediction at design and off-design conditions, parametric study, cavitation analysis, diffuser pump analysis, performance of pump running in turbine mode etc. Unsteady Reynolds-averaged Navier–Stokes equations together with two equation k-ε turbulence model were found to be appropriate for CFD analysis of centrifugal pump. Volute flow study and impeller-volute interaction appeared as an interesting research fields for the further improvement in the pump performance. The most active areas of research and development are the analysis of two phase flow, pump handling non-Newtonian fluids and fluid–structure interaction.

Keywords: CFD, Centrifugal pump, review, flow analysis, paper.

I. INTRODUCTION

The complexity of the flow in any turbomachine is due to the three dimensional blade geometry, turbulence, secondary flows, unsteadiness etc. Computational fluid dynamics (CFD) has successfully contributed to the prediction of the flow through pumps and the enhancement of their design. Various researchers have considerably contributed to reveal the flow mechanisms inside centrifugal pump impellers with spiral volute or vaned diffuser volute aiming to the design of high performance centrifugal turbomachines. [1]

The application of Computational Fluid Dynamics (CFD) in design and optimisation of turbomachinery is state of the art today. Since the flow in a turbomachine is always turbulent, adequate turbulence modelling is necessary in order to obtain accurate predictions of the flow. In almost every industrial CFD simulation a RANS turbulence model is used. The information provided by this approach is sufficient, if only the mean flow is of interest. In general, quite good agreement with experiments can be obtained from classical RANS computations. However these models suffer from predicting details like flow separation or anisotropic turbulence. The question to be answered in this investigation is weather an unsteady RANS model is able to capture the rotating stall cells and their movement. Since the rotating stall is a strongly turbulent phenomenon, a turbulence resolving model is expected to give superior results. Pure LES requires very fine grids especially in boundary layers and the generation of boundary conditions is more complex. For this reason a turbulence resolving URANS model was used in the present investigation. [2]

II. LITERATURE REVIEW

Pump designers are continually being challenged to provide machines that operate more efficiently, quietly and reliably at lower cost. Many investigators have applied CFD as a numerical simulation tool to carry out different investigations on centrifugal pumps. This section describes the research work carried out by the researchers in the centrifugal pumps by CFD approach.

A Cavitation analysis

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.

ZHU Bing et al [3]: This paper aims to clarify the cavitation suppression mechanism of the gap structure impeller based on the analysis of cavitation characteristics in a low specific speed centrifugal pump. In order to obtain reliable and consistent numerical results, some numerical considerations and modeling methodology were demonstrated and researched, and a check of the time and space resolution were also conducted. Hence the predicted cavitation performance of the two centrifugal pumps were investigated and compared with experimental results, and they were in qualitative agreement. It was confirmed that the new gap structure impeller has a very good characteristic of inhibiting cavitation, especially in large flow area, the present numerical method can effectively capture the major internal flow features in the centrifugal pump, through the comparison of the two type impeller flow fields, the cavitation suppression mechanism of the gap impeller may be the combination effects of the small vice blade’s guiding flow and gap tunnel’s auto-balancing of pressure.

B Parametric study

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.

R.Spence et al. [4]: presented form of a parametric study covering four geometric parameters, namely the cutwater gap, vane arrangement, snubber gap and the sidewall clearance. Taguchi methods allowed the number of transient analyses to be limited to a total of 27. Three flow rates were investigated and the pulsations were extracted at 15 different locations covering important pump regions. Taguchi post-processing analysis tools were used to rank the relative importance of the four geometric parameters at each location for each flow rate. The cutwater gap and vane arrangement were found to exert the greatest influence across the various monitored locations and the flow range. A rationalisation process aimed at increased component life and reduced noise/vibration through reductions in pressure pulsations has produced geometric recommendations, which should be useful to designers.

C Pump running in turbine mode:

Shahram Derakhshan et al [5]: studied the best efficiency point of an industrial centrifugal pump running as turbine was achieved using a theoretical analysis. This method tries to estimate hydraulic components of reverse (turbine) mode using direct (pump) mode. In the next step, the pump was simulated in direct and reverse modes by computational fluid dynamics. 3D full Navier–Stokes equations were solved using Fine Turbo V.7 flow solver. Using numerical results, complete characteristic curves of the pump in direct and reverse modes were obtained. For experimental verification of theoretical and numerical results, the pump was tested as a turbine in a test rig. All required parameters were measured to achieve complete characteristic curves of the reverse pump. The theoretical and numerical results were compared with experimental data and some other methods.

Two equations were presented to estimate the complete characteristic curves of centrifugal pumps as turbines based on their best efficiency point. Deviation of suggested method from experimental data were considered and discussed. Finally, a procedure was presented for selecting a suitable pump to work as a turbine in a small hydro-site.

A complete model of a mini hydropower plant was installed in University of Tehran as shown in Fig.2.3. The flow rate and head for each PAT were generated by a proper pump in the experimental setup.

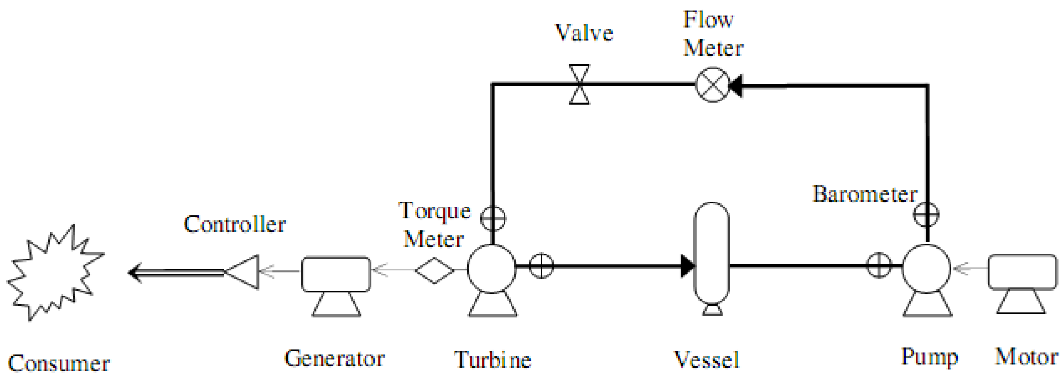


Figure 2.3. The mini hydropower established in the University of Tehran [5]

Punit Singh et al [6] **:** studied the effects of impeller rounding on a combination of radial flow and mixed flow pumps as turbines using experimental data. The impeller rounding is seen to have positive impact on the overall efficiency in different operating regions with an improvement in the range of 1–3%. The behaviour of the two control variables have been elaborately studied in which it is found that the system loss coefficient has reduced drastically due to rounding effects, while the extent of changes to the exit relative flow direction seems to be limited in comparison. The reasons for changes to these control variables have been physically interpreted and attributed to the behaviour of the wake zone at the turbine inlet and circulation within the impeller control volume.

The larger picture of impeller rounding has been discussed in comparison with performance prediction models in pumps as turbines. The possible limitations of the analytical model as well as the test setup are also presented. The paper concludes that the impeller rounding technique is very important for performance optimization and recommends its application on all pump as turbine projects. It also recommends he standardization of the rounding effects over wide range of pump shapes including axial pumps.

D Performance prediction at different operating conditions

Wen-Guang Li et al [7] : Studied Centrifugal pump performances using water and viscous oil as working fluids whose kinematic viscosities are 1 and 48 mm^2/s, respectively, the flows in the centrifugal pump impeller are also measured accurately by using a two-dimensional laser Doppler velocimeter (LDV) in best efficiency and part-loading points, while the pump is handling two kinds of working fluids. The effects of the viscosity on the performance and flow pattern within the impeller are established based on the experimental results. The high viscosity results in rapid increases in the disc friction losses over outsides of the impeller shroud and hub as well as the hydraulic losses in flow channels of the pump. The flow patterns near the impeller outlet are little affected by the viscosity of the fluids, but those near the impeller inlet are greatly affected by the viscosity. There is a wide wake near the blade suction side of the centrifugal pump impeller. The flow pattern is essentially different from the well-known jet/wake model.

The results of the study brought out following finding.

(1) The reason why a centrifugal pump performance goes down when the pump handles high viscosity working fluids, is that high viscosity results in a rapid increase in the disc friction losses over outsides of the impeller shroud and hub as well as in hydraulic losses in flow channels of the pump.

(2) The flow patterns near the impeller outlet are less affected by the viscosity of the fluids in best efficiency and part loading points, but the flow patterns near the impeller inlet are greatly affected by the viscosity.

(3) There is a wide wake near the blade suction side of the centrifugal pump impeller, there is not a jet near the blade pressure side, and the flow pattern is essentially different from the well-known jet/wake model.

B. Jafarzadeh et al [8] : presented a general three-dimensional simulation of turbulent fluid flow to predict velocity and pressure fields for a centrifugal pump. A commercial CFD code was used to solve the governing equations of the flow field. In order to study the most suitable turbulence model, three known turbulence models of standard k–e, RNG and RSM were applied. The complex flow configuration required us to use around 5,800,000 cells, and 12 computational nodes (processors) for parallel computing. Simulation results in the form of characteristic curves were compared with available experimental data, and an acceptable agreement was obtained. Additionally, effect of number of blades on the efficiency of pump was studied. The number of blades was changed from 5 to 7. The results show that the impeller with 7 blades has the highest head coefficient. Finally, it was observed also that the position of blades with respect to the tongue of volute has great effect on the start of the separation. Thus, to analyze the effect of blade number on the characteristics of the pump, the position of blade and tongue should be similar to each other. Investigations of this kind may help to reduce the required experimental work for the development and design of such devices.

Wei Han et al [9]: Presented the solid-liquid two-phase unsteady flow in the screw centrifugal pump is simulated during one revolution by using CFD software. The pressure fluctuations at the volute outlet ，the distribution of radial thrust and axial thrust in the screw centrifugal pump during one revolution are given in the paper.

The results show that the minimum pressure value at the volute outlet occurs when the maximum impeller radius just turned the tongue of volute, and maximum value occurs when the maximum impeller radius just turned between the section VI and VII; the radial thrust on the impeller is assumed the circumference distribution approximately during one revolution. As the solid volume fraction increases, the radial thrust increases gradually. The direction of radial thrust is inverse against the maximum value of pressure in the pump outlet; the maximum radius position has much influe nce on the changes of axial thrust, the axial thrust maximum value occurs when the maximum impeller radius just turned the tongue of volute, and minimum value occurs when the maximum impeller radius is away from the tongue of volute.

Lucius et al [10]: demonstrated the applicability of an eddy resolving turbulence model in a turbomachinery configuration. The model combines the Large Eddy Simulation (LES) and the Reynolds Averaged Navier Stokes (RANS) approach. The point of interest of the present investigation is the unsteady rotating stall phenomenon occurring at low part load conditions. Since RANS turbulence models often fail to predict separation correctly, a LES like model is expected to give superior results. In this investigation the scale-adaptive simulation (SAS) model is used. This model avoids the grid dependence appearing in the Detached Eddy Simulation (DES) modelling strategy. The simulations are validated with transient measurement data. The present results demonstrate that both models are able to predict t the major stall frequency at part load. Results are similar for URANS and SAS, with advantages in predicting minor stall frequencies for the turbulence resolving model.

R. Barrio et al [11]: This paper explores the use of a commercial CFD code to estimate the total radial load on the impeller of two test pumps. The full 3D-URANS equations were solved for several flow rates between 10%–130% of rated conditions. The predictions were validated with experimental data of global characteristics and unsteady pressure distribution round the impeller. The code was used to estimate the total radial load (steady and unsteady components) on the impellers as a function of flow rate. It was observed that the unsteady component can represent about a 40%–70% of the average magnitude when operating at off-design conditions.

A cad model was generated first for the test pumps and discretized subsequently into a mesh. The model comprises three modules, each created and meshed independently: (i) a small pipe portion at the inlet of the pump, (ii) impeller, (iii) volute and outlet duct. Fig. 3.8 presents a detail of the fluid region within the impeller module of pump B (a), showing the geometry of the blades, and also the surface mesh of the volute (b). The numerical model for pump A is very similar to that of pump B, only excluding the differences due to specific geometric details. The mesh was generated by means of tetrahedral cells (see Fig.2.4) to assure an adequate adaptation to the complexity of the geometry.

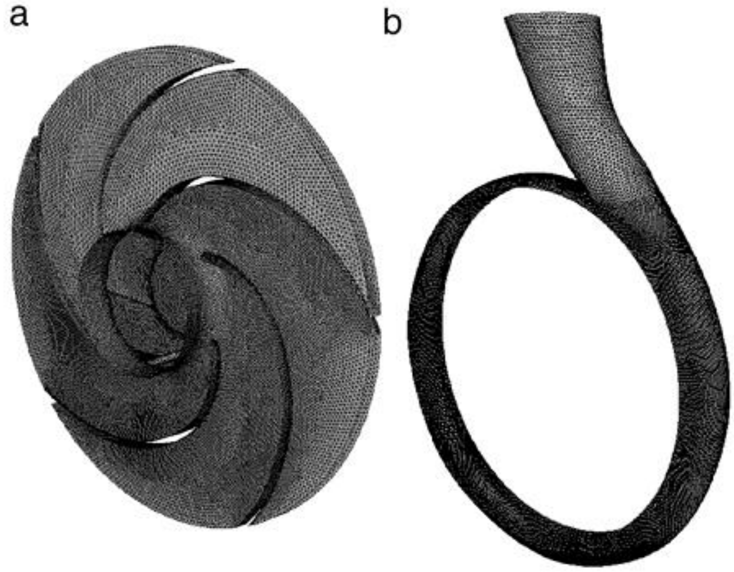
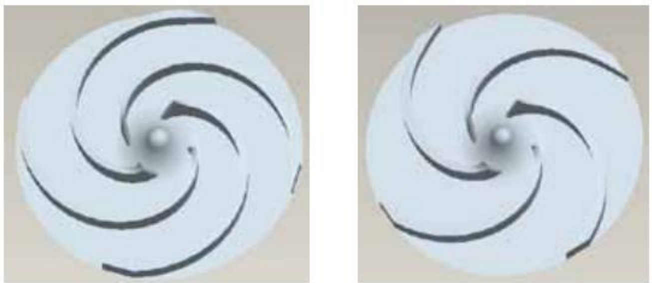


Figure 2.4. Impeller fluid region (a) and volute surface mesh (b) [11]

Li Wei et al [12]: In this paper to achieve the high passing capacity, high efficiency and non-overload performance of a sewage pump, the type of WQS120-60-45 sewage pump is selected as an example to be researched. The theory analyses and numerical simulation were conducted on the internal two-phase flow of the pump. The distribution of the solid particles in impeller is observed, and the influence of impeller outlet angle on pump efficiency and the ability of solid particles passing through is analyzed. The results showed that the solid particles concentrated on pressure surface of impeller. When impeller outlet angle was higher, local medium energy lost seriously. With the angle decreasing, the blade was more suitable for the flow of the solid particles, which has improved the impeller wearing and increased efficiency. This study can provide a certain reference for optimization design of sewage pump impeller.

All researches in this paper are taken out on the WQS120-60-45 sewage pump; the aim is to simulate the internal solid-liquid two-phase flow. Its main operation parameters are presented as: flow rate Q=120m3/h, head H=60m, rotational speed n=1470r/min, specific speed ns=45. Comparative analysis of the two blade outlet angles is conducted between β2=10°and β2=20°, shown in Fig. 2.5.



(a) β1=10° *(b) β1=20°*

Figure: 2.5 Three-dimensional impeller diagrams of different blade outlet angle [12]

John S. Anagnostopoulos [13]**:** In this paper a numerical methodology is developed to simulate the turbulent flow in a 2-dimensional centrifugal pump impeller and to compute the characteristic performance curves of the entire pump. The flow domain is discretized with a polar, Cartesian mesh and the Reynolds-averaged Navier–Stokes (RANS) equations are solved with the control volume approach and the k–e turbulence model. Advanced numerical techniques for adaptive grid refinement and for treatment of grid cells that do not fit the irregular boundaries are implemented in order to achieve a fully automated grid construction for any impeller design, as well as to produce results of adequate precision and accuracy. After estimating the additional hydraulic losses in the casing and the inlet and outlet sections of the pump, the performance of the pump can be predicted using the numerical results from the impeller section only. The regulation of various energy loss coefficients involved in the model is carried out for a commercial pump, for which there are available measurements. The predicted overall efficiency curve of the pump was found to agree very with the corresponding experimental data. Finally, a numerical optimization algorithm based on the unconstrained gradient approach is developed and combined with the evaluation software in order to find the impeller geometry that maximizes the pump efficiency, using as free design variables the blade angles at the leading and the trailing edge. The results verified that the optimization process can converge very fast and to reasonable optimal values.

Mona Golbabaei Asl et al [14]: This paper presents the model generation, static structural analysis, and geometrical modifications performed for a failed volute casing of a real centrifugal-pump. Failure would be examined under hydrostatic test conditions. Finite Element Method is employed in stage of theoretical problem investigation.

To control failure phenomenon, necessary geometrical modifications are applied to the model. Geometrical modifications must have the least effect on hydraulic performance and avoid excessive manufacturing costs. Finally, some test volute casings with new geometry would be built to experimentally validate the analytical results and inspect the hydraulic performance.

III. CONCLUSION

Flow analysis of centrifugal pump is often a challenging task as it requires critical analysis of highly complex flow which is turbulent and three dimensional in nature and having rapidly changing curvature of flow passage. CFD approach has been extensively used in centrifugal pumps as numerical simulation tool for performance prediction at design and off-design conditions, parametric study, cavitation analysis, analysis of interaction effects in different components, prediction of axial thrust, study of pump performance in turbine mode, diffuser pump analysis etc. URANS equations together with two equation k-ε turbulence model were found to be appropriate to get a reasonable estimation of the general performance of the centrifugal pump, from an engineering point of view, with typical errors below 10 percent compared with experimental data. Impeller and diffuser flows have been studied extensively and volute flow study has appeared as an interesting research

REFERENCES

[1] Hyen-Jun Choi, Mohammed Asid Zullah , Hyoung-Woon Roh , Pil-Su Ha , Sueg-Young Oh , Young-Ho Lee , "CFD validation of performance improvement of a 500 kW Francis turbine", Elsevier Journal, Renewable Energy 54 (2013) 111-123.
[2] P.Usha Sri, 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 December 16-18, 2010, IIT Madras, Chennai, India.
[3] ZHU Bing, CHEN Hong-xun, "Cavitating Suppression of Low Specific Speed Centrifugal Pump With Gap Drainage Blades", Elsevier Journal, 2012, 24(5):729-736 DOI: 10.1016/S1001-6058(11)60297-7.
[4] R. Spence, J. Amaral-Teixeira, "A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump", Elsevier Journal, Computers & Fluids 38 (2009) 1243–1257.

[5] Shahram Derakhshan, Ahmad Nourbakhsh, "Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds", Elsevier Journal, Experimental Thermal and Fluid Science 32 (2008) 800–807.
[6] Punit Singh, Franz Nestmann, "Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines" Elsevier Journal, Experimental Thermal and Fluid Science 35 (2011) 121–134.
[7] Wen-Guang Li, "Effects of viscosity of fluids on centrifugal pump performance and flow pattern in the impeller", Elsevier Journal, International Journal of Heat and Fluid Flow 21 (2000) 207-212.
[8] B. Jafarzadeh, A. Hajari, M.M. Alishahi, M.H. Akbari, "The flow simulation of a low-specific-speed high-speed centrifugal pump", Elsevier Journal, Applied Mathematical Modelling 35 (2011) 242–249.
[9] Wei Han ,Wei Ma ,Rennian Li ,Qifei Li, "The Numerical Analysis of Radial Thrust and Axial Thrust in the Screw Centrifugal Pump" , Elsevier Journal, Procedia Engineering 31 (2012) 176 – 181.
[10] A. Lucius , G. Brenner, "Unsteady CFD simulations of a pump in part load conditions using scale-adaptive simulation", Elsevier Journal, International Journal of Heat and Fluid Flow 31 (2010) 1113–1118.
[11] R. Barrio , J. Fernández , E. Blanco , J. Parrondo, "Estimation of radial load in centrifugal pumps using computational fluid dynamics" Elsevier Journal , European Journal of Mechanics B/Fluids 30 (2011) 316–324.
[12] Li Wei, Weidong Shi, Xiaoping Jiang, Bin Chen, Yanlan Wu, "Analysis on Internal Solid-liquid Two-phase Flow in the Impellers of Sewage Pump", Elsevier Journal, Procedia Engineering 31 (2012) 170 – 175.
[13] John S. Anagnostopoulos, "A fast numerical method for flow analysis and blade design in centrifugal pump impellers", Elsevier Journal ,Computers & Fluids 38 (2009) 284–289.
[14] Mona Golbabaei Asl, Rouhollah Torabi, S. Ahmad Nourbakhsh, "Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing", Elsevier Journal, Engineering Failure Analysis 16 (2009) 1996–2003.

