FEA of the Fire Fighting High-Pressure Centrifugal Pump via the Naiver-Stokes Equations

T. Zwane
Department of Mechanical and Industrial Engineering Technology,
University of Johannesburg, South Africa.
thembalihlezwane@gmail.com

P. Zikalala
Department of Mechanical and Industrial Engineering Technology,
University of Johannesburg, South Africa.
200834970@student.uj.ac.za

M. E. Matlakala
Department of Mechanical and Industrial Engineering Technology,
University of Johannesburg, South Africa.
motsiephrey@gmail.com

D. V. V. Kallon
Department of Mechanical and Industrial Engineering Technology,
University of Johannesburg, South Africa.
dkallon@uj.ac.za

Abstract

Centrifugal pumps are the most often used pumping equipment in hydraulics. They are hydraulic machinery that transform mechanical energy into hydraulic energy. A pump is a mechanical device that transports fluids from one location to another. Because of the 3D flow structure involving turbulence, secondary flow, cavitation, and unsteadiness, the flow analysis inside the centrifugal pump is quite complicated. In recent years, advances in the accuracy of numerical methods and increased availability of computational resources have taken turbo machinery computational fluid dynamics technologies from the realm of pure research into the competitive industrial market. In this project calculations for compressible fluids and unsteady flow were done using continuity equations and for incompressible fluid Navier-Stoke equations. The achieved Reynold number shows that the flow is turbulent with all other parameters of volume flow, tangential velocity and specific speed calculated. This study has shown that to avoid cavitation at high flow rates and pump failure, the stator blades must be modified to improve the transfer of the kinetic energy of the water flow to pressure energy.

Keywords: Centrifugal pump, Naiver-Stokes Equation, Compressible fluid, Incompressible fluid, Unsteady flow.

1. Introduction

A centrifugal pump is a mechanical device that moves a fluid by transferring rotational energy from one or more impellers (Anderson Pump and Process, Matlakala et al. 2019a). The impeller raises the velocity and pressure of the fluid and directs it towards the pump exit. The centrifugal pump is straightforward to understand, operate, and maintain because of its simple design. A time-dependent continuity equation for mass conservation, three time-dependent conservation of momentum equations, and a time-dependent conservation of energy equation makes up the Navier-Stokes equations (Hall 2015). Irrigation, water supply plants, steam power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing facilities, and mines all employ centrifugal pumps (Fatsis 2015). The reason for the three-dimensional flow structure that includes turbulence, secondary flow, cavitation, and unsteadiness, is because the flow analysis inside the centrifugal pump is complicated. Computational Fluid Dynamics (CFD) approaches have moved from pure research to competitive industrial markets as the availability of computational resources has increased and the accuracy of numerical methods has improved (Matlakala et al. 2019a, Shah et al. 2013).

CFD aids in the prediction of flow behaviour in various components of hydraulic devices prior to manufacture. Modifications to current systems can be included into a numerical model and their effects may be predicted before they are implemented. CFD analysis aids in the investigation of the impacts of numerous parameters on pump performance, both individually and by building non-dimensional group (Shah et al. 2013). The turbulence viscosity is computed using the SST model (Shear Stress Transport). These transport equations associated with the appropriate boundary conditions are solved by the CFX-13 finite-volume code. The computational domain composed of a single stator and rotor blade passages is discretized by a structured multi-block meshes. It describes in detail the numerical approach, based on URANS (Unsteady Reynolds Average Navier Stokes) to predict the pressure fluctuations. On the other hand, steady state simulations are done to evaluate the performance of centrifugal pump. The unsteady simulations allow access to the temporal behaviour of internal flow and compare the influence of two parameters in particular: mesh and conditions making it difficult to model the pressure fluctuation (Hedi et al. 2012). The pressure fluctuation in the internal flow passage of centrifugal pumps is a key factor affecting the stability of the hydraulic system, especially working at conditions of high temperature and pressure. The paper looks to identify areas and challenges of performance of firefighting centrifugal pump with the aim to improve the performance through experiential and simulation. Calculations will be carried out to determine the performance of the pump and will be valeted through simulations.

1.1 The Fire Fighting High-Pressure Centrifugal Pump

A fire-fighting pump is a centrifugal pump which is used for pumping fire-fighting water (Matlakala et al. 2019a). Firefighting pump can be designed for transportable use, like fire fighting vehicle and portable fire pump (Centrifugal Pump Lexicon). Also fighting pump can be designed for stationary (see Figure 1) use like hydrants and sprinkler systems. Transportable fire-fighting pumps can be installed or mounted on fire-fighting vehicles and, they are used as portable firefight (Centrifugal Pump Lexicon).

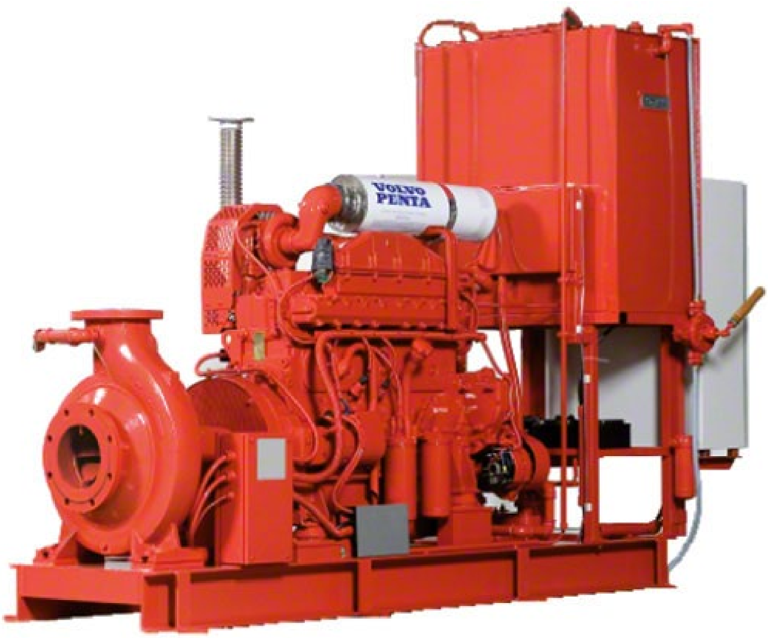

Figure 1:Typical Example of Fire-Fighting Pump (Centrifugal Pump Lexicon)

2. Experimental

The main purpose of the experimental test is to determine the accurate predictions of firefighting high-pressure centrifugal pump. Specifications of the firefighting high pressure centrifugal pump that were used for calculation and simulations are represented in Table 1 (Matlakala et al. 2019b). The specifications were used to calculate the pump speed for design purposes. State of the art CFD solutions consist of the solution of the Reynolds Averaged

Navier Stokes Equations (RANS), for impeller flows as well as for complete pump flows including the impeller – volute interaction. The alternative or supplement to standard RANS turbulence modelling is Large Eddy Simulation (LES). The present numerical model consists of the Navier Stokes equations written in cylindrical coordinates, (r, z, θ), better suited for axisymmetric, rotating geometries than the Cartesian ones. It is assumed that: (a) the impeller flow field has reached steady state conditions; (b) the circumferential component u_θ does not depend on the axial coordinate z.

Table 1. Design Specification of Firefighting High Pressure Centrifugal Pump

Parameters	**Symbol**	**Models**	**Units**
Atmospheric pressure	P_{atm}	101.325	KPa
Rotational Speed	N	1453	Rpm
Mass flow rate	Q	2.1	m^3/sec
Net positive suction head available	$NPSH_{(available)}$	10.02	mm
Net positive suction head required	$NPSH_{(required)}$	9.5	mm
Pump head	H	6	mm
Suction pressure	P_s	58.68	KPa
Discharge pressure	P_d	90	KPa
Suction diameter	D_1	556	mm
Discharge diameter	D_2	345	mm
Efficiency	η	86	%
Impeller outside diameter	d_o	1120	mm
Impeller inner diameter/ hub diameter	d_h	672	mm
Specific Speed	N_s	534	rpm
Number of blades	Z	6	-
Clearance	Δ	1.12	mm
Input Power (Shaft power)	P_s	835.90	kW
Blade outlet angle	β_1	34°	degree
Blade inlet angle	β_1	16°	degree

The fluid pressure enters the impeller/pump, Reynolds number and angled blade rotational speed are calculated through equations 1 and 2.

$$Reynold's\ Number = \frac{Velocity \times Diameter}{Kinematic\ Viscosity\ of\ Water} \tag{1}$$

$$R_e = \frac{\rho R_2 \omega^2}{\mu} \tag{2}$$

$$\omega = 2\times\pi\times1453 = 25.3596\ rad/sec$$

And $\mathrm{R_2}$ − Characteristics Length − 174 mm width of the centrifugal pump

$$\rho = 997\ kg/m^3$$

$$\mu = 8.90\ \times 10^{-4}\ m^2/s$$

$$Re = \frac{997 \times 174 \times 25.3596^2}{8.90 \times 10^{-4}} = 1.23 \times 10^{11}$$

The Reynolds number proves that the flow is a Turbulent Flow entering the impeller.

$$p = \frac{101.325}{997 \times (174 \times 25.3596)^2} = 5.2196 \times 10^{-6} pa$$

The pressure within the pump can be calculated through equation 3.

$$p^* = \frac{g \cdot H}{\frac{1}{2} \cdot \frac{D_2^2}{4} \cdot 4 \cdot \pi^2 \cdot n^2} \tag{3}$$

D_2 Is the impeller inlet and outlet diameter

n – rotational speed

H – Pump head

$$p^* = \frac{9.81 \times 6}{\frac{1}{2} \times \frac{0.345^2}{4} \times 4 \times \pi^2 \times 1453^2} = 4.7466 \times 10^{-5} pa$$

Tangential Velocity

$$u_2 = \pi D_2 n = \pi \times 0.345 \times 1453 = 1574.833\, m/s$$

$$u_1 = \pi D_1 n = \pi \times 0.556 \times 1453 = 2537.992\, m/s$$

Maximum volume flow within the pump casing is calculated using equation 4:

$$\dot{V} = 0.08 \cdot n \cdot D_2^2 \cdot \left(\frac{u_1}{u_2}\right)\left[\left(\frac{g \cdot H_{shut-off}}{0.8 \cdot u_2^2}\right)\left(\frac{n}{1200}\right)\left(\frac{z}{7}\right)\left(\frac{D_1}{D_2}\right)\right] \tag{4}$$

$$V_{max} = 0.08 \times 1453 \times 0.345^3 \times \left(\frac{1574.833}{2537.992}\right)\left[\frac{9.81 \times 6}{0.8 \times 1574.833^2} \times \frac{1453}{1200} \times \frac{14}{7} \times \frac{0.556}{0.345}\right]$$
$$= 0.2128 \times 10^{-3} m/s$$

The pump specific speed is calculated through equation 5:

$$n_q = RPM \frac{\sqrt{\dot{V}}}{H^{0.75}} \tag{5}$$

$$n_q = 1453 \times \frac{\sqrt{0.2128 \times 10^{-3}}}{6^{0.75}} = 5.52889\, m/s$$

Specific speed the fire-fighting centrifugal pump is 5.52889 m/s.

2.1 Analytical

Centrifugal pumps are not usually provided with guiding vanes in front of the propeller wheel. Therefore, total pressure increases. The fluid flow is governed by Euler equation given by equation 6:

$$\Delta p = \rho u_2 c_{2u} \tag{6}$$

where u_2 is outlet blade velocity and c_{2u} is component of outlet absolute velocity c_2 in the direction of blade velocity.

Equation (1) predicts too high power compared to experiments. To add some empirical experience, we used efficiency of diffuser η_{dif} to consider its imperfection to transform kinetic energy into pressure energy. The static pressure obtained by fluid is given by equation 7:

$$\Delta p_{stat} = \Delta p - \rho \frac{c_2^2}{2}(1 - \eta_{dif}) \tag{7}$$

To compare results obtained at different conditions, two dimensionless numbers are used. The flow coefficient φ is a dimensionless quantity used to describe the volume flow rate. It is also referred to as the volume or volume flow coefficient and characterises the flow rate. It is defined as shown in equation 8:

$$\varphi = \frac{Q}{\pi d_2 b_2 u_2} \tag{8}$$

where Q (m^3/s) is volumetric flow rate, d_2 (m) is outlet diameter of the pump wheel and b_2 (m) is outlet width. The head coefficient ψ is a characteristic coefficient derived from the corresponding physical quantity according to the affinity laws and used to characterise the operating behaviour. It characterises the head of the pump:

$$\psi = \frac{\Delta p}{\varrho \frac{u_2^2}{2}} \tag{9}$$

2.2 Numerical
Ansys Fluent software was used to do the numerical analysis. To replicate wheel rotations, we employed two alternative methods: a moving reference frame technique with a stable solver and a dynamic mesh with a transient solver. Instead of the real rotation that is simulated when utilizing the moving reference frame approach, a rotational relative velocity is supplied for volumes of fluid inside the mesh of the wheel.

3. Design of 3D Model
The software that was used to design the impeller, pump and motor is Solidworks Premium 2017. The one-dimensional model is simple and robust, giving accurate performance predictions. It is the basis for comparison of numerical predictions of characteristic lines since it shows good agreement to manufacturer's data and experimental results. It is important that prediction models should include correlations accounting for loss mechanisms, since it was demonstrated that predictions based on the Euler's method and airfoil theory, sometimes produce unrealistic results. The effect of blade exit angle variation on the head and efficiency of a centrifugal pump was investigated using the numerical model. It was concluded that the exit blade angle is influencing mainly at high volume flow rates the pump head and efficiency (Matlakala and Kallon 2019a). A one-dimensional performance analysis has proved to be an effective and important approach on pump design. The pump characteristics depend on geometrical data and on losses in different parts of pump (Matlakala and Kallon 2019b, (Matlakala and Kallon 2020).

3.1. Material Selection
Firefighting centrifugal pumps are mainly used for dewatering processes. Due to this, strong material for the casing should be used primarily because since the impeller rotates at speeds of greater than 730 rpm, the debris or trash such as sand and sludge will impact the pump casing at high velocities, and this might erode the surface of the pump. The most suitable material for the trash centrifugal pump being modelled is cast iron and stainless steel (Trash Pumps Information). This is because the high-velocity fluid cannot rust stainless steel, and cast iron has a tensile strength of 115-700MPa and has other alloys such as silicon, increasing strength (Reliance-Foundry).

3.2. Conceptual Design
The design of the 3D concept of a firefighting centrifugal pump was designed focused on the specific speed of $5.52889\ m/s$. The reliability of a firefighting centrifugal pump casing and impeller is tested using Finite Element Analysis (FEA). The design variables for this pump covered the type of materials used for the centrifugal pump impeller and casing, thickness of the impeller blades, impeller number of blades and direction of the force applied by water on the centrifugal pump impeller blade. For conceptual development for detail design of casing and impeller are represented Figures 2 and 3 respectively.

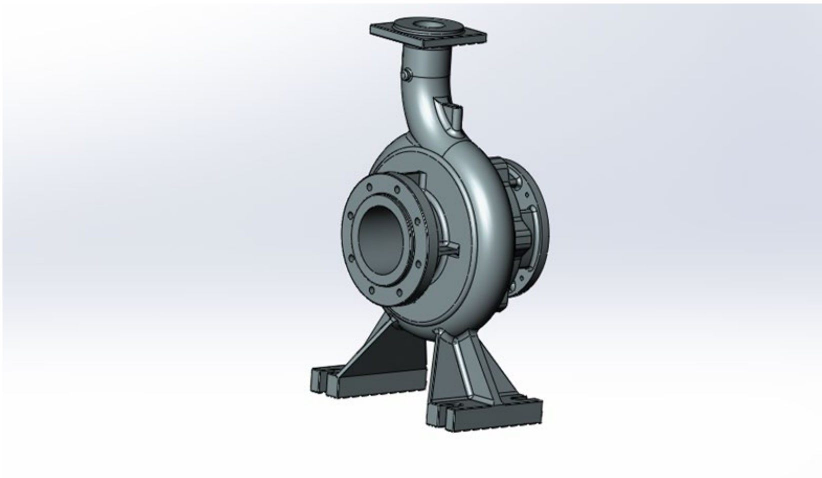

Figure 2. Conceptual development of pump casing

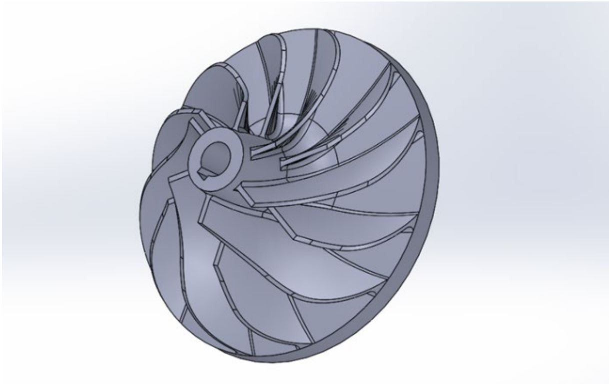

Figure 3. Conceptual development of impeller

4. Simulation

The software that was used for Simulation is ANYS18. Due to the vast applications, it is very important that centrifugal pumps should work efficiently. There have been continuous efforts to improve the performance of centrifugal pumps. The significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. In recent years, a growing availability of computational resources and progress in the accuracy of numerical methods brought Computational Fluid Dynamics (CFD) methods from pure research work into the competitive industrial markets. The advanced CFD commercial software is not suited for a quick assessment of characteristic lines of a series of pumps, due to the detailed impeller blade chamber, thickness and height distribution required for the grid generation (Matlakala et al. 2019a) [2]. Alternatively, numerical models based on one dimensional approach can provide very fast and accurate predictions of the pump performance

4.1. Simulation of the Firefighting Impeller

Simulation of the impeller using ANSYS CFX and Fluent. It includes Mesh, file, physics, and solution reports. The simulations were carried out using ANSYS software. In this project, the main aim was to study the effects of changing the blade number and suction diameter on the velocity and pressure of the centrifugal pump. The study parameter used were as follows: flowrate is 5 m^3/s; gravitational acceleration is 9.81 m^2/s; slurry is used as the fluid with 25 degrees as reference temperature and a density of 1380 kg/m^3. The specific heat is 1006.43 J/kgK, with the molecular weight being 28.966 kg/kmol. The pressure was set at zero.

4.2. Effects of changing the suction diameter

When investigating the effects of changing the suction diameter, the following diameter was used: 75 mm, 85 mm, and 95 mm. An ANSYS (CFX) method was being followed, including the geometry, mesh, setup, solution, and results. The geometry was developed using Autodesk Inventor software. Mesh note and elements are presented in Table 2.

Table 2. Mesh

Number of nodes	218559
Number of elements	1075193

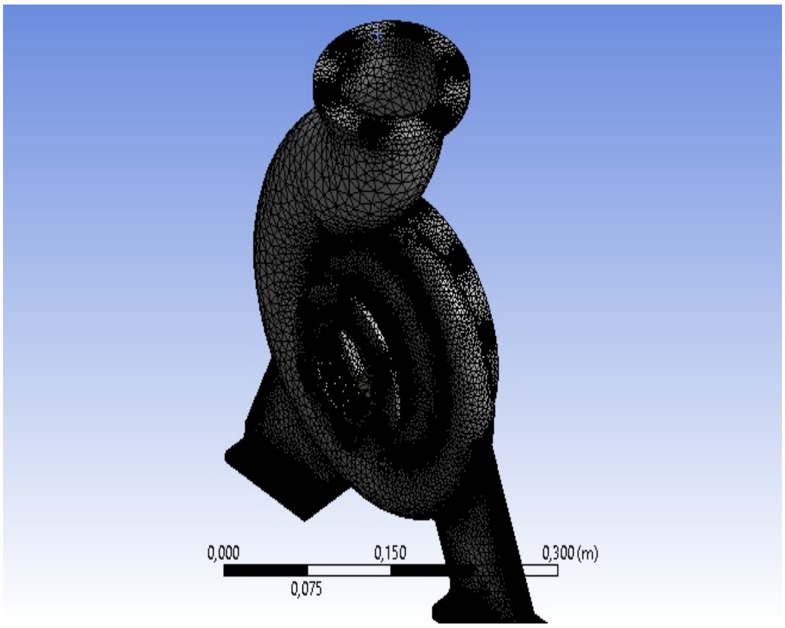

Figure 4. Mesh

Table 2 and Figure 4 show the mesh generated whereby the number of nodes and elements are many, which will help get more accurate results. Figure 5 shows the setup used wherein the boundary conditions were applied. The volumetric flow rate at the inlet and the pressure outlet where the water comes out.

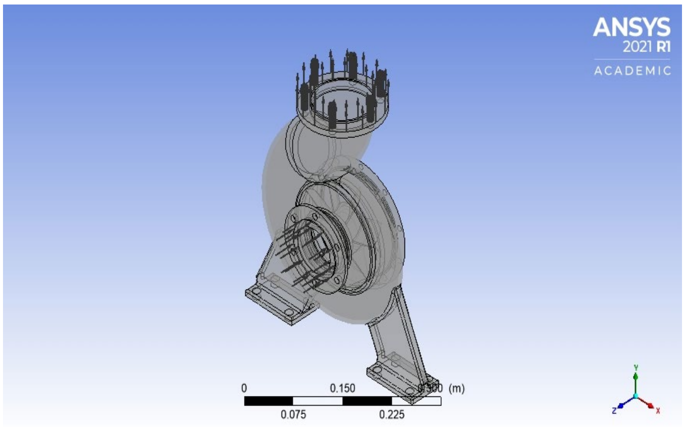

Figure 5.Setup of the Fire-Fighting Centrifugal Pump

From the setting up of the model for the simulation, the results were obtained are represented in Figure 6. The orange colour represents higher pressure and the blue colour represent the low pressure.

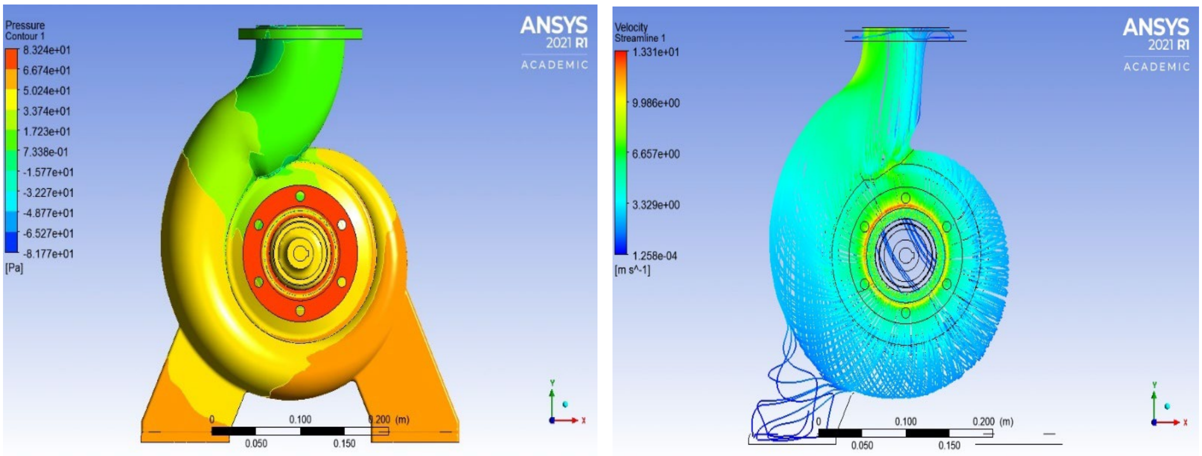

Figure 6. Pressure and Velocity for Simulation Contour for 75 mm

Figure 6 shows the velocity streamlines and the total pressure when using a suction diameter of 75 mm whereby the fluid, which is slurry, flows through the suction and then gets hit up by the impeller as it is rotating, and then the slurry moves up with a certain pressure. The red colour shows where the maximum velocity occurs.

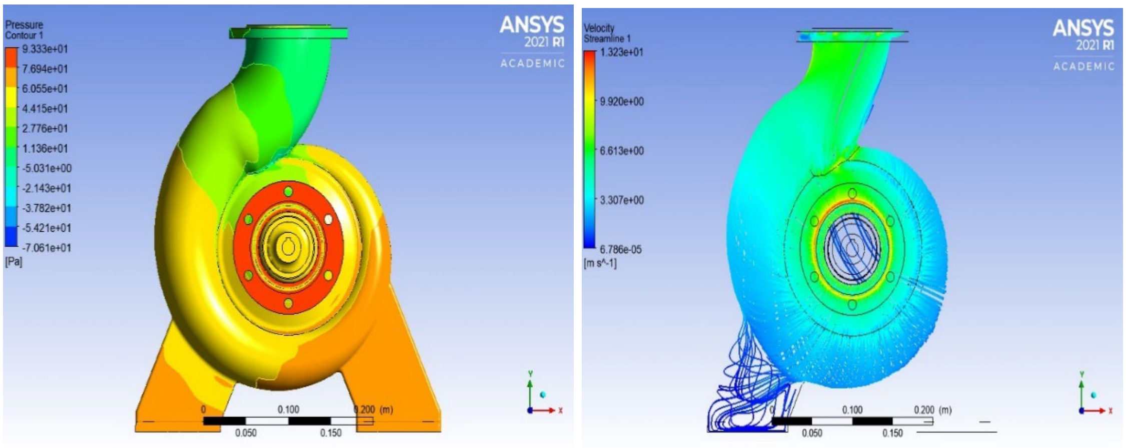

Figure 7. Counter Pressure Total and Velocity for 85 mm

Figure 7 shows the velocity streamlines and the total pressure when using a suction diameter of 85 mm whereby the fluid, which is slurry, flows through the suction and then gets hit up by the impeller as it is rotating, and then the slurry moves up with a certain pressure. The red colour shows where the maximum velocity occurs.

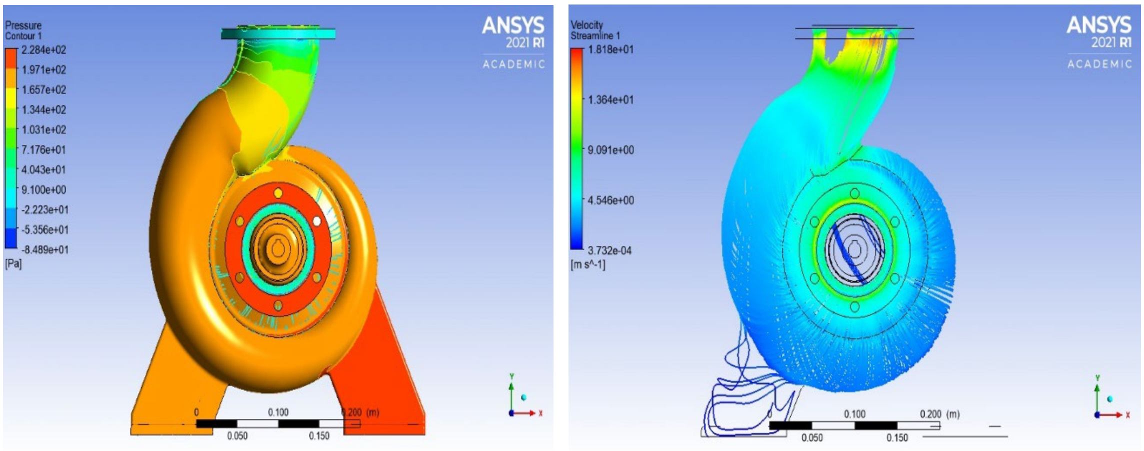

Figure 8. Pressure and Total pressure for 95 mm

The velocity streamlines and the total pressure when using a suction diameter of 95 mm whereby the fluid, which is slurry, flows through the suction and then gets hit up by the impeller as it is rotating, and then the slurry moves up with a certain pressure. Figure 8 shows the pressure and total pressure, which are modelled using ANSYS (CFX). Depicted in the figures is the trash centrifugal pump with different colours, and red represents the maximum while blue represents the minimum. The maximum total pressure is 236.5 Pa, while the maximum contour pressure is 228.4 Pa. The minimum total pressure is -84.89 Pa, while the minimum contour pressure is -84.89 Pa. Figure 10 represent the velocity of the fluid in the pump. It shows that the velocity is higher as leaving the pump and lower as it enters the casing. As increasing the suction, the pump efficiency increases as shown in Figure 9. Table 3 shows the result of the simulation for the suction diameter.

Table 3. Simulation results for suction diameter

Suction Diameter (mm)	Maximum Pressure (Pa)	Maximum Velocity (m/s)	Flow Rate (m^3/s)	Head (m)	Slurry Power (W)	Efficiency (%)

75	90.83	13.23	0.07	1.107	1808.16	39.84
85	100.9	13.33	0.10	1.232	2528.03	55.71
95	236.5	18.18	0.12	1.408	3380.46	74.50

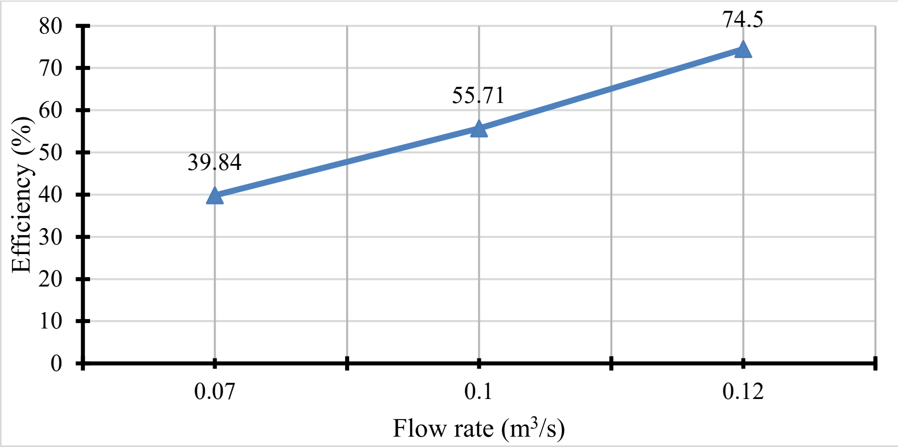

Figure 9. Efficiency Vs Flow Rate graph

5. Discussion of Results

Design and simulation of centrifugal pump was done in Autodesk and ANSYS respectively. The specification used to simulate and perform calculations were as per previous reports (Matlakala et al. 2019a) and are listed in Table 1. The Navier-Stokes Equations are used to find the type of flow at pump entry which is a turbulent flow as Reynold's calculations show. The pressure at pump exit with values was less than normal (Matlakala et al. 2019a, Matlakala and Kallon 2021) which was very low. Theoretically a large pressure is needed to control a low volume flow rate. The tangential velocity of the impeller was determined where it showed the impeller rotating at certain speed values. If the tangential velocity was close to zero it would affect the suction and discharge diameters and rotational speed of the impeller. The pump specific speed was determined to be 5.53 m/s which means the pump is running at a high speed. High speed can result to a lot of noise when pump is operational, shaft failure and cavitation. Method of reducing the speed of the centrifugal pump must be specified.

Simulations were done for the impeller where it cracked as the calculated speed was applied. Practically impeller work on the calculated speed meaning it will fail and cause cavitation on the housing, flow time set was 5s. Due to differences between the calculation and the measurement, it appears that the spaces in front of and behind the wheel, where the fluid can escape through the gap between the rotor and the stator and thus aid the flow in the pump at higher backpressures, should be included in the calculation.

6. Conclusion and Recommendations

Centrifugal Pumps with this impeller will fail due to the high speed of the pump and it will result to shaft failure, cavitation and make a lot of noise when in use. To avoid cavitation at high flow rates, the impeller blades must be redesigned, and the stator blades must be modified to improve the transfer of the kinetic energy of the water flow to pressure energy.

Reference

Anderson Pump and Process. Water Pumps and Their Ability to Make Our Daily Lives Less Stressful. Available: https://andersonpumpprocess.wordpress.com. Accessed June 09, 2021.

Centrifugal Pump Lexicon. Available: https://www.ksb.com/centrifugal-pump-lexicon/fire-fighting-pump/191962/. Accessed February 25, 2022.

Fatsis, A. A Novel Methodology to Predict Centrifugal Pump Characteristics through Navier-Stokes Exact Solutions, International Journal of Engineering Research & Technology (IJERT), vol. 35, pp. 2278-0181, 2015.

Hall, N. Navier-Stokes Equations, Nasa.gov, 2015. Available: https://www.grc.nasa.gov/www/k-12/airplane/nseqs.html. Accessed February 25, 2022.

Hedi, M.L., Hatem, K., and Ridha, Z. Numerical Analysis of the Flow Through in Centrifugal Pumps," International Journal oof Thermal Technologies, vol. 2, no. 4, pp. 216 - 221, 2012.

Matlakala, M.E., Kallon, D.V.V, Simelane, S.P., and Mashinini, P.M. Impact of Design Parameters on the Performance of Centrifugal Pumps, Procedia Manufacturing, vol. 35, pp. 197-209, 2019a.

Matlakala, M.E., Kallon, D.V.V., Mogapi, K.E., Mabelane, I.M., and Makgopa, D.M. Influence of Impeller Diameter on the performance of the Centrifugal pumps, IOP Conference Series: Materials Science and Engineering, Volume 655, Conference of the South African Advanced Materials Initiative (CoSAAMI 2019), pp. 1-10, October 22-25th 2019, Vanderbijlpark, South Africa, 2019b.

Matlakala, M.E., and Kallon, D.V.V. Influence of impeller Blade Count on the Performance of Centrifugal Pump, 2019a.

Matlakala, M.E., and Kallon, D.V.V. Effect of Discharge Diameter on Centrifugal Pump Performance, pp. 721-730, 2019b.

Matlakala, M.E., Kallon, D.V.V., Simelane, S.P., and Mashinini, P.M. A Computational Model for the Efficiency of Centrifugal Pumps. Dissertation Submitted to the University of Johannesburg, 2020.

Matlakala, M.E., and Kallon, D.V.V. Systems Dynamics Modelling of the Water Supply Problem in the Limpopo Province of South Africa, Proceedings of International Conference on Industrial Engineering and Operations Management, pp. 1589 -1597, 2021.

Reliance-Foundry. Introduction to Cast Iron: History, Types, Properties, and Uses. Available: https://www.reliance-foundry.com/blog/cast-iron. 2021. Accessed June 08, 2021.

Trash Pumps Information. Available: https://www.globalspec.com/learnmore/flow_control_flow_transfer/pumps/trash_pumps Accessed May 17, 2021.

Biographies

Mr. Motsi Ephrey Matlakala is a South African holder of a M-Tech in Mechanical Engineering from the University of Johannesburg. Mr. Matlakala is currently working as a Mechanical Engineering Graduate at Rand Water Zuikerbourch Pumping Station since June 2017. During his Master studies, He published eighteen (18) papers and was also selected as a reviewer of paper in four (4) conferences, locally and internationally. Mr. Matlakala is preparing to enrolling PhD in Mechanical Engineering with the University of Johannesburg. Mr. Matlakala is a member of South African Institute of Mechanical Engineering SAIMECH and Professional Engineering Technologist with ECSA. Mr Matlakala's primary research areas are System Analysis and Dynamics, Optimization, Computational Fluid Dynamics, Finate Element Analysis and Water Research.

Dr Daramy Vandi Von Kallon is a Sierra Leonean holder of a PhD degree obtained from the University of Cape Town (UCT) in 2013. He holds a year-long experience as a Postdoctoral researcher at UCT. At the start of 2014 Dr Kallon was formally employed by the Centre for Minerals Research (CMR) at UCT as a Scientific Officer. In May 2014 Dr Kallon transferred to the University of Johannesburg as a full-time Lecturer and later a Senior Lecturer in the Department of Mechanical and Industrial Engineering Technology (DMIET). Dr Kallon has more than twelve (12) years of experience in research and six (6) years of teaching at University level, with industry-based collaborations. He is widely published, has supervised from Masters to Postdoctoral and has graduated seven (7) Masters Candidates. Dr. Kallon's primary research areas are Acoustics Technologies, Mathematical Analysis and Optimization, Vibration Analysis, Water Research and Engineering Education.