Numerical Study of Outlet Blade Angle Effect on Impeller Characteristics of Double Entry Centrifugal Pump

Viktoriia Miltykh(✉) and Mykola Sotnyk

Sumy State University, 2 Rymskogo-Korsakova Street, Sumy 40007, Ukraine
miltykh.viktoriia@gmail.com

Abstract. The characteristics of pump performance are depended strongly on the impeller geometry. Changes in some impeller geometric parameter can improve pump performance. It is well known that outlet blade angle is one of those parameters, which has significant effect on the impeller work characteristics. This work is connected with the effect of blade outlet angle changes on the performance of double entry centrifugal pump impeller. This process is investigated via Ansys computational fluid dynamics (CFD) software. CFD method can predict well the complex internal flows in centrifugal impellers. The present paper describes the simulation of four impeller working process. The outlet blade angle was changed from 26° to 32° while all other geometrical impeller parameters were kept constant. The head-flow rate, hydraulic efficiency-flow rate and shaft power-flow rate curves are compared and discussed for each impeller. The obtained results show that even insignificant changes of the blade outlet angle effect on the impeller performance.

Keywords: Characteristic curve · Impeller geometry CFD · ANSYS-CFX Efficiency

1 Introduction

Pump equipment's designers are challenged continually to provide customers more efficient, reliable and energy saving machines but at lower cost. It is necessary to make detailed understanding of the internal flow within its stationary and rotating passages. That is why it is required the complex internal flows investigations in centrifugal pumps impellers.

Experiment is considered as a preferable investigation method of mechanisms inside centrifugal pumps. However, today the Computational Fluid Dynamics (CFD) approach is quite common because of high cost of experiments. With the aid of CFD, the complex internal flows in centrifugal pumps can be well described. It is worth to pay special attention to internal flow in inter-blade impeller channels, as it is a key element in the entire pump unit.

V. Ivanov et al. (Eds.): DSMIE 2018, LNME, pp. 317–324, 2019.
https://doi.org/10.1007/978-3-319-93587-4_33

2 Literature Review

There are various studies about the effect of impeller geometrical parameters on pump characteristics. Some researchers have investigated the effect of impeller outlet blade angle on centrifugal impeller in order to achieve its better performance.

Bacharoudis et al. [1] describe the flow simulation in the impeller of centrifugal pump. In this study, it is evaluated the performance of impellers with the same outlet diameter and different outlet blade angles. They conclude that increasing of outlet blade angle causes the shape of the curve which becomes smoother and flatter. 50% reduction of the volume rate from its nominal value results to 20%, 25% and 28% drop of the hydraulic efficiency relative to their nominal value for the impellers with outlet blade angle, it is equal to 20°, 30° and 50° respectively. In addition, 50% increase of the volume rate from the nominal point leads to 25%, 15% and 15% drop of the hydraulic efficiency relative to the nominal value for the 3 impellers respectively.

Djerroud et al. [2], have obtained similar study results of the outlet blade angle effect on the head and efficiency and it is noted that the increasing of outlet blade angle entails the increasing in shaft horsepower.

Shojaeefard et al. [3], and Patel et al. [4], have observed the increase in efficiency with outlet blade angle increasing.

Anagnostopoulos [5] has investigated the effect of blade geometry on pump efficiency. It is emphasized that flow rate decreases at low blade angles and increases at high blade angles with increasing the losses.

A sufficient number of such studies were conducted. Nevertheless, the results of those works cannot be generalized because those researches were conducted for different types of pumps that use different working fluid.

The study results of hydraulic performance of the double entry centrifugal pump impeller provide in this paper. The CFD code served as a tool for investigating the outlet blade angle change effect on impeller head, shaft horsepower and hydraulic efficiency.

It was studied four impellers with seven blades. All impellers have the same diameters in suction and pressure side and they vary only on the outlet blade angle, which is 26°, 28°, 30° and 32°, respectively. In the rotational speed (n) of 980 rpm, the normal operation of the original impeller is 3200 m^3/h flow rate (Q), the estimated pump's total head (H) is 75 m and the number of impeller entries (fq) is 2 that results in the value 26 for the specific speed ($n_q = n \cdot (Q/f_q)^{1/2}/H^{3/4}$). Table 1 shows the main design data of the modeled pump impeller.

Table 1. Main data design of the modeled centrifugal impellers.

Geometry variant	Im. 1	Im. 2	Im. 3 (original impeller)	Im. 4
Outlet blade angle, β_2	26°	28°	30°	32°
Impeller outlet width, b_2 (mm)	59,6			
Impeller outlet diameter, D_2 (mm)	740			
Number of blades, z	7			

3 Research Methodology

The ANSYS-CFX code consists of geometry, CFX-pre, CFX-solver, and CFX-post modules. According to the applied ANSYS-CFX code, Fig. 1 shows a block scheme which is used to obtain the numerical simulation results from impeller geometry models to impeller numerical models.

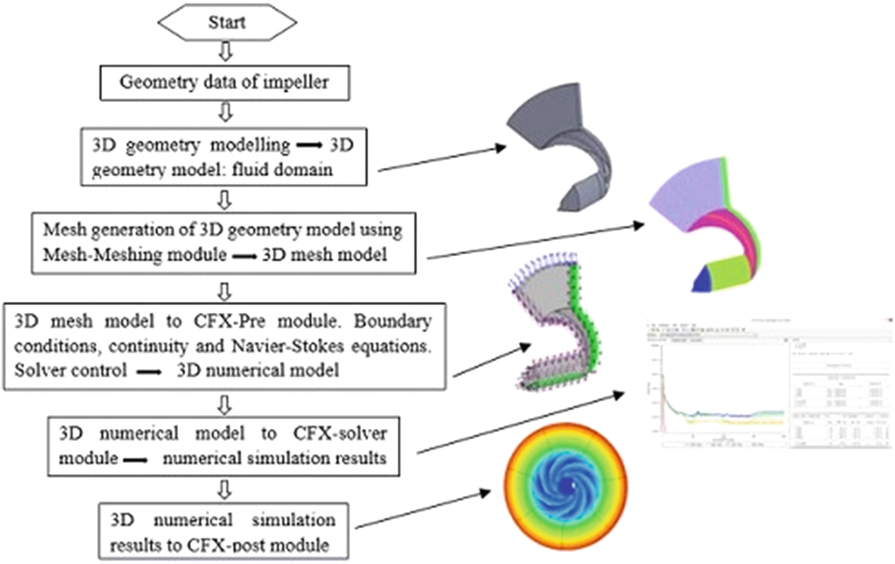

Fig. 1. The steps of study block scheme from geometry models to numerical models for impeller.

3.1 Three-Dimensional Computer Models of the Impellers

To carry out a numerical study, three-dimensional models of impellers with four different outlet blade angles were created, according to the recommendations presented in [6], and use of following assumptions: internal flow is symmetrical; internal flow at the entry to computational domain is axisymmetric; leakage through the impeller seals is neglected.

Due to assumptions above, the computational domain is a single channel of impeller half without gap seals. Inlet and outlet boundaries of the domain are spaced from the control sections at distance of 1…1,5 of impeller outlet diameter in order to get steady flow. A three-dimensional model of the computational fluid domain is shown in Fig. 2.

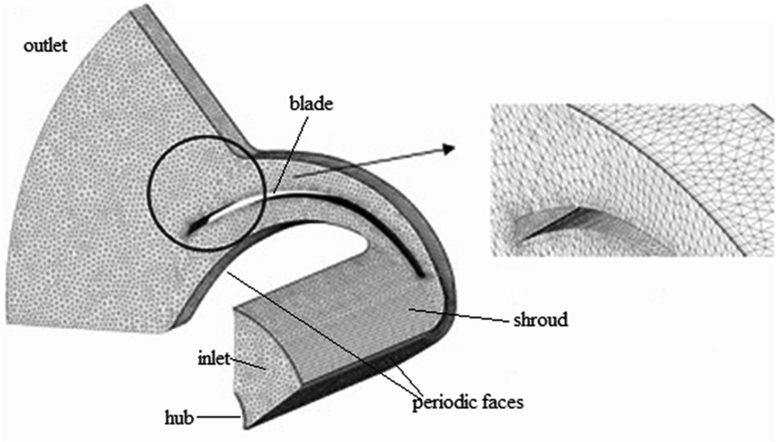

Fig. 2. A sample of the three-dimensional model of the computational fluid domain.

3.2 Mesh Structure and Boundary Conditions

The resulting geometry is used to build a mesh. The meshing is carried out using the Auto Mesh feature. The sample of mesh, which is shown in the Fig. 2, indicates that the tetrahedrons are used as the elements. A grid refinement was studied and adapted to flow morphology. Figure 3 shows the influence of the cell number on the theoretical head (H_t) which was developed by the impeller. According to this figure, the grid with about 1.5 million cells is considered enough sufficiently to ensure mesh independence.

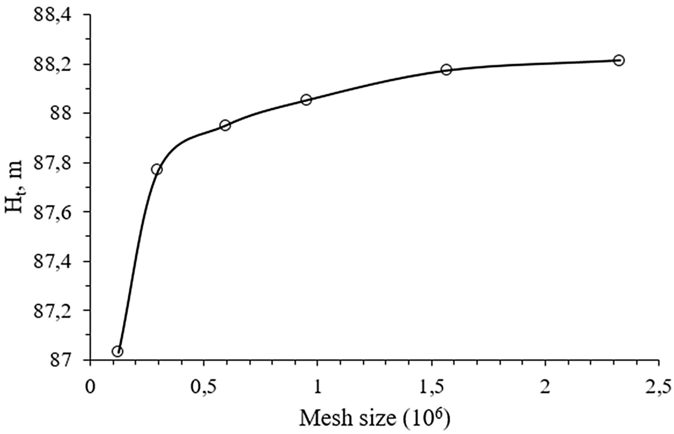

Fig. 3. The influence of grid size on the solution.

Cylindrical velocity components are specified on the computational fluid domain inlet and the static pressure is employed at the impeller outlet. The boundary condition type as “opening” is used at the outlet.

No-slip wall conditions are applied on all walls. Surface roughness is assumed as 6.3 μm for all surfaces of the wall in accordance with material properties. Water at 25 ° C is specified as a working fluid.

Convergence criterion is 10^{-4}. A steady state model is used for all calculations. Inlet and outlet segments were set in stationary frame. However, the impeller is set in rotary frame. Turbulence is modeled with the selection of the standard k-ε model.

We modeled working process of four impellers at the modes from 0,5*Q* to 1,2*Q*, where *Q* is the original impeller flow rate in the optimal point.

4 Results

The performance curves are the results through calculation of internal flow field and successful correlation of local and global parameters. Computational results, converged values are determined for each flow rate and different outlet blade angles, which is shown in Figs. 4, 5 and 6.

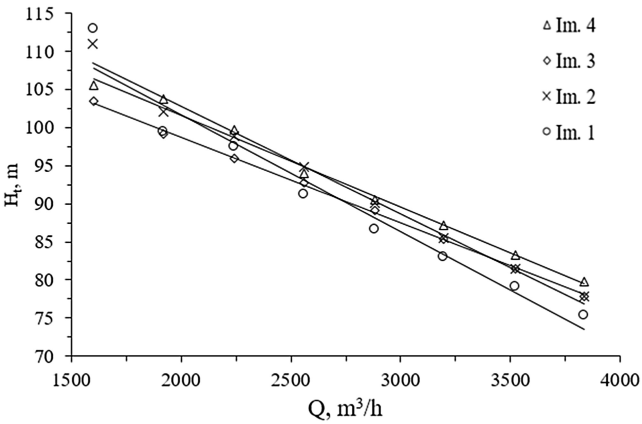

Fig. 4. The effect of outlet blade angles on the theoretical head.

Figure 4 depicts distribution of the calculated theoretical head, which is presented as a function of flow rate and with outlet blade angle as a parameter. Head increases by the increasing of the outlet blade angle. However, the theoretical head is hardly changed and we can clearly see it in flow rate range from 0,9*Q* to 1,2*Q*. The gain in the head is less than 5% when the outlet blade angle increases from 26° to 32°. It proves that the impeller hydraulic loss per outlet blade angle unit differs insignificantly.

With the increase of the outlet blade angle, the impeller pressure significantly increases, according to published conducted research results [1, 4], on this study we don't see a significant increase in head and most likely because of a small range of outlet blade angle variation.

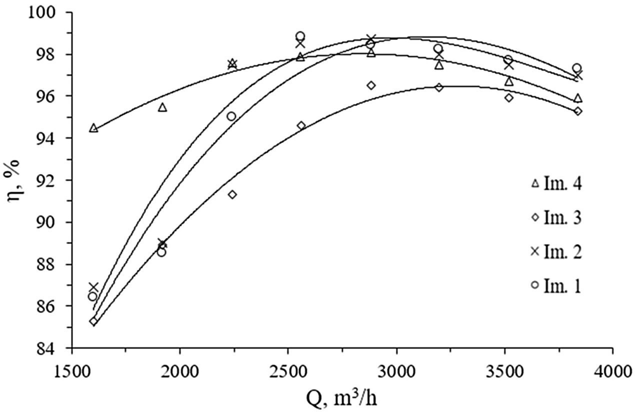

Fig. 5. The effect of outlet blade angles on hydraulic efficiency.

Figure 5 depicts distribution of the calculated hydraulic efficiency, which is presented as a function of flow rate and with outlet blade angle as a parameter. As the outlet blade angle increases, the hydraulic efficiency curves become smoother and flatter for whole range of flow rate. Those curves decrease more rapidly for the impellers with β_2 = 26°, 28° and 30° than for β_2 = 32° at lower flow rate than the Q.

The comparison of the hydraulic efficiency of four impellers shows that the decreasing of outlet blade angle leads to increasing of the hydraulic efficiency value up to 6% for range of flow rate from 0,7Q to 1,2Q. However, the operating regions near to best efficiency point (BEP) become narrower. Moreover, the efficiency curves illustrate that impeller hydraulic efficiency with β_2 = 32° decreases slowly to the left of the BEP.

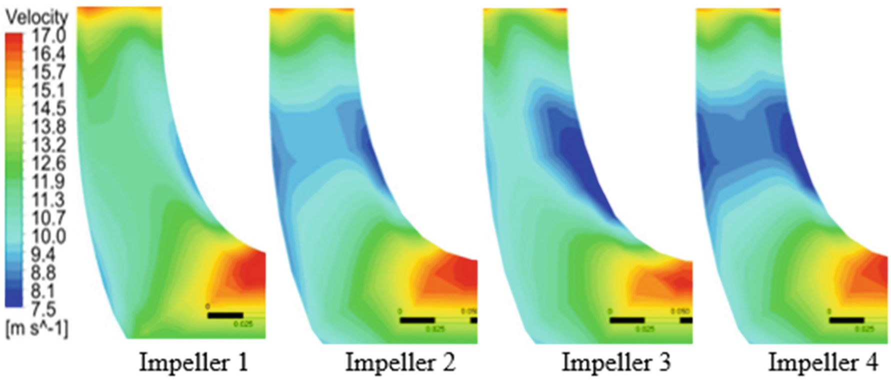

Fig. 6. Meridian view plots of velocity distribution for impellers.

Figure 6 illustrates the velocity contours in the numerical simulations which is performed at the normal operating conditions (3200 m^3/h). It is noticed that zone with low velocity exists in the impeller 2, 3 and 4 but not in the impeller 1. This fact causes higher hydraulic efficiency of impeller 1.

Figure 7 depicts distribution of the calculated shaft horsepower, which is presented as a function of flow rate and with outlet blade angle as a parameter. The curves show that shaft horsepower continuously increases in the flow rate range from 0.5 Q to 1.2 Q. In addition, Fig. 6 illustrates that the shaft horsepower increases relative to the augmenting outlet blade angle for all flow rate range. We observe noticeable increase in shaft horsepower for flow rate range from 0,8Q to 1,2Q.

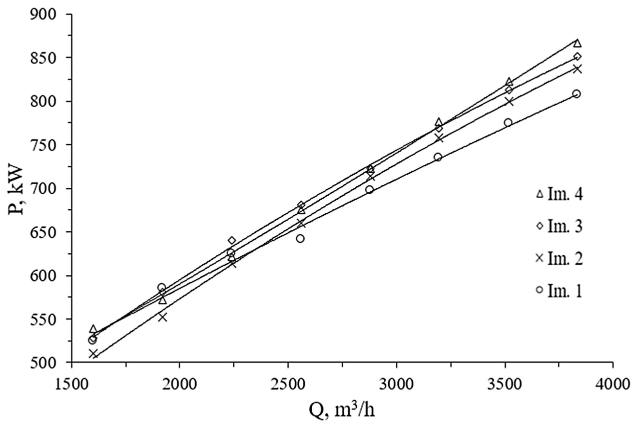

Fig. 7. The effect of outlet blade angles on shaft horsepower.

Nonetheless, the research results of the working process of the centrifugal pump impeller at the outlet blade angle change in the wide range, which is presented in [2], demonstrate a decrease in the pump power consumption at a very small outlet blade angle ($\beta_2 < 14°$) for flow rate larger than the optimal.

5 Conclusions

The investigation of impeller outlet blade angle effect on the centrifugal impeller performance has been done in this study. This study has been carried out by changing the outlet blade angle in range from 26° to 32° while all other parameters were kept constant. The centrifugal impeller hydraulic performance was studied numerically with the CFD code.

We can have following conclusions from the discussion above. The change of outlet blade angle in such small range of specified flow rate range has no significant effect on impeller theoretical head. The gain in head is less than 5% when the outlet blade angle increases from 26° to 32°. However, the decreasing of outlet blade angle leads to the increasing of hydraulic efficiency up to 6% and depends on the flow rate value. However, the operating regions near to BEP become narrower. In addition, we can't dismiss the fact that the shaft power decrease in this case, is insignificant.

It's planned to complete this research work through the investigation the effect of outlet blade angle change in wide range on the impeller and pump hydraulic performance.

References

1. Bacharoudis, E., Filios, A., Mentzos, M., Margaris, D.: Parametric study of a centrifugal pump impeller by varying the outlet blade angle. Open Mech. Eng. J. **2**, 75–83 (2008)
2. Djerroud, M., Dituba Ngoma, G., Ghie, W.: Numerical identification of key design parameters enhancing the centrifugal pump performance: impeller, impeller-volute and impeller-diffuser. International scholarly research network. Mech. Eng. **2011**, 16 (2011)
3. Shojaeefard, M., Tahani, M., Ehghaghi, M., Fallahian, M., Beglari, M.: Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid. Comput. Fluids **60**, 61–70 (2012)
4. Patel, M., Doshi, A.: Effect of impeller blade exit angle on the performance of centrifugal pump. Int. J. Emerg. Technol. Adv. Eng. **3**(1), 702–706 (2013)
5. Anagnostopoulos, J.: A fast numerical method for flow analysis and blade design in centrifugal pump impellers. Comput. Fluids **38**(2), 284–289 (2009)
6. Gűlich, J.: Centrifugal Pumps. Springer, Heidelberg (2010)