M. N. Idris

Application of CFD to Improve on Reliability and Efficiency of Centrifugal Pump for Oil and Gas Processing Plant

idrismn@hotmail.com
Process Refining Technology and Sustainable Energy Development Group, Department of Chemical Engineering, University of Maiduguri, Nigeria.

The link to this publication is **https://ajoeer.org.ng/otn/ajoeer/2022/qtr-1/08.pdf**

Engr. Prof. M. N. Idris is a professor in the Department of Chemical Engineering, University of Maiduguri, Nigeria. He is an expert in oil and gas (upstream, midstream and downstream), CFD modelling and simulation, fluidization, nanotechnology (nanosynthesis), process economies, renewable and sustainable energy development. He holds PhD in Chemical Engineering from the University of Leeds, United Kingdom.
His research interests are on applications of computational fluid dynamics (CFD) in modelling and simulation of process systems and design, reactions engineering, nanotechnology and biotechnology etc. He is a full registered and a senior member of American Institute of Chemical Engineers (MAIChE), a chartered member of the Institution of Engineers (C.Eng., UK), a chartered member of the Institution of Chemical Engineers (MIChemE, UK), Nigerian Society of Chemical Engineers (MNSChE), Nigerian Society of Engineers (MNSE), Council for the Regulations of Engineering in Nigeria (MCOREN) and Professional member of Materials Science and Technology Society of Nigeria (MMSN). He is a fellow of the institute of management consultant (FIMC CWC Global) and a fellow of the institute of management specialists (FIMS UK). For further enquiries or

ISSN: 2635-2974

Application of CFD to Improve on Reliability and Efficiency of Centrifugal Pump for Oil and Gas Processing Plant

M. N. Idris*, A. A. Kois

Process Refining Technology and Sustainable Energy Development Group, Department of Chemical Engineering, University of Maiduguri, Nigeria. Email: idrismn@hotmail.com

ABSTRACT

Centrifugal pump is one of the standard pump specification uses in process industries for qualitative and quantitative fluid transfer. Most industrial application of centrifugal pump has mechanical configuration problems that resulted to pump inefficiency and malfunctions. Fault or bad design pumps are characterized due to quality engineering design which prompted this investigation. In this study, CFD code was used to simulate pump efficiency and compared with real experimental studies. Within the limit of simulation experimental error, it was found that, the centrifugal pump was designed within the constraint of pumping at a flow rate of 0.04 m^3/s and at a Head of 55-70 m. After the design calculations, the system was model and drawn in SOLIDWOKS, it was found that the pump was given the required material from the software material library in line with the design specifications and parameters using local materials. The performance of the pump was simulated and evaluated to have a flow rate of 0.043 m^3/s, velocity of 8.2 m/s with a head of 70 m and various efficiencies ranging from 68.45 - 81.61%. After the analysis was done, 59^0 it was identified to be the optimum impeller outlet blade angle in which volumetric efficiency and hydraulic efficiency is maximum. The efficiency pressure ratio has also increased; therefore the pump head also increase.

Keywords: Flow, pump, efficiency and reliability

1.0 INTRODUCTION

Centrifugal pump are devices simply design to raise, compress, or transfer liquids or gases and is operated by a piston or similar mechanism. The two main parts of pump are the impeller and diffuser. The *impeller* is the only moving part, is attached to a shaft and driven by a motor. Impellers are generally made of bronze, polycarbonate, cast iron, stainless steel as well as other materials. The diffuser (also called as *volute* – spiral shape) houses the impeller and captures and directs the water off the impeller Guelich, 2020. Fluids enter the center (eye) of the impeller and exeunt the impeller with the help of centrifugal force. As fluid leaves the eye of the impeller a low-pressure area is created, causing more fluid to flow into the eye. Atmospheric pressure and centrifugal force cause this to happen. Velocity is developed as the water flows through the impeller spinning at high speed. The water (fluid) velocity is collected by the diffuser and converted to pressure by specially designed passageways that direct the flow to the discharge of the pump, or to the next impeller should the pump have a multi-stage configuration. The pressure (head) that a pump will develop is in direct relationship to the impeller diameter, the number of impellers, the size of impeller eye, and shaft speed. Capacity is determined by the exit width of the impeller. The head and capacity are the main factors, which affect the horsepower size of the motor to be used. The more the quantity of water to be pumped, the more energy is required Anderson, 1994.

1.1 Problem Statement

The continuous concern to improve on the mechanical device of centrifugal pump on flowing fluid activities remain a subject matter. Most industrial application of centrifugal pump has mechanical configuration problems that resulted to pump inefficiency and malfunctions. Fault or bad design pumps are characterized due to quality engineering design.

1.2 Significance of the Study

The significance of this study is to improve on the efficiency, durability and performance of the centrifugal pump. The importance of this work is the applicability of the result which can be scale up to industrial-real life approach.

2.0 BACKGROUND LITERATURES

Several studies were carried out on a new horizon for pump selection; cavitations study, performance enhancement of pumps etc. are summarized. Research gaps are identified and objectives of the present study are framed. The information presented below are Based on the literature review, research gaps are identified and objectives of the present study are framed. Dick et al., 2001 have used CFD-code Fluent 20.4 for the flow analysis of two test pumps of end-suction volute type, one of low specific speed and one of medium specific speed. For both, head as function of flow rate for constant rotational speed is known from experiments. First, the impeller is generated. One impeller channel is meshed and is then rotationally copied the necessary number of times. For the first pump, the impeller is completely two-dimensional. The impeller mesh is made with hexahedra and wedge cells. For the second pump, the impeller channel is much more complex. The mesh is made in a completely unstructured way, mainly using tetrahedron, but other cell forms like pyramids, hexahedra and wedges also occur. Table 1 depicts the summarised version of recent reviewed studies investigated.

Table 1 Summarised version of reviewed investigations

Author(s)	Investigation	Research Benefits	Remarks
Sheth, 2014	Optimization of design parameters using this technique is directly inclined towards economic solution for the turbo machinery industry	Parametric specification are chosen from the ranges where the blower will get the best efficiency	The validations of the CFD results were in fine conformity between the CFD results and the experimental results
Mishra et al., 2016	Performance parameters of centrifugal pump such as overall efficiency, cavitations, slip factor, losses etc. have been evaluated	Open well centrifugal pump namely impeller 165mm, 210 mm, 170 mm, and123 mm were selected for the performance analysis	Overall efficiency of the pump is increases as the flow rate and head decreases whereas the power input is increased.
Singh et al., 2017	Studying the relationships among the impeller eye diameter, vane exit angle and width of the blade at exit	CFD analysis is carried out on the developed models to predict the performance virtually and to verify with the experimental result of the pump.	The objective functions are defined as the total head and the total efficiency at the design flow-rate.

3.0 MATERIALS AND METHODS

3.1 Sample collection and Preparation

Experimental set up: In order to validate the accuracy of numerical computation, it is necessary to conduct a simulation alongside theoretical evaluation to assess the pump performance and efficiency.

3.2 Main Specification of the Centrifugal Pump used:

1. Pump type: - = H 47 Centrifugal Pump
2. Head height (H):- = 10 to 120 m
3. Discharge (Q):- = 0.04 m^3/sec
4. Pump Speed (N):- = 2000 - 2900 RPM (2000)

Impeller Parameters:

$$N = \rho g H Q$$

$$N = 1000 \times 9.81 \times 0.55 \times 0.04 = 215.82 W$$

3.3 Summary of Calculated Data from Existing pumps (Impeller):

1. Nominal diameter at inlet (D): = 50.96mm
2. Torque (T): = 6.76 Nm
3. Shaft diameter (d_s): = 78mm
4. Impeller hub diameter (d_h) : = 116mm
5. Impeller eye diameter (d_e) : = 101.4mm
6. Velocity of fluid at impeller eye (C_0) = 8.39m/s
7. Flow velocity at inlet (Cm_1): = 12.58m/s
8. Inlet blade angle (β_1): = 37^0
9. Outlet blade angle (β_2): = 53^0
10. Outlet diameter of impeller (D_0): = 204.6mm

The detail analysis of these calculations stated above can be found on *Idris and Kois 2020.*

3.4 Design Modification of Existing Impeller to Improve its Efficiency

The existing design of the impeller blade outlet angle is 53^0 and inlet angle is 37^0. The modification in design was done in such a way that the inlet angle is fixed and outlet angle was changed in the step of 2 (two) degrees of incremental. So it was analyzed that at which point efficiency will be maximum. However the blade outlet blade angle is also decreased it check for its effects on the efficiency of the pump.
The CFD code was used and the applications of all the boundary conditions which already given in existing design parameter is added in FLOW EXPRESS MODULE using Finite Element Method in which mainly 3-processes (pre-, processing, and post-) was carried out.

Impeller Parameters:

$$N = \rho g H Q$$
(1)

$$N = 1000 \times 9.81 \times 0.55 \times 0.04 = 215.82W$$

Inlet diameter of impeller is:

$$D_1 = (1.1 \sim 1.15) K_o \sqrt{\frac{Q}{n}}$$
(2)

Where K_o is the constant parameter which value is chosen as 4.5.

$$D_1 = 1.1 \times 4.5 \times \sqrt{\frac{0.04}{308.3}} = 0.05096m = 50.96mm$$

The outlet diameter of impeller is:

$$D_2 = 19.2 \frac{\sqrt{2gH}}{n}$$
(3)

$$D_2 = 19.2 \times \frac{\sqrt{2 \times 9.81 \times 0.55}}{308.3} = 0.20457m = 204.6mm$$

The torsional moment is estimated by:

$$T = 9.65 \frac{N}{n} = 9.65 \times \frac{215.82}{308.3} = 6.76Nm$$
(4)

Shaft diameter d:

$$D = \sqrt[3]{\frac{60 \times T}{\tau \times FOS}} \quad \sqrt[3]{\frac{60 \times 6.76}{247\,\mathrm{E}\,106 \times 2}} = 0.0776m = 78mm$$
(5)

Where τ is the allowable stress of the shaft (mild steel)

τ = 247 MPa = 247 $\times 10^6$ Pa

4.0 RESULTS AND DISCUSSIONS

4.1 Hydraulic Power

The power imparted to the fluid (water) by the pump is called water power or Hydraulic power. To calculate hydraulic power, the flow rate and the pump head must be known. As a result, to provide a certain amount of power to the water a larger amount of power must be provided to the pump shaft. This power is called *brake* power. The efficiency of the pump determines how much more power is required by the shaft to propel.

4.2 CFD Analytical Results for the Variation of Impeller Angles

The following are the steps in which the analysis has been carried out:

- Generation of basic Geometrical model
- Mashing of the model
- Applying the boundary conditions
- Result of the CFD analysis (Pressure Region)

The impeller inlet blade angle is kept constant and outlet angles have been varied in the fixed degrees and its CFD models have been analyzed for its impact on the pressure, hydraulic efficiency and volumetric efficiencies.

4.3 Simulation Results

The conceptual, analytical, impeller and pump casting were presented in Figures 1- 4.

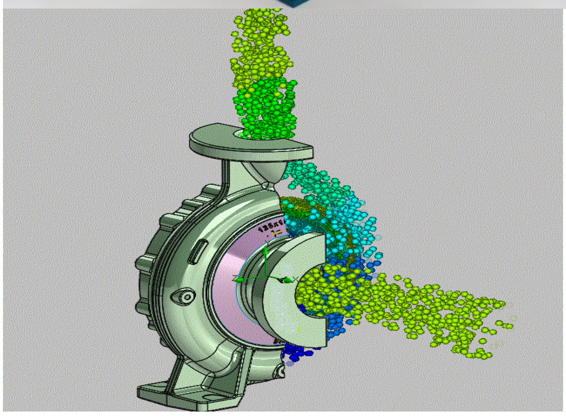
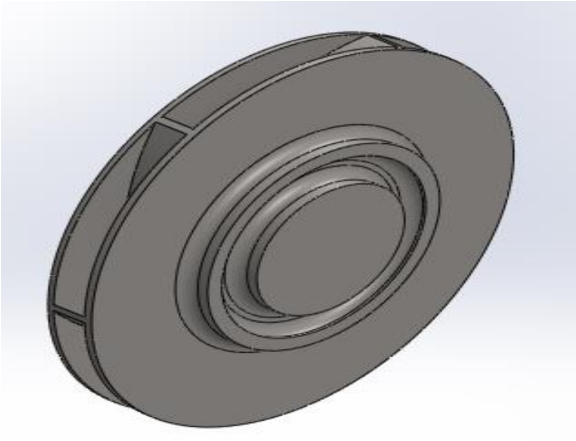


Figure 1: Original Model **Figure 2**: Model Analysed



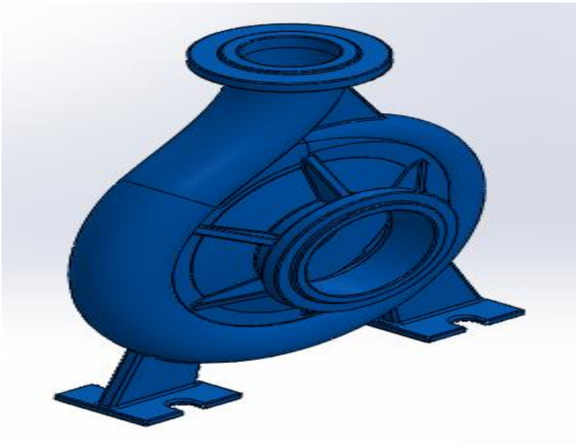
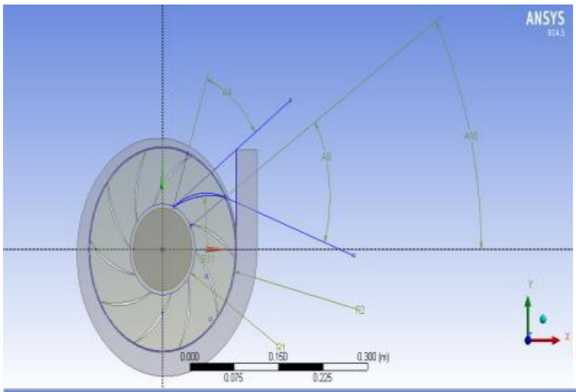


Figure 3: Impeller **Figure 4**: Centrifugal pump casting

Figure 5 – 18 represent the serial configuration of CFD calculations for different experimental runs. This comprises of basic geometry and simulation for delivery (pump exit). ANSYS CFD 2020.

4.3.1 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 49^0



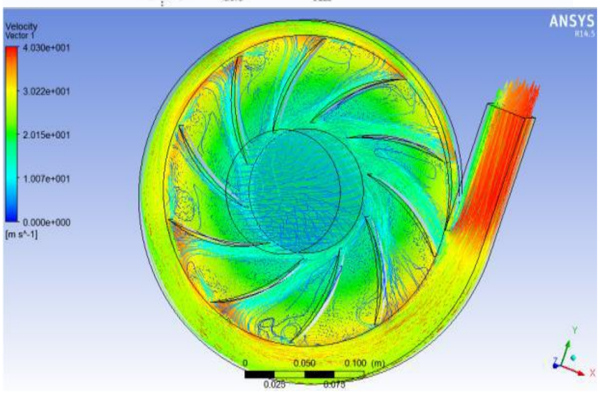
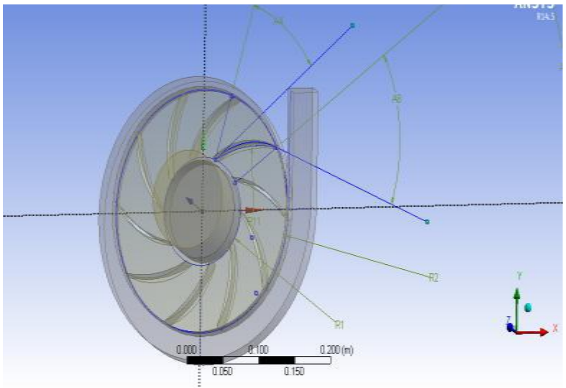


Figure 5: Basic geometry

Figure 6: CFD results (delivery pipe pressure)

4.3.2 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 51^0



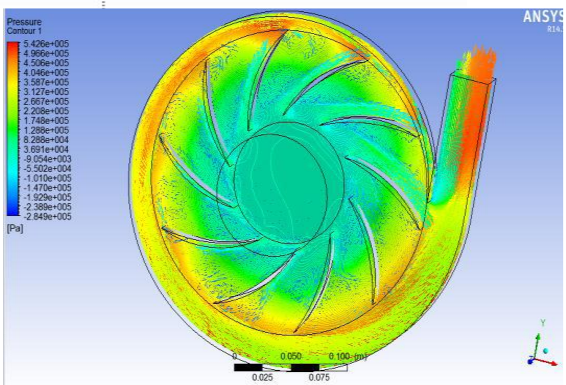
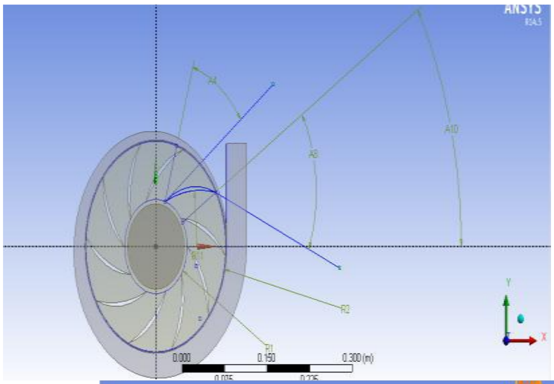


Figure 7: Basic geometry

Figure 8: CFD results (delivery pipe pressure)

4.3.3 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 53^0



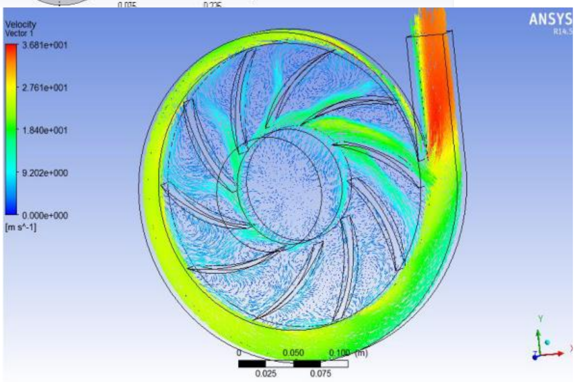
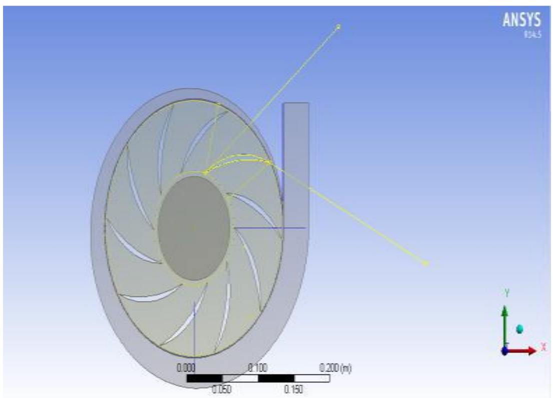


Figure 9: Basic geometry

Figure 10: CFD results (delivery pipe pressure)

4.3.4 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 55^0



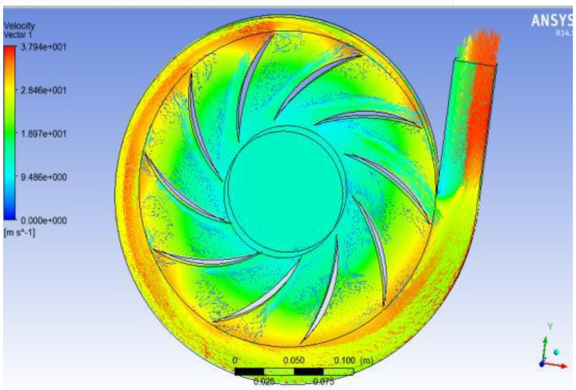
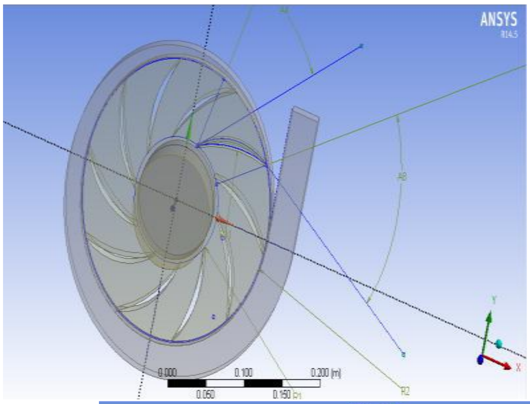


Figure 11: Basic geometry

Figure 12: CFD results (delivery pipe pressure)

4.3.5 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 57^0



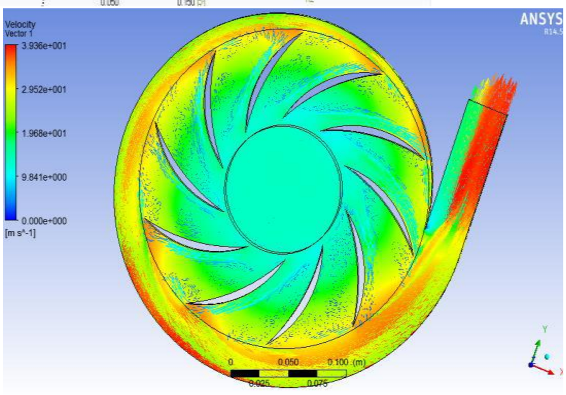
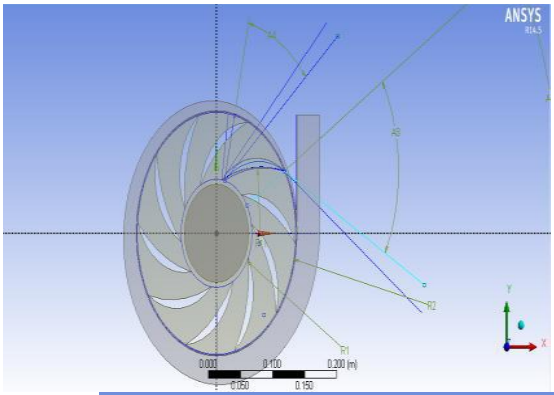


Figure 13: Basic geometry

Figure 14: CFD results (delivery pipe pressure)

4.3.6 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 59^0



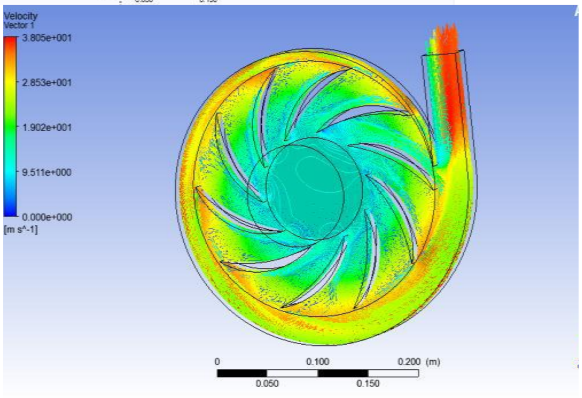
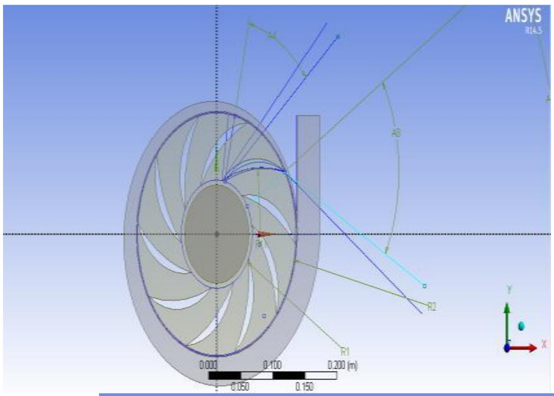


Figure 15: Basic geometry **Figure 16**: CFD results (delivery pipe pressure)

4.3.7 Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 65^0



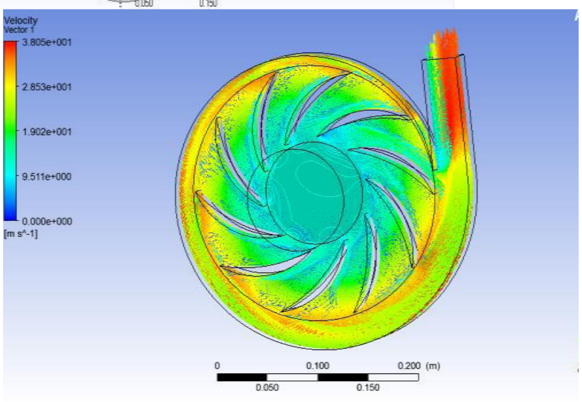


Figure 17: Basic geometry **Figure 18**: CFD results (delivery pipe pressure)

Some of the detail analyses of the Figures 1 – 18 are presented in Table 1 as follows.

4.4 Results Summary

Table 1 depicts the summary of the centrifugal pump CFD calculations for the 7-experiments conducted.

Table 1: Summary of centrifugal pump modelling configuration

S/No.	Inlet Angle	Outlet Angle	Hydraulic Efficiency η_h	Volumetric Efficiency η_v	Pressure ratio (output/ input) (p2/p1)
1	37^0	49^0	39.21%	68.45%	4.23
2	37^0	51^0	41.14%	71.21%	4.49
3	37^0	53^0	42.28%	75.28%	5.43
4	37^0	55^0	48.56%	77.54%	6.18
5	37^0	57^0	53.71%	79.26%	7.23
6	37^0	59^0	57.59%	81.61%	7.49
7	37^0	65^0	52.42%	77.27%	6.99

4.5 Theoretical Calculations of the Volumetric and Hydraulic Efficiency

$$u_2 = \sqrt{\frac{C_{m2}}{2tan\beta_2}} + gH$$

$$= \sqrt{\frac{8.64}{2*tan59°}} + 9.8 \quad = 30.45 \text{ m/sec}$$

$$tan\beta_2 = \frac{V_{f2}}{U_2 - V_{w2}}$$

$$\tan 59° = \frac{8.39}{31.45 - V_{w2}}; \; V_{w2} = 25.25 \text{ m/sec}$$

$$\text{Volumetric Efficiency} = \frac{g\,H}{V_{w2}*U_2} = \frac{9.81*70}{26.25-31.45} = 82.64\%$$

$$Hdraulic\ Efficiency = \frac{actual\ head\ of\ centrifugal\ pump}{total\ head\ of\ centrifugal\ pump} = \frac{70}{120} = 58.1\%$$

Table 2 represents the summary of theoretical calculations and CFD analysis of impeller at 59^0 of the centrifugal pump.

Table 2: Theoretical calculation and CFD analysis

Parameters	Theoretical Calculation	CFD Analysis
Blade Outlet Angle (^{0})	59	59
Hydraulic Efficiency (%)	58.1	57.59
Volumetric Efficiency (%)	82.64	81.61
Difference in Hydraulic Efficiency (%)	0.51	
Difference in Volumetric Efficiency (%)	1.03	

4.5 Analysis of the Pump Impeller

Figures 1 – 18 presents the serial CFD results of the centrifugal pump simulations. It is evidence that the results are in conformity with the experiments of Shetha, 2014; Mishra et al., 2016 and Singh et al., 2017. In analyzing the results for the impeller at 49^0 – 59^0, that is, Figure 5 – 16, it was observed that there were serial changes which culminated with the response of the impeller angle involved. From the results of the analysis it is clearly seen that the increase of the outlet blade angle from 49° to 59° provide a difference in the volumetric efficiency, hydraulic efficiency and pressure ratio in the delivery pipe with an increase in the outlet angle. By varying the outlet angle of the blade from 59° to 90°, that is, Figure 16 – 18, the volumetric efficiency, hydraulic efficiency and pressure ratio decrease with the increase of the outlet blade angle Mishra et al., 2016 and Singh et al., 2017. The optimum outlet blade angle is found at 59° which gives the best possible volumetric efficiency, hydraulic efficiency and pressure ratio for the delivery pipe in the given range of variation the outlet angle of the blade of impeller from 49° to 90° in the selected H47 type of centrifugal pump.

4.5.1 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 49^0 as depicted in Figure 5 and 6. Since the inlet impeller blade is constant at 37^0 for all the simulation case. This is the analytical factor applied in these studies. It is well culminated that the prediction drawn in this simulation is the first prediction which was in line with that of the study work of Chakraborty and Pandey (2011). The outlet delivery of fluid (water) in this case study was very high as depicted in Figure 6. ANSYS CFD 2020.

4.5.2 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 51^0 as depicted in Figure 7 and 8. As the inlet impeller blade remain constant at 37^0, and is the analytical factor applied in this work. It was observed that the predictions at the outlet bade of 51^0 shows well correlations similar to the work of Chakraborty and Pandey (2011); Zhou et al., 2013. The outlet delivery of fluid (water) in this case study was high too, but low when compared with the prediction in Figure 6.

4.5.3 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 53^0 as depicted in Figure 9 and 10. As the inlet impeller blade remain constant at 37^0, and is the analytical factor applied in this work. It was found that a clear profile with defined simulation predictions at the outlet bade of 53^0 was observed. These simulations were also in agreement with the work of Chakraborty and Pandey (2011); Zhou et al., 2013. The outlet delivery of fluid (water) in this case study was high too, but low when compared with the prediction in Figures 6 and 7.

4.5.4 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 55^0 as depicted in Figure 11 and 12. The inlet impeller blade remains constant at 37^0, and is the analytical factor applied in this work. It was found that clear profiles with defined simulation predictions at the outlet bade of 55^0 were observed. These simulations were also in agreement with the work of Okokpujie et al., 2017; Zhou et al., 2013. The outlet delivery of fluid (water) in this case study was high too, but low when compared with the prediction in Figures 6, 7 and 8. ANSYS CFD 2020.

4.5.5 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 57^0 as depicted in Figure 13 and 14. As the inlet impeller blade remain constant at 37^0, and is the analytical factor applied in this work. The simulation profiles identified in this case study were also in agreement with the work of Chakraborty and Pandey (2011); Okokpujie et al., 2017. The outlet delivery of fluid (water) in this case study was high too, but low when compared with the prediction in Figures 6, 7, 8 and 9.
.

4.5.6 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 59^0 as depicted in Figure 15 and 16. As the inlet impeller blade remain constant at 37^0, and is the analytical factor applied in this work. At the outlet blade of 59^0, the simulation profiles identified in this case study were also in agreement with the work of Chakraborty and Pandey (2011); Okokpujie et al., 2017; Zhou et al., 2013. The outlet delivery of fluid (water) in this case study was low when compared with the prediction in Figures 6, 7, 8, and 9. ANSYS CFD 2020.

4.5.7 The Impeller Inlet Blade @ angle 37^0 and Outlet Blade @ angle 65^0 as depicted in Figure 17 and 18. As the inlet impeller blade remain constant at 37^0, and is the analytical factor applied in this work. The simulation profiles at the outlet impeller blade of 65^0, it was identified that the predicted profiles were in agreement with the work of Chakraborty and Pandey (2011); Okokpujie et al., 2017; Zhou et al., 2013. The outlet delivery of fluid (water) in this case study was low when compared with the prediction in Figures 6 – 10.

In addition, the centrifugal pump was designed within the constraint of pumping having the flow rate of 0.04 m^3/s and at a pump Head (ΔH) of the ranges 55-70 m. After the design calculations, the system was model and drawn from the application of SOLIDWOKS. It was found that the pump has the required material from the software material library in line with the design specifications and parameters using local materials. The optimum performance of the pump was simulated and evaluated to have a flow rate Q of 0.043 m^3/s, having the flow velocity υ of 8.2 m/s with a pump head of 70 m. More importantly, the pump has various efficiencies ranging from 68.45 - 81.61%.

5.0 CONCLUSION

Within the limit of simulation experimental error, it was found that, the centrifugal pump was designed within the constraint of pumping at a flow rate of 0.04 m^3/s and at a Head of 55-70 m. After the design calculations, the system was model and drawn in SOLIDWOKS, it was found that the pump was given the required material from the software material library in line with the design specifications and parameters using local materials. The performance of the pump was simulated and evaluated to have a flow rate of 0.043 m^3/s, velocity of 8.2 m/s with a head of 70 m and various efficiencies ranging from 68.45 - 81.61%. After the analysis was done, 59^0 it was identified to be the optimum impeller outlet blade angle in which volumetric efficiency and hydraulic efficiency is maximum. The efficiency pressure ratio has also increased, therefore the pump head also increase.

Recommendations

In this research work, the following were the recommendations:

a) Further researches should be carried out on the design and use of a non-corrosive material for the impeller.
b) Further researches should be carried out on the practical improvement of the pump.
c) Different operating parameters should be used to ascertain the functionality of the material.

Acknowledgement

The authors wish to acknowledge the efforts of the African Journal of Engineering and Environmental Research (AJOEER) for the publication of this article.

REFERENCE

1. ANSYS Inc. CFD-code Fluent 2020. ANSYS Release 20.0, *Southpointe November 2020, 275 Technology Drive 000504 Canonsburg, PA 15317 ANSYS, Inc. is certified to ISO9001:2008.ansysinfo@ansys.com*, http://www.ansys.com
2. Anderson, H. (1994). '*Theory of Centrifugal Pumps'*. United Kingdom
3. Chakraborty, S. & Pandey, K. (2011). *'Numerical Studies on Effects of Blade Number Variations on Performance of Centrifugal Pumps at 4000 RPM'*. International Journal of Engineering and Technology, 3, 410-416.
4. Dick, E., Vierendeels, J., Serbruyns, S. & Vande Voorde, J. (2001). *'Performance prediction of centrifugal pumps with CFD-tools'*. Task Quarterly, 5, 579-594.
5. Guelich, J. (2020). Centrifugal pumps
6. Mishra A., Rehman, A., Paul, A. & Jain, A. 2016. *Performance Analysis of an Impeller of a Centrifugal Pump* using OpenFOAM.
7. **Idris,** M. N. and Kois, A. A **(2020),** *'Experimental Studies to improve on reliability and efficiency of centrifugal pump'.* An unpublished B.Eng. thesis submitted to the Department of Chemical Engineering, University of Maiduguri, Borno State, Nigeria.
8. Okokpujie, I., Okokpujie, K., Enesi, S. & Omoakhalen, A. 2017. *'Design, Production and Testing of a Single Stage Centrifugal Pump'*. International Journal of Applied Engineering Research, 12, 7426-7434.
9. Shetha, S. 2014. '*Parametric Study and Design Optimization of Centrifugal Pump* Impeller'-A Review. 4, 45-49.
10. Singh V, Zinzuwadia M, Sheth S and Desai R, (2017), '*Parametric Study and Design Optimization of Centrifugal Pump Impeller'*. Kalpa Engineering, 1, 507–515.
11. Zhou, L., Shi, W. & Wu, S. 2013. *'Performance Optimization in a Centrifugal Pump Impeller by Orthogonal Experiment and Numerical Simulation'*. Advances in Mechanical Engineering, 2013.