Parametric Study of Complex Liquid Flow in a Centrifugal Pump Consisting of an Impeller, a Volute and a Diffuser

Guyh Dituba Ngoma, Walid Ghie, Nicolas La Roche-Carrier
University of Quebec in Abitibi-Témiscamingue, School of Engineering's Department, 445, Boulevard de l'Université, Rouyn-Noranda, Quebec, J9X 5E4, Canada

Keywords: Centrifugal Pump, Impeller, Volute, Diffuser, Turbulence, CFD, Modeling and Simulation.

Abstract: In this work, the numerical investigation of the complex liquid flow in a centrifugal pump modelconsisting of an impeller, a diffuser and a volute was done to analyze the effects that the blade height, the diffuser blade number, and the volute size had on the pump performance. The continuity and Navier-Stokes equations with the k-ε turbulence model and the standard wall functions based on the logarithmic law were used by mean of ANSYS-CFX code. The results achieved reveal that the selected key design parameters have an impact on the pump head, the brake horsepower and the overall efficiency.

1 INTRODUCTION

Centrifugal pumps are widely used in industrial and mining enterprises. One of the most important components of a centrifugal pump is the impeller (Peng, 2008). The performance characteristics related to the pump comprising the head, the brake horsepower and the overall efficiency rely a great deal on the impeller. To achieve better performance for a centrifugal pump, design parameters must be accurately determined, due to the complex liquid flow through a centrifugal pump. It is therefore important to be aware of the liquid flow's behavior when passing through an impeller. This can be done by accounting for the impeller, the diffuser and the volute in the planning, design, and optimization phases at conditions of design and off-design. Many experimental and numerical studies have been carried out on the liquid flow through a centrifugal pump (Cheah et al., 2007; Djerroud et al., 2011; Ozturk A. et al., 2009). The analysis of previous works clearly demonstrated that research results obtained are specific to the centrifugal pump design parameter values and thus cannot be generalized. In this work therefore a numerical study was performed using a finite volume method according to the CFX code (Ansys inc., 2008) to gain further insight into the characteristics of the turbulent liquid flow through a centrifugal pump consisting of an impeller, a diffuser and a volute, while also considering various flow conditions and pump design parameters: blade height, blade number, and volute size.

2 GOVERNING EQUATIONS

Fig. 1 shows the fluid domain of the considered centrifugal pump model.

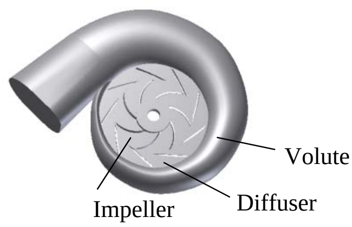

Figure 2: Centrifugal pump fluid domain.

The theoretical analysis of the liquid (water) flow in the considered centrifugal pump model was based on the continuity and Navier-Stokes equations (Ansys inc., 2008). Thus, the continuity equations are expressed by:

$$\nabla.\vec{U} = 0\,, \tag{1}$$

and the Navier–Stokes equations are given by:

$$\rho\nabla.(\vec{U} \otimes \vec{U}) = -\nabla p + \mu_{eff}\,\nabla.(\nabla\vec{U} + (\nabla\vec{U})^{T}) + B \tag{2}$$

where $\vec{U} = \vec{U}(u(x,y,z), v(x,y,z), w(x,y,z))$ is the liquid flow

Dituba Ngoma G., Ghie W. and La Roche-Carrier N..
Parametric Study of Complex Liquid Flow in a Centrifugal Pump Consisting of an Impeller, a Volute and a Diffuser.
DOI: 10.5220/0004035503430346
In *Proceedings of the 2nd International Conference on Simulation and Modeling Methodologies, Technologies and Applications* (SIMULTECH-2012), pages 343-346
ISBN: 978-989-8565-20-4

velocity vector, p is the pressure, ρ is the density (997 kg/m³), μ_{eff} is the effective viscosity accounting for turbulence, $\otimes$ is a tensor product and B is the source term. More particularly, for flows in an impeller rotating at a constant speed ω, the source term can be written as follows:

$$B = -\rho\left(2\vec{\omega}x\vec{U} + \vec{\omega}x\left(\vec{\omega}x\vec{r}\right)\right) \tag{3}$$

where $\vec{r}$ is the location vector.

In addition, μ_{eff} is defined as:

$$\mu_{eff} = \mu + \mu_t \tag{4}$$

where μ is the dynamic viscosity (8.899 x 10^{-4} Pas) and μ_t is the turbulence viscosity (Ansys inc., 2008).

The actual pump head rise is given by:

$$H = \eta_h \mu_s \left(\frac{U_2}{g}\right)\left(U_2 - \frac{Q}{A_2 \tan\beta_{b2}}\right) \tag{5}$$

where η_h is the hydraulic efficiency, μ_s is the slip factor (Peng, 2008), U_2 is the outlet tangential velocity U_2, β_{b2} is the outlet blade angles and Q is the volume flow rate. It is given by $Q = V_{r2}A_2$ with A_2 as the outlet flow passage area normal to the meridional direction.

The overall efficiency of a centrifugal pump can be formulated as:

$$\eta = \frac{P_h}{P_s} \tag{6}$$

where P_h is the centrifugal pump horsepower. It is expressed as $P_h = \rho Q g H$ and P_s is the pump brake horsepower (Peng, 2008).

To solve Eqs. 1 and 2 numerically while accounting for the boundary conditions and the turbulence model k-ε, the ANSYS-CFX code, based on the finite volume method, was used to obtain the liquid flow velocity and the pressure distributions.

3 RESULTS AND DISCUSSION

The main data for the reference impeller were: inlet diameter = 145 mm; outlet diameter = 320 mm; inlet blade angle = 11.69 °; outlet blade angle = 28°; inlet blade width = 12 mm; blade thickness = 4 mm; number of blades = 7; and rotating speed = 1800 rpm.

For the reference diffuser, the main data were: inlet diameter = 320 mm; outlet diameter = 455 mm; blade width = 12 mm; blade thickness = 3.401 mm; inlet blade angle = 11.07°; outlet blade angle = 39.42 °; number of blades = 9.

Concerning the size of the volute, it was characterized by the volute angle as a function of the volute radius (255.17 mm for 0° and 350.35 mm for 360°).

For highest accuracy of numerical simulation results, the convergence criteria based on a RMS (Root Mean Square) residual value of 10^{-4} was used and mesh-independent solution tests were conducted in each case study by finding the number of mesh elements to achieve mesh-independent results.

3.1 Effect of Blade Height

To investigate the impeller and diffuser blade height's effect on the pump performance, the blade heights of 0.012 m, 0.020 m and 0.028 m were selected, while the other parameters were keep constant. Fig. 2 shows the pump head as a function of the volume flow rate with the outlet blade height as a parameter. There, it is observed that the pump head increases with increasing blade height until a certain value of the blade height. This can be explained by the fact that when the volume flow rate is kept constant, the increased outlet blade height leads to the decreasing meridional velocity, which increases the pump head since the outlet tangential velocity and the outlet blade angle remain constant. But when the meridional velocity becomes too small or zero with increasing blade height, its influence to the pump head is negligible.

The curves expressing the pump brake horsepower as a function of the volume flow rate are shown in Fig. 3, illustrating that the brake horsepower increases relative to the increased blade height due to the requested increase in pump shaft torque relative to the increased blade height.

Moreover, Fig. 4 shows the overall efficiency curves as a function of the volume flow rate. It can be seen that the overall efficiency for b_2 = 12 mm decreases rapidly to the right of the best efficiency point (BEP). The overall efficiency curves for b_2 = 20 mm and 28 mm increase with increasing volume flow rate.

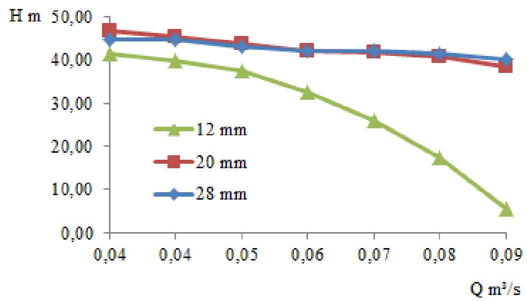

Figure 2: Pump head versus volume flow rate.

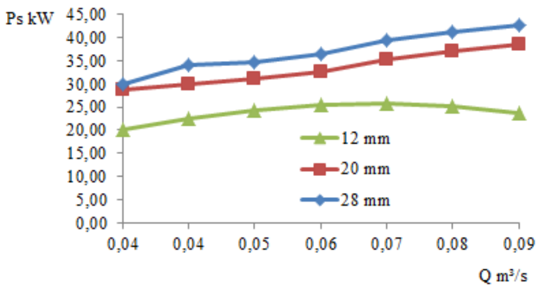

Figure 3: Brake horsepower versus volume flow rate.

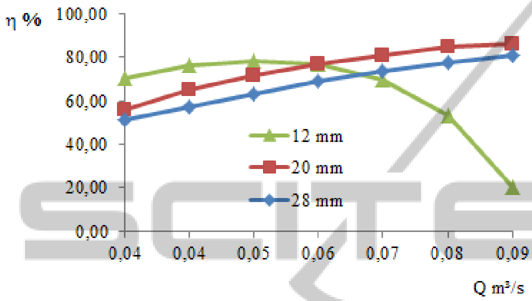

Figure 4: Overall efficiency versus volume flow rate.

3.2 Effect of Diffuser Blade Number

To analyze the effect of the diffuser blade number on the pump performance, a diffuser model without blade and three other diffuser models with blade numbers of 9, 10 and 11 were selected, while the other parameters were kept constant. Fig. 5 shows the pump head as a function of the volume flow rate, where it is observed that the impact of the diffuser blade number on the pump head is small. In addition, Fig. 6 shows that the brake horsepower for the case of a diffuser with blades is higher than the case of a diffuser without blade. This can be explained by the fact that the flow restriction due the blades leads to a higher requested impeller shaft torque. Furthermore, Fig. 7 shows that for the low and the high volume flow rates, the overall efficiency for the diffuser without blade is highest.

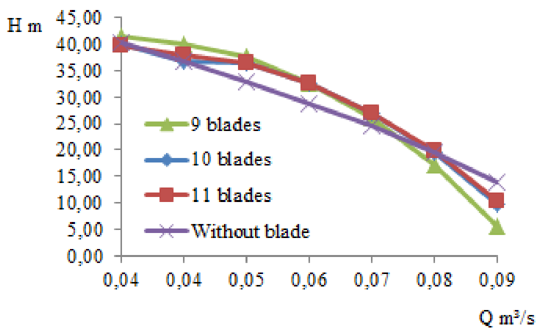

Figure 5: Pump head versus volume flow rate.

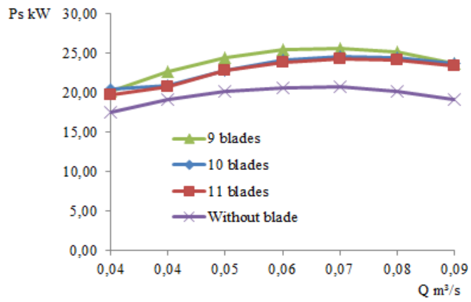

Figure 6: Brake horsepower versus volume flow rate.

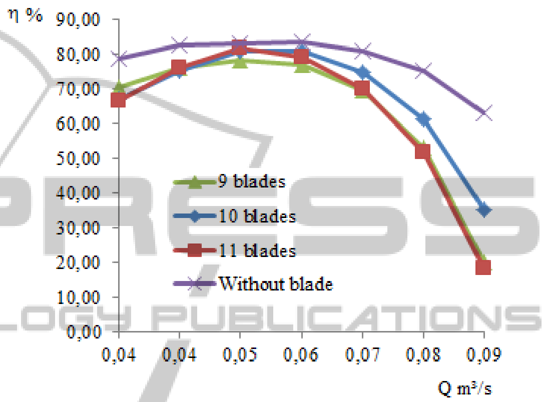

Figure 7: Overall efficiency versus volume flow rate.

3.3 Effect of Volute Size

To investigate the effect of the volute size on the pump performance, three values of 75 %, 100% and 125 % were selected for the volute size, while the other parameters were kept constant. The value of 100% was considered as the volute size reference. Fig. 8 indicates the pump head as a function of the volume flow rate, illustrating that the influence of the volute size on the pump head is small. The corresponding curves for the brake horsepower and the overall efficient are shown in Figs. 9 and 10 respectively, there it can be observed that the pump with a volute size of 75 % requests lowest impeller shaft torque and its overall efficiency is highest.

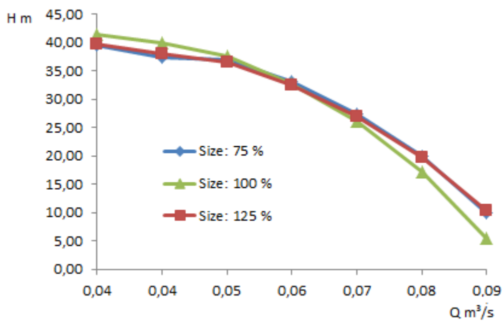

Figure 8: Pump head versus volume flow rate.

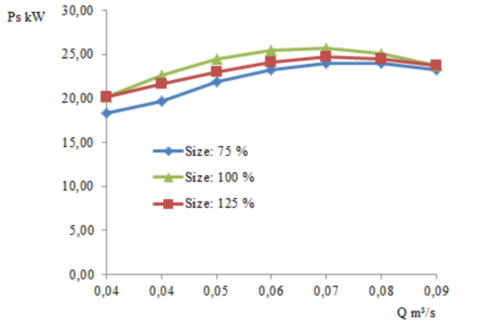

Figure 9: Brake horsepower versus volume flow rate.

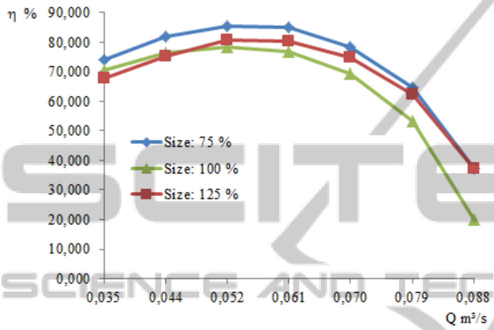

Figure 10: Overall efficiency versus blade number.

4 CONCLUSIONS

In this study, a complex liquid flow model in a centrifugal pump consisting of an impeller, a diffuser and a volute was developed to analyze the effects of the blade height, the diffuser blade number, and the volute size on the pump head, the brake horsepower and the overall efficiency. The obtained results for considered value ranges demonstrate, among others, that the pump head and the brake horsepower increase with increasing blade height. The pump performance is influenced by the variation in volute size. Additionally, the results comparison between the pump model having a diffuser with blades and the pump model with a diffuser without blade reveals that, for the case of a diffuser without blade, the requested impeller shaft torque is lower and the overall efficiency is higher that the case of a pump having a diffuser with blades. Further research work is planned to complete this study comparing numerical simulation results with various experimental values obtained from a pump manufacturer, and optimizing the developed model.

ACKNOWLEDGEMENTS

The authors are grateful to the Foundation of University of Quebec in Abitibi-Temiscamingue (FUQAT) and the company Technosub inc.

REFERENCES

Peng, W, 2008. Fundamentals of turbomachinery. Hoboken, *New Jersey, John Wiley and Sons.*

Cheah, K. W, Lee, T. S., S. H. Winoto, and Z. M. Zhao, 2007. Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions. Hindawi Publishing Corporation *International Journal of Rotating Machinery*, Volume 2007, Article ID 83641, 8 pages.

Djerroud, M., Dituba Ngoma, G., and Ghie, W., 2011. Numerical Identification of Key Design Parameters Enhancing the Centrifugal Pump Performance: Impeller, Impeller-Volute, and Impeller-Diffuser, ISRN Mechanical Engineering, vol. 2011, Article ID 794341, 16 pages, 2011. doi:10.5402/2011/794341.

Ozturk A., Aydin, K., Sahin, B., and Pinarbasi, A., 2009. Effect of impeller-diffuser radial gap ratio in a centrifugal pump. *Journal of Scientific and Industrial Research*, Vol. 68, 203-213.

Ansys inc., 2008. ANSYS-CFX, User Manual USA.