International Journal of Engineering and Applied Physics (IJEAP)
Vol. 3, No. 2, May 2023, pp. 790~798
ISSN: 2737-8071

Dynamic Force Analysis on Blades of Centrifugal Pumps using Computational Fluids Dynamics Simulations

Victor Antwi
Department of Physics Education, University of Education, Winneba, Ghana

Article Info

Article history:

Received Mar 09,2023
Revised Mar 26, 2023
Accepted Apr 15, 2023

Keywords:

Computational Fluid Dynamics
centrifugal pump
tangential force
absolute pressure
flow rate

ABSTRACT

Mechanical stress on the blade of a centrifugal pump is a crucial factor in the design and operation of hydraulic pumps. Therefore, in this study, the tangential force on the blade of a centrifugal pump at different flow rates and directions was investigated using Computational Fluid Dynamics (CFD) simulations. The CFD simulations were performed based on the k-ε turbulence model and were validated through experimental measurements. The study revealed that the tangential force is dependent on the flow rate of the fluid being pumped, and understanding how it changes with varying flow rates is essential for optimizing the performance of the pump and preventing potential fractures. On the other hand, the absolute pressure on the blade was also investigated. The absolute pressure distributions on the blade pressure side, suction side, leading edge, and trailing edge were analyzed for different flow rates. The highest pressure distribution is found on the pressure side than the suction side. On the blade pressure side, a relatively large pressure is found near the trailing edge. Overall, the study provides insights into the complex relationship between flow rate and tangential force in centrifugal pumps and highlights the importance of understanding this relationship for successful pump design and operation.

Corresponding Author:

Victor Antwi,
University of Education, Winneba, Ghana
Email: vantwi@uew.edu.gh

1. INTRODUCTION

Due to the influence of topography and side levels, pumping is an essential method for community and household water supply from water resources such as groundwater basins. Centrifugal pumps are dynamic pumps that work by converting the mechanical energy of a rotating impeller into kinetic energy, which in turn converts to potential energy as the liquid is propelled through the pump. The resulting flow of the liquid can be used for a range of applications such as irrigation, chemical processing, water treatment, and fuel transfer. The basic components of a centrifugal pump include the impeller and volute casing. The impeller is a rotating disc with blades attached to it that is responsible for generating the necessary centrifugal force to move the liquid. The volute casing is a curved chamber surrounding the impeller, designed to capture the liquid from the impeller and direct it towards the discharge outlet. The turbulence flow patterns in centrifugal pumps are caused by the interaction of the impeller blades with the fluid. This turbulence leads to energy losses, increased wear and tear, and reduced efficiency. The degree of turbulence is influenced by the impeller design, the flow rates, and the fluid properties. Usually, turbulence is reduced by optimizing the impeller design, reducing the flow rate. Due aforementioned, centrifugal pumps have attracted a lot of research into the theory and design of the impeller due to the complex flow phenomena.

With the introduction of computational fluid dynamics (CFD), several experiments and numerical simulations have been carried out in the past decades. Yu et al.[1] presented CFD investigations of the hydrodynamic characteristics of a centrifugal pump affected by the impeller-eccentric effect. The study examined the influence of impeller eccentricity on the performance of a pump, including the head, flow rate, and efficiency. The results showed that as impeller eccentricity increases, the head, flow rate, and efficiency of the pump all decrease. The study also further considered the velocity distribution and pressure fluctuations within the pump and found that both are affected by impeller eccentricity. Al-Obaidi [2] conducted a numerical investigation into the effect of various pump rotational speeds on the performance of a centrifugal pump using CFD analysis. The study aimed to investigate the impact of rotational speed on pump performance parameters. The results showed that increasing the pump's rotational speed increased the head and flow rate but decreased the efficiency. The study also analyzed the velocity distribution and pressure distribution within the pump and found that both were affected by the rotational speed. Again, Zhou et al.[3] investigated the flow through centrifugal pump impellers using computational fluid dynamics (CFD). The study considered the flow characteristics within the impeller and investigated the effect of blade geometry on pump performance. The results showed that the blade angle and curvature significantly influenced the flow pattern and pump performance parameters. Furthermore, Muttalli, Agrawal, and Warudkar [4] applied CFD tools to simulate of a centrifugal pump impeller using ANSYS-CFX. The study investigated the flow characteristics within the impeller and evaluate the pump performance. The results showed that the pump's performance was influenced by the impeller's blade angle, blade thickness, and inlet velocity. The study also analyzed the velocity distribution and pressure distribution within the impeller and found that both were affected by the blade angle and thickness. The simulation results provided valuable insights into the flow behavior within centrifugal pump impellers and highlighted the importance of optimizing the impeller design for improved pump performance. Pei et al.[5] investigated the blade force and stress of a centrifugal pump impeller using computational fluid dynamics (CFD) for sewage treatment. The authors conducted simulations to study the fluid flow inside the pump and its effects on the impeller blades. They analyzed the pressure and velocity distribution in the impeller and calculated the blade force and stress. The results showed that the blade force and stress were affected by the fluid flow pattern and the shape of the impeller blades. Cárdenas-Gutiérrez, Valencia Ochoa and Duarte Forero [6] conducted a parametric analysis using CFD to investigate the hydraulic performance of a centrifugal pump with applications to the dredging industry. The study analyzed the velocity distribution and pressure distribution within the pump and found that both were affected by the impeller diameter and blade angle. In 2022, Ramakrishna, Hemalatha, and Rao conducted [7] a study on the analysis and performance of a centrifugal pump impeller. They analyzed the impeller's design and performance using various methods, including CFD and experimental tests. The study found that the impeller's performance was influenced by factors such as its design parameters and operating flow rates. Authors [7-16] also investigated the internal flow characteristics in centrifugal pumps using the CFD tools.

Even though previous works have contributed to the development of centrifugal pumps, they still have some limitations. They can be less efficient than other types of pumps, particularly when handling viscous liquids. They also require a minimum amount of liquid to be present in the volute casing to prevent damage to the impeller. Thus, it is imperative to investigate the unsteady force applied on the impeller blades to prevent damage in centrifugal pumps since they remain a critical component of many industries, and their importance is likely to grow in the future. In the present study, CFD methods are employed to predict the unsteady force on the blade and volute due to the rotor-stator interactions under different flow rates. The findings of the study can provide a physical basis for the design of hydraulic pumps which is critical in determining their performance, efficiency, and reliability. Moreover, understanding the behavior of the unsteady force of the pump can help to identify potential issues that may arise as flow rate changes, such as blade wear or cavitation

2. PUMP MODEL

The main geometric and operational parameters of the 3D centrifugal flow pump in this study are as below: impeller outer diameter = 420 mm, blade number = 6, design flow rate Q_d = 500 $m^3h^{-1,}$ rotational speed, n = 1450 rpm, and design head, H_d = 48.2 m. The 3D geometric model of the whole flow passage is displayed in Figure 1.

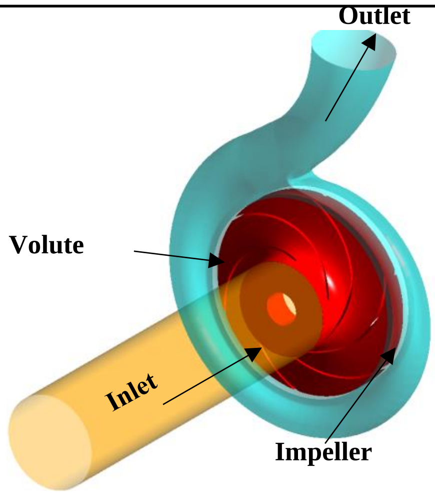


Figure 1. 3D geometric configuration of the investigated centrifugal pump

3. THEORY

The flow in the centrifugal pump is governed by the continuity and momentum equations to determine the force applied on the blades. The conservation equations of mass and momentum for incompressible flow in the pump are written in Cartesian coordinates as follows [17-20]:

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

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial\left[(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]}{\partial x_j} + f_i \tag{2}$$

where f_i is the Coriolis force; ρ is the fluid density, u is the relative velocity, p is the pressure; μ is the viscosity of the fluid; μ_t is the turbulent viscosity. The turbulence flow is solved by using the k-ε based Shear-Stress-Transport (SST) model [8, 21]. This turbulence model is highly accurate for the prediction of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy viscosity. The eddy viscosity is determined by:

$$\mu_t = \frac{\rho \alpha_1 k}{\max(\alpha_1 \omega, SF_2)} \tag{3}$$

here α_1 is the model constant which is given as 5/9; S is the invariant measure of strain rate; F_2 is the blending function; k and ω are respectively the turbulent kinetic energy and the turbulence frequency.

4. CFD METHODS

4.1 Meshing

To start simulating the centrifugal pump impeller, a computational mesh is required. The meshing procedure divided the entire computational domain of the pump into three main sections (Inlet, Impeller, and Outlet). The structured hexahedral topology was used to generate all the computational grids (See Figure 2). Due to the complex nature of the blade and volute tongue, their surfaces were refined. Increasing the number of grids generates more calculations and finer mesh produces more accurate results. The non-dimensional y+ values of the mesh of the walls of the impeller blades and volute are between 20 and 30. At the design flow rate, a mesh independence investigation is carried to improve the numerical reliability. The mesh independence parameters of the mesh used for the entire simulation are as follows: mesh number of entire flow domain 5300000; efficiency 79.51% and pressure head 51.89 m.

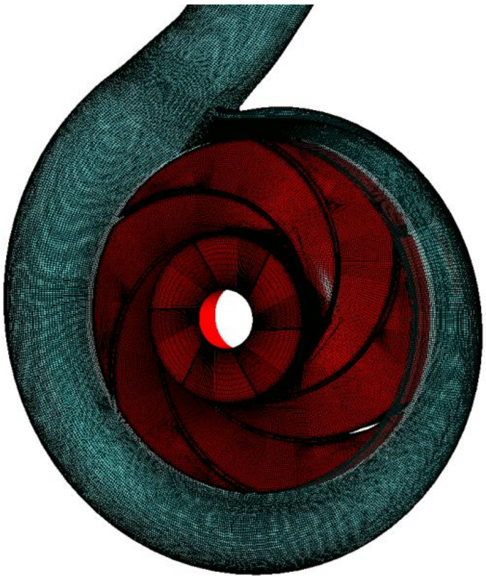


Figure 2. Entire mesh domain of the investigated centrifugal pump

4.2 Physical Boundary Conditions

The ANSYS commercial software was used to perform the numerical simulations based on the Unsteady Reynolds-Averaged-Navier Stokes URANS) equations. In the unsteady flow simulations of the investigated centrifugal pump, water assumed to be incompressible at a temperature of 25°C was used as the material fluid. The inlet boundary condition was defined as the mass flow rate of the flow entering the pump. This is typically set as a time-varying function to simulate the unsteady flow behavior of the pump. The outlet boundary condition was set as static pressure of the fluid exiting the pump. The wall boundary conditions were defined as no-slip condition, which assumes that the fluid velocity is zero at the walls of the pump, and the adiabatic condition, which assumes that there is no heat transfer between the fluid and the walls of the pump. The interface boundary for the steady simulations is set as frozen however upgraded to the transient rotor-stator for the unsteady simulations between the rotating impeller and the stationary volute. These conditions typically include continuity of mass, momentum, and energy across the interface. The time step for the unsteady simulation was computed and set at 1.2×10^{-4} s representing one degree of each impeller turn. The flow equations were set to converge less than 10^{-4}. The reference pressure was fixed at 1 atm with a 5% turbulence intensity

5. RESULTS AND DISCUSSION

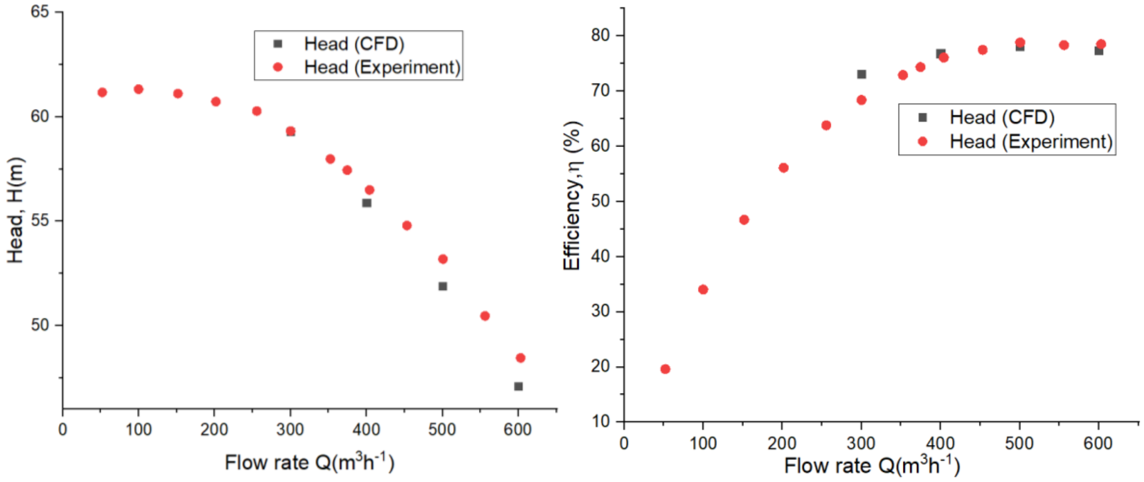
5.1 Verification of numerical simulations

To ensure the robustness and reliability of the CFD simulations, experiments are carried out to determine and compare the head and efficiency of a centrifugal pump through both means. The experimental apparatus included a flow meter, pressure gauges, a thermometer, and a computer for data acquisition. A closed-loop piping system that returns the pumped water to the reservoir was used. The flow rate of the water pumped at different flows by the centrifugal pump needs is measured at a rotational speed of 1450 revolutions per minute. The pressure at the suction or inlet and discharge or outlet of the pump is measured using the pressure gauges. The pressure should be measured at different pump speeds. The thermometer is used to measure the temperature at 25°C. The pressure head (H) and efficiency (η) of the pump is computed using Eqns. [4] and [5].

$$H = \frac{P_{Outlet} - P_{Inlet}}{\rho g} \tag{4}$$

$$\eta = \frac{\rho g Q H}{M\omega} \tag{5}$$

where the P = pressure, ρ = density of water, g = acceleration due to gravity and M = torque



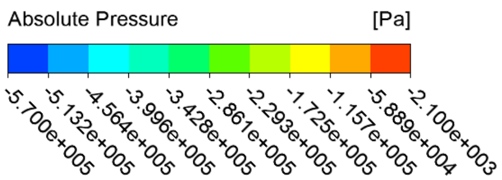
(a) (b)

Figure 3. Performance curves (a) Head (b) Efficiency

Figure 3 presents the graph of the performance of the pump. The performance curves of the pump show the relationship between flow rate, head (pressure), and efficiency. From Figure 3 the CFD results matched well with the experimental data for most of the operating flow rates. This observation agrees with the works of the Authors [3-5, 15]. However, some discrepancies were observed in predicting the head and efficiency of the pump at low flow rates (Q = 300 m^3h^{-1}). This could be due to the limitations of the turbulence model used in the CFD simulations. An acceptable range of maximum deviations of less than 1.0% and 2.5% were recorded between the CFD and experimental results for the head and efficiency curves respectively. Thus, the CFD results demonstrate the potential of CFD simulations in predicting the force and stress on the impeller blades of the investigated pump.

5.2 Pressure Distribution Characteristics

Understanding the absolute pressure in the design and operation of centrifugal pumps is key, as it can impact factors such as the efficiency, cavitation, and overall performance of the pump[22]. The absolute pressure of a centrifugal pump refers to the total pressure at a given point in the pump, including both the static pressure and the dynamic pressure. The absolute pressure can be calculated using Bernoulli's equation, which relates the pressure, velocity, and elevation of the fluid in the pump. The equation states that the sum of the static pressure, dynamic pressure, and potential energy per unit mass of the fluid is constant along a streamline [23]. Figure 4 shows the pressure exerted on the blade induced by the fluid force. The pressure at the center of the impeller is the lowest, where the fluid enters the eye of the impeller. As the fluid moves radially outwards from the center of the impeller, it gains kinetic energy and its pressure decreases. The highest pressure is developed at the trailing edge of the blade, where the fluid is forced to change direction and exit the impeller. The pressure increases gradually along stream-wise direction within the impeller flow passage and thus the pressure on the pressure side of the blade is higher than the suction side. At $0.8Q_d$ flow rate, the pressure distribution on the blade is relatively uniform with the pressure being highest near the trailing edge and decreasing towards the leading edge. As the flow rate increases, the pressure near the trailing edge increases significantly, resulting in a higher pressure gradient across the blade. The pressure gradient is the rate at which pressure changes across the blade, and it is a key factor in determining the efficiency of the pump [9, 24].



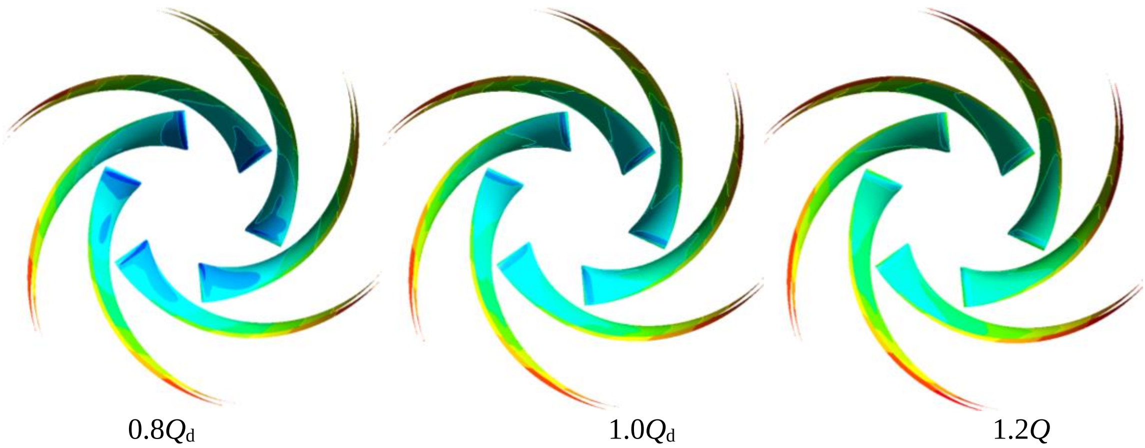


Figure 4. Pressure distribution on the impeller blades

Figure 5 shows the pressure distribution in the entire pump. After the fluid exits the impeller, the fluid pressure gradually decreases as it flows through the volute casing, which is designed to convert the kinetic energy of the fluid into pressure energy. The pressure distribution is affected by the volute tongue shape and the impeller blade. The pressure distribution at the impeller eye is relatively lowest at the flow rate 1.2Q_d and least at the design flow rate. The walls of the volute recorded maximum pressure of about 1.40×10^5 Pa at the design flow rate.

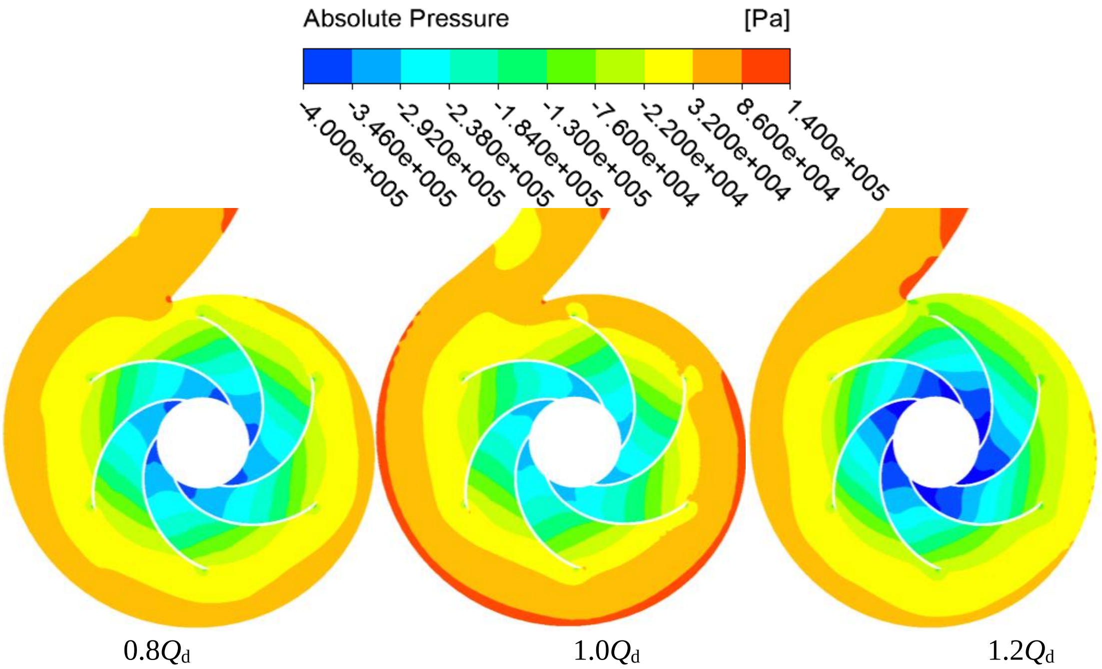


Figure 5. Pressure distribution in the pump

5.3 Tangential Force Characteristics on the Blade

The tangential force is computed using the momentum equation, which relates the force exerted on a control volume of fluid to the change in momentum of the fluid as it flows through the volume. The blade is the rotating component that transfers mechanical energy to the fluid, thereby increasing its kinetic energy and pressure. As the fluid flows through the impeller, it exerts a tangential force on the blades of the impeller [23, 25, 26]. The tangential force, F_t on the impeller of a centrifugal pump is calculated using the Eqn. 6.

$$F_t = \rho Q v r \tag{6}$$

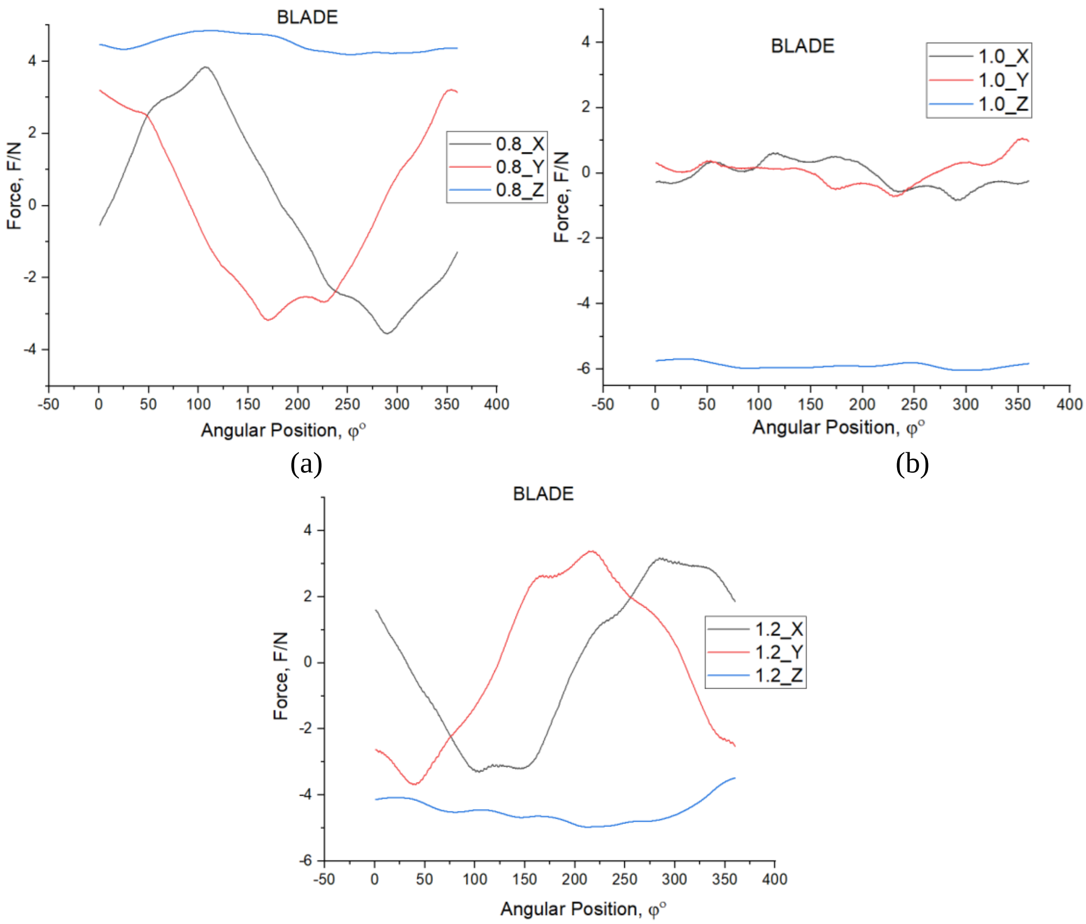
where F_t = tangential force, ρ = density of water, Q = flow rate, v = tangential velocity of the fluid at the impeller blade, and r = distance from the center of the impeller to the point. The tangential velocity of the fluid is calculated using Eqn. 7.

$$v = \omega r \tag{7}$$

where ω = angular velocity of the impeller. Thus, the tangential force on the blade was computed using Eqn. 8.

$$F_t = \rho Q \omega r^2 \tag{8}$$

Figure 6 compares the tangential force exerted on the blade at different angular positions of the impeller. Generally, the tangential force exerted on the blade in x, y and z directions was dynamic with respect to the impeller position and the flow rate. At low flow rate, $0.8Q_d$, the force on the blade recorded relatively high peaks in the x and y directions at 50 and 100 angular positions respectively. On the other hand, the respective low valleys were recorded at 175 and 300 angular positions in the x and y directions. In the z direction, the force was relatively high and steady. A counter observation is revealed at high flow rate, $1.2Q_d$. The force in the z direction was relatively low and steady. Low valleys were recorded at 50 and 100 angular positions in the x and y directions respectively. The sharp peaks were recorded at 225 and 300 angular positions in the x and y directions respectively. This indicates how the flow in the pump is highly turbulent at $0.8Q_d$ and $1.2Q_d$ flow rates. Again, at high flow rates, the force on the blade can become quite significant and can put stress on the pump components. At the $1.0Q_d$ flow rate, the force was steady in all directions and the force exerted on the blade was quite moderate compared to the other two flow rates.

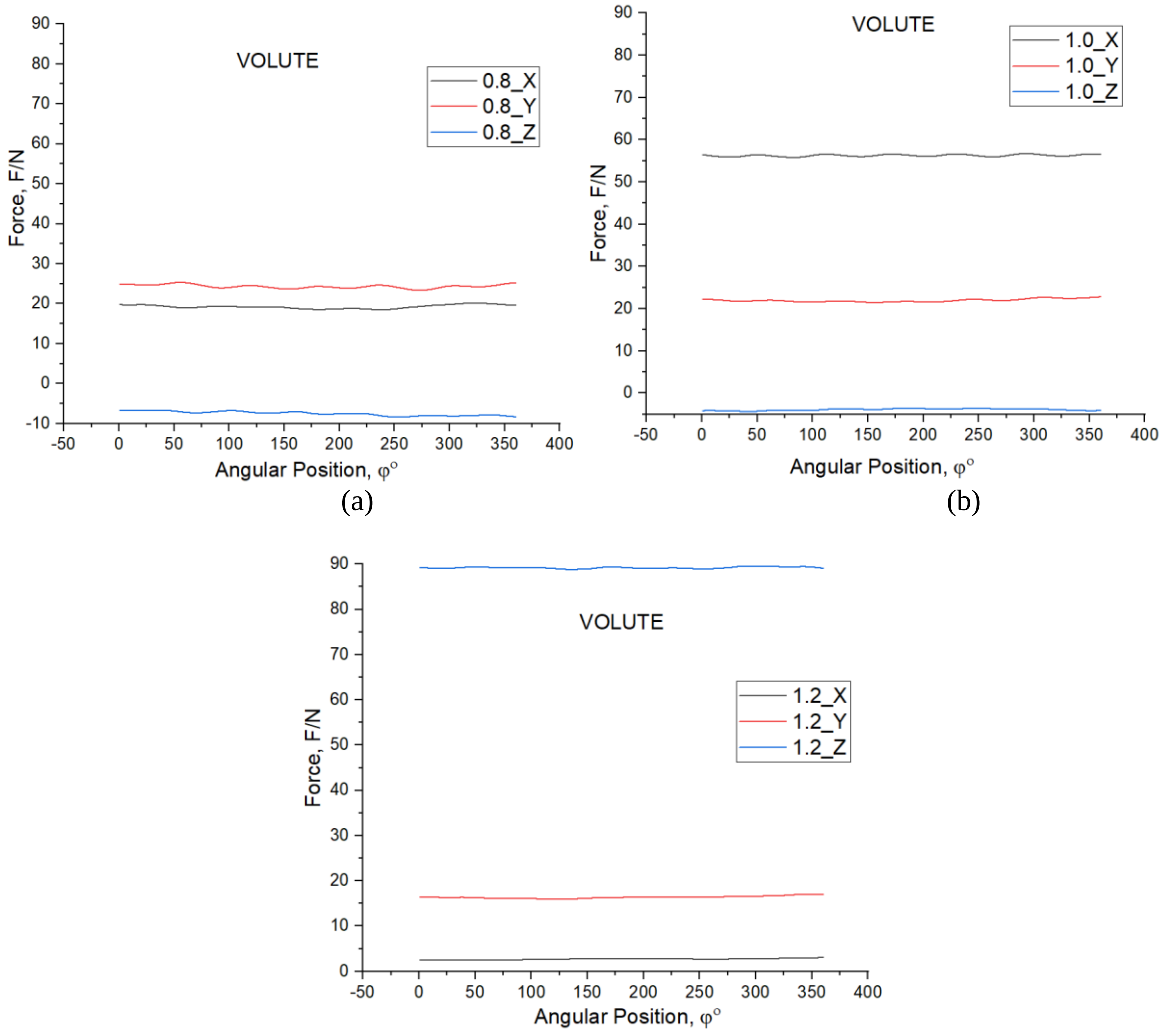


(a) (b) (c)

Figure 6. Tangential force on the impeller blades

5.4 Tangential Force Characteristics on Volute

It is important to consider the tangential force on the volute in the design and operation of centrifugal pumps, as it can impact factors such as the structural integrity of the pump and the efficiency of the fluid flow [10]. Generally, at low flow rates, the force on the volute is relatively low, and as the flow rate increases, the force on the volute also increases. This pattern was also observed in the studies of Authors [5, 27]. The force applied on the volute is greater than the impeller. The variations between the peaks and valleys are steady compared to that of the force profile on the blade. The force on the z direction was the lowest at 0.8Q_d and 1.0Qd except 1.2Q_d which recorded the highest force around 90N.



(a) (b) (c)

Figure 7. Tangential force on the volute

6. CONCLUSION

The study investigated the stress characteristics induced by the flow on the blades of a centrifugal pump at different flow rates and directions. Firstly, the study revealed the pressure exerted on the blade and volute. The absolute pressure distribution on the blade of a centrifugal pump is complex and highest near the trailing edge and decreases towards the leading edge. It is important to state that the pressure distribution was greatly influenced by the operating flow rates. Secondly, the tangential force on the blade is an important parameter to consider during the design of the pump was investigated. It was generally observed that the tangential force is dependent on the flow rate. With higher flow rates greater tangential forces are exerted on the blades and volute. This relationship arises due to the centrifugal force generated by the impeller, which is directly proportional to the flow rate. In conclusion, the study has shed light on the complex and turbulent relationship between flow rate and tangential force in centrifugal pumps and emphasized the importance of considering this relationship when designing and operating these critical components of the pump.

ACKNOWLEDGEMENTS

My sincere gratitude to Dr. Desmond Appiah of University of Education, Winneba-Ghana for his support throughout this study.

REFERENCES

[1] Yu, T., et al., *Numerical study on hydrodynamic characteristics of a centrifugal pump influenced by impeller-eccentric effect.* Engineering Failure Analysis, 2022. **138**: p. 106395.

[2] Al-Obaidi, A.R., *Numerical investigation on effect of various pump rotational speeds on performance of centrifugal pump based on CFD analysis technique.* International Journal of Modeling, Simulation, and Scientific Computing, 2021. **12**(05): p. 2150045.
[3] Zhou, W., et al., *Investigation of flow through centrifugal pump impellers using computational fluid dynamics.* International Journal of Rotating Machinery, 2003. **9**(1): p. 49-61.
[4] Muttalli, R.S., S. Agrawal, and H. Warudkar, *CFD simulation of centrifugal pump impeller using ANSYS-CFX.* International Journal of Innovative Research in Science, Engineering and Technology, 2014. **3**(8): p. 15553-15561.
[5] Pei, J., S. Yuan, and J. Yuan, *Dynamic stress analysis of sewage centrifugal pump impeller based on two-way coupling method.* Chinese journal of mechanical engineering, 2014. **27**(2): p. 369-375.
[6] Cárdenas-Gutiérrez, J.A., G. Valencia Ochoa, and J. Duarte Forero, *Parametric analysis CFD of the hydraulic performance of a centrifugal pump with applications to the dredging industry.* Journal of Engineering Science and Technology Review, 2020. **13**(3 (2020)): p. 8-14.
[7] Ramakrishna, R., S. Hemalatha, and D.S. Rao, *Analysis and performance of centrifugal pump impeller.* Materials Today: Proceedings, 2022. **50**: p. 2467-2473.
[8] Huang, P., et al., *Energy dissipation mechanism of a centrifugal pump with entropy generation theory.* AIP Advances, 2021. **11**(4): p. 045208.
[9] Jia, X.-Q., et al., *Internal unsteady flow characteristics of centrifugal pump based on entropy generation rate and vibration energy.* Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, 2019. **233**(3): p. 456-473.
[10] Kim, E. and A. Palazzolo, *Rotordynamic force prediction of a shrouded centrifugal pump impeller—part I: numerical analysis.* Journal of Vibration and Acoustics, 2016. **138**(3).
[11] Kim, J.-H., et al., *Design techniques to improve the performance of a centrifugal pump using CFD.* Journal of Mechanical Science and Technology, 2015. **29**: p. 215-225.
[12] Nigussie, T. and E. Dribssa, *Design and CFD analysis of centrifugal pump.* International Journal of Engineering Research and General Science, 2015. **3**(3): p. 668-677.
[13] Tan, L., et al., *Numerical study on characteristics of unsteady flow in a centrifugal pump volute at partial load condition.* Engineering computations, 2015. **32**(6): p. 1549-1566.
[14] Wu, C., et al., *Effects of blade pressure side modification on unsteady pressure pulsation and flow structures in a centrifugal pump.* Journal of Fluids Engineering, 2021. **143**(11).
[15] Yan, P., et al., *Computational fluid dynamics-based pump redesign to improve efficiency and decrease unsteady radial forces.* Journal of fluids Engineering, 2017. **139**(1).
[16] Yedidiah, S., *A study in the use of CFD in the design of centrifugal pumps.* Engineering Applications of Computational Fluid Mechanics, 2008. **2**(3): p. 331-343.
[17] Menter, F.R., *Turbulence modeling for engineering flows.* Ansys, Inc, 2011.
[18] Sodja, J., *Turbulence models in CFD.* University of Ljubljana, 2007: p. 1-18.
[19] Wilcox, D.C., *Turbulence modeling for CFD*. Vol. 2. 1998: DCW industries La Canada, CA.
[20] Mishra, P. and K. Aharwal. *A review on selection of turbulence model for CFD analysis of air flow within a cold storage*. in *IOP conference series: Materials science and engineering*. 2018. IOP Publishing.
[21] Bai, Y., D. Appiah, and Y. Tao, *Computational turbulent flow characteristics in a centrifugal pump.* AIP Advances, 2022. **12**(7): p. 075025.
[22] Zhao, X., et al., *Unsteady flow and pressure pulsation characteristics analysis of rotating stall in centrifugal pumps under off-design conditions.* Journal of Fluids Engineering, 2018. **140**(2).
[23] Bachus, L. and A. Custodio, *Know and understand centrifugal pumps*. 2003: Elsevier.
[24] Li, Y., et al., *Investigation of unsteady flow in a centrifugal pump at low flow rate.* Advances in Mechanical Engineering, 2016. **8**(12): p. 1687814016682151.
[25] Gülich, J.F., *Centrifugal pumps*. Vol. 2. 2008: Springer.
[26] Karassik, I. and J.T. McGuire, *Centrifugal pumps*. 2012: Springer Science & Business Media.
[27] Li, W., et al., *Fluid-structure interaction study of a mixed-flow pump impeller during startup.* Engineering computations, 2018.