Design and Analysis of Centrifugal Pump Impeller for Performance Enhancement

*Farah Elida SELAMAT-1**
Wan Hariz Iskandar WAN IZHAN-1
Department of Mechanical Engineering, College of Engineering, Universiti Tenaga Nasional, Malaysia

Bakhtiar Saffuan BAHARUDIN – 2
3Project Oilfield & Operations Services, Uzma Engineering Sdn Bhd, Malaysia

**storm0704@yahoo.com*

ABSTRACT

Centrifugal pump usage has increased over the past year due to its importance and efficiency. Its function is to transport liquid from one place to another using energy applied to the pump. This paper revolves around the idea of design and analysis of centrifugal pump for performance enhancement within the pump specifications. Design and simulation were conducted using ANSYS CFX, using the Navier-Stokes equation. Shear Stress Transport (SST) was chosen for turbulence model for this project. From the simulation results, it can be observed that as the rotation speed of the impeller increases, the pressure within the impeller increases. The pressure increases gradually from impeller inlet to impeller outlet. It can also be observed that the efficiency of the impeller increases as the rotation speed increases. As a result, the performance of the impeller increases as the efficiency increases. Performance of the impellers was compared based on inlet and outlet power, impeller efficiency, pressure distribution, and static head pressure produced.

Keywords: *centrifugal pump, impeller, design, ANSYS CFX, fluid dynamics*.

INTRODUCTION

ISSN 1823- 5514, eISSN 2550-164X

Received for review: 2017-04-05
Accepted for publication: 2017-05-12
Published: 2018-01-31

Mechanical pump is a device used to transport fluid from one point to another by mechanical action [1]. It is commonly used in industries, agricultures, and domestic applications. The basic types of pumps are divided into two which are positive displacement pump and centrifugal pump [2]. The operation of pumps is driven by a mechanism which is supplied by many energy sources such as manual operation electricity, engines and wind power. This energy is used to perform the mechanical work to move fluids inside the pump by rotating the impeller inside the pump. Energy is added continuously in pumps in order to increase the fluid velocity within the machine [3].

Centrifugal pumps are the sub-class of dynamic axisymmetric work-absorbing turbomachinery [4]. The fluids that are being transported by a centrifugal pump use the hydrodynamic energy which is converted from the rotational kinetic energy of the fluid flow. The fluids are transported from a region of low pressure to a higher pressure region via an impeller. Centrifugal pumps produce negative pressure at the pressure of the inlet so the atmospheric pressure pushes the fluid towards the pump [5].

Common problems faced by a centrifugal pump are cavitation, excess vibration, excess noise and heat, leakage, mechanical seal failure, components failure, and many more [6]. Symptoms of cavitation are rattling noise and high level of vibration occurring in the centrifugal pump [7]. Sources of the problems can be seen from the characteristics of flow inside the centrifugal pump. When fluid moves through the impeller, the pressure drop will affect the boiling temperature of the fluid [8]. Therefore, the lower the pressure of fluid, the lower the boiling point of the fluid [9]. Cavitation occurs if the pressure of fluid falls below the vapour pressure [10].

This will cause the vibration and noise of the centrifugal pump. Indirectly, the problems will disintegrate the impeller and create small holes [7]. The head pressure output of the centrifugal pump will also decrease. The pressure drop can be caused by the loss of energy in the fluid which moves from the pump suction line to the rotating impeller. Other considerations that contribute to pressure drop include impeller angles, impeller vanes, and entrance angle which are related to the velocity of the fluid [7].

Air entrainment is another problem faced by the centrifugal pump. It happens when vapour bubble is already present before the fluid enters the pump. The presence of turbulent flow at the suction line is the cause of air entrainment. Turbulent flow exists due to inappropriate piping condition at the suction line [7]. Other than pump mechanism problems, a simple problem such as mechanical seal failure can massively contribute to pump shutdown [11]. It fails due to wear and tear and inadequate operations such as poor selection and installation error.

This situation happens at Uzma Engineering Sdn. Bhd., an international oil and gas service company providing a variety of services for the upstream sector to the downstream sector for facilities and plant construction, operations, and maintenance. This is the major problem to the company as the centrifugal pump is expensive and having pumps in downtime could lead to financial deficiency.

A centrifugal pump failure can be identified by knowing when it happens, whether during start-up of the pump, after 2-3 weeks of operation or 3-4 months of operation. These symptoms are caused by the operation and design of pump that failed to meet the requirements, parts failure due to wear and tear, or the pump is damaged from the fluid being transported through the pump. The only way to solve these problems is by doing maintenance work in order to keep up with the operation schedule. The maintenance repair could take weeks depending on the level of damage of that particular centrifugal pump.

Another proposition to overcome these problems is to design a new centrifugal pump impeller. With the new impeller design prior to the given pump specification, the pump can operate at high efficiency and can operate as the operation phase, thus reducing the cost of maintenance.

Hence, the objective of this study is to provide an impeller design with better flow and pressure distribution by studying the effect of RPM on efficiency and static head. The design is done by performing CFD analysis on the impeller with various rotating speeds.

METHODS

Research Setting

Uzma Engineering Sdn Bhd is a service company, which provide its services to upstream sector. In this research, the location of the data collected is at one of the oil and gas platforms in offshore Terengganu, which has the highest number of centrifugal pump malfunctions.

Design Tools

The design tool used for this study is the ANSYS software. ANSYS is an engineering analysis software to predict a wide range of analysis including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods, and heat transfer. It is one of the well-established and convenient analysis tools that has been successfully applied to solve engineering problems. This advance technology allows the calculation of the simulations faster, more accurate and more efficient [12].

In this study, the model construction and flow simulation were done in the ANSYS Workbench and ANSYS CFX, respectively. The flow simulations were done within 1000 iterations and all of the simulations were converged within 1000 iterations. The convergence criterion chosen for this study is criterion-type 1, which is the relative convergence criterion. Grid independence test was not conducted due to the time constraint to complete this study. Instead, the mesh was auto generated using the ANSYS Meshing platform in the ANSYS Workbench which produced the most appropriate mesh for a specific analysis with a single mouse click. Fine tuning (choosing regions to use finer mesh) option is also available for the authors to obtain optimum mesh size.

Pump Specifications

Pump specifications need to be studied before the impeller can be designed because the performance of the radial flow pump is hugely affected by the geometry of the impeller. Table 1 below shows the pump specifications for this project.

Table 1 :- Description of the pump parameters

Parameters	Dimension
Pump Head	24 m
Flowrate	114 m^3/h = 0.032 m^3/s = 500 US GPM
Motor Speeds	1150, 1750, 2500 rpm
Shaft Torque	80 Nm
Outlet Mass Flow Rate	35 kg/s
Number of Blades	8

Model Construction

The three-dimensional model of the impeller was created using the ANSYS software. The fluid volume was split into a fixed wall, rotating fluid volume, and inlet and outlet fluid. The flow was assumed to be fully developed as it left the inlet and outlet ducts. The impeller was constructed with eight blades. Figure 1 below shows the isometric view of the impeller while Figure 2 shows the front view of the impeller.

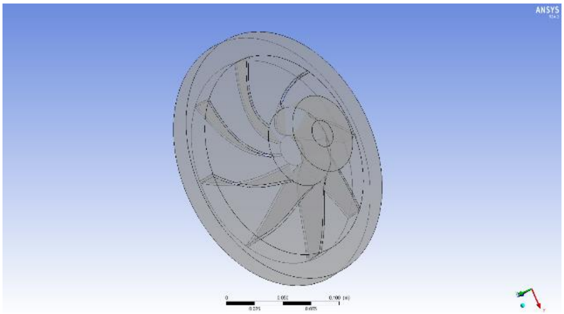

Fig. 1. Isometric View of Impeller

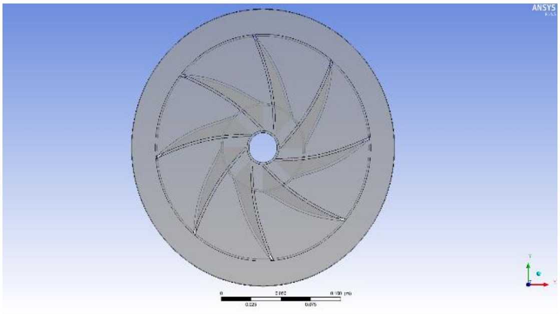

Fig. 2. Front View of Impeller

Meshing

In analysing the fluid flow of the impeller, a mesh is required to split the flow domain into smaller parts. Thus, the type of mesh used is a combination of triangle and quadrilateral meshes. The triangle mesh is for accuracy purpose while the quadrilateral mesh is for reducing simulation render time. With the triangle mesh, the resolutions are better, thus, increasing the accuracy of the simulation. Figure 3 shows the isometric view of the meshed impeller. There are 75223 nodes, 108676 triangle meshes, and 17083 quadrilateral meshes. Figure 4 shows a denser mesh at the leading edge of the impeller.

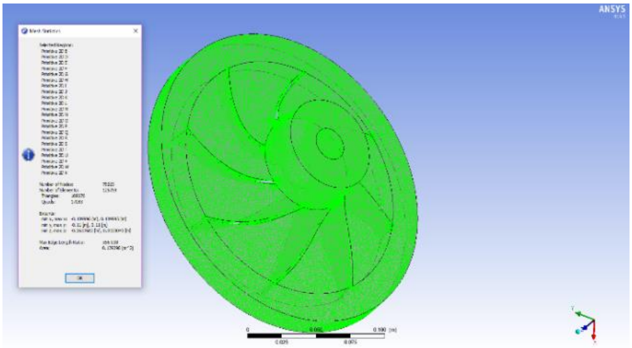

Fig. 3. Isometric View of Impeller Mesh

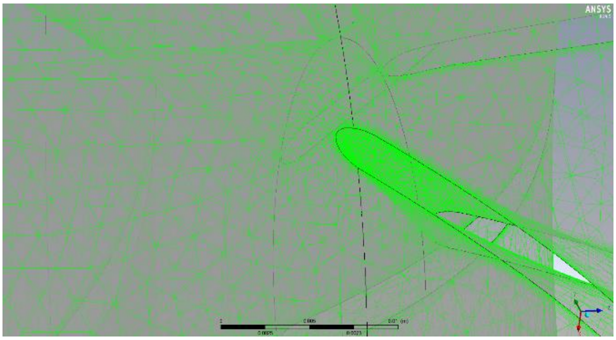

Fig. 4. Denser Meshing at Leading Edge

Boundary Condition

At the eye of the impeller, the water flow outwards centrifugally through the blades of the impeller. The mass flow rate was specified to calculate the domain for each computation. The global rotating reference frame defines the hub and back of the impeller as the boundary conditions. The inlet velocity of the impeller is set to be the inlet boundary condition (Figure 5), while the outlet mass flow rate is defined as the outlet boundary condition (Figure 6). The outlet boundary condition defines the exit of the impeller. The inlet pressure is set to be 0 Pa.

In this study, the radial flow pump impeller has various operating speeds of 1150, 1750, and 2500 rpm to be investigated. The rotation speeds were chosen based on the operation specification of the centrifugal pump on the platform. There are settings at the software for changing the RPM value before running the simulation. At the outlet, the mass flow rate is set to be 35 kg/s. The working fluid in the impeller is water with 1000 kg/m^3 density at 25°C with Shear Stress Transport (SST) as the turbulence model. When the results have converged, the numerical calculation is done and the flow within the centrifugal flow pump is obtained.

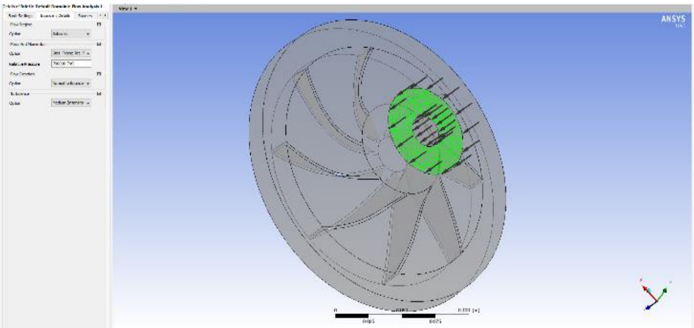

Fig. 5. Isometric View of Inlet Boundary Condition

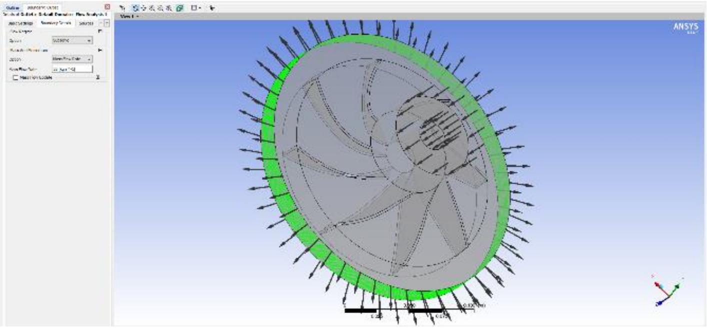

Fig. 6. Isometric View of Outlet Boundary Condition

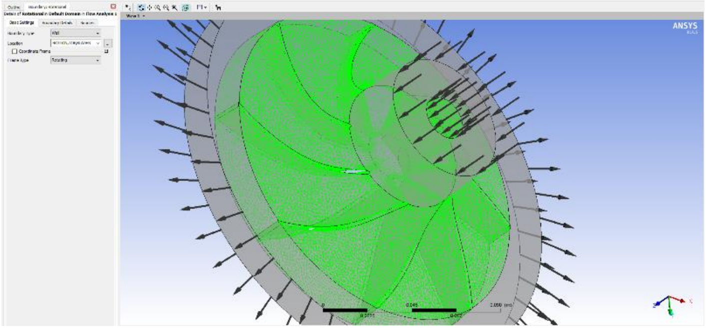

Fig. 7. Isometric View of Rotational Boundary Condition

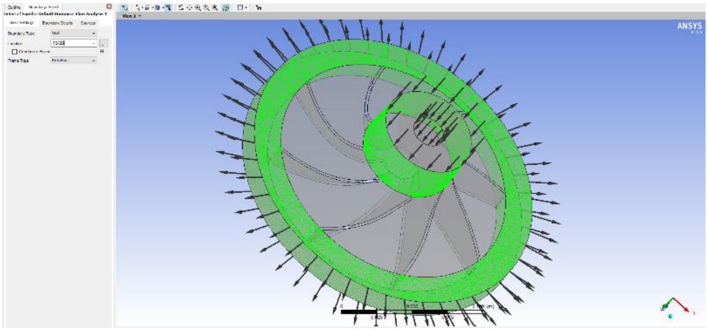

Fig. 8. Isometric View of Fixed Boundary Condition

Figure 5, 6, 7 and 8 show the boundary conditions for inlet, outlet, rotational and fixed respectively. These boundary conditions are necessary in order to proceed with the flow simulation in ANSYS CFX.

RESULTS

The impeller geometry was constructed in the ANSYS Workbench and the flow simulation was carried out using the ANSYS CFX. To initialise and run the simulation, a hybrid initialisation was used to obtain the results. The results of the simulations were in terms of pressure contour. After the analysis was carried out, the results were obtained from the software once the simulation has converged. The simulation iterations were set in 1000 times and all of the simulation results were converged within the range.

For the flow simulation, various rotating speeds were selected to be the variable factors in this project. The aim is to come out with an impeller design which has a gradually increasing pressure distribution in order to be chosen as the most efficient impeller. This study focuses on the pressure distribution of the centrifugal pump impeller by varying the rotation speed at 1150 rpm, 1750 rpm, and 2500 rpm.

The results of the simulations were obtained from the CFD software and compared among the three impellers. A normal and steady pressure distribution is important to show the impeller is in good condition when the pump is running with the impeller at the given rotation speed. Cavitation will occur if the pressure is too low and this will damage the impeller, thus, reducing its life cycle. The input and output powers were calculated in order to obtain the impeller efficiency. The following results were taken in varying axis and cross sections.

Pressure Distribution of Centrifugal Pump Impeller A (1150RPM)

Figure 9 and 10 show the overall pressure distribution of the impeller when the speed of rotation was set at 1150 rpm.

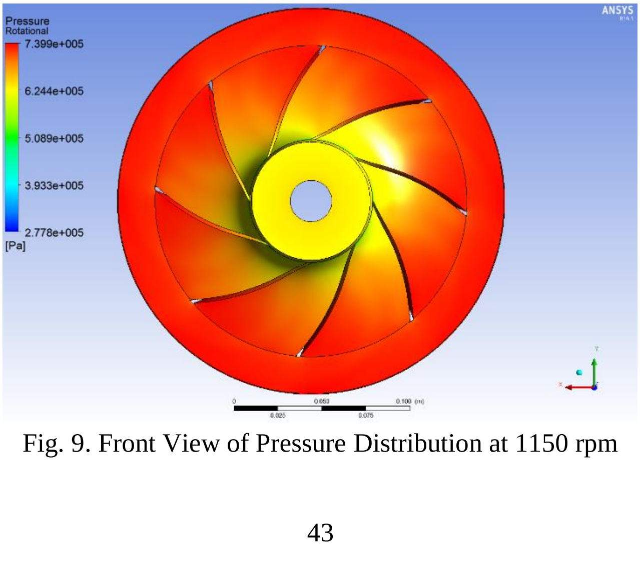

Fig. 9. Front View of Pressure Distribution at 1150 rpm

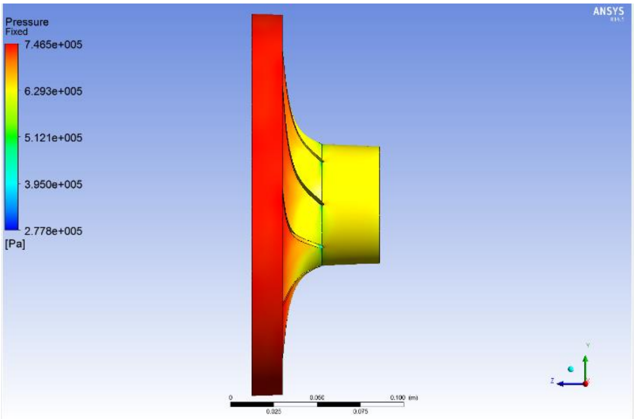

Fig. 10. Side View of Pressure Distribution at 1150 rpm

The simulation results are shown at the left side of Figure 9 and 10, where the impeller has a maximum pressure of 740 kPa and a minimum pressure of 270 kPa. The pressure at the inlet of the impeller is 624 kPa. The outlet of the impeller has a pressure of 740 kPa.

Pressure Distribution of Centrifugal Pump Impeller B (1750 RPM)

Figure 11 and 12 show the overall pressure distribution of the impeller when the speed of rotation was set at 1750 rpm.

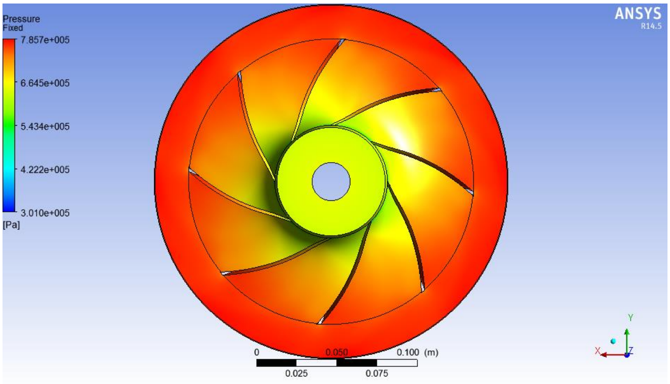

Fig. 11. Front View of Pressure Distribution at 1750 rpm

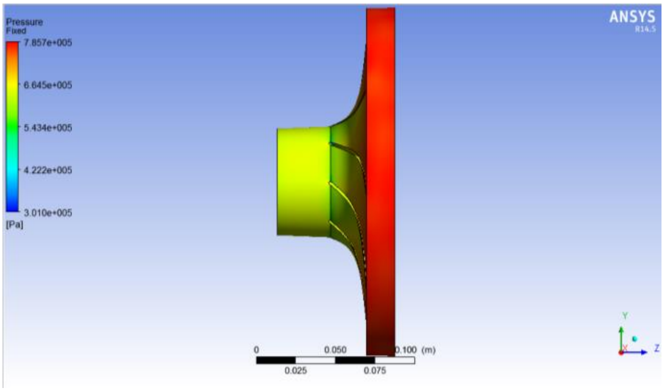

Fig. 12. Side View of Pressure Distribution at 1750 rpm

From the observation of the above figures, the impeller has a maximum pressure of 785 kPa and a minimum pressure of 310 kPa. The pressure at the inlet of the impeller is 543 kPa. The outlet of the impeller has a pressure of 785 kPa.

Pressure Distribution of Centrifugal Pump Impeller C (2500 RPM)
Figure 13 and 14 show the overall pressure distribution of the impeller when the speed of rotation was set at 2500 rpm.

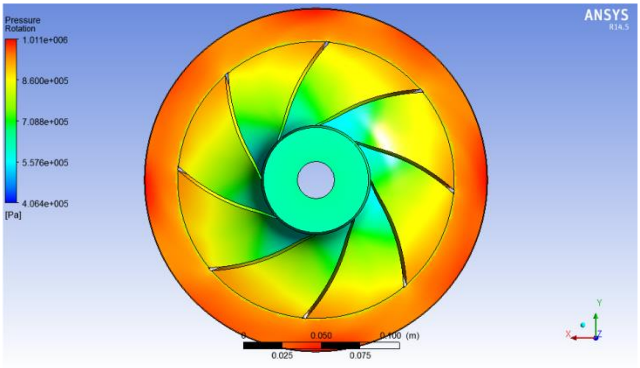

Fig. 13. Front View of Pressure Distribution at 2500 rpm

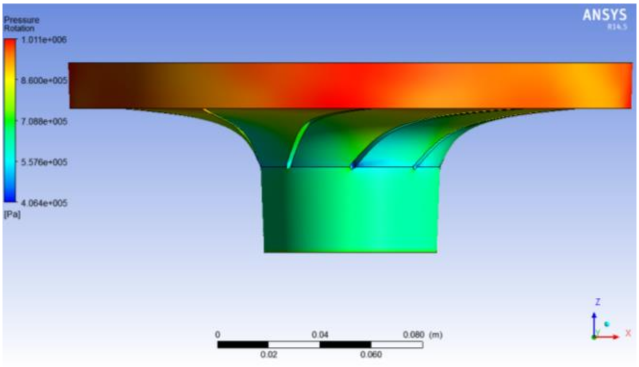

Fig. 14. Side View of Pressure Distribution at 2500 rpm

From the observation of Figure 11 and 12, at 2500 rpm, the impeller has a maximum pressure of 1011 kPa and a minimum pressure of 406 kPa. The pressure at the inlet of the impeller is 557 kPa. The outlet of the impeller has a pressure of 1011 kPa.

Pressure Distribution Evaluation

Table 2 below indicates the minimum and maximum pressure values in kilo Pascal (kPa) for each impeller.

Table 2: - Pressure properties of the 3 impellers (A, B, C)

Pressure Properties	Pressure (kPa)		
	Impeller A (1150 rpm)	Impeller B (1750 rpm)	Impeller C (2500 rpm)
Minimum	270	310	406
Maximum	740	785	1011

Impeller A operates at 1150 rpm, has a minimum pressure of 270 kPa, and a maximum pressure of 740 kPa. Whereas Impeller B operates at 1750 rpm, and the minimum pressure shown is 310 kPa while the maximum pressure is 785 kPa. Lastly, Impeller C has the highest pressure among the three impellers with 1011 kPa and the minimum pressure for Impeller C is 406 kPa with speed rotation of 2500 rpm.

Table 3: - Pressure at inlet and outlet regions

Pressure Region	Pressure (kPa)		
	Impeller A (1150 rpm)	Impeller B (1750 rpm)	Impeller C (2500 rpm)
Inlet Pressure	624	543	557
Outlet Pressure	740	785	1011

Table 3 above shows the pressure in each of the impeller at different regions. Impeller A has an inlet pressure of 624 kPa and outlet pressure of 740 kPa. For Impeller B, the pressure at the inlet is 543 kPa and the pressure at the outlet is 785 kPa. Lastly, for Impeller C the pressure at the inlet is 557 kPa and the pressure at the outlet is 1011kPa. It is shown in Table 2 and 3 that the outlet pressure value of each impeller has the same value as the maximum pressure.

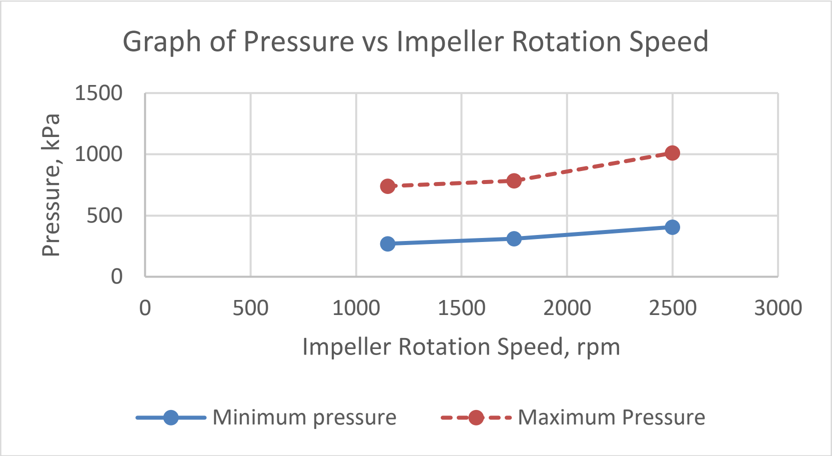

Fig. 15. Graph of Pressure vs Impeller Rotation Speed

Figure 15 shows the relationship of pressure with impeller rotation speed. As the rotation speed of impeller increases, the minimum and maximum pressures also increase.

Impeller Power and Efficiency Calculation

With the pressure values obtained from the simulation results, the efficiency and static head of the respective centrifugal pump impeller can be calculated. The input and output power are calculated in order to obtain the impeller efficiency. The impeller with highest efficiency and best pressure distribution will be employed in the centrifugal pump.

$$\text{Inlet power} = \frac{2\pi NT}{60\times1000} \quad (1)$$

Where,

N – Rotation speed
T – Shaft torque

Outlet power $= \dfrac{(P_0 - P_1)Q}{1000}$ (2)

Where,

P_0 – Pressure outlet
P_1 – Pressure inlet
Q – Flow rate

Efficiency $= \dfrac{Outlet\ Power}{Inlet\ Power} \times 100$ (3)

Head $= \dfrac{P_0 - P_1}{\rho g}$ (4)

Where,

P_0 – Pressure outlet
P_1 – Pressure inlet
g – Gravitational force
ρ – Water density

Sample Calculation (Impeller A at 1150 rpm)

- Inlet power $= \dfrac{2\pi NT}{60 \times 1000}$

  $= \dfrac{2\pi(1150)(80)}{60 \times 1000} = 9.63$ kW

- Outlet power $= \dfrac{(P_0 - P_1)Q}{1000}$

  $= \dfrac{(740 \times 10^3 - 624 \times 10^3)(0.032)}{1000} = 3.71$ kW

- Efficiency $= \dfrac{Outlet\ Power}{Inlet\ Power} \times 100$

  $= \dfrac{3.71}{9.63} \times 100 = 38.55\%$

- Static Head $= \dfrac{P_0 - P_1}{\rho g} = \dfrac{740 - 624}{1000\ x\ 9.81} = 11.72$ m

Using the same equations as shown above, the inlet and outlet power, efficiency and static head values for Impeller B at 1750 rpm and Impeller C at 2500 rpm is calculated.

Summary of results of each impeller

Table 4 below shows the power at the inlet and outlet of the impellers, and the overall efficiency of the impellers. The efficiency of the impeller can be calculated based on both powers. Impeller A, which operates at 1150 rpm, has an inlet power of 9.63 kW and outlet power of 3.71 kW. With both inlet and outlet powers, the efficiency of Impeller A is 38.55%. On top of that, Impeller B which rotates at 1750 rpm, has an inlet and outlet powers of 14.66 kW and 7.74 kW, respectively. The efficiency of Impeller B is 52.80%. Lastly, Impeller C has 20.94 kW of inlet power and 14.53 kW of outlet power at the rotation speed of 2500 rpm. The efficiency of Impeller C is 69.39%.

Table 4: – Efficiency values for each impeller

Impeller	Power (kW)		Efficiency (%)
	Inlet	Outlet	
Impeller A (1150 rpm)	9.63	3.71	38.55
Impeller B (1750 rpm)	14.66	7.74	52.80
Impeller C (2500 rpm)	20.94	14.53	69.39

The following table (Table 5) shows the static head produced by each of the impeller at various rotation speeds. At 2500 rpm, Impeller C has the highest static head produced which is 46.28m, followed by Impeller B at 1750 rpm, with 24.67m of static head. Impeller A has the lowest static head of 11.72m at the speed of rotation of 1150 rpm.

Table 5: - Head produced by each impeller at various rotation speed

Impeller	Static Head (m)
Impeller A (1150 rpm)	11.72
Impeller B (1750 rpm)	24.67
Impeller C (2500 rpm)	46.28

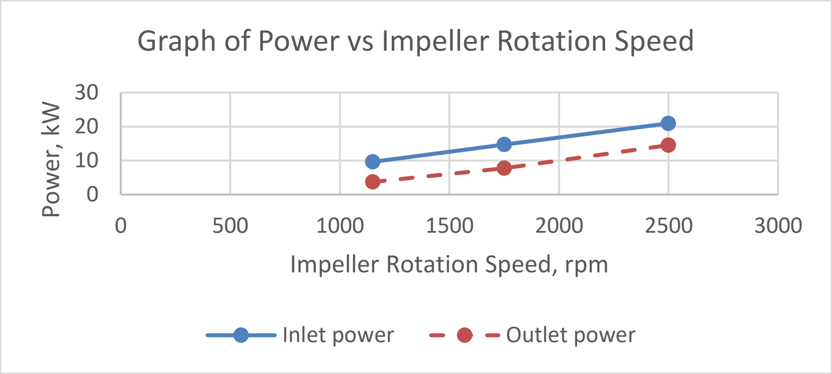

Fig. 16. Graph of Power vs Impeller Rotation Speed

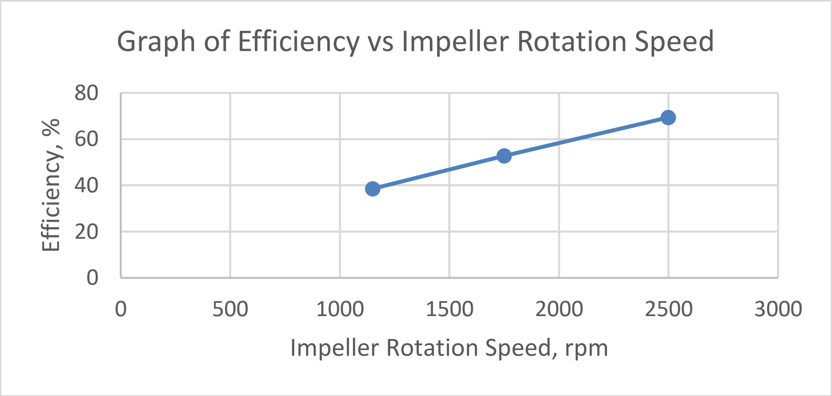

Fig. 17. Graph of Efficiency vs Impeller Rotation Speed

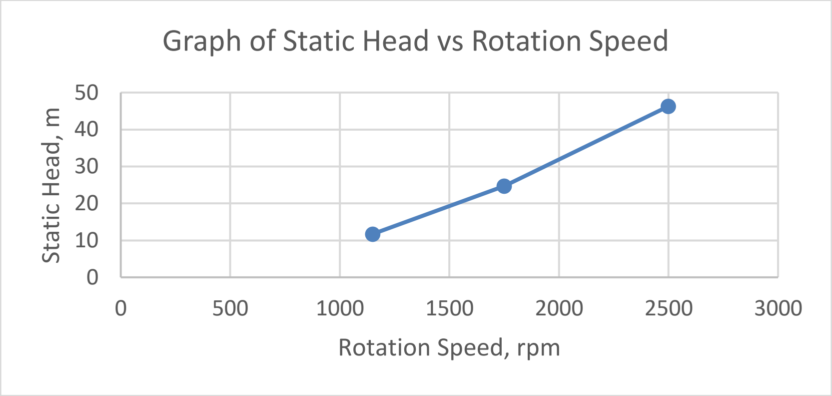

Fig. 18. Graph of Rotation Speed vs Static Head

Based on Figure 16, as the impeller rotation speed increases, the power of input and output increase linearly. The maximum outlet power recorded is 14.53 kW by Impeller C and minimum inlet power recorded is 9.63 kW by Impeller A.

Figure 17 shows that as the impeller's rotation speed increases, the efficiency of the impeller increases as well. The highest efficiency is recorded by Impeller C with 69.39%, whereas the lowest efficiency is recorded by Impeller A with 38.55%.

Figure 18 shows the relationship of rotation speed with static head. It is clear that as the rotation speed increases, the static head also increases. The highest static head recorded is 46.28m by Impeller C with rotation speed of 2500 rpm and the lowest static head is produced by Impeller A with 11.72 m at 1150 rpm.

CONCLUSION

To conclude, the design of the centrifugal pump impeller is set at three different rotation speeds that have been determined. Based on the simulations, a radial centrifugal pump impeller was designed and analysed using CFD software and the pressure contours were obtained. The results of the flow field simulation are presented in terms of pressure distribution of the impeller. At the eye of the impeller, the low pressure region is observed. The pressure contours show a continuous pressure rising from the eye of the impeller to the edge of the impeller due to the dynamic head developed by the rotating pump impeller.

According to the results, the impellers undergone simulations with various rotation speeds, noting that Impeller C, which operates at 2500 rpm, has the best pressure distribution and the highest efficiency at 69.39%. This is due to the fact that Impeller C has gradual pressure distribution from the inlet to the outlet than other impellers. It can be concluded that the performance of the impeller increases as the rotation speed of the impeller increases. By increasing rotation speed, the efficiency of the impeller will also increase.

The design and analysis methods are useful to generate performance and flow predictions. Therefore, the design can be optimised to reduce energy consumption, increase pump operating life, and provide better system flexibility. With these results, it is the authors’ hope that Uzma Engineering Sdn Bhd will apply the proposed impeller design to their centrifugal pump with favourable outcome.

Pump related research should be intensified in the Malaysian oil and gas industry. A more detailed study on the relationship between machine’s operation and design should be conducted. Thus, problems related to pump issues such as cavitation, wear and tear, and inadequate design can be controlled, and as a result, production environment can be attained.

ABBREVIATIONS

ANSYS - American Computer Aided Software
ANSYS CFX - CFD Software Tool
CFD - Computational Fluid Dynamics
Sdn. Bhd. - Sendirian Berhad/Private Limited
SST - Shear Stress Transport

ACKNOWLEDGEMENTS

The authors would like to thank Uzma Engineering Sdn. Bhd. for their assistance. Not to forget, our sincerest gratitude to all of those who took part directly and indirectly in this research.

REFERENCES

[1] P. L. Fraenkel, "Water lifting devices", *Fao.org* (1986). [Online]. Available: http://www.fao.org/docrep/010/ah810e/AH810E05.htm#5.3.1 [Accessed: 02- Jan- 2017].

[2] R. Ragoth Singh and M. Nataraj, "Design and analysis of pump impeller using SWFS", World Journal of Modelling and Simulation, vol. 10, no. 2, pp. 152-160 (2014).

[3] M. Ajith and D. M Issa, “Design and Analysis of Centrifugal Pump Impeller using ANSYS FLUENT”, International Journal of Science, Engineering and Technology Research (IJSETR), vol. 4, no. 10, pp. 3640-3643 (2017).

[4] K. Kumar Yadav, K. Mendiratta and V. Gahlot, “Optimization of the Design of Radial Flow Pump Impeller through CFD Analysis”, International Journal of Research in Engineering and Technology, vol. 05, no. 11, pp. 157-161 (2016).

[5] S. Rao, “Reliability-based design”, 1st Ed. New York: McGraw-Hill (1992).

[6] L. Bachus and A. Custodio, “Know and understand centrifugal pumps”, 1st ed. Oxford: Elsevier (2006).

[7] R. Mackay, "Troubleshooting Pump Problems and Failures", *Pumpsandsystems.com* (2008). [Online].

Available:
http://www.pumpsandsystems.com/topics/pumps/pumps/troubleshooting-pump-problems-and-failures?page=2
[Accessed: 29- Dec- 2016].

[8] K. Schoenheit, "Detecting Pump Cavitation", *Modern Pumping Today®* (2013). [Online].
Available:
http://www.modernpumpingtoday.com/detecting-pump-cavitation/
[Accessed: 09- Dec- 2016].

[9] "Boiling", *Chem.purdue.edu*. [Online]. Available:
https://www.chem.purdue.edu/gchelp/liquids/boil.html
[Accessed: 18- Jan- 2017].

[10] R. S Muttalli, S. Agrawal and H. Warudkar, "CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX", International Journal of Innovative Research in Science, Engineering and Technology, vol. 03, no. 08, pp. 15553-15561 (2014).

[11] A. R. Budris, "Back To Basics: Mechanical Seals for Water and Wastewater Pumps", *Waterworld.com*. [Online].
Available: http://www.waterworld.com/articles/print/volume-27/issue-7/departments/pump-tips-techniques/back-to-basics-mechanical-seals-for-water-and-wastewater-pumps.html
[Accessed: 30- Jan- 2017].

[12] S.S. Rao, "The finite element method in engineering", *Elsevier*. (2010).