Progress in Engineering Application and Technology Vol. 2 No. 2 (2021) 844–853
© Universiti Tun Hussein Onn Malaysia Publisher's Office

PEAT

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/peat
e-ISSN : 2773-5303

Simulation Study of Various Blade's Numbers and Performance's Effect Inside Impeller for Centrifugal Pump System

Helmy Faidzhi Zainal[1], **Rahmah Mahmudin**[1]*

[1]Department of Mechanical Engineering Technology, Faculty of Engineering Technology,
Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, MALAYSIA

* Corresponding Author Designation

DOI: https://doi.org/10.30880/peat.2021.02.02.076
Received 13 January 2021; Accepted 01 March 2021; Available online 02 December 2021

Abstract: The Impeller of Centrifugal Pump is one of most Critical Component to be designed. The design of enclosed impeller is done by using various number of blades to determine the pressure and velocity of the fluid. The CFD analysis of Enclosed impeller has been carried out at the same inlet and outlet of impeller blade angles to investigate the changes in head generated. The three number of different blades impeller with 14 degree of inlet angle and 20 degree of outlet angle was simulated and extracted with data on the pressure and velocity contained within. The highest value for pressure generated is 1340730 Pa which is with the number of blades is 6. Meanwhile in the scope of velocity of fluid, the 3 blades of impeller can produce the highest value which is 9.57 ms^{-1} also with the most the head generated 56.04 m. Therefore 3 numbers of blades impeller had better fluid velocity and head generated compared with others. For the future research, we recommend to modify the angle of inlet and outlet for better performance of this impeller. Angle of inlet and outlet can play importance role in deliver fluid transfer in pump with better optimization to reduce energy consumption, increase pump operating life, and provide better system flexibility

Keywords: Impeller Design, Solidworks, Blades, Pressure Distribution, CFD

1. Introduction

A centrifugal pump is a functional mechanism designed to shift a liquid from one or more powered rotors, by the transmission of rotational energy. The fluid joins the quickly revolving impeller along its axis and is thrown back through the vane tips of the impeller by centrifugal force along its diameter.[1] The impeller's motion raises the velocity and friction of the fluid and thus drives it into the outlet of the pump. The pump housing is designed primarily to constrict the fluid from the pump suction, direct it into the impeller and then slow down the fluid before release. The main feature of a centrifugal pump is the impeller. It consists of a set of vanes which are curved. Normally, they are sandwiched between

*Corresponding author: rahmah@uthm.edu.my

two discs. Open or semi-open impeller for fluids of entrained solids. At its axis, fluid joins the impeller and leaves along the circumference between the vanes. In the opposite side of the eye, the impeller is attached to a motor by a drive shaft and rotated at high speed (typically 500-5000 rpm) [2].

The impeller's rotational motion accelerates the fluid through the pump casing through the impeller vanes. The impeller, driven by the blower shaft adds the velocity component to the fluid by centrifugally casting the fluid away from the impeller vane tips. The key idea here is that the energy created is kinetic energy. The amount of energy given to the fluid corresponds to the velocity at the edge or vane tip of the impeller. Mechanical pumps serve in a wide range of applications such as pumping water from wells, aquarium filtering, pond filtering and aeration. In the car industry for water-cooling and fuel injection, in the energy industry for pumping oil and natural gas or for operating cooling towers and other components of heating, ventilation and air conditioning systems.[3] In the medical industry, pumps are used for biochemical processes in developing and manufacturing medicine, and as artificial replacements for body parts, in particular the artificial heart and penile prosthesis. When a casing contains only one revolving impeller, it is called a single-stage pump.

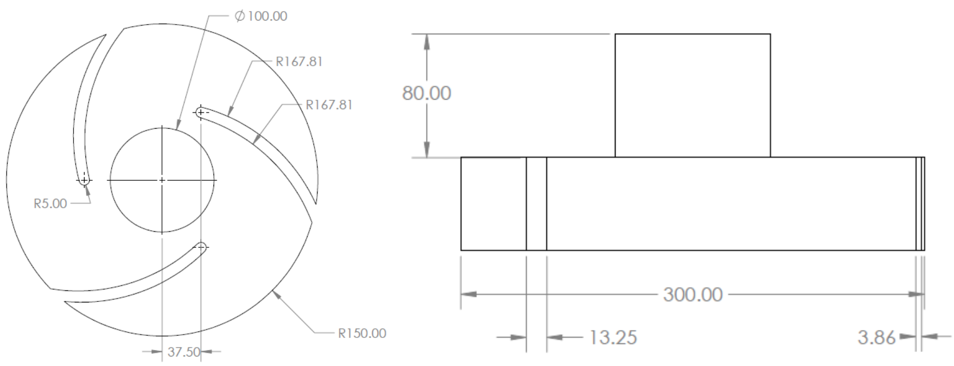
E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris (2008) in their study, the performance of impellers with the same outlet diameter having different outlet blade angles is thoroughly evaluated. The One-dimensional approach along with empirical equations is adopted for the design of each impeller.[4] The predicted performance curves result through the calculation of the internal flow field. Head-discharge curve play important role into different outlet angles. The influence of the outlet blade angle on the performance is verified with the CFD. The performance curve becomes smoother and flatter with the increase with the increase outlet blade angle. At nominal capacity, when the outlet blade angle was increased from 20 ° to 50 °, the head was increased by more than 6.00 % but the hydraulic efficiency was reduced by 4.50 %. However, at high flow rates, the increase of the outlet blade angle caused a significant improvement.[5]

2. Methodology

The present study used the impeller turbine model in reference. The reference model was then modified into a new design to study the pressure and velocity output performance. All the reference and modified models were depicted as in Figure 1 (a) and (b).

2.1 Design Model

For reference model, the inlet diameter was 100 mm, outlet diameter was 150 mm, inlet and outlet angle of the blade was 14 degree and 20 degrees respectively where as modified model attributed different number of blades which was 4, 5, 6 and 7. The height of both inlet and outlet channels were 80 mm and 60 mm respectively. The impeller turbine was installed with three bladed backward type rotors. The thickness of blades was 10 mm respectively.[6]



(a) – Top view of the impeller **(b)** – Side view of the impeller

Figure 1: Impeller Dimension

In this section, a flowchart will be constructed as a guide throughout this thesis conducted. The completion for this study is to conduct a parameter study in order to collect the data from the simulated impeller whose test will be performed in this project which the Ansys CFX Software. The experiment on the impeller will be run only on one type of experiment which is 3 blades of impeller. When the experiment data have been collected, the result will be compared and discussed. Then 4, 5, 6 and 7 blades of impeller will be design with suitable parameter in order to study which is to get the higher velocity of water delivery and pressure.

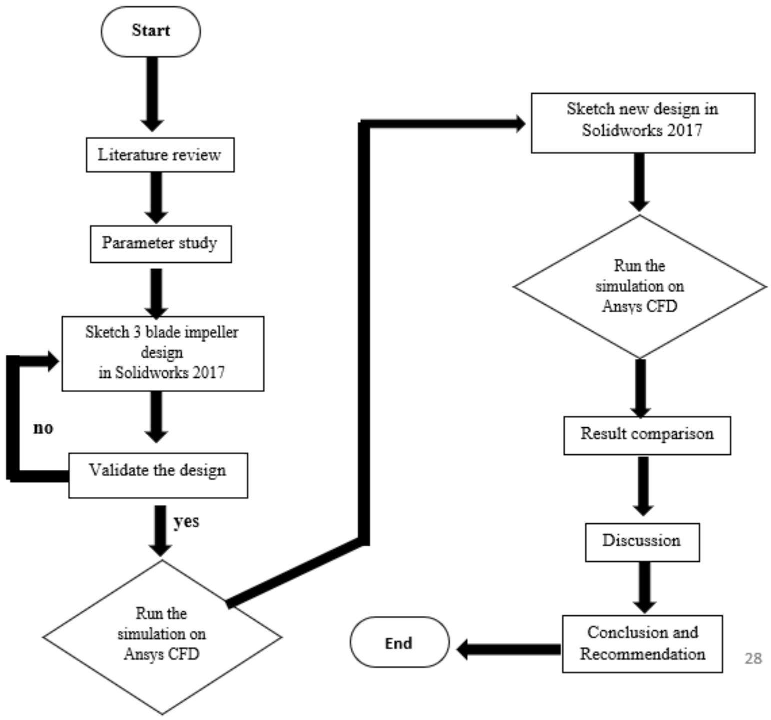


Figure 2: Analysis Flowchart

2.2 Simulation Setup

The global rotating reference frame defines the hub and back of the impeller as the boundary conditions. The inlet relative pressure is set to 700000 Pa of the impeller is set, while the outlet mass flow rate is 35 kg s-1 defined as the outlet boundary condition. The outlet boundary condition defines the exit of the impeller. The reference pressure is set to be 0 Pa. In this study, the radial flow pump impeller has constant operating speed of 2500 rpm and the various numbers of the blades which is 3, 4, 5, 6 and 7 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 mesh of the impeller 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 as the turbulence model. When the results have converged, the numerical calculation is done and the flow within the centrifugal flow pump is obtained.

Table 1: Boundary Condition

Item	Parameter Name	Variable Value	Unit or Dimension
1	Mass Flow Rate	35	Kg/s
2	Impeller Speed	2500	rpm
3	Inlet Relative Prssure	700000	Pa
4	Density	1000	kg m-3
5	Inlet Angle	14	Degree
6	Outlet Angle	20	Degree

2.3 Simulation Process

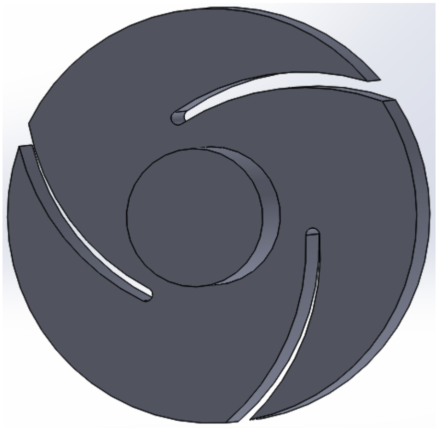


Figure 3: Cavity Impeller Design

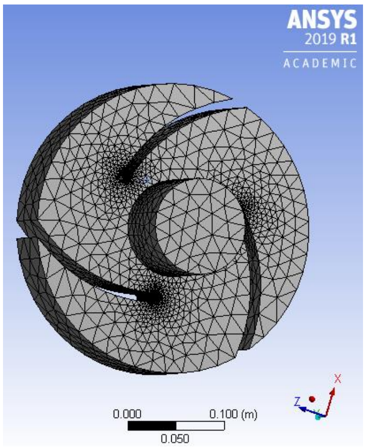


Figure 4: Mesh Setup

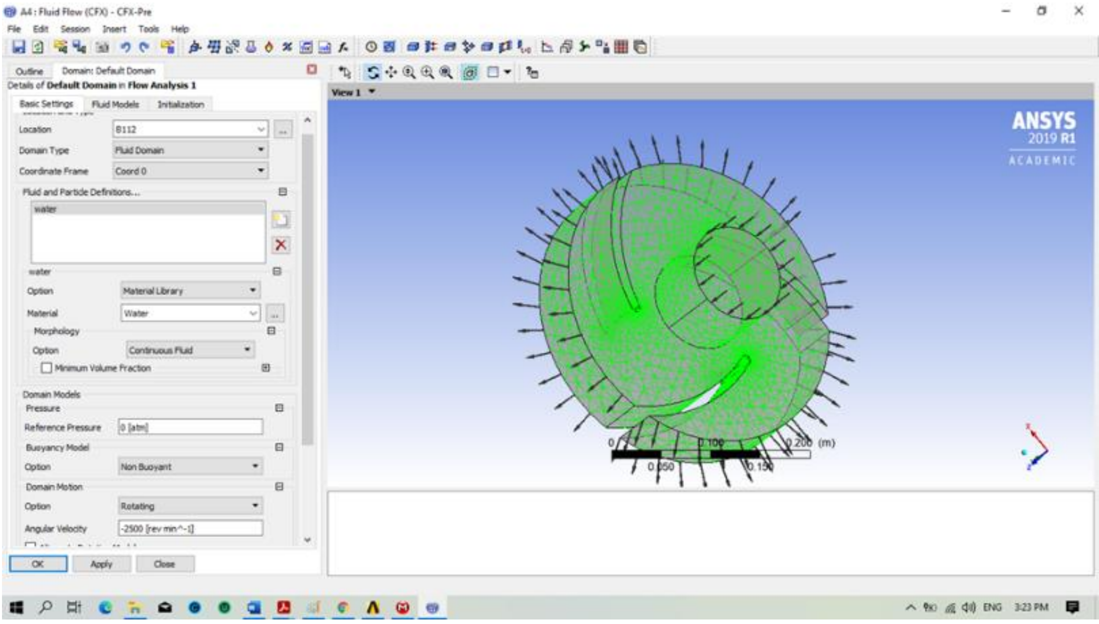


Figure 5: CFX Pre Setting

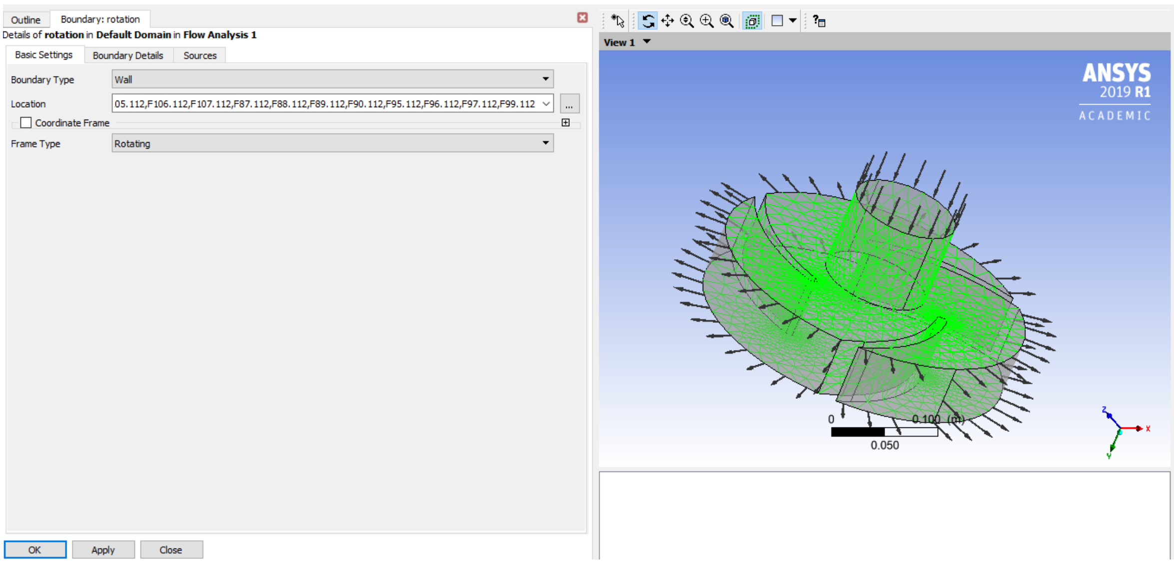


Figure 6: Define Hub as Rotating Wall

1. Meshing of Impeller as **Figure 4**, Meshing Type: 3D, Type of Element: Automatic, No. of Nodes: 18658, No. of Elements: 96737
2. The "CFX Pre" windows will come out. Click on default domain and click edit **(Figure 5)**. Then go to the basic setting and create water fluid type for this impeller. Set the material analysis as water and the type is continuous fluid. The reference pressure will be set to 0 atm and the domain motion will be set as rotating and the angular velocity is 2500 rpm.
3. Define Hub as a Rotating Wall as **Figure 6**, Wall Roughness: Smooth Wall, Heat Transfer: Adiabatic
4. Set the location of the inlet by click at the body of the impeller. Then set the boundary detail for mass and momentum "Stat.Frame.Tot.Press" then set the relative pressure at 700000 Pa.
5. Now do add the next boundary which is the outlet for the impeller. Select the location of the outlet then set the mass flow rate with the value 35 kg/s-1.
6. Next, go to the material specification and then set the material properties as water and the density of the fluid is 1000 kg m-3.
7. Click on the "Solver Control" to adjust the turbulence numeric as high rotation. Set the convergence control with 1000 max iteration and the length scale option as aggressive. Also, the value for the conservation target as 1% imbalance.
8. Now click on the solution section then edit. The define run windows will come out and click on the double precision. Set the run mode as INTEL MPI LOCAL PARALLEL and set partition as 4. After that, click on start run. This method of the step will be repeated for the rest 4, 5, 6 and 7 blades of impeller.

3. Results and Discussion

3.1 Verification and comparison between Reference Model and Modified Model

The pressure and velocity contour of 3 blades impeller for reference and modified model were compared and analyzed as depicted in Figure 1 (a) and (b). The reference and modified model were exhibited maximum pressure of 1305000 Pa and 1285420 Pa respectively.

3.2 Pressure Contour Analysis

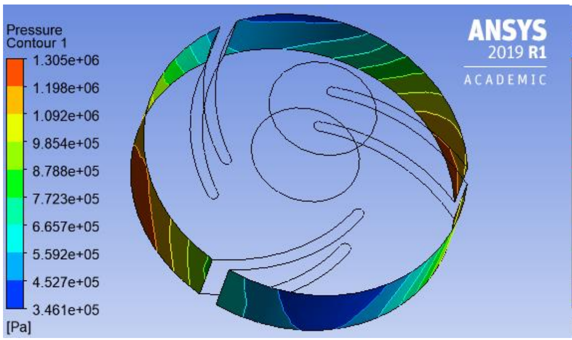


Figure 7: Pressure Contour for 3 blades

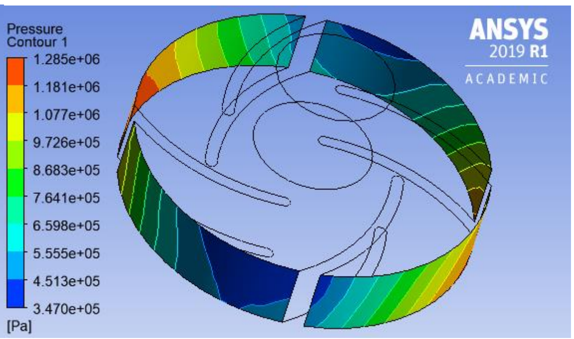


Figure 8: Pressure Contour for 4 blades

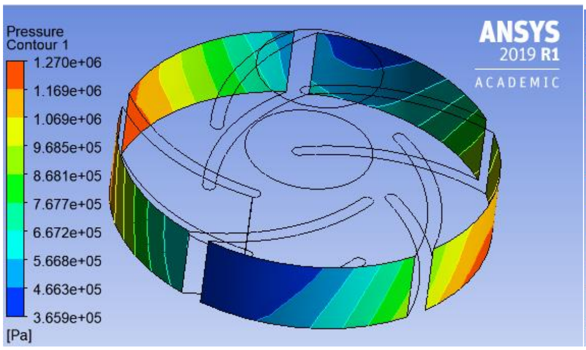


Figure 9: Pressure Contour for 5 blades

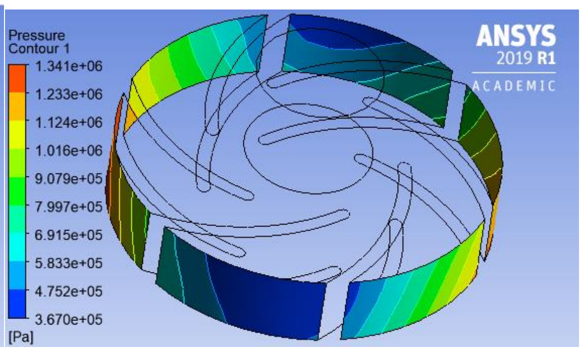


Figure 10: Pressure Contour for 6 blades

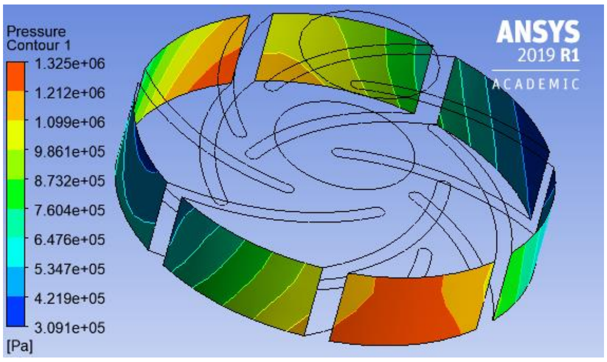


Figure 11: Pressure Contour for 7 blades

Figure 7 shows the pressure contour distribution of 3 blades impeller with minimum value is 346115 Pa and maximum value is 1305000 Pa. The analysis also compared the difference of pressure contour for 4 blades impeller with minimum value is 347012 Pa and maximum value is 1285420 Pa (Figure 8). Meanwhile, Figure 9 shows the pressure contour for 5 blades of impeller with minimum value is 365895 Pa and maximum value is 1269870 Pa. Figure 10 show the pressure contour for 6 blades of impeller with minimum value is 366953 Pa and maximum value is 1340730 Pa. Then, Figure 11 show the pressure contour for 7 blades of impeller with minimum value is 309082 Pa and maximum value is 1324570 Pa.

3.3 Comparison between values of Pressure contour

Table 2: Minimum and Maximum Pressure Values

No. of blade	Minimum values (Pa)		Maximum values (Pa)	
	Inlet	Outlet	Inlet	Outlet
3	634969	346115	755242	1305000
4	623133	347012	810121	1285420
5	641278	365895	838084	1269870
6	644927	366953	842147	1340730
7	639375	309082	777959	1324570

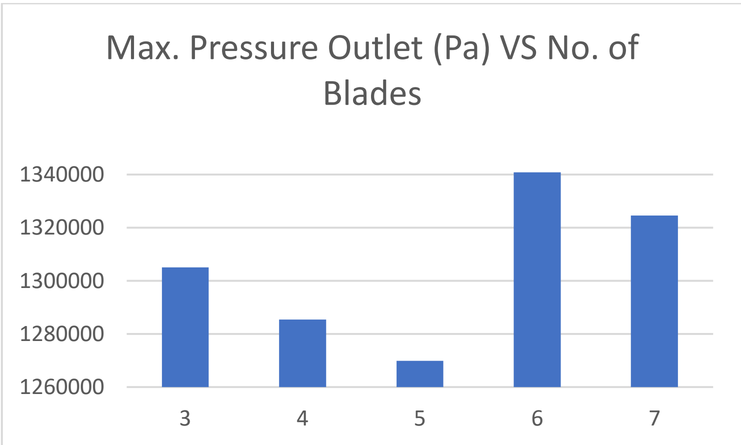


Figure 1: Max. Pressure Outlet VS No. of Blades

The highest value for the outlet pressure at 1340730 Pa which is at impeller with six blades. Meanwhile, the second highest value is 1324570 Pa which at impeller with seven blades with 1.00 % decrease compare to the six blades of impeller. Now, the third highest pressure value is 1305000 Pa which is at impeller with three blades with 2.00 % less then pressure value at impeller with six blades. The fourth highest pressure value is 1285420 Pa with 4.00 % less pressure value compared with 6 blades. The lowest value goes to impeller with five blades which is 1269870 Pa with 5.00 % less than the highest velocity value. This means the impeller with six blades has the highest-pressure value compared with others and it is the best and suitable for this type of impeller in the scope of pressure.

3.4 Velocity Contour Analysis

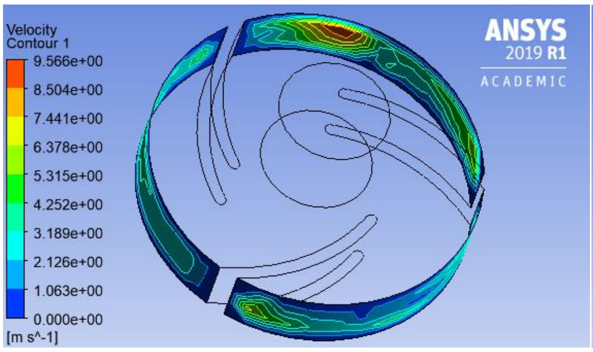


Figure 12: Velocity Contour for 3 blades

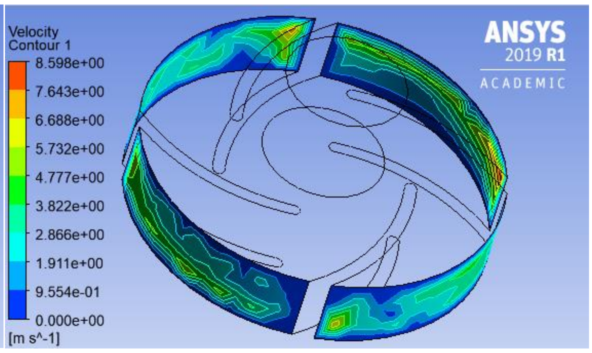


Figure 13: Velocity Contour for 4 blades

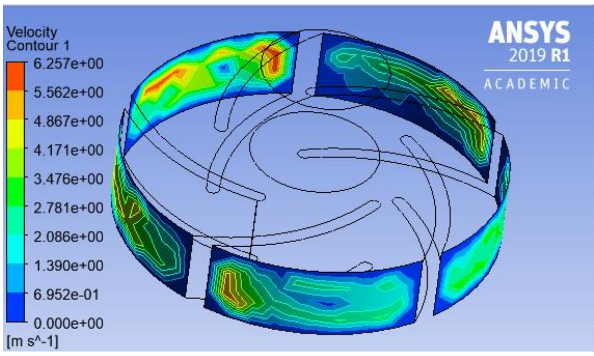


Figure 14: Velocity Contour for 5 blades

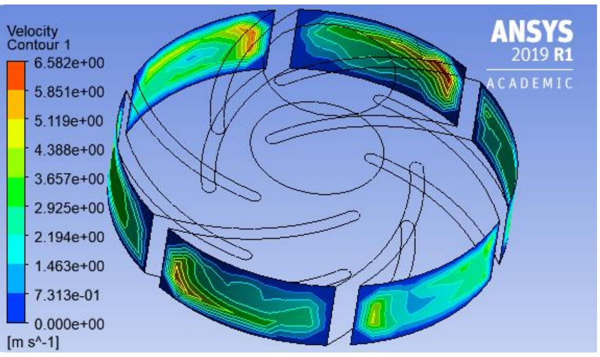


Figure 15: Velocity Contour for 6 blades

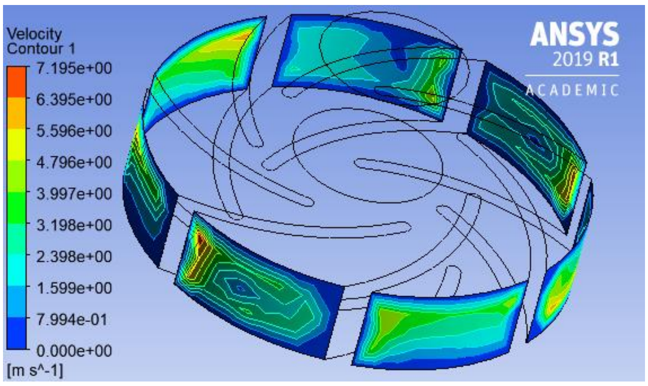


Figure 16: Velocity Contour for 7 blades

Figure 12 shows the velocity contour distribution of 3 blades impeller with minimum value is 0 ms^{-1} and maximum value is 9.56649 ms^{-1}. The analysis also compared the difference of velocity contour for 4 blades impeller with minimum value is 0 ms^{-1} and maximum value is 8.59845 ms^{-1}

(Figure 13). Meanwhile, Figure 14 shows the velocity contour for 5 blades of impeller with minimum value is 0 ms^{-1} and maximum value is 6.25721 ms^{-1}. Figure 15 show the pressure contour for 6 blades of impeller with minimum value is 0 ms^{-1} and maximum value is 6.58204 ms^{-1}. Then, Figure 16 show the pressure contour for 7 blades of impeller with minimum value is 0 ms^{-1} and maximum value is 7.19471 ms^{-1}.

3.5 Comparison between values of velocity contour

Table 3: Minimum and Maximum Velocity Values Comparison

No. of blade	Minimum values (ms^{-1})		Maximum values (ms^{-1})	
	Inlet	Outlet	Inlet	Outlet
3	0.581947	0	24.6793	9.56649
4	1.04109	0	27.4021	8.59845
5	1.07423	0	26.4086	6.25721
6	0.548394	0	26.3951	6.58204
7	0.738928	0	25.9743	7.19471

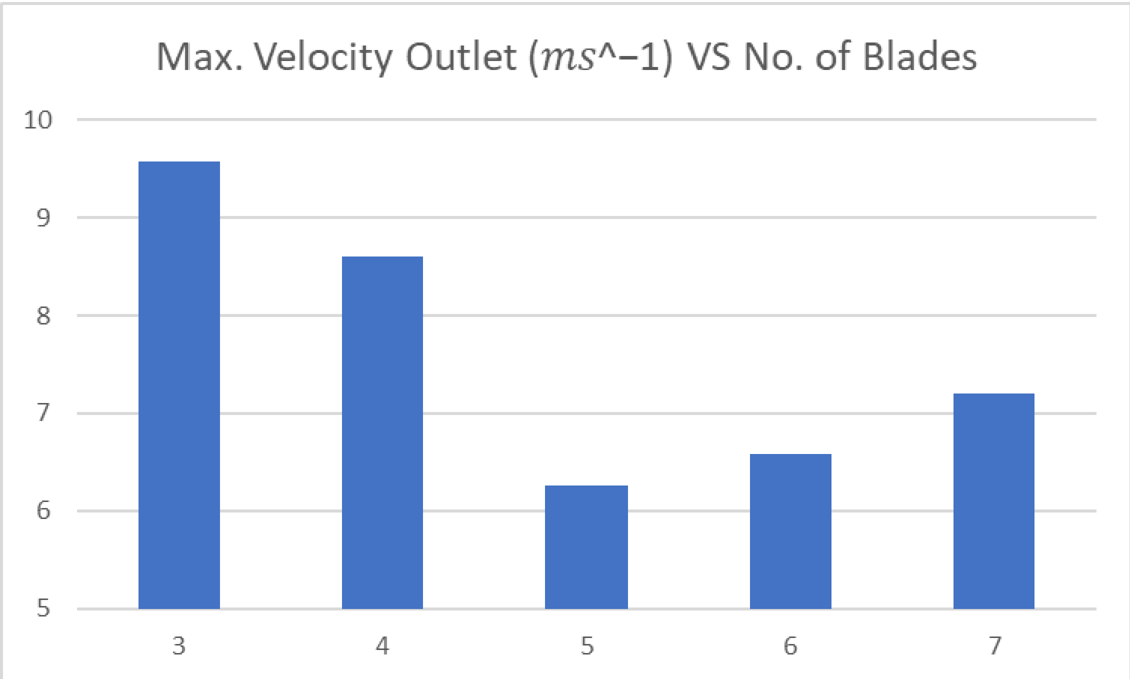


Figure 17: Max Velocity Outlet VS No. of Blades

The highest value for the outlet velocity at 9.57 ms^{-1} which is at impeller with three blades. Meanwhile, the second highest value is 8.59 ms^{-1} which at impeller with 4 blades with 10% decrease compare to the 3 blades of impeller. Now, the third highest velocity value is 7.29 ms^{-1} which is at impeller with 7 blades with 24% less then velocity value at impeller with 3 blades. The fourth highest velocity value is 6.58 ms^{-1} with 31% less velocity value compared with 9.57 ms^{-1}. The lowest value goes to impeller with five blades which is 6.26 ms^{-1} with 34% less then the highest velocity value. This means the impeller with 3 blades has the highest velocity value compared with others and it is the best and suitable for this type of impeller in the scope of velocity.

3.4 Head Generated

In order to get the head generated for each of blade number, we need to calculate the values using equation.

$$\text{Head} = \frac{(Max\ Outlet\ Pressure\ -\ Max\ Inlet\ Pressure)}{\rho\ .\ g}$$

$$= \frac{(1305000\ -\ 755242)}{1000\ .\ 9.81}$$

$$= \frac{(549758)}{9810}$$

$$= 56.04\ \text{m}$$

The head value for the next 4, 5, 6 and 7 blades of the impeller was calculated with the same equation and the results is shown at Table 4.

Table 4: Head Generated Values

No. of Blades	Head Generated (m)
3	56.04
4	48.45
5	44.01
6	50.82
7	55.72

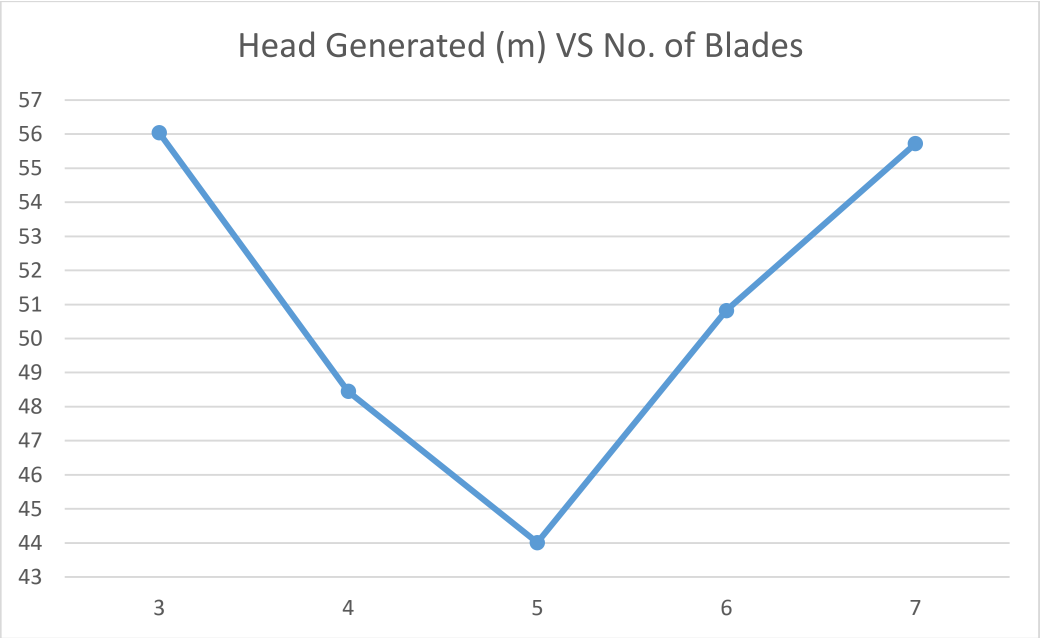


Figure 18: Head Generated VS No. of Blades

4. Conclusion

In this section, we will make conclusion based on the results of tested impeller with various numbers of blades with specific categories which is pressure, velocity and the head generated by the impeller. The highest value for pressure generated is 1340730 Pa which is with the number of blades is 6. 6 blades of impeller can produce higher numbers of pressure compared with others. Meanwhile in the scope of velocity of fluid, the blades of impeller with 3 can produce the highest value which is 9.57 ms^{-1} also with the most the head generated 56.04 m. It is clear that the 3 numbers of blades impeller had better fluid velocity and head generated compared with others. The design and analysis methods are useful to generate performance and flow predictions. Therefore, the design can be optimized to reduce energy consumption, increase pump operating life, and provide better system flexibility. For the future research, we recommend to modify the angle of inlet and outlet for better performance of this impeller. Angle of inlet and outlet can play importance role in deliver fluid transfer in pump with better optimization.

Acknowledgement

The authors would like to thank the Faculty of Engineering Technology, Universiti Tun Hussein On Malaysia (UTHM) for its support.

References

[1] Sujoy Chakraborty, Kishan Chaudhary, Prasenjit Dutta, “Performance prediction of Centrifugal pumps with variations of blade number”, Journal of science and Industrial research, vol.72, June 2013, pp. 373-378

[2] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, “Parametric Study of a Centrifugal Pump Impeller by Varying th e Outlet Blade Angle,” The Open Mechanical Engineering Journal, no 2, 75-83, 2008

[3] Erik Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, “Performance prediction of centrifugal pumps with cfd -tools,” Task Quarterly 5, no 4, 579–594, 2001

[4] Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao and Wang Kai, “Effects of Blade Number on Characteristics of Centrifugal Pumps,” Chinese journal of mechanical engineering, Vol. 23, 2010

[5] T. Brunschwiler et al., “Formulation of percolating thermal underfills using hierarchical self-assembly of microparticles and nanoparticles by centrifugal forces and capillary bridging,” J. Microelectron. Electron. Packag., vol. 9, no. 4, pp. 149–159, 2012, doi: 10.4071/imaps.357

[6] S.Rajendran and Dr.k.Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS-CFX”, International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 3, May – 2012, ISSN: 2278-0181