Blade Exit Angle Impact on Turbulent Fluid Flow and Performance of Centrifugal Pump Using CFD

Abdelmadjid CHEHHAT *
Dept. of mechanical engineering, fac. of engineering
Abbes Laghrour University
Khenchela, Algeria
achehhat@gmail.com

Mohamed SI-AMEUR
LESEI laboratory, Dept. of mechanical engineering
El Hadj Lakhdar University
Batna, Algeria
msiameur@yahoo.fr

Abstract— **Rotor is a working part that adds energy to the fluid; its geometry plays a fundamental role in the centrifugal pump performance. Any change in the rotor geometry would have an effect on the impeller inlet or exit velocity triangles, which may cause a significant performance change. Hence the blade exit angle have very important role in the performance of the centrifugal pump. In this paper three impeller designs that differ in their outlet blade angles, with the casing and other geometric parameters keep constant, are considered to investigate its effect on the flow parameters and turbulence intensity as well as on the performance of the centrifugal pump. For this purpose, a three-dimensional flow is simulated solving Reynolds averaged Navier Stokes (RANS) equations by using FLUENT code. Standard k-ε model is used for the turbulent closure of steady incompressible flow. The CFD analysis shows that the blade exit angle has influence on the head and efficiency. Rise in head and hydraulic efficiency have been observed with increasing blade exit angle. The research results are helpful for energy efficiency and hydraulic design of centrifugal pump.**

Keywords-CFD; turbulent flow;k-ε model;centrifugal pump

I. INTRODUCTION

The use of CFD-tools to analyze the flow field in centrifugal pumps and to predict performance parameters has gained enormous popularity in recent years. There have been continuous researches to improve the performance of centrifugal pumps. There are still many unknown issues associated with improving the performance in these pumps, which need to be investigated. Some of the key works are based on the change of pump geometry, especially impeller and diffuser. Since the impeller is a moving part that provides energy to the fluid, its geometry plays a major role in the centrifugal pump performance. Any modification in the impeller geometry would have an impact on the impeller inlet or exit velocity triangles, which may result in significant performance change. In this study different blade exit angles are examined to investigate its effect on the performance of the centrifugal pump [4]. Especially the verification of such a parameter using experiments is quite difficult and expensive. Therefore numerical simulation based on mathematical modeling is preferred and more useful. Several research works have been carried out on the centrifugal pump flow simulation. M. G. Patel et al [4] have studied the effect of impeller blade exit angle on the performances of centrifugal pump using MATLAB programming for the calculations based on the model presented by Gulich, and they found that the pump efficiency and head increase with increasing in the blade exit angle. W. G. Li [5] has investigated the influence of blade exit angle on the performances of a standard industrial centrifugal oil pump, using experiments at various viscosity conditions, and he found that the influence is equal on the head, shaft power and efficiency. M. H. Shojaee et al [13] have studied the impact of various blade outlet angles on the centrifugal pump operating conditions when handling viscous fluids, the results of this study show that, when the outlet angle increase, the centrifugal pump performance handling viscous fluids improves. The study of Liu Houlin et al [7] has been made using CFD tools to investigate the effect of blade number on centrifugal pump characteristics, the results of this study show that, blade number has an important effect on the area of low pressure region behind the blade inlet and jet-wake structure impellers. W. Zhou et al [16] have used the CFD to investigate the internal flow through a centrifugal pump impeller. It was found that the predicted results relating to twisted-blade pumps were better than those relating to the straight-blade pump, which suggests that the efficiency of a twisted-blade pump will be greater than that of a straight-blade pump. From these studies it is clear that, impeller is an active part that adds energy to the fluid; its geometry plays a crucial role in the centrifugal pump performance. Any change in the impeller geometry would have an impact on the impeller inlet or exit velocity triangles, which may result in significant performance change. The variation of blade exit angle have very important role in the performance of the centrifugal pump. In present study three impeller designs that differ in their outlet blade angles, with the volute and other geometric parameters keep constant, are considered to investigate its effect on the flow field parameters (velocity and pressure), and turbulence intensity, as well as on the performance of the centrifugal pump. For this purpose, a three-dimensional flow is simulated solving Reynolds averaged Navier-Stokes (RANS) equations by using FLUENT CFD code. Standard k-ε two

978-1-4673-7894-9/15/$31.00 ©2015 IEEE

equation model is used for the turbulent closure of steady incompressible flow. The computational fluid dynamics analysis shows that the blade exit angle has influence on the centrifugal pump head, output power and efficiency.

II. PUMP GEOMETRY MODELING

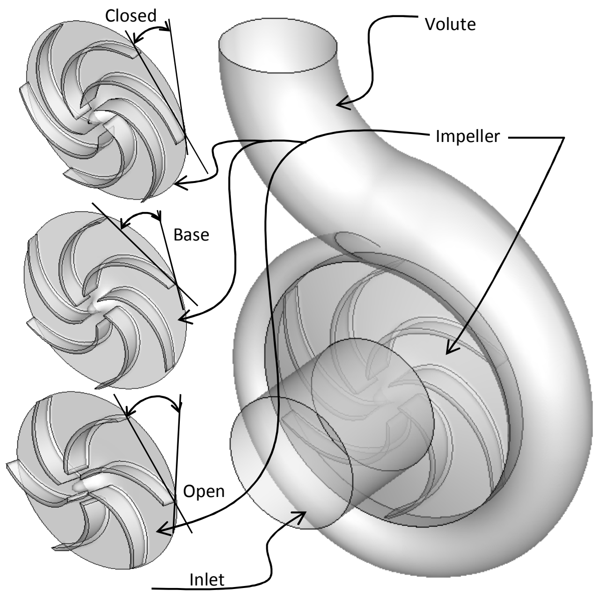

Figure 1. Centrifugal pump geometry modeled using GAMBIT

The centrifugal pump geometry development is carried out on GAMBIT software the various components of the pump are designed individually and all are assembled. The blade profiles are generated with help of coordinates that have been introduced, a single blade profile is made and then the blades are arranged by further rotating over an angle 360°/z , z is number of blades. For generating the volume of impeller inside the shroud volume, the blade volume must be subtracted from the shroud volume, a single rotational volume is obtained, it is limited by the impeller inlet surface, outlet surface, and hub and shroud surfaces, including the blade passage. [2]

TABLE I. CENTRIFUGAL PUMP IMPELLER GEOMETRICAL DATA

Description	Dimension
Impeller inlet diameter	50 mm
Impeller outlet diameter	100 mm
Height of blade exit	15.7 mm
Height of blade inlet	8.11 mm
Blade number	6
Blade thickness	2.7 mm
Blade inlet angle	30°
Blade exit angle	Open, Base, Close

The pump dimensions are shown in the table 1. The volute is modeled by assembling a casing of symmetrical cross section beginning from 0° ending at 360° (when a tongue is located) and a conic pipe discharging in radial direction.

III. MATHEMATIC MODEL AND NUMERICAL PROCESS

A. Governing equations

Mass is conserved for the incompressible flow at steady state as:

$$\frac{\partial}{\partial x_i}(u_i) = 0 \tag{1}$$

The following form of Navier-Stokes equations was used for the fluid analysis in the entire pump at the steady state:

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\right)\right] + \frac{\partial}{\partial x_j}\left[\mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i}\delta_{ij}\right)\right] \tag{2}$$

Where ρ, P and u_i denote fluid density, mean static pressure, and fluid velocity respectively.

The standard k-ε model, the most frequently used turbulence model in CFD codes was used in the simulation for turbulent kinetic energy k;

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \tag{3}$$

Where G_k is the kinetic energy production, given by:

$$G_k = \left[\mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial u_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_m}{\partial x_m}\right)\delta_{ij}\right]\frac{\partial u_j}{\partial xi} \tag{4}$$

For dissipation rate ε;

$$\frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

δ_{ij} is the Kronecker delta =1 if i=j , otherwise δ_{ij}=0, μ is the laminar viscosity(kg/ms) and μ_t is the turbulent viscosity(kg/ms) given by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{6}$$

σ_ε is ε - Prandtl number and σ_k is k- Prandtl number , the default values for k-ε model constants are taken; $C_{1\varepsilon}$=1.44; $C_{2\varepsilon}$=1.92; C_μ =0.9; σ_k=1.0; σ_ε=1.3

The high complexity of the flow in the rotating impeller makes the CFD modeling very difficult, only steady state flow is investigated, the governing equations implemented in a commercial CFD code Fluent 6.3.26 are solved using the finite volume method, with the most appropriate discretization scheme (pressure based implicit solver), the momentum and energy equations are solved using first order scheme, the turbulent kinetic energy and dissipation rate equations are solved using the power low differencing scheme (PLDS) the coupling velocity-pressure correction is treated with SIMPLE (Semi-Implicit Pressure Linked Equation) algorithm. The under relaxation factor for the pressure is taken 0.2 it seems very conservative, a common value of 0.5 is taken for the momentum, energy, and turbulence k-ε model equations. The convergence criteria of 10^{-4} are used for all the governing equations.

B. *Grid and boundary conditions*

The meshed fluid volume has only one flow inlet (the pump inlet) and one flow outlet (the volute exit). Mass flow inlet boundary condition was imposed on pump inlet position. The velocity was specified to be normal to the boundary and it is defined with reference to the absolute frame. The turbulence intensity for all conditions is considered 1%. Out flow boundary condition was imposed at outlet with a flow rate weighting of 1. Outer walls were stationary but the inner walls were rotational. There were interfaces between the stationary and rotational regions. Also non-slip boundary conditions have been imposed over the impeller blades and walls, the volute casing and the inlet wall. A constant angular velocity of 2900 rpm was imposed for rotating fluid. [17]

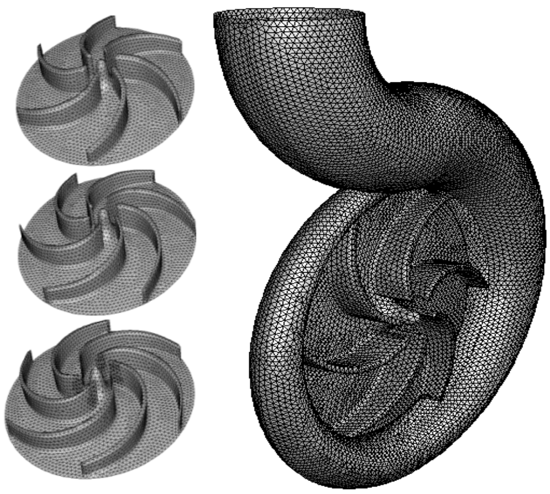

Figure 2. Computational mesh with different blade exit angle

The entire computational domain is meshed with tetrahedral grids due to the complexity of the pump geometry, the clearance gap between the blades and the shroud is included, and the detail for each element of the pump is shown in fig. 4; it contains a number of cells as shown in Table 2.

TABLE II. GRID CELLS NUMBER FOR EACH PUMP ELEMENT

angle volume	inlet	impeller	volute
Base	93903	74371	208623
Closed	94082	69484	208623
Open	93903	62972	208535

IV. RESULTS AND DISCUSSION

A. *Overall performances*

The pump characteristic curves, efficiency and output power for different blade exit angles, at 2900 rpm, are shown

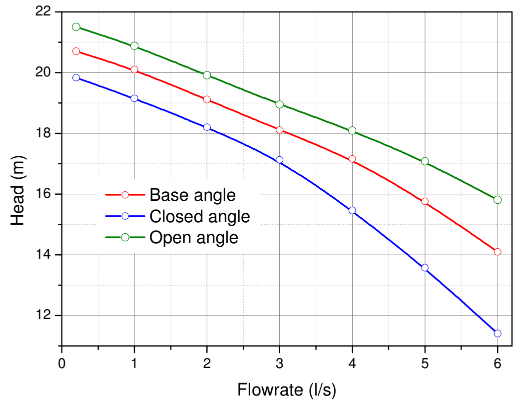

Figure 3. Comparison of head-flow characteristic

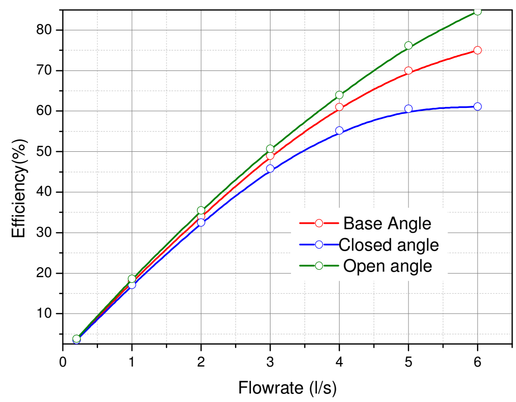

Figure 4. Comparison of efficiency-flow characteristic

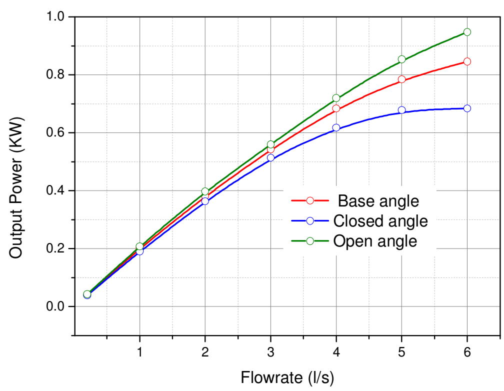

Figure 5. Com parison of Power-flow characteristic

respectively in Figs (3, 4, 5). It is found that the decrease of pump head with increasing flow rates is remarkable, and the effect of the blade exit angle is clear, especially at high flowrates. The best pump head is obtained for the highest (open) blade exit angle, we also note that the decrease of the pump head for the three selected impellers is slow, which prevents us to consider the range of selected flow rates as a stable operating range of the pump.

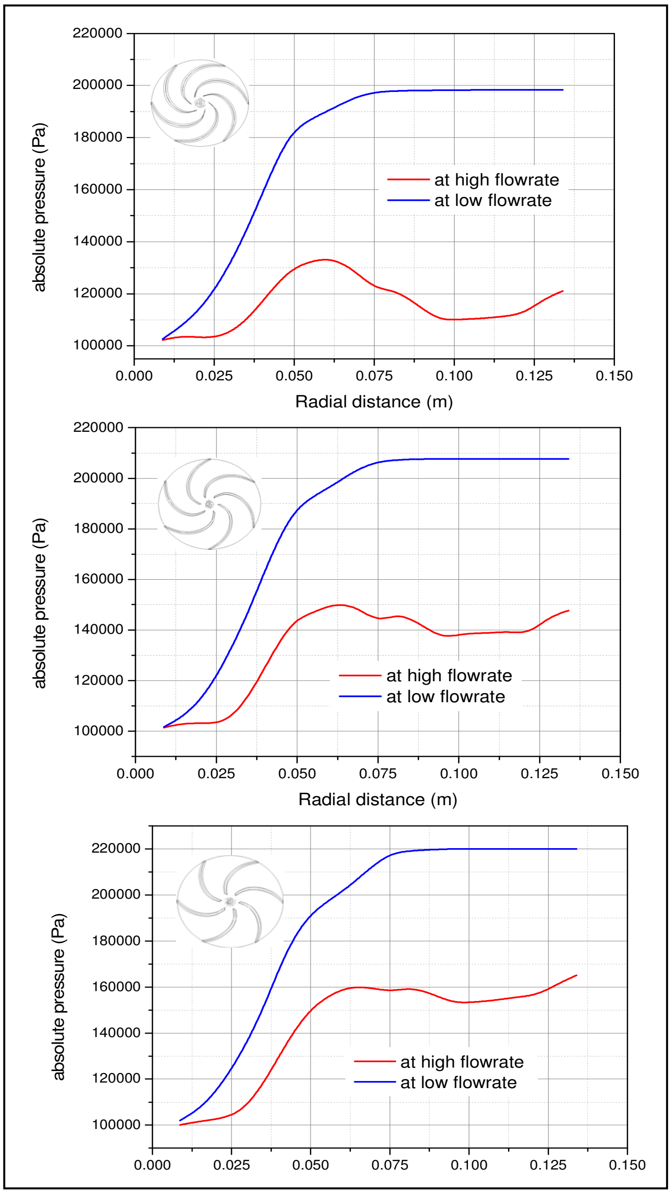

Figure 6. Circumference average of absolute pressure for three selected impeller designs

It is also found that the increase of pump efficiency and output power with increasing flow rates is remarkable, and the effect of the blade exit angle is clear, especially at high flow rates. Similarly to pump head, the best efficiency and output power are obtained for the highest (open) blade exit angle.

B. Pressure and velocity Variation

The circumferential average of absolute pressure as function of the radial direction of the pump, at high and low flow rates, for different blade exit angle, is shown in Fig. 6. It can be seen that the pressure increase with increasing radius and the pressure uniformity is affected by the flow rate and the blade exit angle. More pressure uniformity is obtained for less flow rate and less blade exit angle especially in the pump volute region. The contour plot of variation of absolute Pressure at nominal flow rate for three blade exit angles is shown in Fig. 7. It can be seen from the figures that, absolute pressure inside impeller and volute is asymmetry distributed. The maximum pressure area appears at volute tongue and

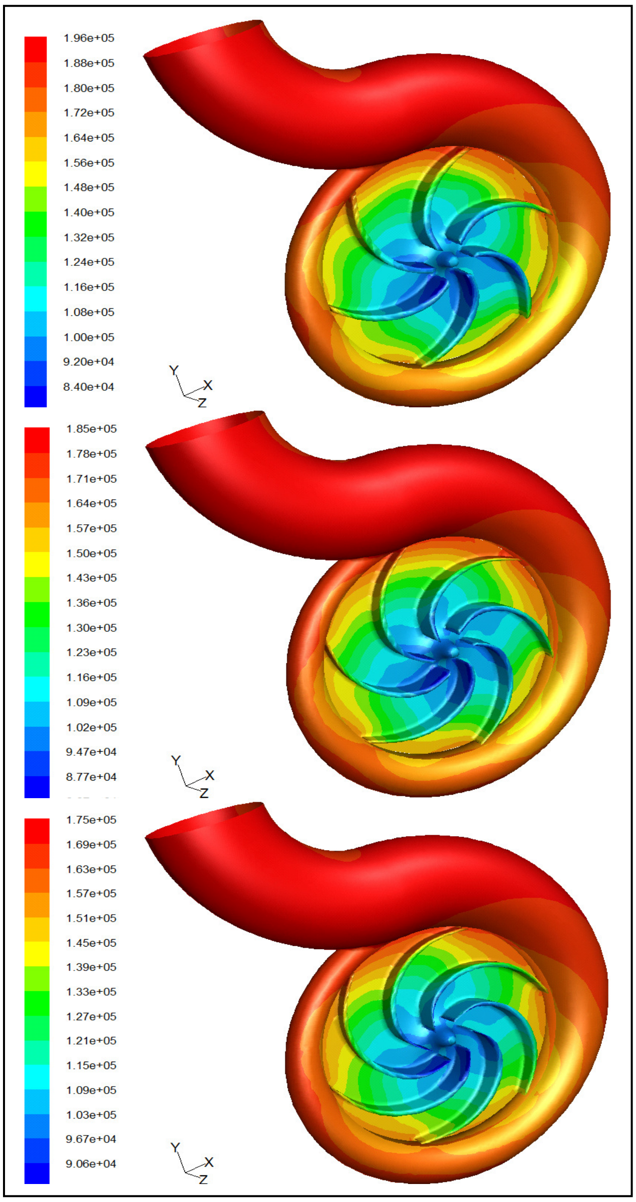

Figure 7. Contour plot of absolute pressure for three selected impeller designs at nominal mass flowrate

outlet regions and the minimum one at the back of blade at impeller inlet region. It can also be observed from the figures that, the pressure increases gradually from impeller inlet to outlet. The absolute pressure on pressure side is evidently

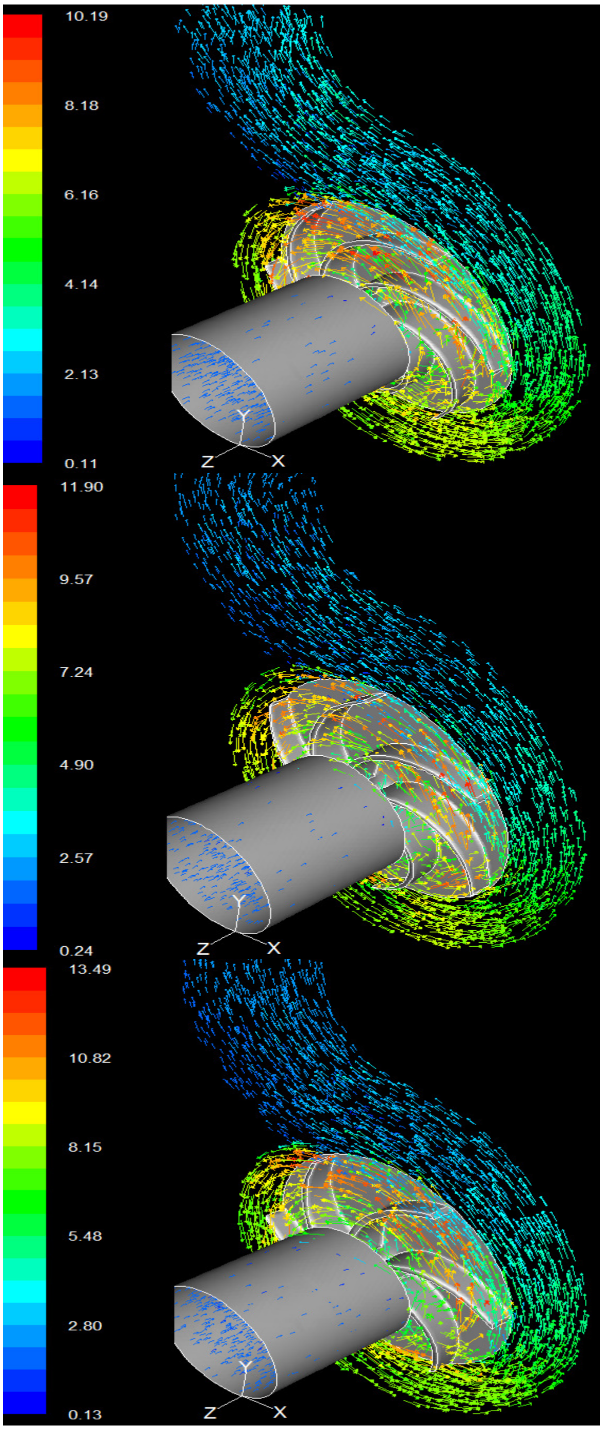

Figure 8. Velocity vectors for three selected impellers

larger than that on suction side at the same impeller radius. It can be remarked from the figures that pressure changes oppositely with flow rate. As the flow rate goes on increasing the pressure area at the suction side of the blade inlet at small flow rate, as the flow increases the area gets close to the middle of the blade suction side. In both cases low and high flow rates, the pressure at the pump outlet increases with increasing blade exit angle.

Fig. 8 shows the velocity vectors colored by velocity magnitude, through the entire configuration of the pump, where it can be seen that the region of acceleration of the fluid takes place in the impeller, as far as the diameter increases, the

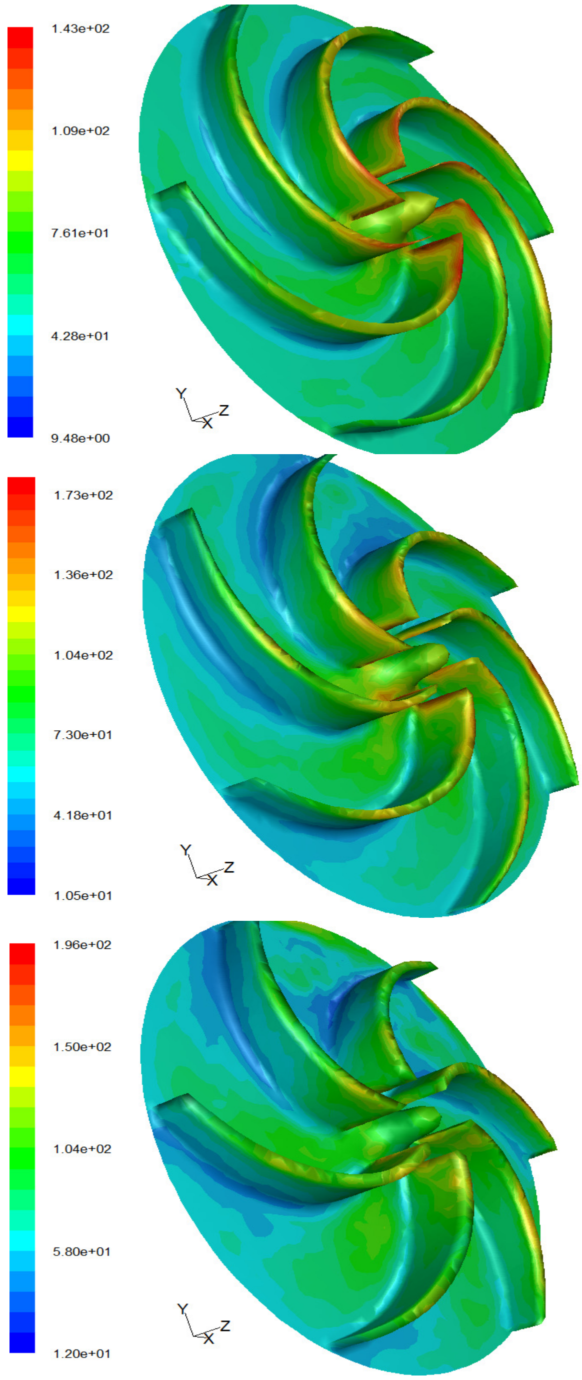

Figure 9. Contour plot of turbulent intensity

limit of acceleration zone in the volute is affected by the blade exit angle, this limit comes back as well as the blade exit angle increases. It can be seen clearly how the velocity magnitude decreases from volute inlet to outlet. It is also notable that no reverse flow is observed at the volute exit for all cases. Turbulent intensity distribution

The turbulence intensity $I = \frac{\sqrt{\frac{2k}{3}}}{U_{ref}}$ is presented as an indicator of turbulent agitation relative to the kinematics field of the mean flow, through the visualization of turbulence intensity we can then locate the regions of strong velocity fluctuation, and consequently of large turbulent kinetic energy k. The contour plot of turbulent intensity is shown in Fig. 9. As shown in the figures, the fluid flow near the shroud wall presents a high turbulent intensity comparing with the hub wall, especially at impeller inlet region, where the mean flow changes suddenly its direction from axial to radial; this is more observed when closing the blade exit angle. Whereas approaching to the volute inlet, the turbulent intensity goes on decreasing.

V. CONCLUSION

In this work, a steady state computational fluid dynamics analysis of a 3D-RANS turbulent flow through a six blades centrifugal pump is carried out considering three different impeller geometries, which differ in their blade exit angles. The contour and vector plots of pressure, turbulent intensity and velocity distribution in the flow passage are displayed. Besides, the operating characteristics and overall performances of the pump are also computed from Fluent numerical results. Although specific experimental results are not available for the pump considered for this study, the results agree well with most of the available results obtained by different authors for a similar pump. From the CFD analysis it can be concluded that:

- There is a low pressure area at the suction side of blade inlet at low flow rates, as the flow rate increases, the area gets close to the middle of blade suction side. The static pressure also increases on diffusion section of the volute outlet markedly at small flow rate while the static pressure on the same place decreases at higher flow rate.
- The simulation results for flow rate and head are also compared with analytical formulae used to predict the flow rate and theoretical head. There is a good agreement between the results and this also shows the accuracy of the analysis.
- The blade exit angle has influence on the pump characteristics and performances, especially at high flow rates. Rise in head, hydraulic efficiency and output power have been observed with increasing blade exit angle.

REFERENCES

[1] S. Yahia Turbines Compressors and Fan, 2nd edn. Tata Mc Graw Hill,2005

[2] Chehhat A., Si-Ameur M. and B. Boumeddane, “CFD Analysis of the Volute Geometry Effect on the Turbulent Air Flow through the Turbocharger Compressor”, Energy Procedia, Elsevier, Vol. 36, 2013, PP. 746-755

[3] O. Petit, H. Nilsson, “Numerical Investigations of Unsteady Flow In A Centrifugal Pump With A Vaned Diffuser”, International Journal of Rotating Machinery, Hindawi Publishing Corporation, Vol. 2013

[4] M.G. Patel, A.V.Doshi, “Effect of Impeller Blade Exit Angle on The Performance of Centrifugal Pump”, International Journal of Emerging Technology And Advanced Engineering, India,Vol. 3, Iss. 1, 2013, PP. 702-706

[5] W.G. Li, “ Blade Exit Angle Effects on Performance of A Standard Industrial Centrifugal Oil Pump”, Journal of Applied Fluid Mechanics, Iran, Vol. 4, No. 2, Issue 1, 2011, PP. 105-119

[6] S. Yang, F. Kong, B. Chen ,” Research on Pump Volute Designmethod Using CFD”, International Journal of Rotating Machinery, Hindawi Publishing Corporation, Vol. 2011

[7] L. Houlin, W. Yong, Y. Shouqi, T. Minggao, W. Kai, “Effects of Blade Number on Characteristics of Centrifugal Pumps”, Chines Journal of Mechanical Engineering , Vol. 23,A2010, PP. 1-6

[8] Bilus, A. Predin, “Numerical And Experimental Approach To Cavitation Surge Obstruction In Water Pump”, International Journal of Numerical Methods For Heat & Fluid Flow , Emerald Group Publishing Limited, Vol. 19, No. 7, 2009, PP. 818-834

[9] R. Spence, J. Amaral-Teixeira, “A CFD Parametric Study of Geometrical Variations on The Pressure Pulsations And Performance Characteristics of A Centrifugal Pump”, Computers & Fluids, Elsevier, Vol.38, Iss. 6, 2009, PP. 1243-1257

[10] Y. Shouqi, N. Yongyan, P. Zhongyong, Y. Jianping, “Unsteady Turbulent Simulation And Pressure Fluctuation Analysis For Centrifugal Pumps”, Chines Journal of Mechanical Engineering, Vol. 22, No. 1,A2009, PP. 1-6

[11] Jose Caridad , Miguel Asuaje , Frank Kenyery , Andrés Tremante , Orlando Aguillón , “Characterization of A Centrifugal Pump Impeller Under Two-Phase Flow Conditions”, Journal Of Petroleum Science And Engineering, Elsevier, Vol.63, 2008, PP. 18-22

[12] S. Derakhshan , A. Nourbakhsh, “ Theoretical, Numerical And Experimental Investigation of Centrifugal Pumps In Reverse Operation”, Experimental Thermal And Fluid Science,Elsevier, Vol. 32, 2008, PP. 1620–1627

[13] M. H. Shojaee Fard And F.A. Boyaghchi, “Studies on The Influence of Various Blade Outlet Angles In A Centrifugal Pump When Handling Viscous Fluids”, American Journal of Applied Sciences, Science Publications, Vol. 4, N. 9, 2007, PP. 718-724

[14] M. Asuaje, F. Bakir, S. Kouidri, ” Numerical Modelization of the Flow in Centrifugal Pump: Volute Influence in Velocity and Pressure Fields, International Journal of Rotating Machinery,:Vol. 3, 2005, PP. 244–255

[15] K M Guleren, A Pinarbasi, “Numerical Simulation of The Stalled Flow Within a Vaned Centrifugal Pump”, Proc. Instn Mech. Engrs Vol. 218 Part C: J. Mechanical Engineering Science, 2004, PP. 1-11

[16] W. Zhou, Z. Zhao, T. S. Lee, And S. H.Winoto, “Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics”, International Journal of Rotating Machinery, Taylor & Francis,Vol. 9, N. 1, 2003, PP. 49–61

[17] B. Jafarzadeh, A. Hajari, M.M. Alishahi, M.H. Akbari, “ the flow simulation of a low-specific–speed high- speed centrifugal pump”, Applied Mathematical Modelling, Elsevier .Vol 35, Is.1, 2011, PP. 242–249

[18] Li Xian-Hua; Zhang Shu-jia; Zhu Bao-lin; Hu Qing-bo, « The study of the k-ε turbulence model for numerical simulation ofcentrifugal pump », Computer-Aided Industrial Design and Conceptual Design, 2006. CAIDCD '06. 7th International Conference , IEEE, 2006,Pages: 1 - 5

[19] Fluent 6.3 user’s guide, (Fluent Inc., Lebanon, New Hampshire) 2006

[20] GAMBIT 2.2 user’s guide, (Fluent Inc., Lebanon, New Hampshire) 2004