Parametric performance evaluation of a hydraulic centrifugal pump

M W Heo¹, K Y Kim¹, S B Ma¹, I S Yoo², W C Choi², J H Kim³, Y S Choi³

¹ Department of Mechanical Engineering, Inha University, Korea.
² Extreme Mechanical Engineering Research Division, Korea Institute of Machinery and Materials, Korea.
³ Thermal & Fluid System R&B Group, Korea Institute of Industrial Technology, Korea.

E-mail: kykim@inha.ac.kr

Abstract. Parametric study of a hydraulic centrifugal pump with backward curved blades has been performed numerically using three-dimensional Reynolds-averaged Navier-Stokes equations. The shear stress transport turbulence model was used for analysis of turbulence. The finite volume method and an unstructured grid system were used for the numerical solution. The optimal grid system in the computational domain was selected through a grid dependency test. Tested parameters were related to the geometry of the impeller and volute: seven variables defining the hub and shroud contours and the blades angle of impeller, and two variables defining the inlet width and expansion angle of volute. The effects of these parameters on the hydrodynamic performance of the centrifugal pump have been investigated. It was found that the centrifugal water pump with the twisted blades has the enhancing efficiency compared to the straight blades pump.

1. Introduction

A centrifugal pump is a rotating machine in which flow and pressure are generated dynamically. This pump has been used widely for the agricultural and water and sewage, etc. Flow in centrifugal pumps produces generally a complex three-dimensional flow phenomenon involving turbulence, secondary flows, and unsteadiness because of complexity and asymmetry of volute shape [1]. This complex flow phenomenon directly affects the hydraulic performance of the centrifugal pump, and many studies have been carried out to investigate the flow characteristics in the centrifugal pump. Particularly, computational fluid dynamics (CFD) analysis is being used widely in hydrodynamic analysis of pumps.

Gonzalez et al. [2] performed a numerical analysis of a centrifugal pump to capture the dynamic interaction between the flows leaving the impeller and volute tongue. Zhou et al. [3] carried out numerical analysis to investigate the flow characteristics of three different centrifugal models, viz., one pump with four straight blades and two pumps with six twisted blades. Also, Jafarzadeh et al. [4] studied the effects of turbulence model and blade number on the prediction of flow field and efficiency of a high speed centrifugal pump, respectively. Guleren and Pinarbasi [5] carried out a numerical analysis of a centrifugal pump, and it was observed that the vane diffuser significantly affects the pump performance. Anagnostopoulos [6] developed a numerical model for the numerical analysis of a centrifugal pump by solving the Reynolds-averaged Navier-Stokes (RANS) equations with k-ε turbulence model.
In this study, a parametric study on a centrifugal pump has been carried out by using three-dimensional RANS equations. To investigate the effects of impeller and volute geometries on the hydraulic performance of the centrifugal pump, centrifugal impellers having different geometries have been tested. Tested parameters were related to the geometry of the impeller and volute: seven variables defining the hub and shroud contours and the blades angle of impeller, and two variables defining the inlet width and expansion angle of volute. The effects of these parameters on the hydrodynamic performance of the centrifugal pump have been investigated.

2. Preliminary Design of a Centrifugal Pump

The centrifugal pump model considered in this work is preliminary designed using CFTurbo [7] at the prescribed specific speed as shown in Fig. 1. The centrifugal pump consists of an impeller with five blades and a volute with a specific speed of \( N_s = N \cdot \frac{Q^{0.5} \cdot H^{0.75}}{150} = 150 \) at the best efficiency point (BEP). The volumetric flow rate and the total head at the design point of the reference pump model are 4 \( \text{m}^3/\text{min} \) and 68 \( \text{m} \), respectively, with an efficiency of 80%. The additional design specifications are listed in Table 1.

3. Numerical Approach Description

Three-dimensional RANS equations were solved for analysis of fluid flow in the computational domain using ANSYS-CFX 14.5 [8], which employs an unstructured grid system. The numerical analysis was carried out through the finite volume method to discretize the RANS equations. Blade profile creation, computational mesh generation, boundary condition definitions, flow analysis and post processing were performed by using Blade-Gen, Turbo-Grid, ICEM-CFD, CFX-pre, CFX-Solver, and CFX-Post, respectively.

| Design volumetric flow rate, \( \text{m}^3/\text{min} \) | 4 |
| Rotational speed, \( \text{r/min} \) | 1750 |
| Total head, \( \text{m} \) | 68 |
| Number of blades | 5 |
| Total power, kW | 56 |
| Efficiency, % | 80 |

Table 1. Design specification of centrifugal pump.

**Figure 1.** Preliminary design of centrifugal pump.
The shear stress transport (SST) turbulence model used in this analysis, works by solving a turbulence/frequency-based model ($k-\omega$) in near wall region and a $k-\varepsilon$ model in the rest region. A blending function ensures a smooth transition between these two models. Bardina et al. [9] showed that the SST model more effectively captures the flow separation under an adverse pressure gradient than other eddy viscosity models, and thus precisely predicts the near-wall turbulence that plays a vital role in the accurate prediction of flow separation. A formulation called automatic wall functions employed in ANSYS-CFX 14.5 switches automatically between a low-Reynolds number formulation and a wall-function treatment with a high-Reynolds number model according to the distance of the first grid from the wall ($y^+$).

The computational domain, where the present simulations were performed, is shown in Fig. 2. The computational domain for the numerical analysis consists of a rotating impeller domain with five blades and a stationary volute domain. The total pressure and the designed mass flow rate are set at the inlet and outlet of the computational domain, respectively. Water is considered as the working fluid, and the solid surfaces in the computational domain are considered to be hydraulically smooth with no-slip and adiabatic conditions. The frozen-rotor method is applied for the connection between the rotating impeller domain and two stationary domains (inlet duct and volute domains).

A hexahedral grid system was employed to generate the mesh in the impeller and inlet duct domain, with O-type grids near the blade and H/J/C/L-type grids in the other regions, and a tetrahedral grid system in constructed in the volute domain. Through the grid dependency test results, 1,050,000 was determined as optimal number of grids which consists of 580,000, 225,000, and 245,000 nodes in the rotating impeller, inlet duct, and volute domains, respectively.

Root-mean-square (RMS) values of the equation residual for convergence criteria were specified to be at least $1.0E-5$ for all equations. The solver finished a single simulation in approximately 1,000 iterations. The present numerical simulations were performed by an Intel Core i7 CPU having clock speed of 2.94 GHz. Each calculation was subdivided into eight tasks; the data transfer was performed using the MPICH2 [10]. Parallel calculation by eight CPUs helped us to reduce the computational time by as much as 35.6% compared to serial calculation. The computational time was typically 12~15 h, and depends on the geometry considered and the convergence criteria.

4. Parametric Study
Nine variables related to the geometry of the centrifugal pump were tested to investigate their effects on the pump’s efficiency. Tested parameters are seven variables defining the hub and shroud contours...
Figure 3. Variation of the efficiency with the impeller geometry.
(\(CP_{1h}, CP_{2h}, CP_{1s}, CP_{2s}\)) and the blades angle (\(\beta_{1s}, \beta_{1h}, \beta_{2}\)) of impeller and two variables defining the inlet width (\(b_{l}\)) and expansion angle (\(V_{\theta}\)) of volute, as shown in Fig. 1. In the present parametric study of the centrifugal pump, efficiency was used to estimate the hydraulic performance of the centrifugal pump. The efficiency of the centrifugal pump is defined as:

\[
\eta = \frac{\rho g H Q}{P}
\]

where, \(\rho, g, H, Q,\) and \(P\) indicate the density, acceleration of gravity, total head, volume flow rate, and power, respectively.

5. Results and Discussion

In this work, nine variables related to the geometry of impeller and volute in a centrifugal pump were used to investigate their effects on the pump’s efficiency. The efficiencies of the centrifugal pump having different geometries are compared in Fig. 3. In the case of \(\beta_{1h}\), the centrifugal pump with \(\beta_{1h} = 20^\circ\) shows best efficiency among the tested cases of the angle as shown in Fig. 3(a). Figure 4 shows the velocity contours at the mid-span of impeller for different inlet angles of blade hub. In case of \(\beta_{1h} = 17^\circ\) and \(19^\circ\), the low velocity region is observed near the blade inlet region. However, in case of \(\beta_{1h} = 20^\circ\), the flow velocity in this region (dashed ellipse) increases in comparison with the other cases.

The efficiency of the centrifugal pump increases with an increase in the \(\beta_{1s}\) and \(CP_{2h}\) as shown in Figs. 3(b) and (d), respectively. The low velocity region generated at the blade inlet is decreased as \(\beta_{1s}\) and \(CP_{2h}\) increase, as shown in Figs. 5 and 7, respectively.

The efficiency of the centrifugal pump decreases with an increase in the \(CP_{1h}, CP_{1s},\) and \(\beta_{2}\) as shown in Figs. 3(c), (e) and (g), respectively. Figure 6 shows the velocity contours at the mid-span of impeller for different \(CP_{1h}\). The low velocity region (red arrows) at the blade inlet is increased with the increases in \(CP_{1h}\). Also, in the case of centrifugal pump with different \(CP_{1h}s\), the flow velocity near the blade suction side (dashed ellipse) is reduced with increases in \(CP_{1h}\) as shown in Fig. 8. In case of \(\beta_{2} = 25^\circ\), it is observed that higher pressure region is generated near the cut-off (dashed circle) in comparison with the other cases as shown in Fig. 10.

In Fig. 3(f), a sharp decrease in efficiency is represented in the centrifugal pump with \(CP_{2s}\) of 20° and 95°. In case of \(CP_{2s} = 60\%\), the high velocity region (dashed ellipses) is more widely distributed near the pressure side of blade compared to the other cases as shown in Fig. 9.

Figure 11 represents the variation of efficiency of centrifugal pump with different volute geometries. There are few changes in efficiency of centrifugal pump as the volute geometries change.

![Figure 4. Velocity contours at the mid-span of impeller for different inlet angles of blade hub.](image)
Figure 5. Velocity contours at the mid-span of impeller for different inlet angles of blade shroud.

Figure 6. Velocity contours at the mid-span of impeller for different \( CP_{1h} \)s.

Figure 7. Velocity contours at the mid-span of impeller for different \( CP_{2h} \)s.
Figure 8. Velocity contours at the mid-span of impeller for different $CP_{1s}$.

Figure 9. Velocity contours at the mid-span of impeller for different $CP_{2s}$.

Figure 10. Pressure contours at the mid-span of impeller for different outlet angles of blade.
6. Conclusion

In this work, a parametric study has been performed to analyse the effects of geometry of the impeller and volute on the hydraulic performance of a centrifugal pump by solving three-dimensional RANS equations. Tested parameters were related to the geometry of the impeller and volute: seven variables defining the hub and shroud contours and the blades angle of impeller, and two variables defining the inlet width and expansion angle of volute. It was observed that the efficiency of a centrifugal pump is more sensitive to the impeller geometry than to the volute shape. A study to optimize the shape of the impeller with systematic optimization techniques needs to be performed in the future based on the results of this parametric study.

Acknowledgments

This research was supported by the Korea Institute of Industrial Technology Evaluation and planning (ITEP) grant funded by the ministry of Trade, Industry and Energy (No. 10044860).

References

[1] Brennen C E 1994 Oxford University Press and CETI Inc.
[2] Gonzalez J, Fernandez J, Blanco E and Santolaria C 2002 ASME J. Fluids Eng. 124 348
[3] Zhou W, Zhao Z, Lee T S and Winoto S H 2003 Int. J. Rotat. Mach. 9 49
[4] Jafarzadeh B, Hajari A, Alishahi M M and Akbari M H 2011 Appl. Math. Model. 25 242
[5] Guleren K M and Pinarbasi A 2004 J. Mech. Eng. Sci. 218 425
[6] Anagnostopoulos J S 2006 Numerical Calculation of the Flow in a Centrifugal Pump Impeller Using Cartesian Grid Proc. of the 2nd WSEAS Int. Conf. on Applied and Theoretical Mechanics
[7] CFTurbo 9 2011 User manual for CFTurbo 9 software
[8] ANSYS CFX-14.5 2012 Theory Guide of ANSYS CFX 14.5 ANSYS Inc
[9] Bardina J E, Huang P G and Coakley T J 1997 AIAA J. 35 1997.
[10] Thakur R, Gropp W and Toonen B 2005 Int. J. High Perform. C. 19 119.