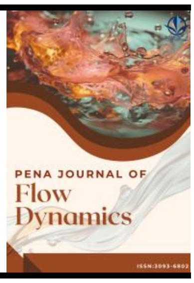
Pena Journal of Flow Dynamics

Journal homepage:
https://penacendekia.com.my/index.php/pjfd/
ISSN: 3093-6802



Analysis of Flow Characteristic and Pump Efficiency at KKTDI Residential College

Nor Adrian Nor Salim[1], Mohamad Faizal Ibrahim[1], Mohamad Shafiq Jamaludin[1], Aliff Asyraf Redzwan[1], Nik Husnun Zain Nik Abdullah[1], Nur Atiqah Zulkefle[1], Ishkrizat Taib[1,*], Ahmad Mubarak Tajul Ariffin[1], Adam Kamsani[1], M. Saddam Kamarudin[1]

[1] Faculty of Mechanical and Manufacturing Engineering Universiti Tun Hussein Onn Malaysia (UTHM), 86400 Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 14 December 2024
Received in revised form 17 January 2025
Accepted 21 February 2025
Available online 31 March 2025

Keywords:
Centrifugal pump; fluid flow characteristics; pump efficiency; pressure; velocity

ABSTRACT

A centrifugal pump is a mechanical device for moving a fluid from a lower to a higher location, or a lower to a higher-pressure area. This study focuses to investigate the fluid flow characteristic and the efficiency of centrifugal pumps. A Centrifugal pump is functioning to convert rotational energy, often from a motor to energy in a moving fluid. For the experimental data, the analysis of the fluid flow characteristics and efficiency of the centrifugal pump was conducted by using the simulation to observe its characteristics and efficiency in the centrifugal pump that cannot be observed by experimental. The simulation on CFD modelling is acceptable which has the difference between experimental and simulation below 20%. The efficiency of the simulation was 29% meanwhile the efficiency for the experimental was 45%. Then, the pressure and velocity for simulation at 7m3/h flowrate were 421Pa and 35.0m/s, respectively. However, the pressure and velocity for pump specification at 12m3/h were 1090Pa and 47.8m/s, respectively. Therefore, it can be concluded that the fluid flow characteristic which is pressure and velocity, and their efficiency can be identified.

1. Introduction

In this research, the aim in conducting this project is to investigate fluid flow characteristic in the centrifugal pump and to determine the efficiency of pump. Water tank is an indirect system which means the water supply is supplied at the water meter through the service pipe to the water tank [1]. The function of water tank is to supply the water to various plumbing in Tun Dr Ismail Residential College such as toilet, bathroom, laundry room and tap pipes [2]. For water supply in college requires a pumping system as usually the pressure in the public pipe is insufficient to reach the water storage tank. The storage tank that used at Tun Dr Ismail Residential College have capacity about 320 kilo gallons.

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/pjfd.1.1.5363

For pumping system, it requires sufficient energy to deliver water throughout the pipeline. A pump is a mechanical device for moving a fluid from a lower to a higher location, or from a lower to a higher-pressure area [3]. A pump can be further defined as a machine that uses several energy transformations to increase the pressure of a liquid. Regenerative turbine pump is a type of centrifugal pump that uses a rotating impeller to increase the velocity of the fluid. These pumps work on a very simple mechanism. A centrifugal pump converts rotational energy, often from a motor to energy in a moving fluid.

There are two main parts that responsible for the conversion of energy are the impeller and casing. The impeller is the rotating part of the pump, and the casing is the airtight passage which surrounds the impeller. In a centrifugal pump, the fluids enter the casing and falls on the impeller blades at the eye of the impeller. Then it is whirled tangentially and radially outward until it leaves the impeller into the diffuser part of the casing. While passing through the impeller, the fluid is gaining both velocity and pressure.

In college also provide suction tank that is a tank installed in combination with a pump. The require energy for the system is provided by the pump [4]. If the energy is not enough to deliver the water, it is providing disruption of water supply in the pipelines such as the current problem happen at Tun Dr Ismail Residential College. Hence, our group decided to study the energy required to pump through a pipeline. The focus is the use of computer software and data modelling to simulate and analyze the structure and efficiency of centrifugal pump.

The various parameters affect the pump performance and energy consumption [5]. In this study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. The numerical solution of 3D, incompressible Navier-Stokes equations over an unstructured and non- uniform grid is accomplished with a commercial CFD finite volume code. At high flow rates, the increase of the outlet blade angle causes a significant improvement of the hydraulic efficiency. So, through this paper one can know about the effect of outlet blade angle on performance of centrifugal pump. The numerical simulation of centrifugal pump having 200 m3/hr capacity using commercial CFD package FLUENT [6].

The simulations were carried out at six different operating points between 30% to 110% discharge, to cover the wide range of operation. It was also observed that the pressure rise was quite gradual and uniform at rated and overrated discharge, but it was non- uniform at partial discharge. The variation of velocity is quite uniform at rated discharge compared to part load & overrated discharge. It was found that k-ω SST turbulence model provides better results compared to RNG k-Ɛ model. With increase in discharge, Head decreases, Power input increases and Efficiency of Pump increases. Efficiency is maximum at duty point, after this point as discharge increases, efficiency of pump decreases.

It modelled a centrifugal pump impeller and solved using computational fluid dynamics, the flow patterns through the pump, performance results, circumferential area averaged pressure from hub to shroud line, blade loading plot at 50 % span, stream wise variation of mass averaged total pressure and static pressure, stream wise variation of area averaged absolute velocity and variation of mass averaged total pressure contours at blade leading edge and trailing edge for designed flow rate are presented [7]. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller.

It shows role of CFD in analysis of Centrifugal pump. CFD technique has been applied to carry out different investigations on centrifugal pumps viz. performance prediction at design and off-design conditions, parametric study, cavitation analysis, diffuser pump analysis, performance of pump running in turbine mode etc. Unsteady Reynolds Average Navier Stokes (URANS) equations together with two equation k-ε turbulence model was found to be appropriate to get a reasonable estimation of the general performance of the centrifugal pump, from an engineering point of view, with typical errors below 10 percent compared with experimental data [8].

Impeller and diffuser flows have been studied extensively and volute flow study has appeared as an interesting research field for further improvement of the pump performance. The most active areas of research and development are the analysis of 2-phase flow (cavitation and slurry flow), pump handling non- Newtonian fluids and fluid–structure interaction. CFD approach provides many advantages compared to other approaches; however due to the empirical nature of solution technique validation with experimental results is usually recommended. Based on our site visit at the TDI residential college, the pump that had been used there is Regaling pump with the model RGV8-4. Pipe flow under pressure is widely used and have a lot of purpose. As we can see in the industries, the huge flow of transport is necessary to get continuous flow of the fluid from the different processing unit. The Regaling type of pump is manufactured from the stainless steel and had using the latest technology to provide a single pump solution to high pressure applications.

Its precise engineering ensures easy installation with no alignment problems and is ideal for energy saving. Furthermore, it can operate with the capacity of 2-160m3/h, normal temperature is between -15˚C to 70˚C, maximum operating pressure is 10 bar and the altitude can be achieved up until 1000m. This pump has been introduced to many applications such as industrial, fire protection, booster system, air conditioning and boiler system. However, the calculation performed will not be exactly accurate as the functioning pipe at the site. This is because there are another factor affecting the flow in the pipe such as head losses.

2. Methodology

In the simulations analysis by using ANSYS 19.2, there have several steps that must be followed to make sure the analysis running smoothly without facing any problem. To make sure that we get the accurate value for the simulation’s analysis, we must get the certain important of parameters that might be used in the analysis when we try to run the simulation.

This chapter comprises study on method of implementing the project entirely. In addition, this section will focus on the procedures and method used to determine the efficiency of a pump. The methodology was design to achieve the objectives.

To determine the fluid flow characteristic, there are required a specific computer software and certain fluid dynamic equation to measure and identify the flow characteristic. Computational fluid dynamic software plays the main role to determine the flow of fluid. It can help the researchers to determine the fluid flow in the flowrate in a pump by the simulation. This experiment is conducted at Tun Dr Ismail Residential College (TDI) in UTHM and focusing on water pumping system. The main objective in this experiment is to identify the effectiveness of water pumping system in TDI. This experiment is related to the formula as shown below [9,10]:

$$Pump\ Efficiency, \eta = \frac{Water\ housepower}{Housepower\ of\ pump} \tag{1}$$

$$Water\ housepower, WHP = \frac{HQ}{3690} \tag{2}$$

Where:
H = Change in pressure measured in height of water in feet
Q = Water flowrate in gallons per minute

To get all the information, this study needs to make a site visit. The equipment's have been used to get the data have stated as Figure 1 below:

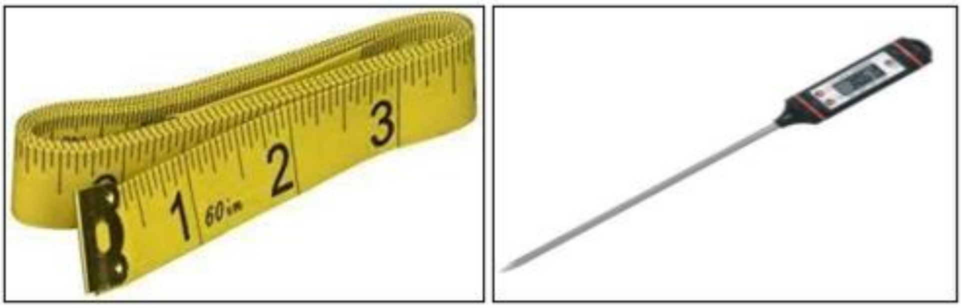


Fig. 1. The equipment (a) Measuring tape (b) Digital thermometer

Measuring tape is used to measure the dimension of pipe. To get an accurate drawing in Solid works software, the pump needs to be measured detailed. This type of measuring tape is used because it flexible and suitable to measure the pump. thermometer was used to measure the temperature of water inside and outside the pump. The temperature will be used as the boundary conditions in simulation. To get the temperature reading inside the pump, a valve needs to open and allowed it to flow out. The thermometer will put at the flow water and the reading will be recorded.

2.1 Geometry Creation

The first step involved in this stage is to define a suitable geometry in which the fluid problem is to be analyzed. Defining the geometry can be achieved by replicating a known dimension such as length and all the importance parameters follow with the actual project of the pump through SOLIDWORKS software.

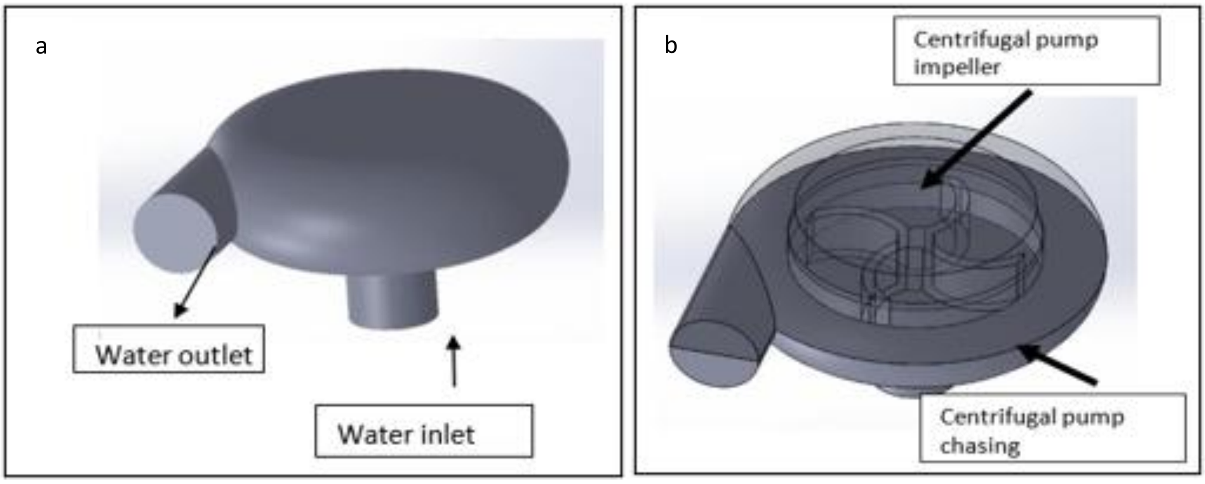


Fig. 2. Geometry (a) 3D modelling of centrifugal pump (b) Centrifugal pump details

2.2 Meshing

Meshing plays an important role for any simulation to be done in the computer simulation. The quality of the mesh is crucial since it will determine the results where it influence the accuracy, convergence, and the speed of the solution. There are few methods to obtain the most quality number of meshing [11].

In this analysis started with global meshing control which is based on the geometry study. Automatics method was chosen as meshing method since it gives the shortest time in meshing. Next is, control the meshing sizing which by selecting the body of the geometry and the element sizing was determine by applying grid independence test (GIT), to produce more accurate meshing process. On the meshing, we get the node about 7231while the total of element is 23600. The face meshing for this centrifugal pump that involve is eight faces. For physics preferences, it set in the Computational Fluid Dynamics (CFD) and the solver preference is in fluent.

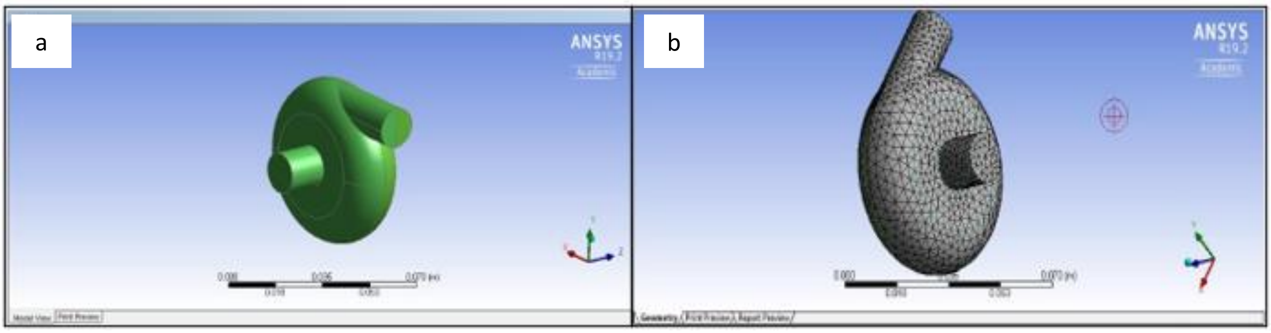


Fig. 3. Geometry (a) The geometry of centrifugal pump (b) The meshing

2.3 Boundary Condition

The model that has been defined the physical modelling is then be applied by boundary conditions to cell faces in the form of a flux adjacent to an inlet or outlet or wall such as temperature, pressure, and velocity profile. This involves specifying the fluid behaviour and properties at the boundaries of the problem.

To run the simulation, the value of parameter setting need to know first. In the parameter are included density, velocity, pressure, area, revolution per minutes (RPM) of inlet pump. This value needs to be calculate using the information from specification of the centrifugal pump.

Before defining the value of parameter to run the simulation, the information of the centrifugal pump is show in Table 1.

The volume flow rate of the centrifugal pump is 0.0194 m3/s generated by 2,900RPM of the blade motor. The area of boundary is only involved inlet and outlet of centrifugal pump.

Table 1
Information of REGALINE water pump RGV8-4

Rotational of blade ω (RPM)	2900
Temperature of water °C	Inlet: 30
	Outlet: 29.5
Q flowrate, volume (m^3/s)	0.0194
Pressure, P (kPa)	20
Time taken to fill up tank (1 tank), t	4 hours
Diameter of inlet pump, d1 (m)	0.048

From the information at Table 3, we can determine the inlet velocity using formula:

$$Velocity, v_1 = \frac{Q}{A} \tag{3}$$

$$Area\ of\ inlet\ centrifugal\ pump, A_1 = \pi r^2 \tag{4}$$

Where:
Q = Volume flowrate (m3/s)
A = Area of inlet centrifugal pump (m2) r = Radius of inlet centrifugal pump (m)

From Equation 4, we can obtain area of inlet centrifugal pump as expressed below:

$$A_1 = \pi(0.024^2)$$
$$= 1.809 \times 10^{-3}$$

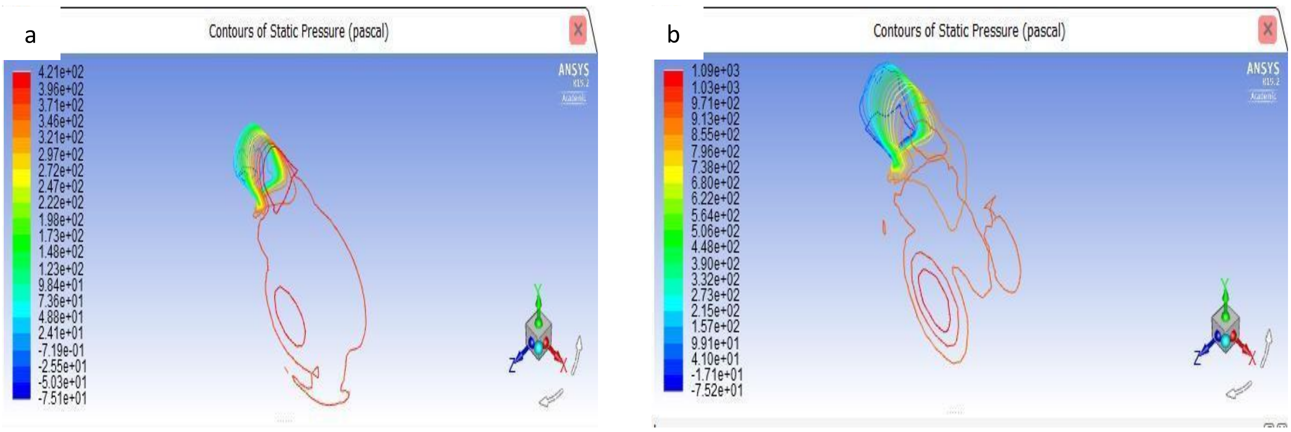
Therefore, substitute value of into Equation 3:

$$v_1 = \frac{0.0194}{1.809 \times 10^{-3}}$$
$$= 10.71\ ms^{-1}$$

3. Result and Discussions

The results and discussions of this research work are presented here in this chapter. The discussion has been discovered with the explanation to summarize the results obtained from the simulation. This discussion is further with several sets of data from the simulation to enable us to solve the residential college pump performance. Data about pressure and velocity are extracted in the simulation for the centrifugal pump and being discussed in this chapter.

From the result obtained from the simulation, we get the value of pressure from the results in range of minimum that is -75 Pa and for the maximum we get 421 Pa. The value of the pressure obtained from the simulation are in the range and are proved to be accepted. The Figure 4 below shows the simulation results of pressure contour distribution in the centrifugal pump. This result used the flowrate, Q = 7 m3/h as the inlet boundary condition. Based on Figure 4, the result obtained on the specs simulation, we get the value of the pressure in the range of -75 Pa and for the maximum is 1090 Pa. The maximum pressure for the specs simulation is quite high compared to the simulation result may be due to its different boundary condition. This simulation used the flowrate, Q = 12 m3/h as the inlet boundary condition.



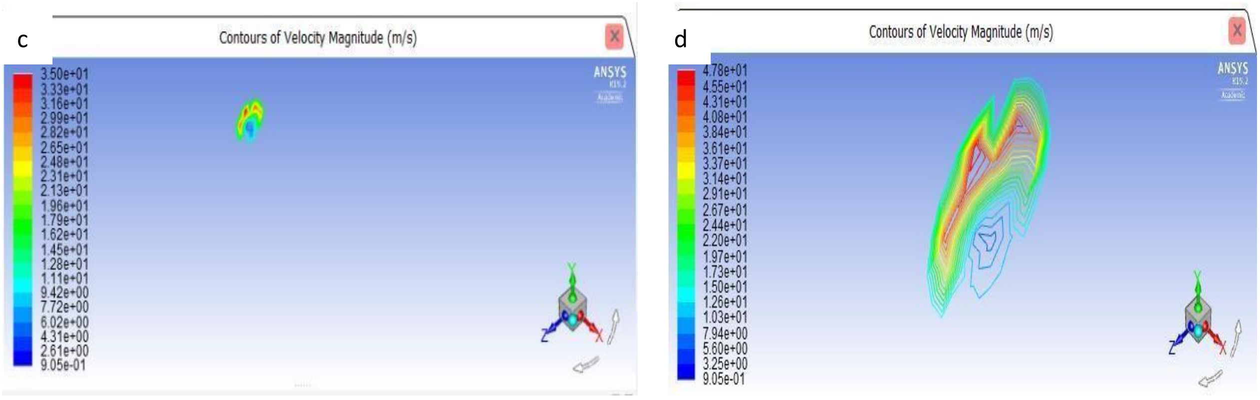


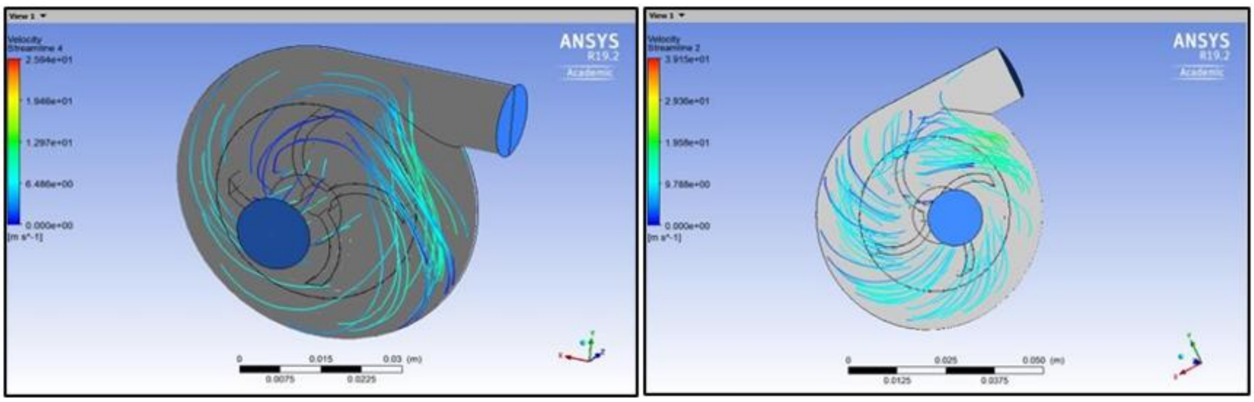
Fig. 4. Result (a) Pressure contour for centrifugal pump (b) Specs simulation (c) Velocity contour for centrifugal pump and (d) Velocity contour for specs simulation

Table 2
Validation by comparing simulation and specs results

Parameter	Simulation (Q=7 m^3/h)	Specs (Q=12m^3/h)
Pressure	421 Pa	1
Velocity	35.0 m/s	2

3.1 Streamline of the Simulation

The streamline of the simulation is a crucial element in this study because it shows the result obtained are accurate. Other than that, it shows the flow behavior inside the centrifugal pump itself. From the result obtained, it illustrates the flow of the fluid inside the pump from inlet to the outlet as our study focus on velocity on the outlet. For the velocity, we get 6.486 m/s in simulation compared to the specs velocity which is 9.788 m/s. Figure 5 shows the velocity streamline of the centrifugal pump.



(a) (b)

Fig. 5. Result (a) Velocity streamline for 3D view (Simulation) (b) Velocity streamline for specs simulation

3.2 Pressure, Turbulence and Velocity Contour for Flowrate, Q=7m^3/h

The simulation has been done by using the flowrate, Q = 7 m^3/h. Based on the result, it shows that the range for static pressure of the pump is between -75 Pa until it reaches the maximum value 421 Pa. The result of static pressure is illustrated in Figure 6(a). Figure 6(b) shows the turbulence result of the pump by using the 7m^3/h flowrate. It shows that the minimum value is 0 until it achieves the highest state which is 53.9. The value for the turbulence of pump is in the range and are proved to be accepted. Based on the Figure 6(c), it illustrates the velocity contour of the centrifugal pump. The difference of range between minimum and maximum value of velocity is slightly difference. From the simulation, it shows that the value for velocity is between 0.95 m/s to 35 m/s when it achieves the maximum state.

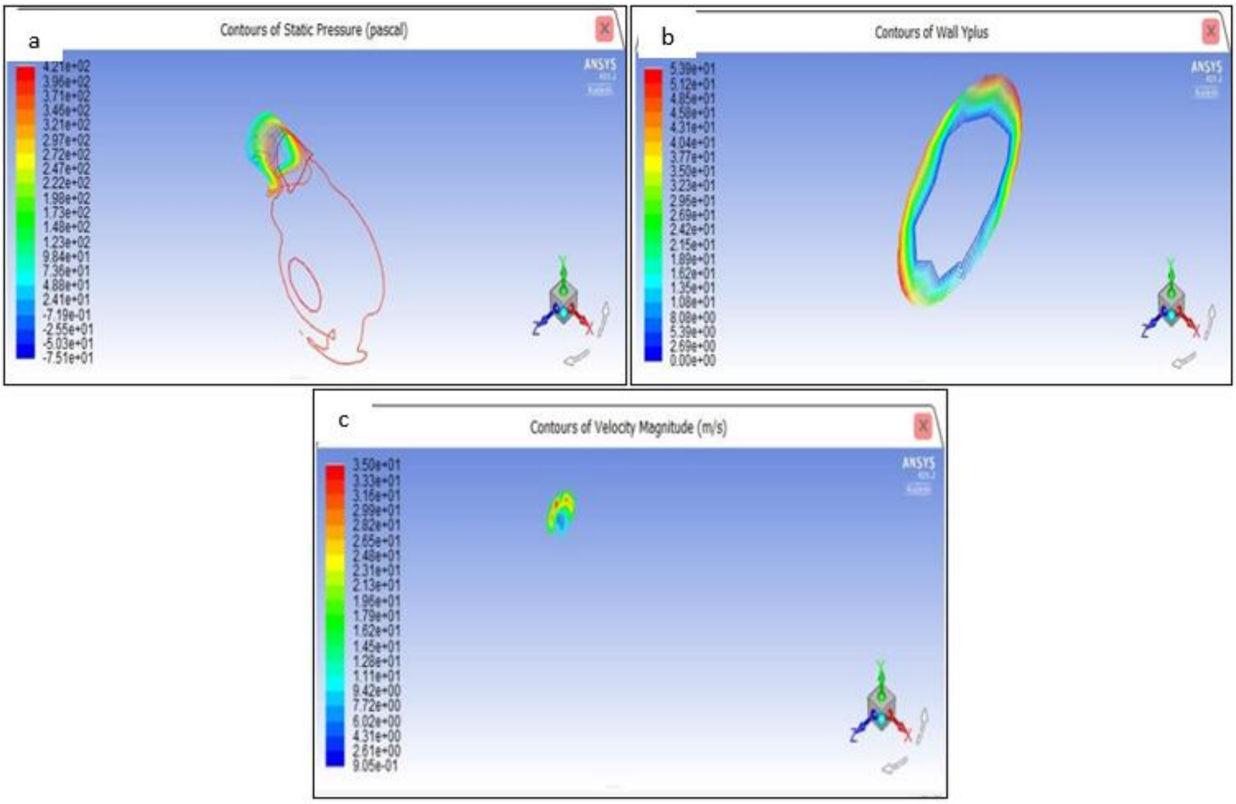


Fig. 6. Result (a) Static pressure of the pump (b) Turbulence result of pump and (c) Velocity of the pump

3.3 Pressure, Turbulence and Velocity Contour for Flowrate, Q=9m3/h

The simulation has been done by using the flowrate, Q = 9 m3/h. Based on the simulation, it shows that the static pressure is in the range of -67 Pa to 674 Pa. Figure 7(a) illustrate the pressure contour for simulation result. Figure 7(b) shows the turbulence result of the pump by using the 9m3/h flowrate. It shows that the minimum value is 0 until it achieves the highest state which is 60.9. The value for the turbulence of pump is in the range and are proved to be accepted. Based on the Figure 7(c), it illustrates the velocity contour of the centrifugal pump. The difference of range between minimum and maximum value of velocity is only slightly difference. From the simulation, it shows that the value for velocity is between 0.95 m/s to 41.1 m/s when it achieves the maximum state.

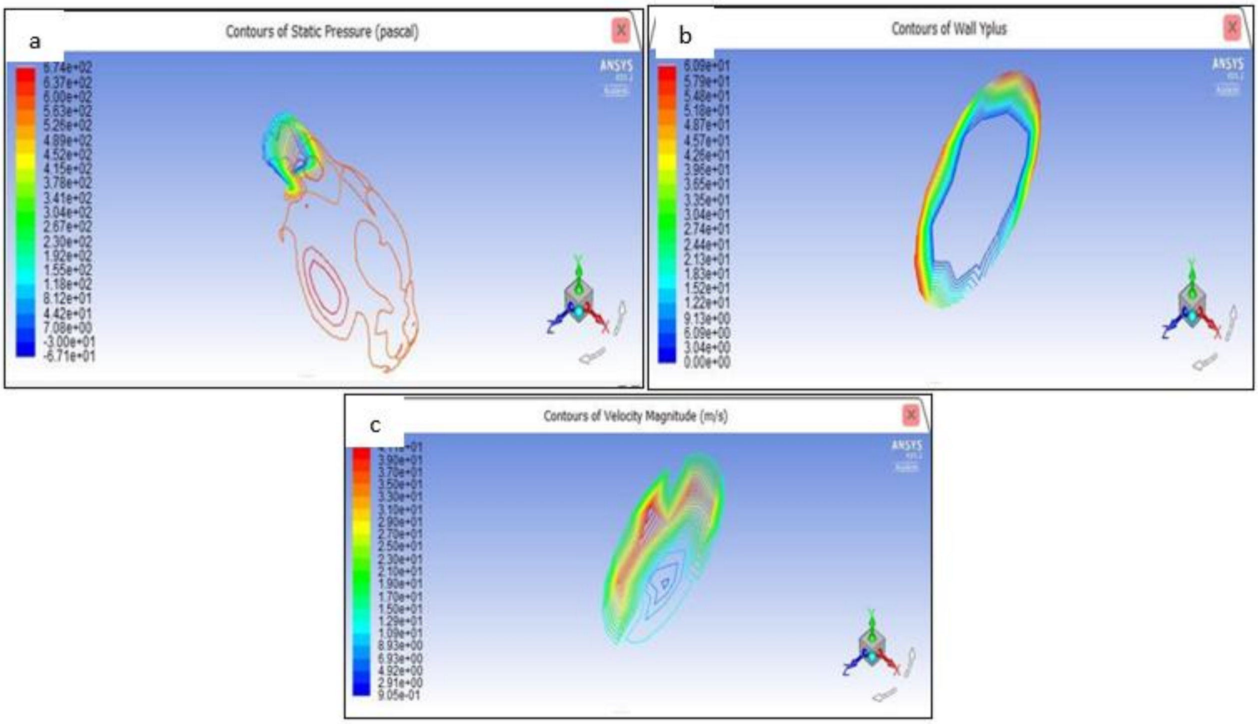


Fig. 7. Result (a) Static pressure of the pump (b) Turbulence result of pump and (c) Velocity of the pump

3.4 Pressure, Turbulence and Velocity Contour for Flowrate, Q=11m3/h

The simulation has been done by using the flowrate, Q = 11m^3/h. Based on the simulation, it shows that the static pressure is in the range of -64.7 Pa to 987 Pa. The difference of pressure between minimum and maximum is quite high maybe due to several factor.

Figure 8 illustrate the pressure contour for the simulation result. Table 3 shows the turbulence result of the pump by using the 11m^3/h flowrate. It shows that the minimum value is 0 until it achieves the highest state which is 68.4. The value for the turbulence of pump is in the range and are proved to be accepted. Based on the Figure 4-15, it illustrates the velocity contour of the centrifugal pump. The difference of range between minimum and maximum value of velocity is only slightly difference. From the simulation, it shows that the value for velocity is between 0.95 m/s to 46.6 m/s when it achieves the maximum state.

Table 3 shows the results for the different flowrate used in this simulation and for the flowrate of 12 as the reference that comes from the specs. From the results obtained, it shows that the value of pressure, turbulence and velocity increased when the number of flowrates increased.

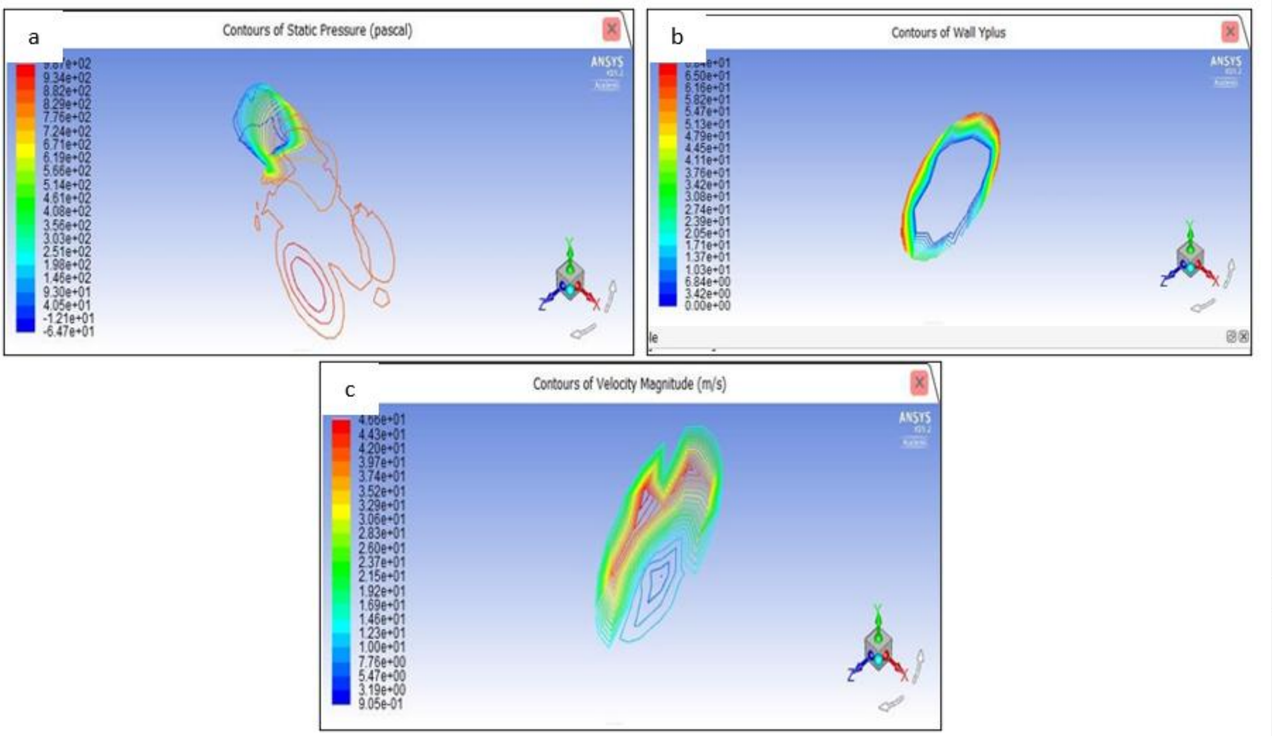


Fig. 8. Result (a) Static pressure of the pump (b) Turbulence result of pump and (c) Velocity of the pump

Table 3
Result for different flowrate

Flowrate, Q (m^3/h)	Velocity, V (m/s)	Turbulence	Pressure,P (Pa)	Percentage difference (%)
7	35.0	53.9	421	26.8
9	41.1	60.9	674	14.0
11	46.6	68.4	987	2.5
12	47.8	69.9	1090	(reference)

3.5 Pump Efficiency

From the results of simulation, we obtained the pump efficiency is 29 % while for the experimental results for pump efficiency is 45 % after 5 years used in the residential college.

Pressure, p = 2 bar
= 200 kPa

Flowrate, Q = $7m^3/h$
= $0.00194444m^3/s$

Power, P = 1.5 kW

$P = T\omega$
$1500 = T\ (2900)$
$T = 0.517$

V = 6.486 (from simulation) converts to ω= 2580.691rpm

$$\eta = \frac{PQ^{\cdot}}{T\omega}$$

$$= \frac{(200kPa)(0.00194444)}{(0.517)(2580.691)} \times 100\ \%$$

$$= 29\ \%$$

4. Conclusions

The model of centrifugal pump has been developed to investigate the fluid flow inside pump at Tun Dr Ismail Residential College. It was simulated to investigate the structure in centrifugal pump according to the parameter. The data for pressure and velocity of centrifugal pump were obtained from the simulation. From the simulation, the pressure and velocity of flowing fluid in the centrifugal pump was in the acceptable range. Then, we also obtained that the efficiency of the centrifugal pump was 45% after 5 years use it.

Thus, the model has been successfully captured the fluid flow in centrifugal pump. The present work has made a useful contribution to study the flow of fluid in the centrifugal pump at Tun Dr Ismail Residential College. Therefore, it can be concluded that this study has achieve the objective which to investigate the fluid flow characteristic and its efficiency in the centrifugal pump.

Acknowledgement
This research has been supported by Universiti Tun Hussein Onn Malaysia (UTHM) through the TIER 1 research grant (H926) lead by Dr. Nor Adrian Nor Salim.

References

[1] Dona Rose, K. J., M. Sreekumar, and A. S. Anumod. "A study of overhead water tanks subjected to dynamic loads." *International Journal of Engineering Trends And Technology (IJETT)* (2015).
[2] T. Yogesh, K. Yogesh, K. Sagar, and K. Sanchit, "International Journal of Engineering Sciences & Manangement Analysis of Water Tank on Sloping Ground," 2017.
[3] Gurupranesh, P., R. C. Radha, and N. Karthikeyan. "CFD Analysis of centrifugal pump impeller for performance enhancement." *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN* (2010): 2278-1684.
[4] Song, Wen-wu, Li-chao Wei, Jie Fu, Jian-wei Shi, Xiu-xin Yang, and Qian-yu Xu. "Analysis and control of flow at suction connection in high-speed centrifugal pump." *Advances in Mechanical Engineering* 9, no. 1 (2016): 1687814016685293. https://doi.org/10.1177/1687814016685293
[5] Bacharoudis, E. C., A. E. Filios, M. D. Mentzos, and D. P. Margaris. "Parametric study of a centrifugal pump impeller by varying the outlet blade angle." *Open mechanical engineering journal* 2, no. 5 (2008): 75-83. https://doi.org/10.2174/1874155X00802010075
[6] Shah, S. R., S. V. Jain, R. N. Patel, and V. J. Lakhera. "CFD for centrifugal pumps: a review of the state-of-the-art." *Procedia Engineering* 51 (2013): 715-720. https://doi.org/10.1016/j.proeng.2013.01.102
[7] S.Rajendran and Dr.K.Purushothaman, "(PDF) Analysis of a centrifugal pump impeller using ANSYS-CFX," vol. Volumw 1, no. Issue 3, 2012.
[8] Muttalli, Raghavendra S., Shweta Agrawal, and Harshla Warudkar. "CFD simulation of centrifugal pump impeller using ANSYS-CFX." *International Journal of Innovative Research in Science, Engineering and Technology* 3, no. 8 (2014): 15553-15561.
[9] Stringam, Blair. *Understanding Horsepower and Water Horsepower Efficiency and Fuel Consumption Costs for Irrigation Pumps*. NM State University, Cooperative Extension Service, 2013.
[10] Joe Evans, "Pump Efficiency—What Is Efficiency?," 2012.[Online].Available: https://www.pumpsandsystems.com/topics/pumps/pumps/centrifugal-pump-efficiency-what-efficiency. [Accessed: 20-Apr-2019].
[11] I. Sadrehaghighi, "Mesh Generation in CFD," CFD Open Ser., 2017.