RESEARCH ARTICLE OPEN ACCESS

CFD Analysis of Centrifugal Pump: A Review

Narayan P. Jaiswal
PG student, A. D. Patel Institute of Technology, New V. V. nagar, Anand, Gujarat, India

Abstract
The main objective of this work is to understand role of the computational fluid dynamics (CFD) technique in analyzing and predicting the performance of centrifugal pump. Computational Fluid Dynamics (CFD) is the present day state-of-art technique for fluid flow analysis. The critical review of CFD analysis of CFD analysis of centrifugal pump along with future scope for further improvement is presented in this paper. Different solver like ANSYS-CFX, FLUENT etc can be used for simulations. Shear stress transport model has been found appropriate as turbulence model. Study of pressure contours, velocity contours, flow streamlines etc can be studied by CFD techniques. Unsteady Reynolds Averaged Navier Stokes (URANS) equations are solved by solver to get flow simulation results inside centrifugal pump. CFD results has to be validated with testing results or with performance characteristics curves. Performance prediction at design and off-design conditions, parametric study, cavitation analysis, diffuser pump analysis, performance of pump running in turbine mode etc. are possible with CFD simulation techniques.

Keywords:- Centrifugal pump, CFD, Numerical simulation, Validation, Performance Characteristics Curve.

I. INTRODUCTION

Centrifugal pump is a most common pump used in industries, agriculture and domestic applications. The centrifugal pump creates an increase in pressure by imparting mechanical energy from the motor to the fluid through the rotating impeller. The fluid enters from impeller eye and flows along its blades. In this centrifugal force developed due to rotation of impeller. This centrifugal force increases fluid velocity and then kinetic energy is transformed to pressure. Figure 1 shows different components of centrifugal pump.

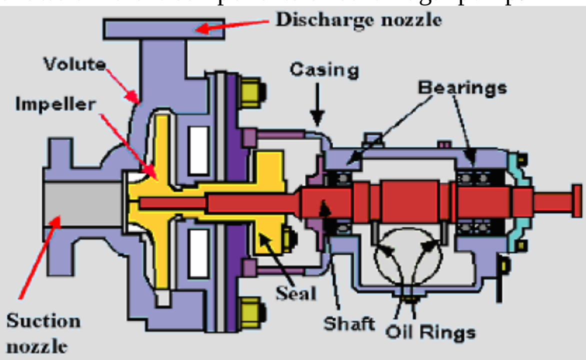

Figure 1. Components of single stage centrifugal pump

Bacharoudis et al.[1] studied the performance of impellers with the same outlet diameter having different outlet blade angles and conclude that at high flow rates, increase in outlet blade angle causes improvement in hydraulic efficiency. Suthep et al.[2] shows that hydraulic efficiency is different at different discharge. It is maximum at $Q/Q_{design} = 0.7$ which is 98%. Shah et al.[3] studied pressure and velocity contours at part load, rated and over rated conditions. Rajendran et al.[4] study of The flow pattern, pressure distribution in the blade passage, blade loading and pressure. Pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. Shah et al.[5] explain role of CFD in performance prediction and analysis of centrifugal pump. CFD technique has been applied 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. Lei et al.[6] explain influence of wrap angle on performance of centrifugal pump.

CFD procedure has been shown in figure 2.

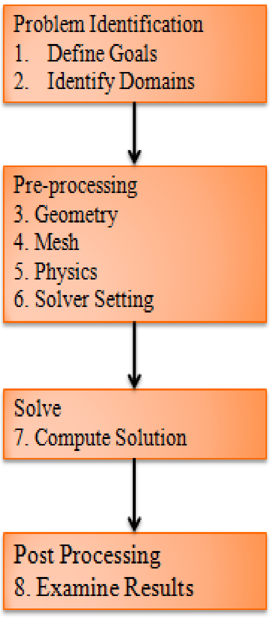

Figure 2. CFD Procedure

Following equation is general form of conservation equation.

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi dV}_{\text{unsteady}} + \underbrace{\oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A}}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{diffusion}} + \underbrace{\int_V S_\phi dV}_{\text{generation}}$$

The fluid region is decomposed into a finite set of control volumes. General conservation equations for mass, momentum, energy etc. are solved on this set of control volumes.

II. LITERATURE REVIEW

Bacharoudis et al.[1] [2008] states that various parameters affect the pump performance and energy consumption. In this study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. The numerical solution of 3D, incompressible Navier-Stokes equations over an unstructured and non uniform grid is accomplished with a commercial CFD finite volume code. For each impeller, the flow pattern and the pressure distribution in the blade passages are calculated and finally the head-capacity curves are compared. When the pump operates off-design, the percentage raise of the head curve, due to the increment of the outlet blade angle, is larger for high flow rates and becomes smaller for flow rates $Q/Q_N<0.65$.When pump operates at nominal capacity, the gain in the head is more than 6% when the outlet blade angle increases from 20^o to 50^o. At high flow rates, the increase of the outlet blade angle causes a significant improvement of the hydraulic efficiency. So through this paper one can know about the effect of outlet blade angle on performance of centrifugal pump.

Kaewnai et al.[2] [2009] represents to use the computational fluid dynamics (CFD) technique in analyzing and predicting the performance of a radial flow-type impeller of centrifugal pump. The duty point of impeller analyzed is : flow rate of 528 m3/hr, speed of 1450 rpm and head of 20 m. In first stage meshing and its refinement in some domain has been done. The second stage different domain identification and its material properties, boundary conditions and convergence criteria of mesh-equipped module. In last stage, various results are calculated and analyzed for different factors affecting impeller performance. The results indicate that the total head rise of the impeller at the design point is approximately 19.8 m. The loss coefficient of the impeller is 0.015 when 0.6 < Q/Qdesign < 1.2. Maximum hydraulic efficiency of impeller is 0.98 at Q/Qdesign = 0.7. Based on the comparison of the theoretical head coefficient and static pressure rise coefficient between simulation results and experimental data. The surface roughness value was found to have a high effect on loss.

Shah et al.[3] [2010] presented numerical simulation of centrifugal pump having 200 m^3/hr capacity using commercial CFD package FLUENT. The simulations were carried out at six different operating points between 30% to 110% discharge, to cover the wide range of operation. It was also observed that the pressure rise was quite gradual and uniform at rated and over rated discharge but it was non-uniform at partial discharge. the variation of velocity is quite uniform at rated discharge compared to part load & over rated discharge. It was found that k-ω SST turbulence model provides better results compared to RNG k-Ɛ model. With increase in discharge, Head decreases, Power input increases and Efficiency of Pump increases. Efficiency is maximum at duty point, after this point as discharge increases, efficiency of pump decreases.

Rajendran et al.[4] [2012] modeled a centrifugal pump impeller and solved using computational fluid dynamics, the flow patterns through the pump, performance results, circumferential area averaged pressure from hub to shroud line, blade loading plot at 50 % span, stream wise variation of mass averaged total pressure and static pressure, stream wise variation of area averaged absolute velocity and variation of mass averaged total pressure contours at blade leading edge and trailing edge for designed flow rate are presented. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. Near leading edge of the blade low pressure and high velocities are observed due to the thickness of the blade. Near trailing edge of the blade total pressure loss is observed due to the presence of trailing edge wake.

Shah et al.[5] [2013] shows role of CFD in analysis of Centrifugal pump. CFD technique has been applied 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 Average Navier Stokes (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 field for further improvement of the pump performance. The most active areas of research and development are the analysis of 2-phase flow (cavitation and slurry flow), pump handling non-Newtonian fluids and fluid–structure interaction. CFD approach provides many advantages compared to other approaches; however due to the empirical nature of solution technique validation with experimental results is usually recommended.

Lei et al.[6] [2014] represents direct and inverse method of designing centrifugal pump. The existing research on improving the hydraulic performance of centrifugal pumps mainly focuses on the design method and the parameter optimization. The traditional design method for centrifugal impellers relies more on experience of engineers that typically only satisfies the continuity equation of the fluid. In this study, on the basis of the direct and inverse iteration design method which simultaneously solves the continuity and motion equations of the fluid and shapes the blade geometry by controlling the wrap angle, three centrifugal pump impellers are designed by altering blade wrap angles while keeping other parameters constant. The three-dimensional flow fields in three centrifugal pumps are numerically simulated, and the simulation results illustrate that the blade with larger wrap angle has more powerful control ability on the flow pattern in impeller. The three pumps have nearly the same pressure distributions at the small flow rate, but the pressure gradient increase in the pump with the largest wrap angle is smoother than the other two pumps at the design and large flow rates. The pump head and efficiency are also influenced by the blade wrap angle. The highest head and efficiency are also observed for the largest angle. An experiment rig is designed and built to test the performance of the pump with the largest wrap angle. The test results show that the wide space of its efficiency area and the stability of its operation ensure the excellent performance of the design method and verify the numerical analysis. The analysis on influence of the blade wrap angle for centrifugal pump performance in this paper can be beneficial to the optimization design of the centrifugal pump. So wrap angle is one of the important parameter that can be optimized to improve performance of centrifugal pump.

III. PARAMETRIC ANALYSIS

3.1 Effect of outlet blade angle

Bacharoudis et al.[1] presents effect of blade angle on performance of pump. Figure 3(a-b) shows head predicted and hydraulic efficiency at different outlet blade angles that as the outlet blade angle increases the performance curve becomes smoother and flatter for the whole range of the flow rates. at high flow rates, the increase of the outlet blade angle causes a significant improvement of the hydraulic efficiency.

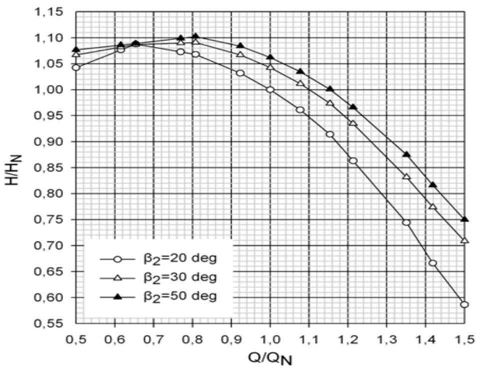

Figure 3(a). Predicted head curves for the examined pump impellers[1]

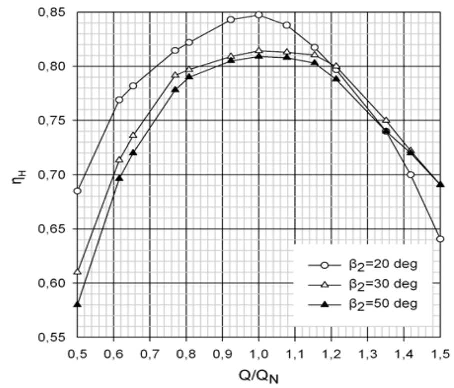

Figure 3(b). Predicted hydraulic efficiency curves for the examined pump impellers[1]

3.2 Effect of turbulence model

Kaewnai et al.[2] represents simulation of centrifugal pump using three different turbulence models viz. k-Ɛ, k-ω, RNG k-Ɛ model. Figure 4. shows head predicted at different turbulence model. The prediction of impeller performance indicated that the loss coefficient was minimum and the hydraulic efficiency was maximum at a volumetric flow rate Q/Qdesign = 0.7.

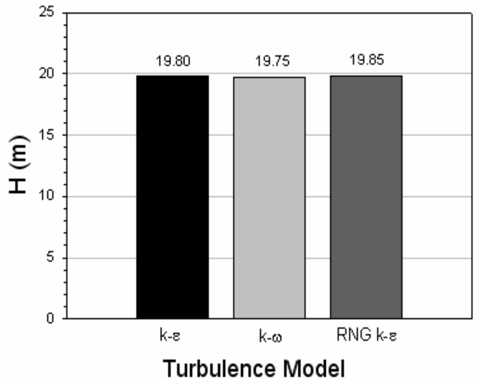

Figure 4. Total head rise of impeller obtained from various turbulence models[2]

3.3 Effect of wrap angle

Lei et al.[6] [2014] represents direct and inverse method of designing centrifugal pump. The pump head and efficiency are also influenced by the blade wrap angle. The highest head and efficiency are also observed for the largest angle.

IV. RESULT AND DISCUSSION

CFD plays an important role in Fluid flow analysis of Centrifugal pump and performance prediction. Different solver like ANSYS, FLUENT etc. can be used for simulation of Centrifugal Pump. Blade angles, wrap angle, number of blades, turbulence model used etc. have deep effect on performance of centrifugal Pump. As discharge increases, head decreases, power input of pump shaft increases, & efficiency of pump increases. Efficiency is maximum at rated conditions, beyond this as discharge increases, efficiency decreases. Pressure continuously increases as the mechanical energy imparted in form of impeller rotation is converted into the pressure energy. Shear stress transport (SST) turbulence model provides better result than any other turbulence model. At high flow rates, increase in outlet blade angle causes improvement in Hydraulic efficiency. At low flow rates very high recirculation of flow takes place in suction side of the blade. With increase in wrap angle head and efficiency of pump increases. Performance prediction at design and off-design conditions, parametric study, cavitation analysis, diffuser pump analysis, performance of pump running in turbine mode etc. are possible with CFD simulation techniques.

NOMENCLATURE

dA	:	Elemental area, m^2
H	:	Head, m
n	:	Revolution per minute
Q	:	Dischalrge, m^3/h
$S\phi$	:	Rate of generation ofquantity per unit volume
V	:	Velocity of fluid, m/s
dV	:	Elementary control volume, m^3
ϕ	:	Quantity per unit mass that is transported through control volume
ρ	:	Density of fluid, kg/m^3
Γ	:	Generalized diffusion coefficient

REFERENCES

[1] Bacharoudis E. C. , A. E. Filios, M. D. Mentos and D. P. Margaris, "Parametric study of a Centrifugal pump Impeller by varying the Outlet Blade angle", Open Mechanical Engineering Journal,vol. 2, 75-83, 2008.

[2] Suthep Kaewnai, Manuspong Chamaoot and Somchai Wongwises, "Predicting performance of radial flow type impeller of Centrifugal pump using CFD“, Journal of Mechanical Science & Technology,1620-1627, 2009.

[3] Shah S. R. , Jain S. V. and Lakhera V. J.,"CFD based flow analysis of Centrifugal pump",Proceedings of 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, 2010.

[4] Purushothaman K. and S. Rajendran, "Analysis of a Centrifugal pump Impeller using ANSYS-CFX", International Journal of Engineering Research & Technology, May-2012.

[5] Shah S. R., Jain S. V., Patel R. N. and Lakhera V.J., "CFD for Centrifugal pumps: a review of the state-of-the-art", Journal of ELSEVIER, vol.51, 715-720, 2013.

[6] Lei Tan, Baoshan Zhu, Shuliang Cao, Hao Bing, Yuming Wang, " Influence of blade wrap angle on centrifugal pump performance by numerical and experimental study", Chinese Journal of Mechanical Engineering , vol. 27, 171-177, 2014