Progress in Engineering Application and Technology Vol. 2 No. 1 (2021) 882–891

PEAT

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

CFD Simulation to Enhance the Efficiency of Air Compressor

Nur Arif Aiman Md Akhir[1], Mohd Faizal Esa[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.2021.02.01.084
Received 13 January 2021; Accepted 01 March 2021; Available online 25 June 2021

Abstract: The project is about analyse the fluid flow inside the air compressor from one energy to another. For air compressor, the main factor that effect the efficiency is the design of the vane which transfer the amount of energy from the motor to the tank pressure. The simulation analysis of existing air compressor, study can be made to improve the air compressor efficiency on different geometry design. Centrifugal compressor consists of several parts which is impeller, diffuser and volute. One of the critical problems in centrifugal compressor design is the diffuser-impeller interaction. The impeller discharge mixing process appear to be proved experimentally only at low tip speeds, which effect the efficiency. The aim of this project is to study the flow inside the air compressor by analysing the efficiency of air compressor on different geometry design. All the geometry of the model was generated by using SolidWorks 2017 and the numerical solution was run by ANSYS CFX. At the end of study, impeller design with 10 number of blades and 7 axis ratio edge of blade achieves more performance of efficiency compared to other design. The simulation results can be more precise and more details by adding more lines/point of impeller design during the simulation process. However, the simulation process will be time consuming. Therefore, 5 lines/point of this simulation would be enough to produce good result.

Keywords: Compressor, Increase Efficiency, Simulation Analysis, Design

1. Introduction

Centrifugal compressors, known as radial compressors, are a sub-class of dynamic axisymmetric work-absorbing turbomachinery. The idealized compressive dynamic turbo-machine achieves a pressure rise by adding kinetic energy/velocity to a continuous flow of fluid through the rotor or impeller [1]. This kinetic energy is then converted to an increase in potential energy/static pressure by slowing the flow through a diffuser. The pressure rise in impeller is in most cases almost equal to the rise in the diffuser section [2]. Generally, the centrifugal compressor has many advantages, it has high pressure ratio per stage with reasonable efficiency, and high resistance to foreign object damage.

**Corresponding author: faizalesa@uthm.edu.my*

In addition, its production is easy, and it has a smaller number of parts than the axial compressor. The nature of the flow inside the centrifugal compressor is a three-dimensional, turbulent, and viscous flow it represents a challenge to the designer of such part. In addition, existence of many secondary flows represents another difficulty to be added to the design process. The design usually starts by using the basic fluid equations to layout the main dimensions, and then the process continued using 3D CFD solvers to determine the flow parameters and dimensions more accurately [3]. The centrifugal compressor in an automotive turbocharger is required to have a high-pressure ratio, high efficiency, and in particular, wide operating range. Performance improvement has been carried out utilizing computational fluid dynamics and experiments, and further performance improvement through conventional design methods has become increasingly difficult, requiring a significant amount of time for aerodynamic design. Therefore, we developed an optimized design method with computers using a genetic algorithm and artificial neural network in place of the conventional design method. This method was applied to the design of the centrifugal compressor impeller. The performance results of the two designed impellers attained higher efficiency and a significant extension of operating range, respectively, compared with the baseline impeller [4].

2. Literature review

In this chapter, the discussion and explanations a literature review was carried out to gain information and skills necessary to complete this project. The information comes from the journal, the paper, the internet, theories, the previous research and findings are included. With reference from various sources such as journal, thesis, reference books, literature review has been carried out to collect all information related to this project. A compressor is a mechanical device capable of efficiently handling fluids the energy transfer to the fluid medium so that large quantities of it can be delivered at rising conditions of pressure. Compressors have multiple applications that range from household appliances such as refrigerators in the aircraft and processing industries and air conditioners. There are various compressor types, each suitable for a specific application [5]. Moreover, compressors are mechanical devices which are used to in a range of compressible fluids, or gases, boost pressure, of these, the most popular is air. Compressors are used all over the industry, providing air to the shop or instrument for powering air instruments, paint sprayers, and equipment with abrasive blast. As with generators, compressors are split into centrifugal compressors types of and positive-displacement, but where pumps are mainly represented by Compressors are more commonly of the positive- displacement form, centrifugal types. From the fits-in-a-glovebox unit that inflates tires to the tires, they can vary in size from in pipeline operation, giant reciprocating or turbo compressor machines were found. Further, positive-displacement compressors It can be broken into reciprocating forms where the style of the piston predominates, and forms of rotation, such as the helical screw and rotary vane [6].

2.1 Effect of Blade Shape on Performance (Impeller)

Class blades in which it is also possible to categorize centrifugal compressors as backward, radial and forward curved blades based on the shape of the rotor exit blade, as in Figure 4, as shown. To research the effect of the shape of blades One resorts to velocity triangles for efficiency. Besides that, as shown in Figure 5, they have different theoretical head flow relationships with each other. The forward-curved head is the largest, the head features of all the impellers are similar to the backward-curved impeller in actual practice [13]. The figure clearly shows the flow in theory, compressor head rate rises or the pressure ratio for backward-curved blades decreases, for radial exit compressors stays approximately the same, and for forward-curved blades increases. In the petrochemical industry, the majority of impellers are bent radially or backwards [14].

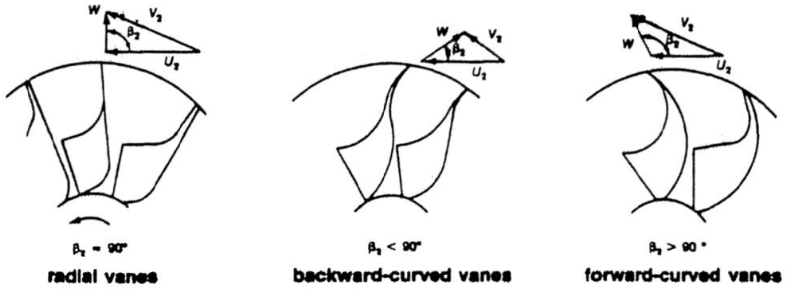


Figure 1: Velocity triangle of various type of impeller balding

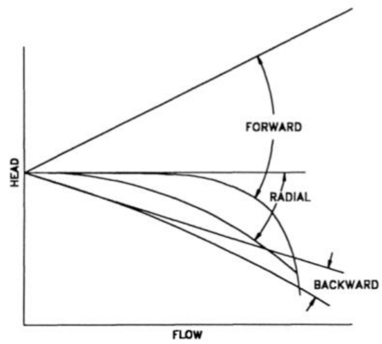


Figure 2: Head and flowrate characteristics of various impeller blading

2.2 Compressor performance

The output curves of the compressor consist mainly displaying a plot. The compressor's variation or pressure ratio of the head and performance at various constant rpm conditions at various mass flows. This will improve the rotor speed, which increases the compressor flow rate. The compressor pressure ratio can be increased at a particular rotor speed mass flow rate by elevating the compressor. That is predictable, as an increase in compressor pressure could be anticipated if flow resistance is available. Otherwise, the compressor like a fan or a blower is more like [5]. In a comparable way, rpm can be accomplished by an improvement in flow rate at a constant resistance reduction at the exit of the compressor at constant delivery pressure, the flow of the compressor can be increased. Moreover, not only could this condition be fulfilled by changing the exit throttle but also by increasing the compressor's rpm. Minimum and maximum acceptable flow rates are called surge rates and at constant rpm choke limits. A line which follows all the stall points of constant rpm lines is call a surge line. Begin with, the locus of the working sections of the compressor at different rpm is call the operating line. That must be the compressor rotor constructed in such a manner in which its efficacy peaks. Near the surge line. When the compressor pressure ratio is full at constant rpm [15].

2.3 Compressor Efficiencies

That's a compressor work there is continuous entropy under the ideal the circumstances. The dotted line marks the actual work conducted. Isentropic efficiency is the compressor in terms can be articulated of the complete alterations in enthalpy.[20].

$$\eta_{ad_c} = \frac{isentropic\ work}{actual} = \frac{(h_{2t} - h_{1t})id}{(h_{2't} - h_{1t})act} \qquad Eq.\ 1$$

Equation can be rewritten for a thermally and calorific ally perfect gas in terms of total pressure and temperature as follows:

$$\eta_{ad_c} = [(\frac{P_{2t}}{P_{1t}})^{\frac{\gamma-1}{\gamma}} - 1] / [\frac{T_{2t}}{T_{1t}} - 1] \qquad Eq.\ 2$$

The process between 1 and 2' can be defined by the following equation of state:

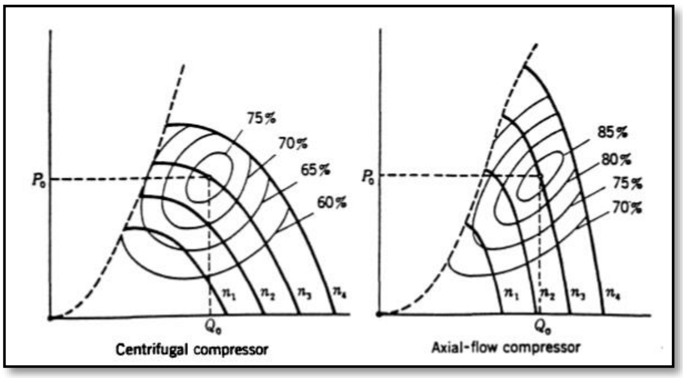


Figure 3: Compressor performance curve

3. Methodology

3.1 Simulation steps conducting

Figure 1 below show the flow chart for simulation setup. The flow shows the important step by step of the process before get the result data. In each step has specific procedure.

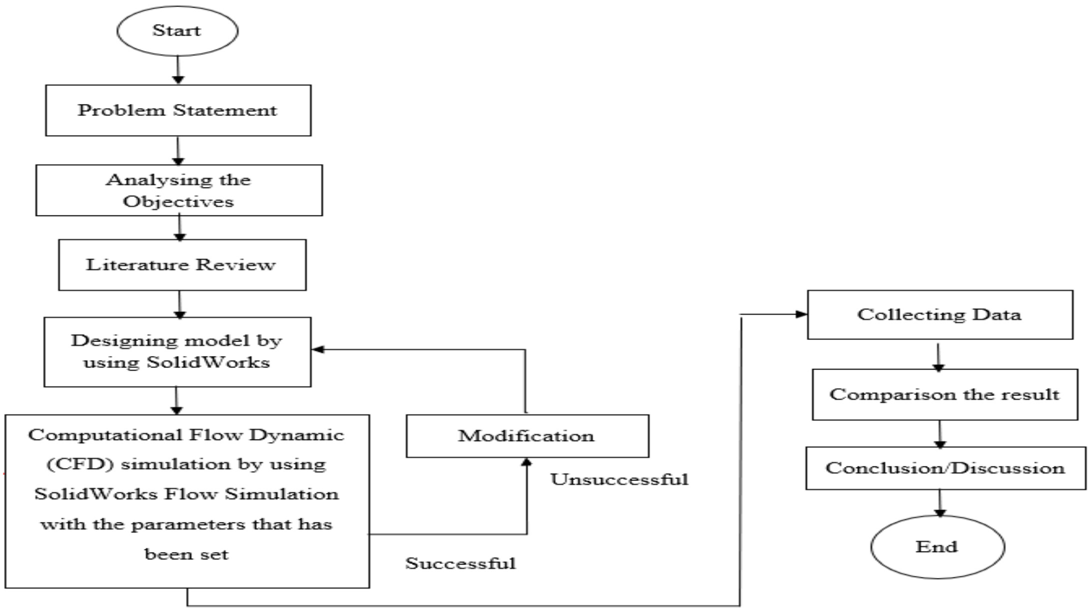


Figure 4: Project Process Flow

3.2 Geometrical modelling

All the geometry of the model was generated by using SolidWorks 2017 and the numerical solution was run by ANSYS CFX. The details on the geometry of the model will be discussed in this section. SolidWorks is a solid modelling computer-aided design (CAD). Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Building a model in SolidWorks usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are added to the sketch to define the size and location of the geometry.

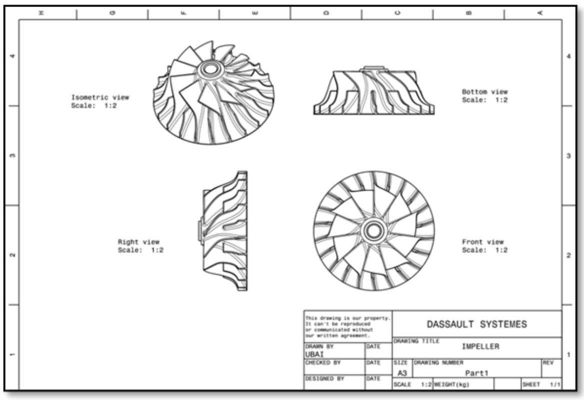


Figure 5: Isometric view impeller design

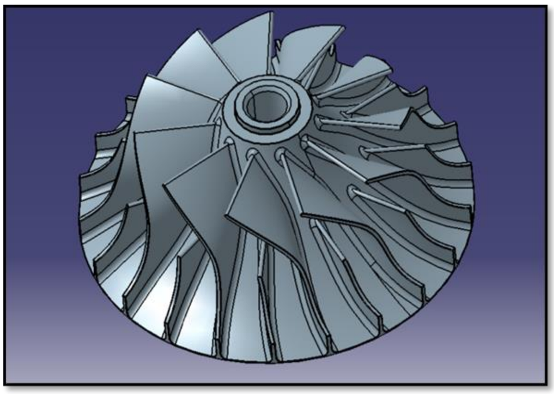


Figure 6: Full 3D impeller design

3.3 CFD Turbomachinery software

Turbomachinery CFD is an open-source CFD software package. It is the final outcome of many years of development of CFD support team of engineers and developers. Turbomachinery CFD was created to enable a quick and efficient design optimization of turbomachinery components. This workflow covers complete process from basic (usually CAD) data over CFD analysis to significant engineering results. Turbomachinery CFD is not dependent on other software but it is fully compatible with standard other software packages. It was originally designed for simulating rotational machines, nevertheless it can be used for a wide range of various CFD simulations. Turbomachinery CFD simulations are handled by CFD Processor. The report can be updated anytime during the simulation run.

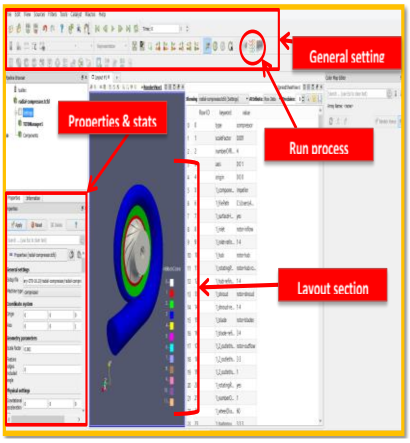


Figure 7: Turbomachinery CFD process

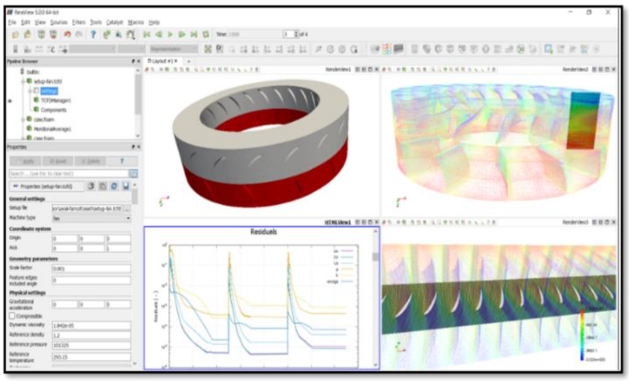


Figure 8: Turbomachinery CFD software setting

3.4 Boundary Condition

It is important to set specific boundary conditions for the computational domain. The computational domain around the model is considered as the actual size. Computation of the impeller main dimensions: hub diameter, suction diameter, impeller diameter, outlet width.

Table 1: Boundary Condition

Inlet		Outlet	
Average inlet velocity	4.7 m/s	Outlet mer. Velocity	3.6 m/s
Average inlet velocity (net)	4.5m/s	Outlet mer. Velocity (net)	3.5 m/s
Inlet Cir. Velocity	0 m/s	Outlet circ. Velocity	13.3 m/s
Inlet rel. velocity	9.9 m/s	Outlet rel. velocity	13.7 m/s

3. Results and Discussion

3.1 Velocity distribution

In the analysis, the nature of the flow field and its structure are visualized and examine by velocity streamlines. Figure 5 shows the representation of velocity streamlines as it flows around the blade. Region of high flow and low flow velocity indicates the differences in the flow characteristic as a result of flow separations phenomena.

Based on Figure 5, high-velocity streamlines can be seen moving at the end of blades. The highest velocity recorded for the simulation is 650 m/s where it occurred most at the end section of blade. It is obvious to see that the flow behaviour becomes more disorganized as the flow passes through the blade's midsection. These phenomena happened as the blade angle changed at the highest value.

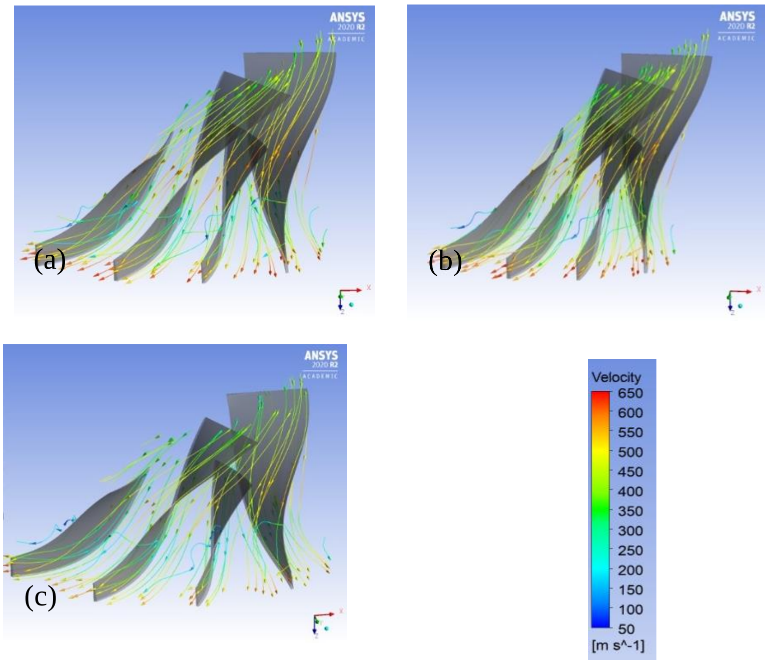


Figure 9: The velocity streamlines at blade (a) 8 blades (b) 10 blades (c) 12 blades

3.2 Number of blade test

The simulation test referring to the number of impeller blade followed by the number of splitters. The number of blades must be in even number and cannot in odd number in order to maintain the balance of impeller as shown in Figure 10.

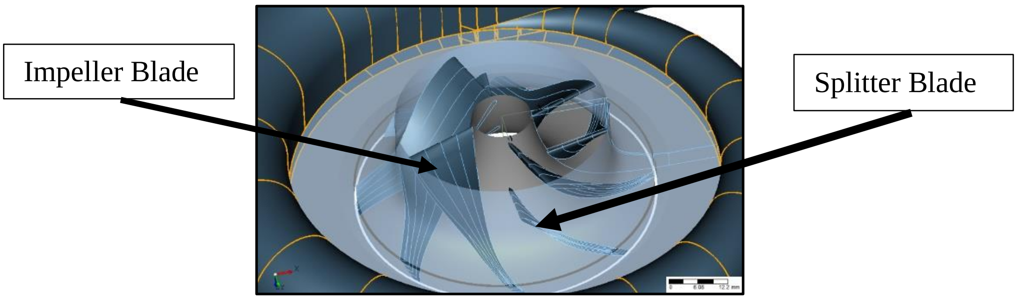
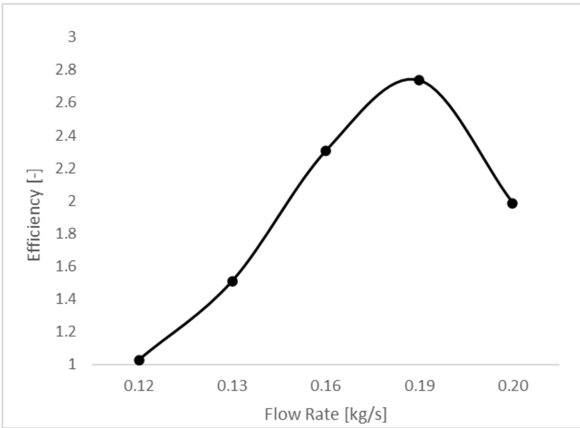
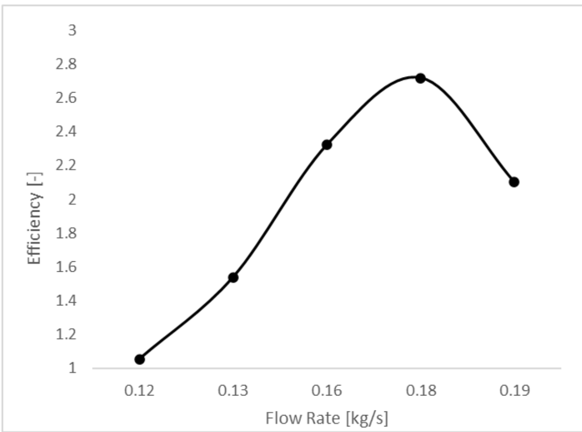


Figure 10: The number of impeller blade and splitter blade

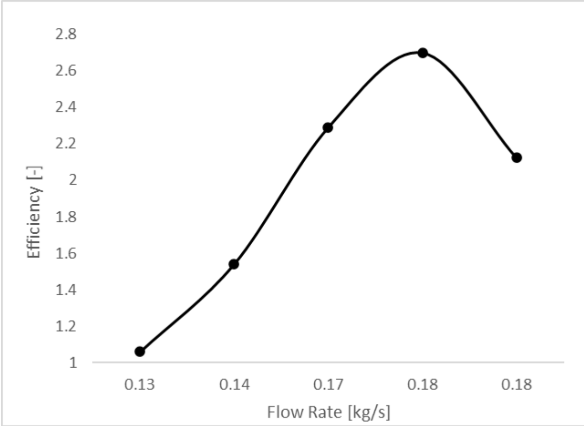
The efficiency of the blade was analysed with different number of blades, which is 8, 10 and 12 blades respectively. The result shown in the Figure 11 of flow rate [kg/s] versus efficiency [-] is based on 5 line/point of simulation process.



(a)



(b)



(c)

Figure 11: The efficiency with different flow rate (a) 8 blades (b) 10 blades (c) 12 blades

Figure 12 shows the result of average efficiency for different number of blades according to the result as shown in Figure 11. The result shows the highest result achieve at 10 number of blades with the efficiency is 1.946768.

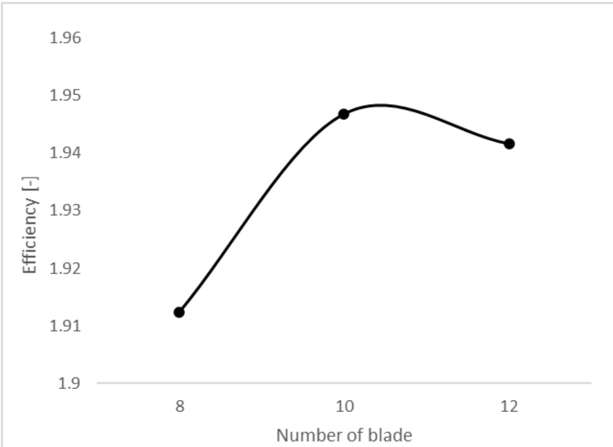


Figure 12: The average efficiency with different number of blades

3.3 Edge of blade test

The simulation test referring to the axis ratio of edge impeller blade. The number of axis ratio is applied on the blade and splitter, the axis ratio is referring the sharp edge of the tip blade. The result show that sharper the edge of the blade the more increase the efficiency because of aerodynamic factors. The test then was analysed with different axis ratio edge of blades, which is 4, 5, 6 and 7 axis ratios respectively and was plotted in Figure 13.

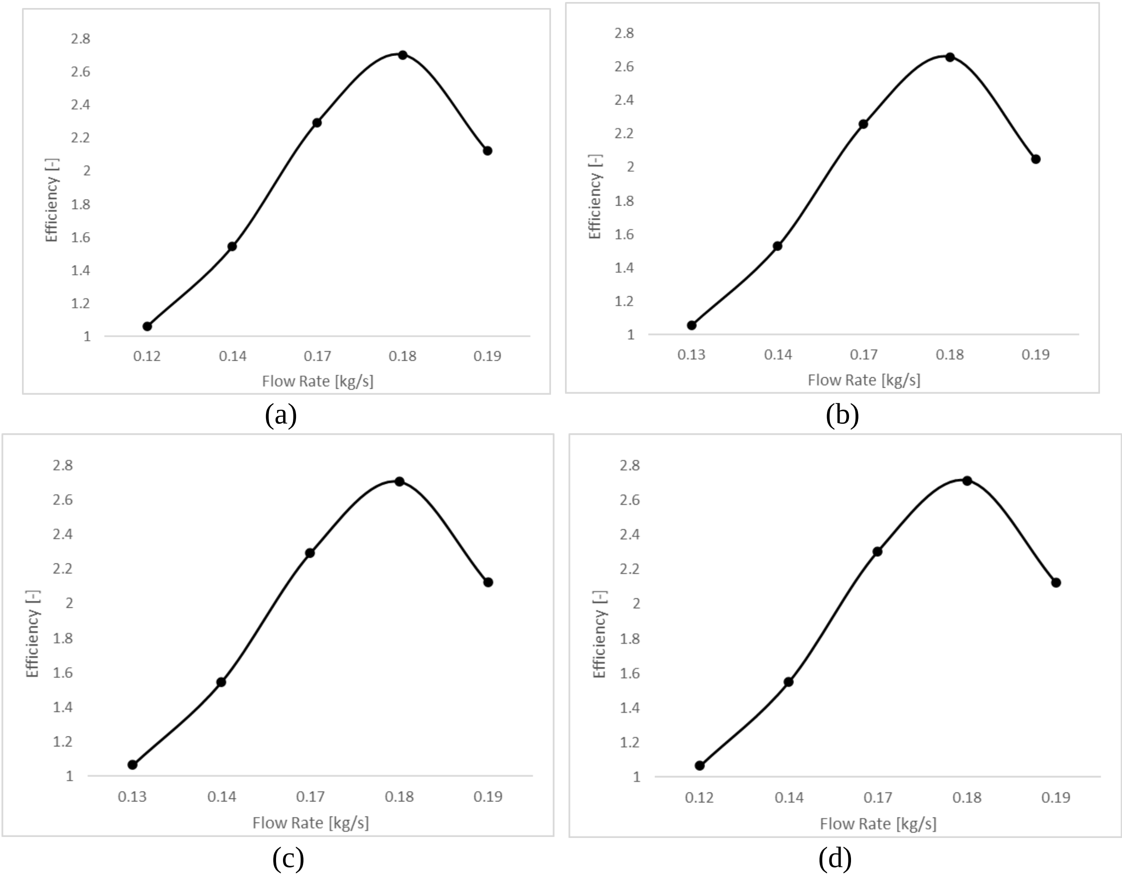


Figure 13: The efficiency with different flow rate (a) 4 axis ratio (b) 5 axis ratio (c) 6 axis ratio (d) 7 axis ratio

Figure 14 shows the result of average efficiency for different number of axis ratio of edge blade according to the result as shown in Figure 13. The result shows the highest result achieve at 7 axis ratios with the efficiency is 1.951316.

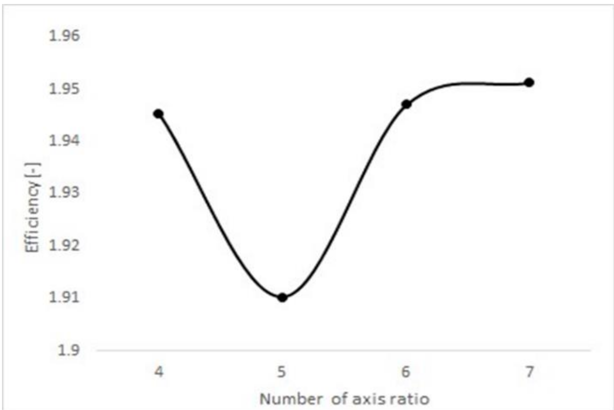


Figure 14: The average efficiency with different axis ratio edge of blades

5. Conclusion

The performance of centrifugal compressor was studied by simulating the several software based on the theoretically in the present research. The design was referred to several factors that can be conclude into this simulation which is the number of blade and the axis ratio of the edge blade. Results showed that the impeller design that has 10 number of blades achieved more performance of efficiency compare to the 8 and 12 number of blades. Meanwhile, with 7 axis ratio edge of blade which is the highest axis ratio tested will also give more performance of efficiency compared to other numbers. The simulation results can be more precise and more details by adding more lines/point of impeller design during the simulation process. However, the simulation process will be time consuming. Therefore, 5 lines/point of this simulation would be enough to produce good result.

Acknowledgement

The authors would like to thank Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Sarbu, Ioan, and Calin Sebarchievici. Ground-Source Heat Pumps: Fundamentals, Experiments and Applications. Academic Press/Elsevier, 2016.

[2] About: Centrifugal Compressor, dbpedia.org/page/Centrifugal_compressor.

[3] Eldalil, Khaled. (2008). Theoretical and Experimental Study of Centrifugal Compressor Impellers Elnashar, H Amr, Eldalil, M Khaled, Hashim, A., Ali, Abdelrahman, M Mohamed

[4] Design and optimization of the impeller on centrifugal compressor in fuel cell vehicle. Guangyu Wul, Sichuan Xul, Yu Wanl.2011

[5] Principles of operation and performance estimation of centrifugal compressors. Dr. Meherwan P. Boyce.2000

[6] Person. “Understanding Compressors.” Types, Applications and Selection Criteria, Thomas, www.thomasnet.com/articles/machinery-tools-supplies/understanding-compressors/.

[7] Bansal, R. K. (2014). A textbook of fluid mechanics and hydraulic machines. New Delhi: Laxmi Publications (P).

[8] Centrifugal Air Compressor Controls and Sizing Basics. (n.d.). Retrieved June 23, 2020, from https://airbestpractices.com/technology/air-compressors/centrifugal-air-compressor-controls-and-sizing-basics

[9] Waleed, A., & Pilidis, P. (2015). Review of Efficiency and Stable Operating Range Enhancements Options of Centrifugal Compressors. Journal of Applied Mechanical Engineering, 04(05). doi:10.4172/2168-9873.1000181

[10] Admin. (2018, February 13). Centrifugal Compressor: Principle, Construction, Working, Types, Advantages, Disadvantages with its Application. Retrieved June 23, 2020, from https://www.mech4study.com/2017/11/centrifugal-compressor.html

[11] Diffusers. (n.d.). Retrieved July 06, 2020, from https://www.sciencedirect.com/topics/engineering/diffusers

[12] Sawyer, J. W. (1966). Gas turbine engineering handbook. Stamford, CT: Gas Turbine Publications.

[13] Centrifugal Fan. (n.d.). Retrieved July 11, 2020, from https://www.sciencedirect.com/topics/engineering/centrifugal-fan

[14] Wood, B. D. (1981). Applications of thermodynamics. Taipei: Southeast.

[15] Moshfeghian, D. (n.d.). How to Estimate Compressor Efficiency? Retrieved July 13, 2020, from http://www.jmcampbell.com/tip-of-the-month/2015/07/how-to-estimate-compressor-efficiency/