THE EFFECT OF NUMBER OF BLADES ON COMPRESSOR CASCADE BLADESPERFORMANCE BY USING 3D CFD

SITI NURAZEERA BINTI AZLI

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

THE EFFECT OF NUMBER OF BLADES ON COMPRESSOR CASCADE BLADESPERFORMANCE BY USING 3D CFD

SITI NURAZEERA BINTI AZLI

A report submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Thermal-Fluids) with Honour

Faculty of Mechanical Engineering

UNIVERSITITEKNIKAL MALAYSIA MELAKA

2017

DECLARATION

I declare that this project report entitled "The Effect of Number of Blades on Compressor Cascade Blades Performance by using 3D CFD" is the result of my own work except as cited in the references

Signature : …………………………

Name : Siti Nurazeera binti Azli

Date : ………………………...

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Thermal-Fluids) with Honour.

Signature : ....

Name of Supervisor: **Dr. Yusmady bin Mohamed Arifin**

Date : ....

DEDICATION

To my beloved mother and father

ABSTRACT

A compressor is one of the mechanical devices in gas turbine engine that compresses air to increases its pressure as well as changing its temperature. In order to design the efficient gas turbine engine, high efficiency of engine is required. Smoothness of airflow in gas turbine has an impact on engine performance. Compressor efficiency is one of the parameter should be considered to maintain that airflow. Hence, compressor blade design was play important role to obtain high efficiency of compressor. A computational investigation has been carried out on the effect of number of blade on 3-Dimensional of compressor cascade blade performance. In order to design the good performance of compressor cascade blade, higher efficiency of cascade is required. It is give benefit to the compressor efficiency, hence produce the efficient turbomachine. The study of flow field is analyzed with help of Computational Fluid Dynamics using the FLUENT software. The flow considered steady with inlet velocity of 30 m/s. NACA 65-206 chosen as blade profile with angle of attack of 10^{o}. The compressor cascade blade is modelled by using SolidWorks software. The blade is twisted. The analysis has been made on different number of blade, namely 3, 5, 7 and 9 with fixed geometry and parameter. The performance of compressor cascade blade have been shown in results of pressure distribution, velocity profile, lift and drag coefficient, pressure different and cascade efficiency. The results have been shown in 2-Dimensional taken at four ratio of position plane. The pattern of velocity profile around the cascade blade and pressure distribution on the surface of cascade blade followed the Bernoulli's principle stated that the velocity on the suction side of airfoil higher than pressure side of airfoil. The acceleration of air creates the lower pressure system and vice versa. Thus, the pattern of pressure distribution is inversely to the velocity profile of cascade blade. Nine blade of compressor have the higher pressure different of inlet and outlet between the blade passages with value of 5.7095 Pa. In order to design the compressor cascade blade, maximum lift and minimize drag should be considered. As result of the effect of number of blade to the lift and drag, it is showed that lift coefficient is higher that drag coefficient for various number of blades. However, five blades of compressor have better result of efficiency of cascade and lift to drag ratio with 75% and 7.857, respectively. It is shown that the five blades of compressor have the best performance compared to the other number of blades.

ABSTRAK

Pemampat adalah salah satu daripada peranti mekanikal dalam enjin turbin gas yang memampatkan udara untuk kenaikan tekanan serta perubahan suhu. Dalam usaha untuk mereka bentuk enjin turbin gas yang cekap, kecekapan tinggi enjin diperlukan. Kelancaran aliran udara di dalam turbin gas mempunyai kesan ke atas prestasi enjin. Kecekapan pemampat adalah salah satu parameter yang perlu dipertimbangkan untuk mengekalkan aliran udara itu. Oleh itu, reka bentuk bilah pemampat adalah memainkan peranan penting untuk mendapatkan kecekapan pemampat yang tinggi. Kajian pengiraan perisian komputer telah dijalankan ke atas kesan bilangan bilah 3 Dimensi prestasi bilah pemampat lata. Untuk merangka prestasi baik bilah pemampat lata, kecekapan lata yang tinggi diperlukan. Ia memberi manfaat kepada kecekapan pemampat , dengan itu menghasilkan mesin turbo yang cekap. Kajian medan aliran dianalisis dengan bantuan Pengiraan Bendalir Dinamik menggunakan perisian FLUENT. Aliran dalam keadaan tetap dengan halalaju masuk 30 m/s. NACA 65-206 dipilih sebagai profil bilah dengan sudut serang 10^{o}. Bilah pemampat lata dimodelkan dengan menggunakan perisian SolidWorks. Bilah adalah berpintal. Analisis telah dibuat ke atas bilangan bilah yang berbeza iaitu 3, 5, 7, dan 9 dengan geometri dan parameter yang tetap. Prestasi bilah pemampat lata telah ditunjukkan dalam keputusan pengagihan tekanan, profil halalaju, daya angkat dan pekali seretan, perbezaan tekanan dan kecekapan bilah lata. Keputusan telah ditunjukkan dalam 2 Dimensi di ambil pada empat kedudukan nisbah. Corak profil halalaju di sekeliling bilah lata dan taburan tekanan pada permukaan bilah lata adalah sejajar dengan prinsip Bernoulli yang menyatakan halalaju di bahagian sedutan airfoil lebih tinggi berbanding bahagian tekanan airfoil. Pecutan udara mewujudkan sistem tekanan yang rendah dan sebaliknya. Oleh itu, corak taburan tekanan adalah songsang dengan profil halalaju bilah lata. Sembilan bilah pemampat mempunyai perbezaan tekanan di antara tekanan masuk dan tekanan keluar yang tinggi dengan nilai 5.7095 Pa. Dalam usaha untuk mereka bentuk bilah pemampat lata, maksimum daya angkat dan minimum pekali seretan perlu dipertimbangkan. Sebagai hasil daripada kesan bilangan bilah terhadap daya angkat dan pekali seretan, ia menunjukkan bahawa daya angkat adalah lebih tinggi daripada pekali seretan untuk pelbagai bilangan bilah. Walau bagaimanapun, lima bilah pemampat mempunyai hasil kecekapan lata yang lebih baik dan nisbah daya angkat dan pekali seretan dengan masing-masing 75% dan 7.857. Ia menunjukan bahawa lima bilah pemampat mempunyai prestasi yang terbaik berbanding bilangan bilah yang lain.

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to the following important people who are supported me, not only during the course of this project, but throughout my Bachelor's degree. Secondly, I would also like to express my gratitude to my supervisor Dr. Yusmady Mohamed Arifin for his essential supervision, support and encouragement towards the completion of this project report and my second examiner Dr. Cheng See Yuan that give comment and judge during seminar which helped me to improve my studies.

Thirdly, I would also like to thanks technician who are in charges of CAE laboratory for allowing me use the laboratory to run my project using the ANSYS software also give the support and solve the solution immediately regarding the computer problem during running the simulation.

And finally, I would like to thank all my friends and family who are sharing their knowledge, understanding the struggles and giving comment to the project in order to make this project are completely finished.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

			Unit
t	-	Thickness	m
a	-	Maximum displacement of camber from leading edge	m
b	-	Maximum displacement from the chord line	m
λ	-	Angle of attack	^{o}
s	-	Spacing	cm
c	-	Chord length	cm
i	-	Incidence angle	^{o}
C_L	-	Lift coefficient	-
L	-	Lift forces	$kg.m/s^2$
C_D	-	Drag coefficient	-
D	-	Drag forces	N
ρ	-	Density	kg/m^3
V	-	Velocity	m/s
l	-	Length	m
η	-	Efficiency	-

LIST OF ABBEREVATIONS

NACA	National Advisory Committee for Aeronautics
CFD	Computational Fluid Dynamics

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND OF STUDY

A compressor is one of major part in a gas turbine engine other than turbine and combustor. It is function to increase the pressure of incoming air before it is enter the combustion section. This component is important to design as it is affecting the efficiency and performance of engine. Lebele-Awala and Jo-Appah (2015) have shown compressor work was increase 30 percent in increasing of ambient temperature for gas turbine power plant analysis. In manufacturing the gas turbine engine, high efficiency is required to obtain the best design of gas turbine engine which produced high thrust and good performance. Gas turbine engine, mostly used for power generation, is a combustion engine that can convert natural gas or other liquid fuels to mechanical energy which drives a generator to produces electrical energy. There are several parameters that affect aerodynamics performance of gas turbine including the compressor compression ratio, combustion inlet temperature and turbine inlet temperature. To achieve the efficient gas turbine, improvement in compressor blade design is essential since overall efficiency of the gas turbine cycle depends primarily upon the pressure ratio of the compressor. In development of the highly efficient axial flow compressor, the study of the two and three dimensional flow through a compressor cascade of aerofoil has played an important role.

Figure 1.1 shown the nomenclature of compressor blade, shaped in aerofoil. The dotted line indicated the camber line, or it is defined as skeleton of the aerofoil. A thickness, t is distributed over the camber line with the leading and trailing edge circles that finally form an aerofoil. ‘a’ is maximum displacement from the leading edge for maximum camber. ‘b’ is the maximum displacement from the chord line. Upper side of airfoil is known as suction surface while lower side is called pressure surface of airfoil.

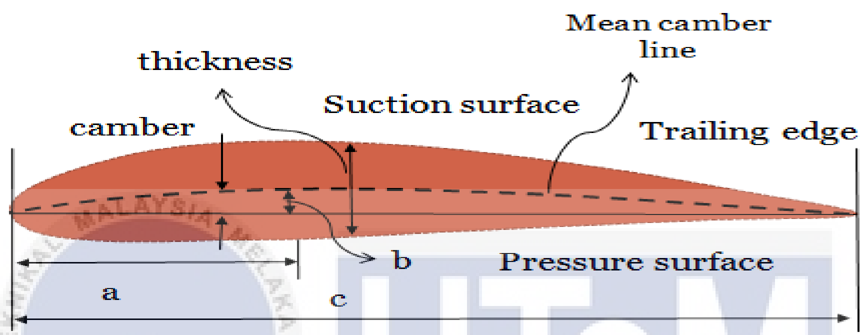

Figure 1.1 Basic nomenclature airfoil for NACA-65 (Pandey, *el at.*, 2012)

Compressor cascade blade, means by set of blades comprises a number of identical blades, equally spaced and parallel to one another defined in Figure 1.2. In modelling the blade profile, the cascade geometry is defined by the aerofoil specification such as angle of attack (λ), spacing (s), chord length (c) and incidence angle (*i*).

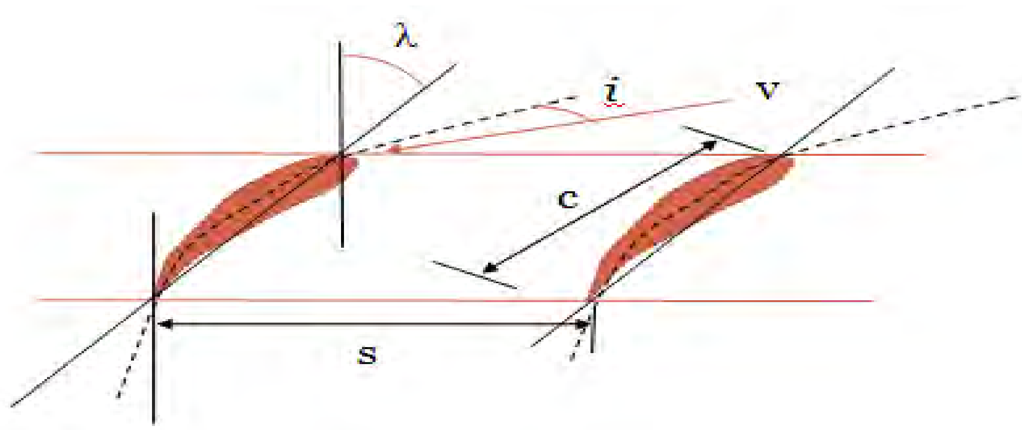

Figure 1.2 Geometrical of cascade airfoil for NACA-65 (Pandey, *el at.*, 2012)

1.2 PROBLEM STATEMENT

High efficiency is required in order to produce gas turbine engine in the most efficient manners. This is to minimize the fuel consumption in the process of converting the mechanical energy to produce the power, which engine efficiency is the ratio between work output and fuel energy input. Smoothness of airflow influenced the efficiency of compressor, means that design of the compressor in gas turbine engine has to be considered. This is to reduce the airflow losses due to friction and turbulence. Thus, shaped of aerofoils are play important role in the compressor to maintain the smoothness of airflow. Instead of incident angle and angle of attack that influence the efficiency, number of blades on compressor cascade blades also has an effect on the compressor performance. The performance of compressor cascade blade will be shown in result of pressure distribution, velocity profile, lift and drag coefficient, pressure different and efficiency of cascade at four position plane in 2-Dimensional for four case study with different number of blades.

1.3 OBJECTIVE

This study presents the investigation of the effect of number of blades on compressor cascade blades performance by using 3D CFD. The aim of this study is to obtain cascade performance for compressor cascade blade with various number of blades. The specific research questions are as follows:

i. How does number of blades affect compressor cascade performance?
ii. How does twisted blade affect the compressor cascade blade?

1.4 SCOPE OF PROJECT

The scopes of this project are:

i. The flow direction of compressor is parallel to the axial, called as axial flow compressor and the blade is twisted. The arrangement of blade in test section is linear.

ii. The geometry of blade such as angle of attack, blade profile, chord, angle of twist is fixed for four case study. Similar to the parameter setting in ANSYS such as velocity inlet, meshing and so on are fixed. The test section with dimension of $50\text{ cm} \times 50\text{ cm} \times 100\text{ cm}$ was used. Thus, Reynolds number and Mach number are same with various numbers of blades.

iii. Realizable k-ε (epsilon) viscous model is used in this analysis and ANSYS Fluent code as the type of solver preference.

iv. Then working fluid is air at 15ºC and the flow is considered in steady state and incompressible flow.

v. The performance of cascade blade is shown in pressure distribution, velocity profile, lift to drag ratio, pressure different and efficiency of cascade. The losses of energy and 3D flow such as secondary flow are negligible.

CHAPTER 2

LITERATURE REVIEW

2.1 TURBOMACHINERY

2.1.1 Basic of Gas Turbine

Gas turbine has been used in aerospace and industrial applications in many years. Mostly, gas turbine engine drives for power generation and power aircraft. Figure 2.1 depict the gas turbine and how the device works. Development of gas turbine can be divided in two field of technology, namely steam turbine and internal combustion engine. Recent research on Gas Turbine (Hackert, 2014) has shown the first steam turbine was built by Sir Charles Parsons and on 1903, gas turbine thesis are established by Dr. Sanford Moss .Year by year, gas turbine is upgraded and it is accomplished to installed at Oklahoma for electric power generation purpose.

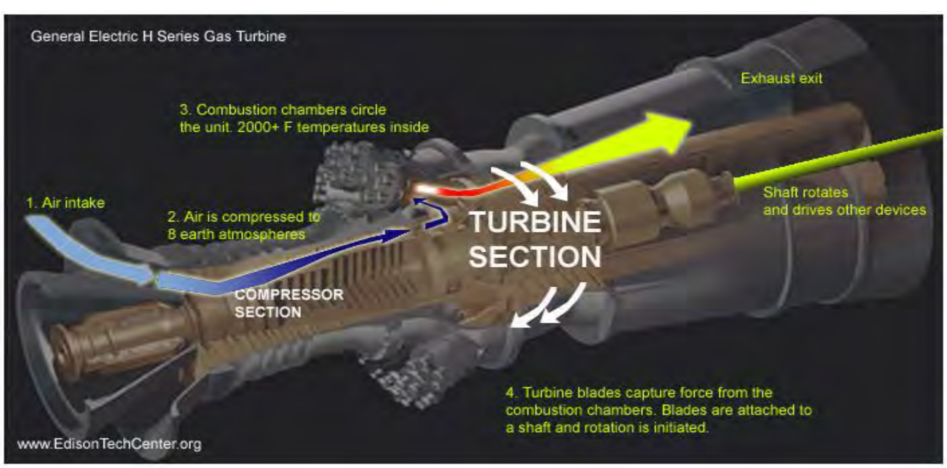

Figure 2.1 Gas turbine and devices work (source: Edison Tech Center website)

The compressor required the power provided by turbine to compress the air in the engine compressor. Turbine and compressor are the main component in the gas turbine. Otherwise the combustor, function to be combustion section by burning the large amount of fuel and air to release the heat. The turbine and compressor have different functional, where turbine was extracted the work from the flow while compressor was supplied the air at high pressure. In the other words, the inlet air at ambient pressure is compressed to obtain the air at high pressure.

2.1.2 Performance of Gas Turbine

Kurz (2005) stated that gas turbine performance is influenced with several ambient conditions such as ambient temperature, inlet and exhaust pressure losses, fuel, ambient pressure and relative humidity. The power turbine reduces as more work required to increase the pressure at higher temperature inlet. With increasing the temperature will lessen the pressure ratio of the compressor at constant speed. Therefore, component efficiencies in gas turbine have an effect on changes in the ambient temperature. As well as ambient pressure, this condition give impact on the power output when the air density is decreased resulting on the impact of operating the engine at lower ambient pressure. Other than that, it is found that increasing the tip clearances, changing in airfoil geometry and airfoil surface quality has major effect on compressor performance. Anoop and Onkar (2014) have shown that there is a change in specific work output, thermal efficiency and saving of fuel when the compressor inlet temperature decreased from 318K to 282K which is 10.12, 3.45 and 3.43 percent, respectively.

Arangi, *el at.* (2015) has analysed that the efficiency and power output has dependency on inlet air temperature of compressor. The compressor will produces high temperature of air as increasing of pressure. Low inlet temperature is advantages as it is give the maximum of efficiency and maximum of power output of gas turbine. Hence, the cooler is provided to lower the temperature also give the better performance in the gas turbine. Naeim, *el.at*. (2013) have been investigated the effect of ambient temperature on the performance of gas turbine power plant by comparing the ambient temperature for two years. The result shows variation of efficiency and electric-power output of gas turbines has an impact on electricity production, fuel consumption and plant incomes.

2.2 AXIAL FLOW COMPRESSOR

2.2.1 Introduction of compressor

Axial flow compressor as shown in Figure 2.2, where the air entering the axial compressor is parallel to the axis of shaft. Pressure ratio and temperature act as the basic parameter that measure the efficiency of gas turbine. Table 2.1 shows the axial flow compressor characteristic for two applications, namely industrial and aerospace. Both are used the compressor for power generation (Boyce, 2011). Axial flow compressor has an advantage of potential for higher pressure ratio, higher efficiency and larger flow rate possible at given frontal area.

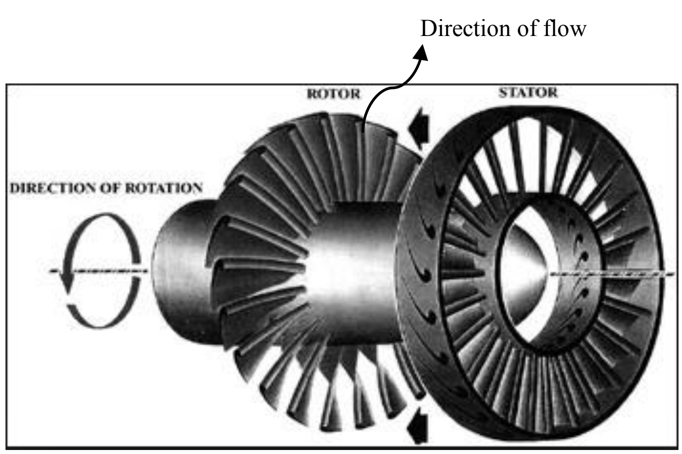

Figure 2.2 Isometric view of an axial flow compressor stage (source: IEEE website)

Table 2.1 Axial Flow Compressor Characteristic (source: Gas Turbine Engineering Handbook)

Type of Application	Type of Flow	Inlet Relative Velocity Mach Number	Pressure Ratio per stage	Efficiency per stage
Industrial	Subsonic	0.4-0.8	1.05-1.2	88% - 92%
Aerospace	Transonic	0.7-1.1	1.15-1.6	80% - 85%

According to Table 2.1, both type of application for gas turbine have different characteristic in term of type of flow, Mach number, pressure ratio and efficiency. This is because they depend on the gas turbine utilization as industrial more to generate electricity, while aerospace is for power aircraft.

2.2.2 Compressor cascade blades

Many scientists have done their research on cascade wind tunnel and publish their research. Some are collected as reference for this project. Compressor cascade blade profile was designed and it is analysed by Pradhapraj, *el at.* (2016).The model was designed based on two dimensional flows through a cascade of aerofoil study. Testing cascade model will provide better result and good operating condition compare to single blade. Difference angle of blade with various angle of attack of compressor cascade have been done to obtain the variation of pressure in order to measure the efficiency of compressor cascade. However, the experimental analysis result was compared to the numerical analysis which carried out using Computational Fluid Dynamics. Both analysis shows that pressure distribution increased with increase angle of attack. Thus, efficiency increased. In future, the researches expected increasing the number of blades may be improving the efficiency.

Mohsinali and Vimal (2014) has been optimized the number of blades in high pressure compressor. In designing the compressor, three major design parameter is important including the number of stages, rotational speed and number of blades. By assumed the rotational speed 6000 to 40000 in range with same boundary condition for 2000 modules, the result shows the best module with 91 percent of efficiency and 11400 rpm for rotational speed. With help of Axstream software, number of blade has been selected in rotor and stator. This number of blades helps the researcher to understanding some parameter such as outlet pressure, enthalpy, temperature and velocity. Efficiency of compressor is measured in term of energy losses. Losses occur during the air compression in the compressor, it might be due to friction and flow separation. Heat loss created by high loaded and higher pressure ratio compressor is the main losses in this study. Thus, it is shows that increasing the number of blade in rotor would increase the efficiency as well as power requirement.

Cascade blades are defines as number of blades is assembled in parallel or annular with similar stagger angle and pitch to one another of blades. The flow is deflected through the cascade blades due to the loss in stagnation pressure. Research found that the performance of compressor cascade blade is depending on exit angle with optimum inlet angle and stagnation pressure loss across on it. Rhoden, *el.at.* (1942)has been measured the distribution of static pressure over the central cross section of the middle blade, traverse static pressure and angle inlet and outlet flow in the plane of the central cross section in experimental. The different camber angle of axial flow cascade compressor blades with same chord/pitch ratio and same stagger angle is used in the experiment to obtain the result on effect of Reynolds number on inlet and outlet air angles, efficiency of blade against Reynolds number and graph of static pressure over the surface of the middle blade. Incidence flow in cascade has been the important parameter in process of design turbomachine blades. This has been investigated by

Zenouze, *el at.* (2015) in his studied between numerical and experimental of effect on surface distribution of axial compressor blades. The changes in incidence blade may due to rotational speed of the compressor and inlet flow condition. This proved by seen the result of pressure coefficient on suction and pressure side have slight different with comparison between two values of Reynolds number.

Salim (2013) has reported that Reynolds number, incidence angle and blade angle of cascade influenced the aerodynamic performance of axial compressor. It is results in graph plot of variation of angle blade and angle of incidence at two Reynolds numbers where incidence angle decrease with blade angle. In fact, blade angle are determined with different of cascade. In addition, static pressure rise coefficients, total pressure loss coefficient, drag and lift coefficient is observed as behaviour of aerodynamic flow parameter in blade cascade. At lower Reynolds number, significant changes in variant of static pressure rise and pressure loss with different in blade angle whereas invariant is shown at higher Reynolds number. Lift coefficient seem to decrease against the blade angle and incidence angle meanwhile drag coefficient decrease at zero degree of angle of blade with lower degree of incidence angle for lower Reynolds number. Thus, aerodynamic parameters are related to obtain the axial cascade of performance.

Research has been observed the relation between angle of attack with lift coefficient and drag coefficient (Boyce, 2011). It is shown that lift coefficient increased with variation angle of attack. On the other hand, drag coefficient will decrease. However, at some point stall happened when exceed the angle of attack and drag will start to increased. Thus, optimum angle of attack has to define in order to maximize lift coefficient and minimize drag coefficient. Sathiyalingam, *et al.* (2012) are analysed the flow properties of compressor by taking the static pressure distribution over the compressor cascade blade. Experimental is conducted and then the results of pressure distribution are compared with the simulation approach by using analysis software. Losses are estimated using theoretical method as aerodynamic losses occur in cascade blades analysis. Profile losses are expressed in term of drag coefficient as it is total loss in stagnation pressure across the cascade blade.

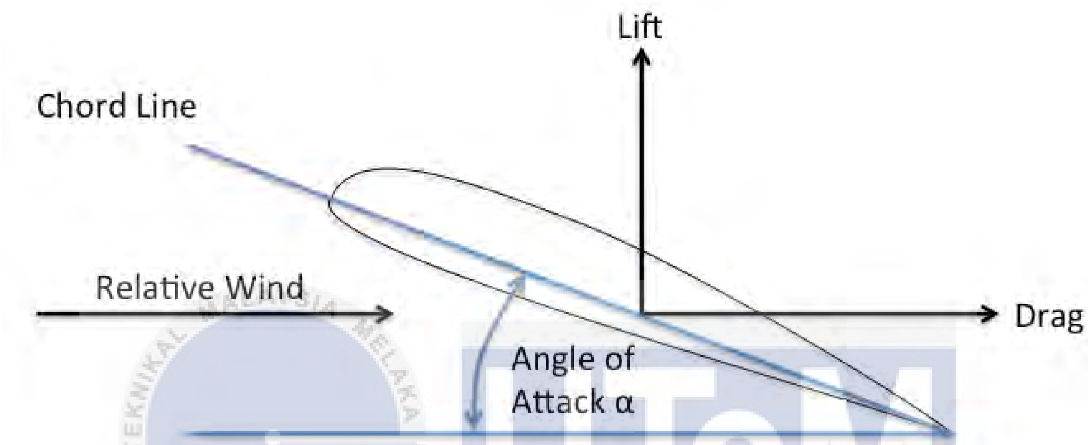

Figure 2.3 Lift and Drag Forces illustrations (source: code7700 website)

According to the Rathakrishnan (2013), lift and drag forces acting on the blade due to the pressure distribution on pressure side and suction side of aerofoil. The lift force is perpendicular to the direction of air flow, meanwhile drag force is the component of forces acting opposite of direction of motion as shown in Figure 2.3. Sullivan (2010) stated that the lift force said to be occur when the pressure side of aerofoil have a higher pressure than suction side. The friction between air molecules and surface of aerofoil produce skin friction drag. Theoretically, the coefficient of lift and drag can be measured as equation (2.1) and (2.2) .The lift coefficient equation is given by

$$C_L = \frac{L}{\frac{1}{2}\rho V^2 l} \quad (2.1)$$

And, the drag coefficient equation is given by

$$C_D = \frac{D}{\frac{1}{2}\rho V^2 l} \qquad (2.2)$$

Turek (2010) has been studied the effect of flow field in the compressor cascade blade NACA 65-100. It is done by using ANSYS Fluent software to compute the results. The effect has been shown in static pressure contours and velocity magnitude. With obtained the cascade data from previous studies, two section of flow, transonic flow with some characteristic and transition model consists of laminar and turbulent flow is analysed.

2.2.3 Compressor cascade performances

Determination of compressor cascade performance is the main objective in this study. In turbomachinery, compressor is function to supply air at high static pressure. Hence, the air flows from a low static pressure at inlet to higher static pressure at exit within the compressor blades. Velocities, flow at exit angle and stagnation pressure loss are the vital parameter that measured the performance of cascade. The deflection of flow occurs due to loss in stagnation pressure across the cascade blades and increment of entropy. The performance of cascade has an effect with losses. Total pressure losses changes in variation of incidence angle. Angle of incidence is formed in between the direction of flow motion and chord line profile, also known as angle of attack. Another loss occurs in an axial compressor is viscous losses, shock losses, mixing losses and 3-D effect on cascade blades. Viscous losses consist of profile losses, annulus losses and end wall losses. Meanwhile, Secondary flows and tip leakage flows will be discussed in 3-D effects. Many researches are studied in 2-D cascade losses in compressor blades which only involve profile loss, wake mixing losses, shock losses and trailing edge loss. In previous studied the performance of axial compressor cascade blades is determined with affect the surface roughness on it.

Jafar, *et al.* (2015) on his research was studied the performance of cascade in 2-D flow which is involved the profile loss. With the variation of stagger angle and blade roughness, these are against the drag coefficient and lift coefficient in graph plotted. Besides, efficiency of blades decreases with variation of stagger angle. Results shown the pitching moment coefficient suddenly increased to higher values at critical stagger angle due to occurrence of the flow separation at blade suction side. Hence, performance of compressor cascade blades is limited by the growth and separation of blade surface boundary layer.

Back, *el at.* (2010) are observed the profile loss in compressor cascade affected on surface roughness and the deviation flow in compressor cascade. This result of experiment is shown in blade pressure distribution, deviation distribution and loss coefficient. Concluded that loss is the sensitive parameter to changes in blades roughness. On the other hand, the similar research has been studied by Back, *el at.* (2012). In this case study, Reynolds numbers have been changes to be varies. The result of performances for both case studies is presented in term of loss coefficient, static pressure coefficient and blade loading. It seems, the profile loss depend upon Reynolds number, pressure gradient, Mach number and the resulting of unsteady boundary layers. The blade parameter that affected the flow are stagger angle, solidity and blade roughness. Three design blades was compared in this study, namely original blade, twist blade and twist/stagger blades.

Chun, *el at*. (1981) has found that the performance of transonic compressor has been improved by reduction of three degree blade of twist/stagger design blade. Chun said that, the distance between trailing edge and maximum chamber has been move to downstream due to changes in stagger of angle.

2.3 CFD analysis

Sheikh, *el at.* (2015) have been analysed the compressor blade with the help of Computational Fluid Dynamic software. Two dimension of blade is created by using CATIA to study the flow behaviour through the linear cascade blade. Contour of pressure distribution is appeared after meshing done. Also, three dimensions is created purpose of comparison. The computational results have been showed that static pressure is lower at suction side of leading edge of aerofoil in two dimension of cascade blade. Meanwhile, the different result obtained for three dimensions where the critical pressure is occurred at inlet of the blade when the flow passes through the blades.

Pandey, *el at.(*2012) found that the static pressure for a compressor cascade increased along the cascade airfoil inversely to the velocity, which decreased in varying angle of incidence. The analysis of compressor cascade flow has been done by using CFD. The results have been shown that +2 to +6 degree of incidence angle was advantageous.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter describes the methodology in present study to obtain the cascade performance data for four case studies with different number of blades of compressor cascade. The geometry of each blade were fixed and identical for four case studies. In the present study, SolidWorks software is used for modelling the compressor cascade blades in three dimensions. ANSYS 16.0 CFD package software is used for the flow simulation. 3D analysis type is used since the geometry is three dimensional and Fluent is used as the solver preferences to simulate the flow behaviour in the cascade compressor blades.

3.2 RESEARCH METHOD

The flow chart shows in Figure 3.1 enable the investigation are organized well. According to the flow chart, the title is determined at first. Then, the aim and scope of project is clarified. The purpose of this study should be clear and recognized. In order to obtain the knowledge about this study, the literature review is studied. Reliable journal, conferences, trusted website and book is referred as references on this study. This is because there a lot of previous study regarding this project even are not similar title are conducted. More than 20 references from previous study is referred to understand this study. The scope of references study included experimental, numerical and simulation analysis. The data for this study is collected and is compared with other data from different collection of study. The three dimension compressor cascade blades is modelling by SolidWorks, and then imported to the ANSYS Design Modeller in geometry. Meshing is done to separate the domain into number of cell. The boundary condition, material, model and solution method is setting up in solver purpose to run the simulation. Fluent is chosen as type of solver that used in this analysis. According to the Figure 3.1, the analysis of data will be run repeatedly if the result are not achieved the objective. The results of pressure distribution, velocity profile, lift and drag coefficient, also efficiency of blade cascade are discussed on discussion section. The comparison between result analysis is determined.

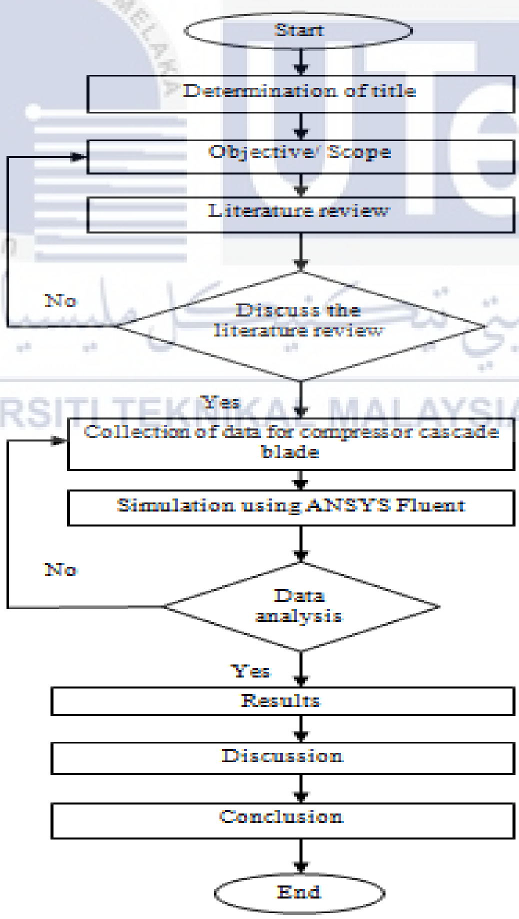

Figure 3.1 Research Method Flow Chart

3.3 BLADE PROFILE

In this study, NACA 65-206 Blade Profile, shown in Figure 3.2 is chosen as a fixed blade profile to compare the performance of cascade with different number of cascade blades (Airfoil Tools, 2017). The selection of blades profile is made according to the general quantities of blades profile; the blade maximum thickness; the position of the blade maximum thickness; the chamber line; the position of the blade maximum chamber and the thickness of the leading and trailing edges. Instead of five quantities, the position of the maximum thickness and maximum chamber are considered as it is the major variables in controlling the aerodynamics performances. Thus, the position of maximum chamber 50 per cent chord from leading edge and the position of maximum thickness 40 per cent chord of the leading edge of blades profile is selected as it is the best that improved the performances of blades (Carter, 1957). Modifications of blade profile and blade shape have been observed able to improve in efficiency of gas turbine (Katoh, el at., 1996). Inlet flow will give the different of performance when it comes to higher subsonic region and transonic range. Shock wave can be avoided, pressure lost in the blade rows can be minimized and thus total efficiency can be increasing, if velocity on the suction side is increased. Besides that, the advantages of 6-series of aerofoil are it is high maximum lift coefficient and having a very low drag over a small range of operating conditions other than optimized for high speed. But, this 6 series has poor stall behaviour and has disadvantages for roughness. Most application using this aerofoil series is jet. For example business jets, supersonic jets, jet trainers, also piston-powered fighters.

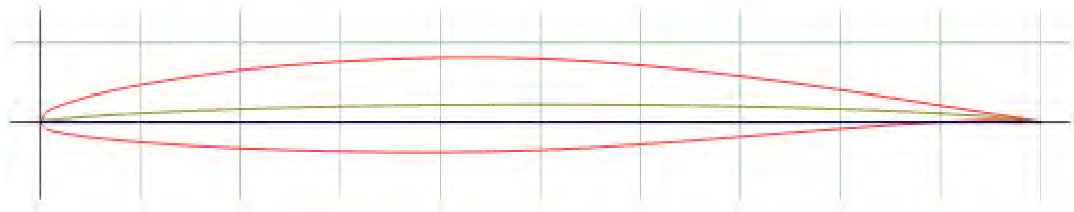

Figure 3.2 NACA 65-206 Blade Profiles (source: Airfoil Tools website)

Each blade in cascade has similar cascade geometry such as chord length, aspect ratio, angle of attack, solidity and spacing. NACA 65 series, which is NACA, stands for National Advisory Committee for Aeronautics. The shape of the NACA aerofoils is described using a series of digits following the word "NACA". For this case, 6 denoted the series and indicated it has greater laminar flow. The next digit is 5, show minimum pressure in tenths of chord (0.5c). Digit 2 after the dash is a minimum lift coefficient design and the two last digits indicated the maximum of thickness in percentage of chord. In details, the chosen NACA has 6 per cent of maximum thickness at 40 per cent of chord value and 1.1 per cent of maximum camber at 50 per cent of chord value.

3.4 CASCADE GEOMETRICAL DATA

Cascade geometrical data is one of the important things to consider in the process of designing the blades. Cascade data being the measurement to define the variation of losses, pressure rise, surface pressure distribution and outlet flow angle in such experimental and computational analysis. Thus, the cascade data in this case study should meet the aerofoil selected. The geometric parameter of cascade is listed in Table 3.1.

Table 3.1 Geometrical cascade used in present study

Geometry	Data			
Number of Blade	3	5	7	9
Spacing (cm)	12.5	8.33	6.35	5.0
Blade solidity (c/s)	0.8	1.2	1.6	2
Aspect ratio (S/c)	5			
Chord length(cm)	10			
Angle of attack	10º			
Spanwise (cm)	50			
Twisted angle	18º			
Tip clearance (cm)	0.1			

Since no experimental axial compressor cascade blade data is available in the open literature related to the effect of number of blades on compressor cascade performance, therefore similar experimental investigation regarding the axial compressor cascade performance is referred. Even the case study is unrelated to the present study, yet the objective of the case study is similar which is to obtain the performance data of axial compressor cascades blades. The geometry parameters used in this study are determined by referring to the previous study. The chord length and blade height are 10 cm and 50 cm, respectively. The fluid domain is 100 cm in length, 50 cm in height and 50.1 cm in width. The additional of 0.1 cm for width of domain because blades are bounded by end wall with tip clearance of 0.1 cm. The clearance gaps used to prevent the physical contact between rotating tip blade and wall in axial compressor. According to the Jingjun, *et al*. (2013), it is show that 1% of chord length is better than 1.5% and 2% for the tip clearance. This because at 1.5% and 2% has no improvement in total loss coefficient even though the strength of the tip leakage is reduced. The measurement of test section depends on the size of blades. The spacing between the blades is decreasing with addition of number of blade since the height of test section is fixed. Blade solidity is the ratio of blade chord length to spacing. The compressor cascade blade has an aspect ratio of 5 with 10 degree of angle of attack. Aspect ratio is defined as the ratio of blade height to the chord length. Typical angle of twist for compressor blade are ranges in 10 to 20 degree. Thus, 18 degree is selected for blade twisted. For this case study, the Reynolds number based on the inlet velocity, the blade chord and the kinematic viscosity of air at 15^{o} C is 2.046×10^{6}. This is calculated by using general formula of Reynolds number. The flow is considered turbulence flow. The Mach number computed with the inlet velocity and the speed of sound in air at 15^{o} C is 0.09 and therefore, the flow is considered steady and incompressible. The velocity of air enter the channels is 30 m/s. The test section is considered as low speed with subsonic flow.

3.5 COMPUTATIONAL FLUID DYNAMICS METHODOLOGY

In present, the three dimension cascade compressor model are designed using the SolidWorks and the analysis is carried out using CFD Fluent software. Figure 3.3 show the typical CFD methodology flow chart. This method is repeatedly used for each case study with different number of blade. There are four phase in order to obtain the result. The first step is to identify the problem. The problem is identified based on objective and scope of this project. Then, blade model, meshing and set up the parameter is done under pre-processing. The simulation is run in solver to compute the solution. Finally, result is shown in post processing. The process is return to the pre-processing if the results are not achieved the target.

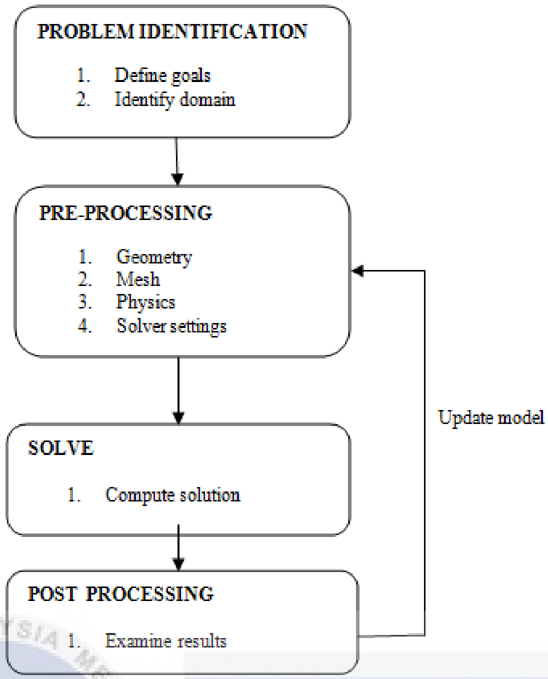

Figure 3.3 CFD Flow Chart

3.5.1 Modelling the cascade blade

Figure 3.4 shows the coordination data of NACA 65-206 (Airfoil Tools, 2017). It is then imported to the SolidWorks to begin the sketching of blade profile. For instance, three dimension of compressor blade model is shown in Figure 3.5. The blade is twisted. The solid cascade blade from SolidWorks is imported to the ANSYS modeller. Test section is constructed. According to the dimension, the test section is extruded. Then, Boolean operation is selected to the test section as target bodies and the cascade blade as tool bodies by using subtract operation. Both cascade blade and test section is adding frozen for their operation so that can work on them individually. The process of modelling the cascade blade in ANSYS modeller is similar for each case study.

NACA65-206 - Notepad

File Edit Format View Help

```
1        0        0
0.95009  0.00511  0
0.90018  0.01027  0
0.85024  0.01538  0
0.80027  0.02029  0
0.75028  0.02489  0
0.70026  0.02907  0
0.65022  0.03276  0
0.60016  0.03589  0
0.55009  0.03836  0
0.5      0.04003  0
0.4499   0.04078  0
0.39981  0.04069  0
0.34971  0.03982  0
0.29962  0.03824  0
0.24953  0.03592  0
0.19945  0.03277  0
0.14939  0.02869  0
0.09936  0.0234   0
0.07437  0.02012  0
0.04939  0.01625  0
0.02444  0.0114   0
0.012    0.00822  0
0.00706  0.00642  0
0.0046   0.00524  0
0        0        0
0.0054   -0.00424 0
```

Figure 3.4 Coordination data of NACA 65-206 (source: Airfoil Tools website)

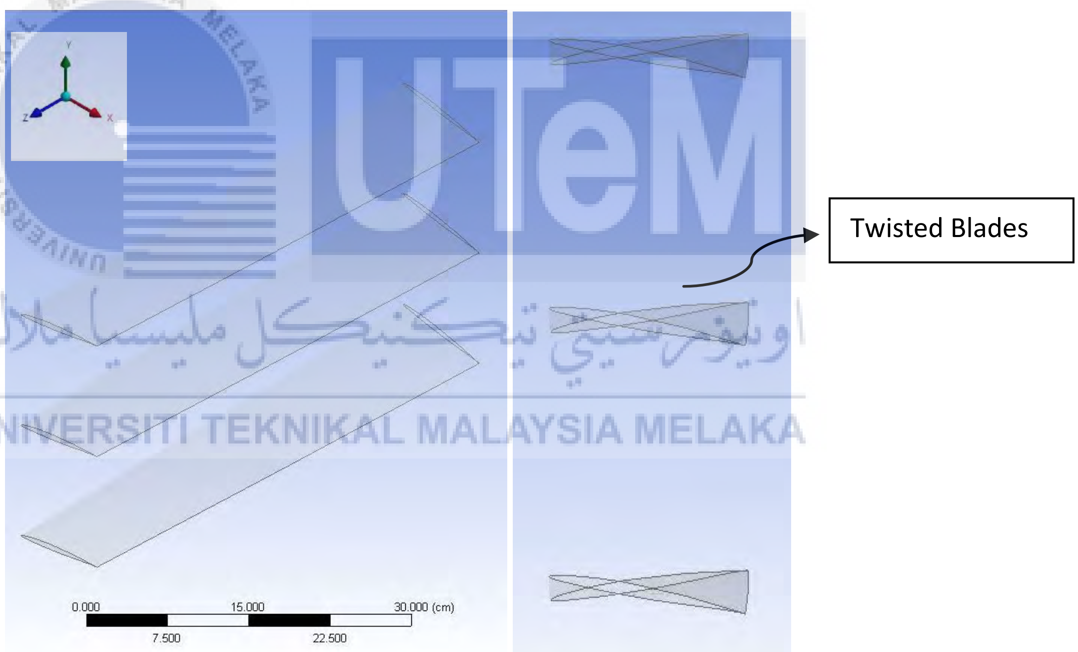

Figure 3.5 Compressor cascade blade with three blades

3.5.2 Meshing

Meshing is generated after model the compressor cascade blade in ANSYS modeller. The purpose of mesh generation is to allow the analysis of flow behaviour in the domain. Some shape are formed on the computational domain selected to be mesh such as triangle, quadrilaterals, tetrahedral and hexahedra. The domain is converted into sub-domain when the element is divided into many of cell. The shape of three dimensional compressor cascade blade is tetrahedral. During the meshing process, names selection is added for the surface of body involved. The names selection namely as inlet, outlet and wall. The purpose of naming is to define the boundary condition in the fluid solver. Figure 3.6 shows the meshing generation and boundary condition for three blades.

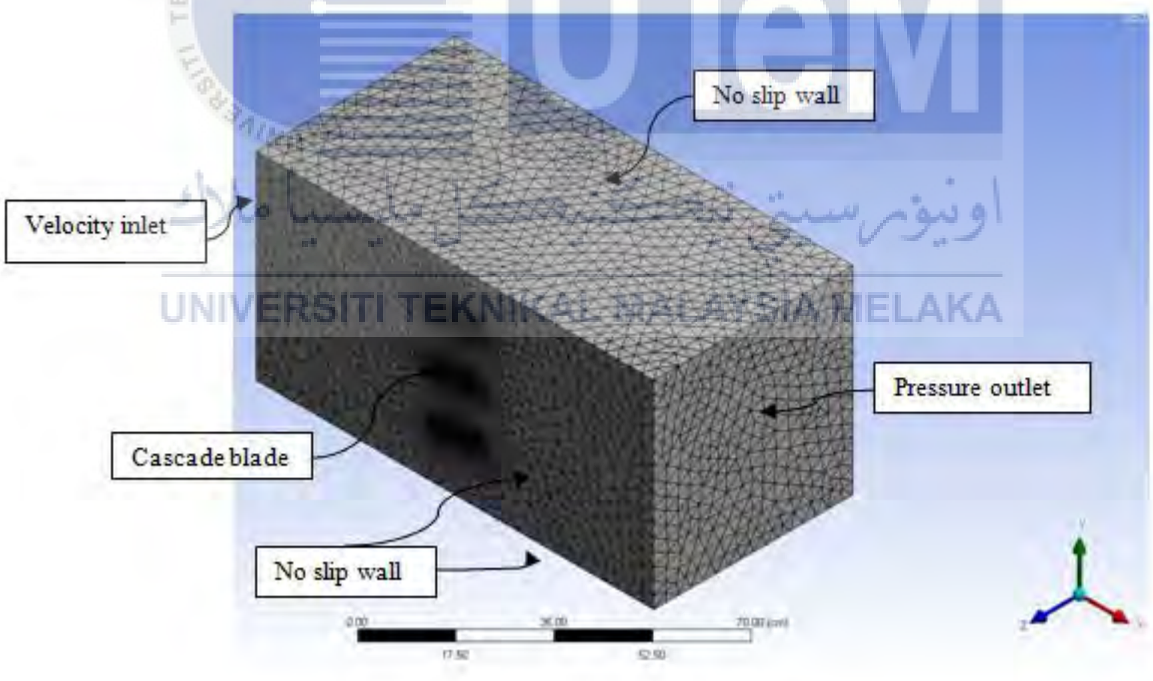

Figure 3.6 Meshing generation and boundary condition for three blades

Sizing will be control the size of distribution mesh to be growth. There is two ways to control sizing on the model, which is global and local sizing. For the global sizing, the proximity and curvature is on. This is to determines the edge and face size based on curvature normal angle. The relevance center for sizing is set to medium and other parameter is set up as default. However, edge sizing and face sizing of mesh is adjusted on local sizing. Edge of tip, hub and trailing edge is selected. Number of division is vary according to the conformity with hard behaviour. The number of division is different for each case study. This is because there are some changes in the cascade blade modelling. The number of division and element size is determined according to the number of element and nodes generated by the meshing. In fact, number of element is limited to generate because the server and licence of ANSYS Fluent software are used only for educational which below than 510000 of element. Thus, it is affects the meshing quality of model.

The number of nodes and element for meshing generation is shown in the Table 3.2. This statistic of meshing influence the quality of mesh. In order to specify the quality of mesh, skewness is measured and determined how close to ideal a face or cell. Table 3.3 shows the skewness range and cell quality (Metin, 2014). The skewness of meshing then referred to the rate of skewness shown in Table 3.3. Based on the Table 3.2, it is found that the maximum skewness of four cascade blade mesh is in bad quality which ranges 0.95 to 0.96. This is because of the limited number of element that can generate from mesh. Otherwise, the quality of skewness can be improved.

Table 3.2 Number of nodes and element

Number of blade	Nodes	Element	Skewness
3	86455	477474	0.96343
5	90060	493995	0.95405
7	92453	495246	0.9657
9	84662	450196	0.96774

Table 3.3 Skewness range and cell quality (Metin, 2014)

Skewness	0.98-1.00	0.95-0.97	0.80-0.94	0.50-0.80	0.25-0.50	0-0.25
Cell Quality	Unacceptable	bad	Acceptable	Good	Very good	excellent

3.5.3 Fluent

In order to run the simulation, some parameters need to set up. The setup consists of model, material, boundary condition and cell zone condition. On this general setup, the pressure-based solver type is selected. This is because this kind of solver is enabling solved for mass and momentum of governing equation. Otherwise, it is applicable for flow analysis. The flow is considered as steady state as the flow does not change with time. The model is turbulence with two equations. Thus, realizable k-ε (epsilon) of viscous model is the best selected since it is focus on the mechanism of turbulent kinetic energy affect. The reason why selected realizable model is realizable k-epsilon model have better improvement in term of flows performance involving boundary layer under strong adverse pressure gradients, also strong streamline curvature compared to the other type of model RNG and Standard. The working fluid used is air at 15ºC. The boundary condition is set up. There are three boundary condition namely velocity inlet, pressure outlet and wall. Some properties are adjusted in boundary condition. Under the velocity inlet, the velocity value is set as 30 m/s at temperature of air, 15ºC. The outlet is present as pressure outlet because the velocities are influenced by pressure gradient. For the solution method, SIMPLE scheme is selected as pressure-velocity coupling. The selection is made because the model is just simple model and steady. For spatial discretization, the gradient is default least squares cell based and second order upwind is set because it is more accurate even though difficult to converge compared to the first order upwind. The different between them are basically the number of point used for the computation. The solution is initialization by using hybrid method and it is computed from the inlet. Finally, 500 iteration is set to run the simulation in order to obtain the results.

CHAPTER 4

RESULT AND DISCUSSION

4.1 RESULTS

In this chapter, the result that has been obtained from the simulation is discussed. There are four study cases that have been done. The parameter and geometry used in these case studies are fixed except the number of blades for the compressor cascade. The analysis was carried out in three dimensional and the blades are twisted at 18^{o} with angle of attack of 10^{o}. The inlet velocity is 30 m/s. The working fluid is air at 15^{o}C. The numbers of blades for each case are 3, 5, 7 and 9. Velocity profile, pressure distribution between the cascade blade and lift and drag coefficient are the result that obtained from the simulation. The result obtained in two dimensional with different position of blade plane. The position plane is represented in ratio. Figure 4.1 shows the measurement of plane position ratio for compressor blade.

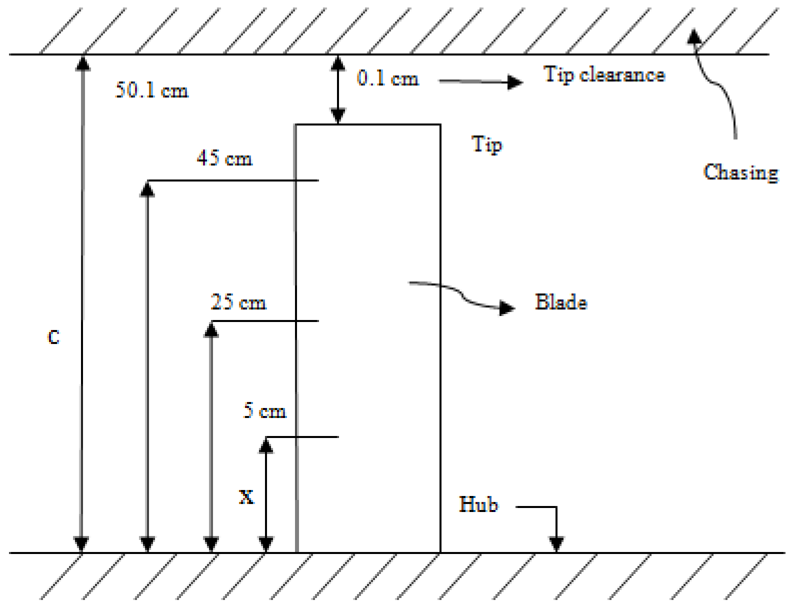

Figure 4.1 Plane position ratio of compressor blade

There are four location that represented by ratio. The first ratio is 0.1 which is 1% from hub. Second and third position is 50% and 90% from the hub and the last one is 0.99 which is the ratio of tip clearance. The position of tip clearance plane is in the middle between the wall and blade tip.

There are two description for each result obtained for pressure distribution and velocity profile. First, the result of pressure distribution and velocity profile for each case study with different number of blades at four ratio position plane. Second, Comparison of pressure distribution and velocity profile of four case studies with different number of blade at each ratio position plane. Another result is lift coefficient and drag coefficient. Both coefficients are resultant from the aerodynamic forces. Pressure distribution around the blade aerofoil causes the lift and drag forces acting on the blade. Last but not least, pressure different between leading edge and trailing edge of the blade passage. The pressure occurs at leading and trailing edge is different since the flow pattern is different at both edges.

4.1.1 Pressure Distribution of Case Study

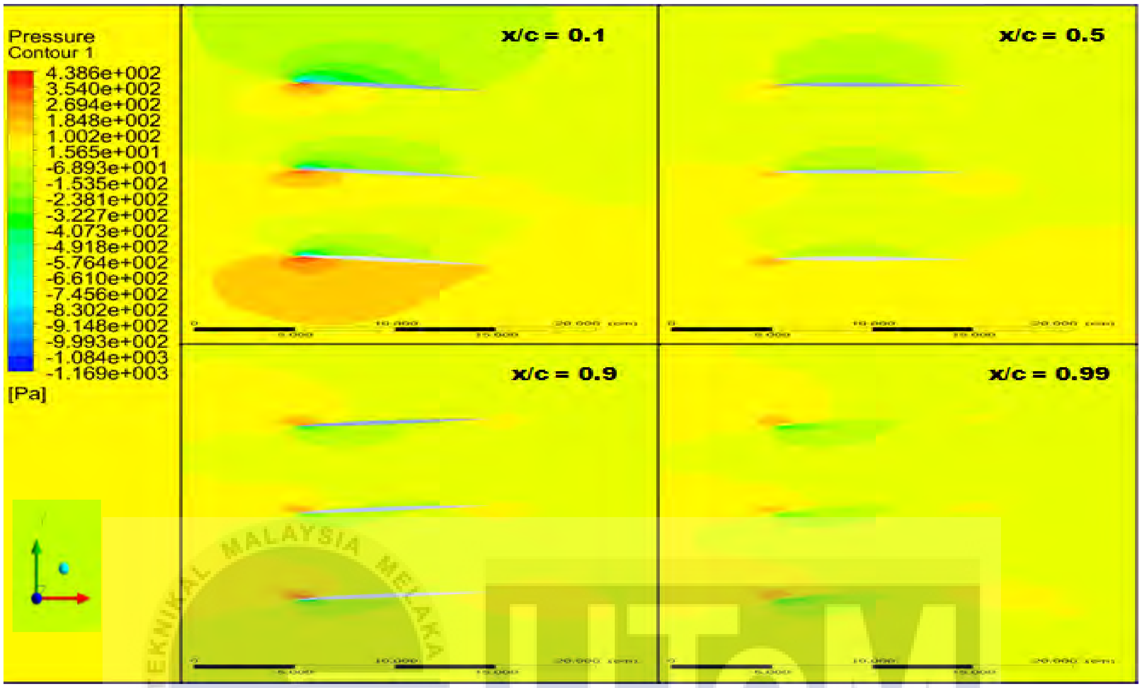

Figure 4.2 Pressure contour for 3 Blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

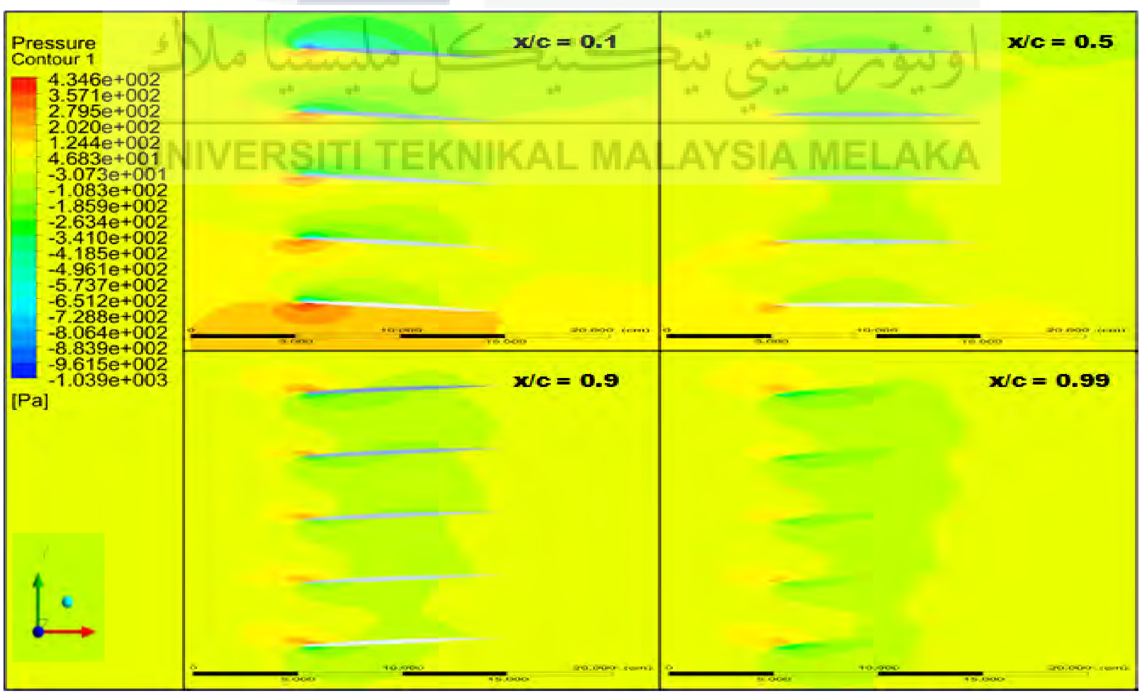

Figure 4.3 Pressure contour of 5 blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

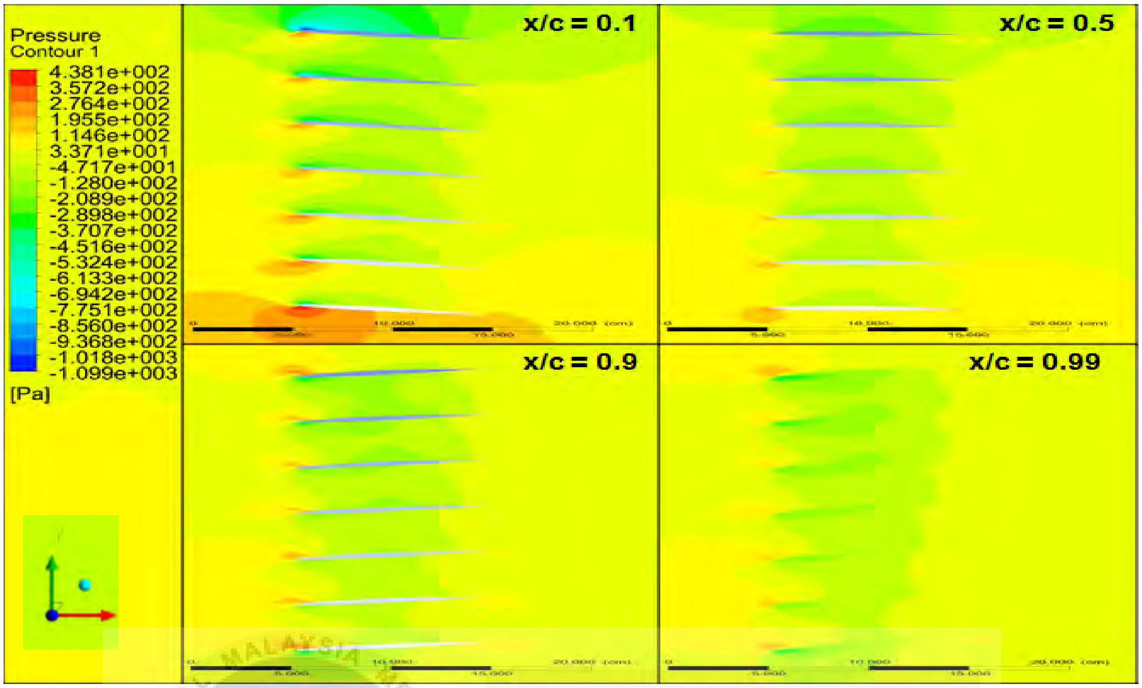

Figure 4.4 Pressure contour of 7 blades
(AOA: 10^{o}, Angle of twist: 18^{o}, Inlet velocity: 30 m/s, Working fluid: Air at 15^{o}C)

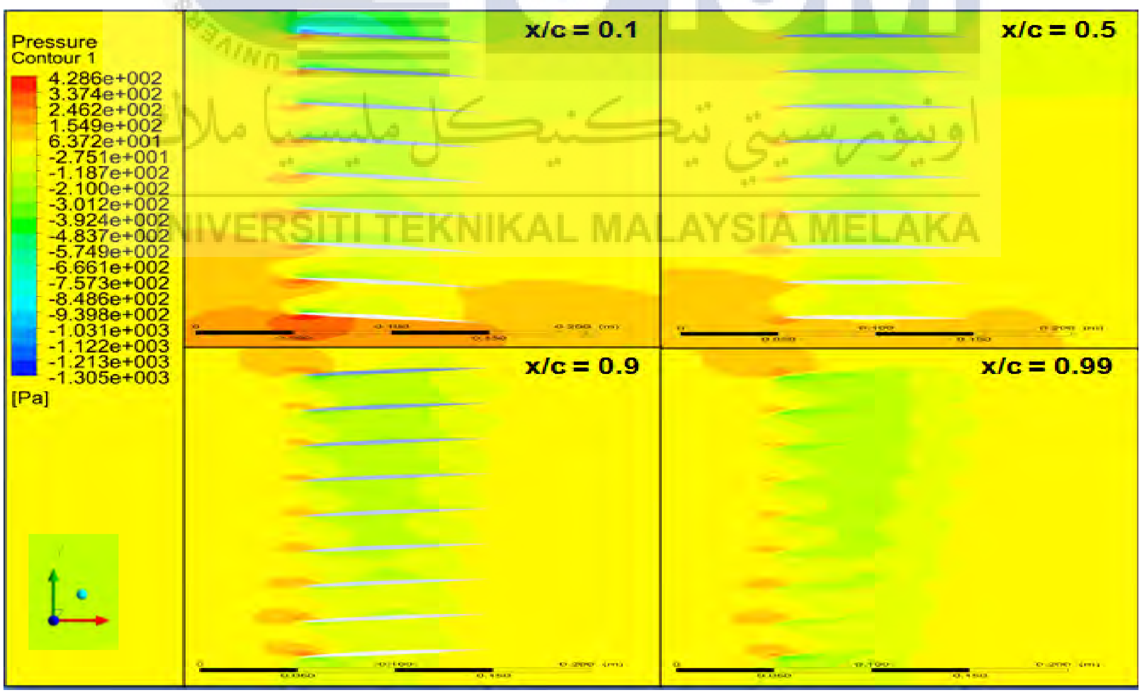

Figure 4.5 Pressure contour of 9 blades
(AOA: 10^{o}, Angle of twist: 18^{o}, Inlet velocity: 30 m/s, Working fluid: Air at 15^{o}C)

Figure 4.2 shows the pressure distribution of three number of blade. The blade are twisted, hence pressure along the blade are not same. It is different following their location. Angle of attack of each plane ratio is varies due to the blade twisted. At ratio of 0.1, it is show that the pressure distribution are higher on the pressure side of airfoil which represented by red colour. The pressure is maximum on the pressure side of leading edge of airfoil with value of 438.6 Pa. Meanwhile, pressure is lower on the suction side of airfoil and in between of cascade blade range in -407.3 Pa to 100.2 Pa. The region of red became small when the ratio plane is increasing to 0.5, 0.9 and 0.99. In addition, the pattern of pressure contour on the airfoil is inverted at ratio plane of 0.9 and 0.99, where the pressure is higher on the suction side and lower on the pressure side. This might be due to the changing of angle of attack from +10° to -10°. The pressure on the pressure side of trailing edge is quite high when the air is pass through the cascade blade which represented in yellow colour.

Figure 4.3 show the pressure distribution for five number of blades. It is shows that the pressure is maximum on the pressure side of leading edge of airfoil with value of 438.1 Pa. The pattern of pressure distribution is similar to the other case study which the pressure on the pressure side of cascade airfoil is high and it is lower on the suction side of cascade airfoil. At plane ratio of 0.1, the pressure on the suction side of leading edge of blade near to the wall is –1039 Pa. The pressure between the cascade airfoil is range in between -47.17 Pa to -370.7 Pa, which represented by green colour. The red region became smaller when the plane ratio increases from hub to tip along the cascade blade.

Pressure contour of cascade blade for seven and nine number of blades is shown in Figure 4.4 and Figure 4.5, respectively. The similar pattern of colour region are shown for both case study, which is the red colour occurred on the leading edge of pressure side and also suction side at ratio of 0.9 and 0.99 due to the negative angle of attack. The red colour represented of high pressure region and blue colour is low pressure region. The maximum pressure of seven number of blade is 438.1 Pa higher than nine number of blade of maximum pressure which is 428.6 Pa. The spacing between the cascade airfoil is influenced the pressure distribution of blade where, it is lower when the air is passing through the cascade blade.

4.1.2 Comparison of Pressure Distribution

Figure 4.6 Pressure contour of four cases study at ratio of 0.1
(AOA: 10^o, Angle of twist: 18^o, Inlet velocity: 30 m/s, Working fluid: Air at 15^{o}C)

Figure 4.6 shows the pressure distribution at ratio plane of 0.1 for four case studies with various numbers of blades. In overall, the results shows the pattern of the maximum pressure occurred are identical, which the high pressure region hit the leading edge of pressure side of cascade airfoil. The red region is widely distributed near the wall of pressure side airfoil. In addition, the pressure along the suction side of cascade airfoil is lower. Hence, the pressure between the blades is also considered lower compared to the pressure surrounding the cascade blade. The maximum pressure for three numbers of blades is 438.6 Pa; similarly with seven number of blades which approximately 438.1 Pa. However, the maximum pressure number of blades of five and nine are lower with value of 434.6 Pa and 428.6 Pa, respectively.

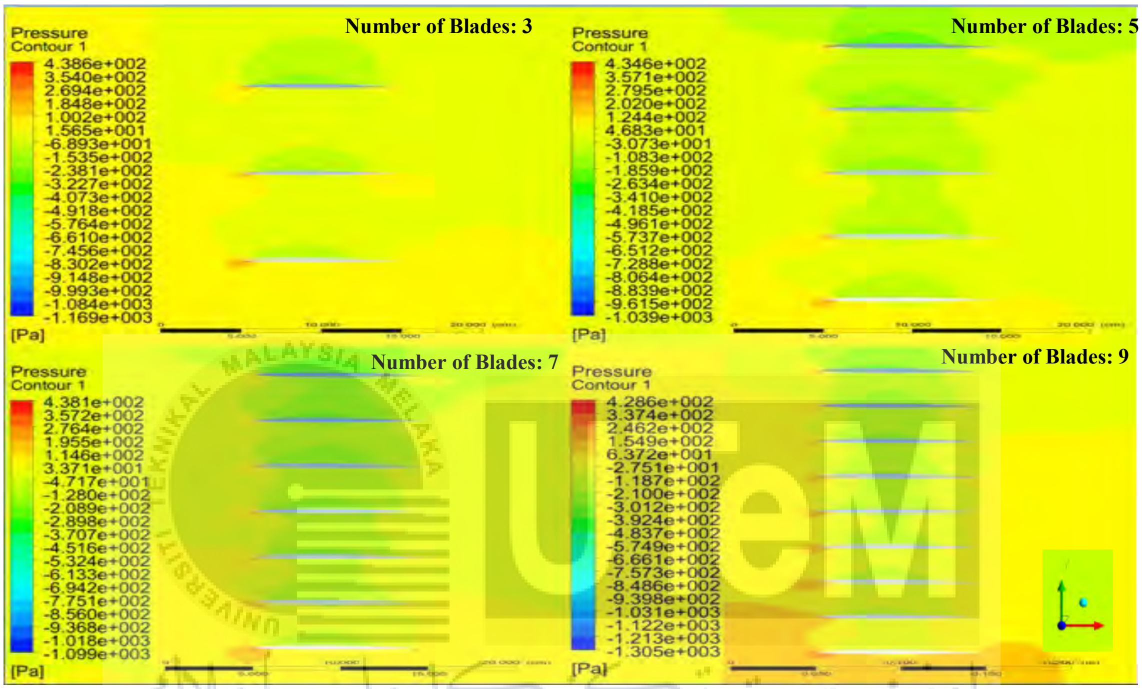

Figure 4.7 Pressure contour of four cases study at ratio of 0.5
(AOA: 10°, Angle of twist: 18°, Inlet velocity: 30 m/s, Working fluid: Air at 15°C)

Figure 4.7 shows the pressure distribution of compressor cascade blade at ratio of 0.5. This ratio plane is located at the middle of blades from the hub. Therefore, the angle of attack of airfoil are varies near to the zero. In comparison of maximum pressure from the ratio of 0.1, it is shows that the red region occurred at the leading edge of cascade airfoil became smaller and the pressure are decreased as the blade near to the upper wall. The pressure between blades on the suction side are maintain lower, but range of pressure distributed represented in green colour of nine blades are the most lower in range of -483.7 Pa to -27.51 Pa.

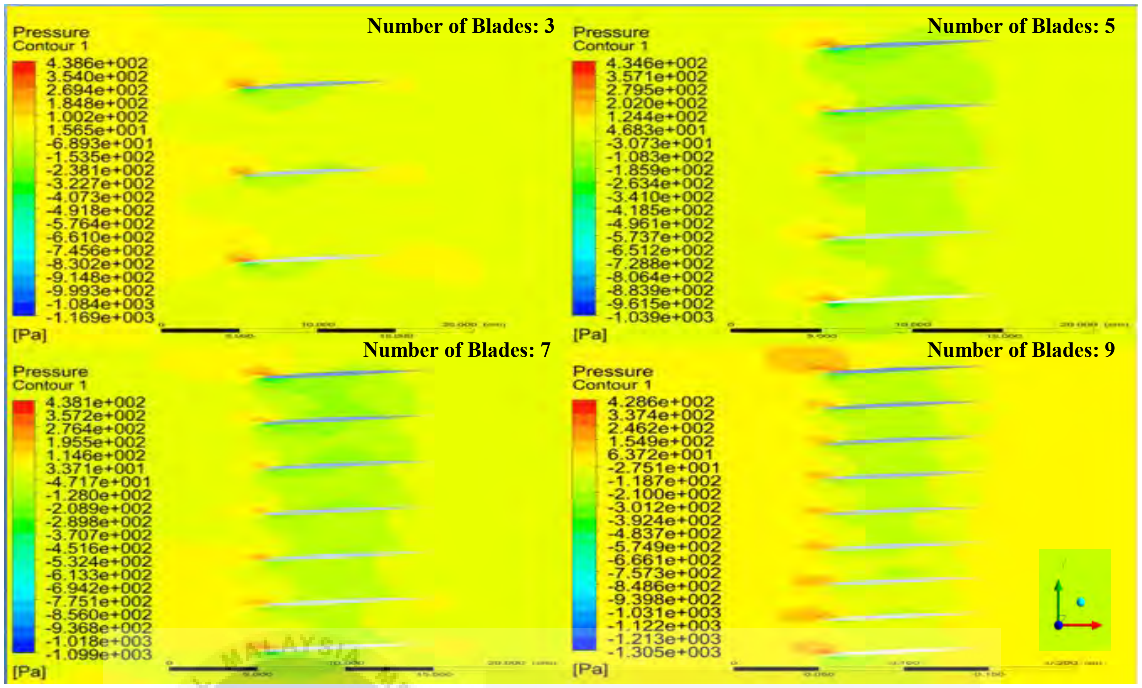

Figure 4.8 Pressure contour of four cases study at ratio of 0.9
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

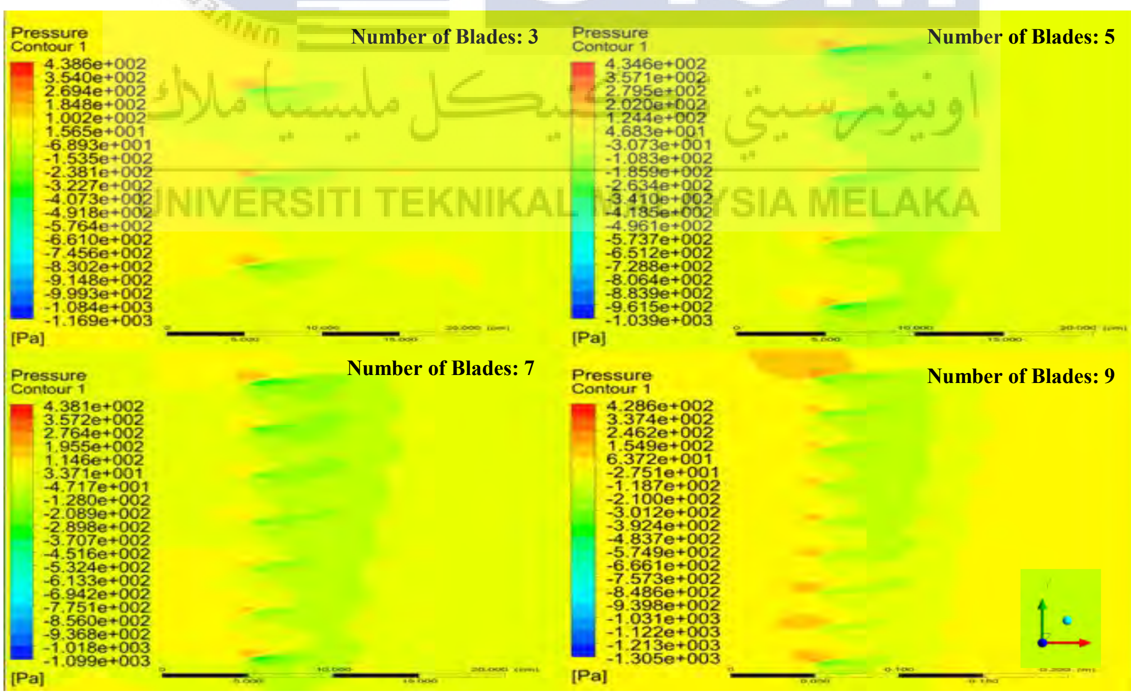

Figure 4.9 Pressure contour of four cases study at ratio of 0.99
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

Next is pressure distribution of cascade blade at ratio of 0.9 as shown in Figure 4.8. In this result, the angle of attack of cascade airfoil in negative. Thus, the pattern of pressure distribution is affected. It is show the maximum of pressure is occurred on the leading edge at the suction side. For the first case with three number of blades, it is show that pressure at the inlet of cascade blade and trailing edge of is quite higher in range of 15.65 Pa to 100.2 Pa. Different with nine blade number, which the inlet and outlet constantly higher than the pressure across the cascade blade. This might be due to the spacing between the blades and location of plane of blade which near to the tip of blades.

Figure 4.9 shows the pressure distribution at ratio plane of 0.99.This plane located between the tip of blade and wall called as tip clearance. The pattern of pressure distribution quite similar with the pressure pattern at ratio 0.9. The pressure still occurred due the air flow across the tip clearance. Further, the distance of tip clearance are too small, hence it does not affect the pressure at ratio plane of 0.99.

4.1.3 Velocity Profile of Case Study

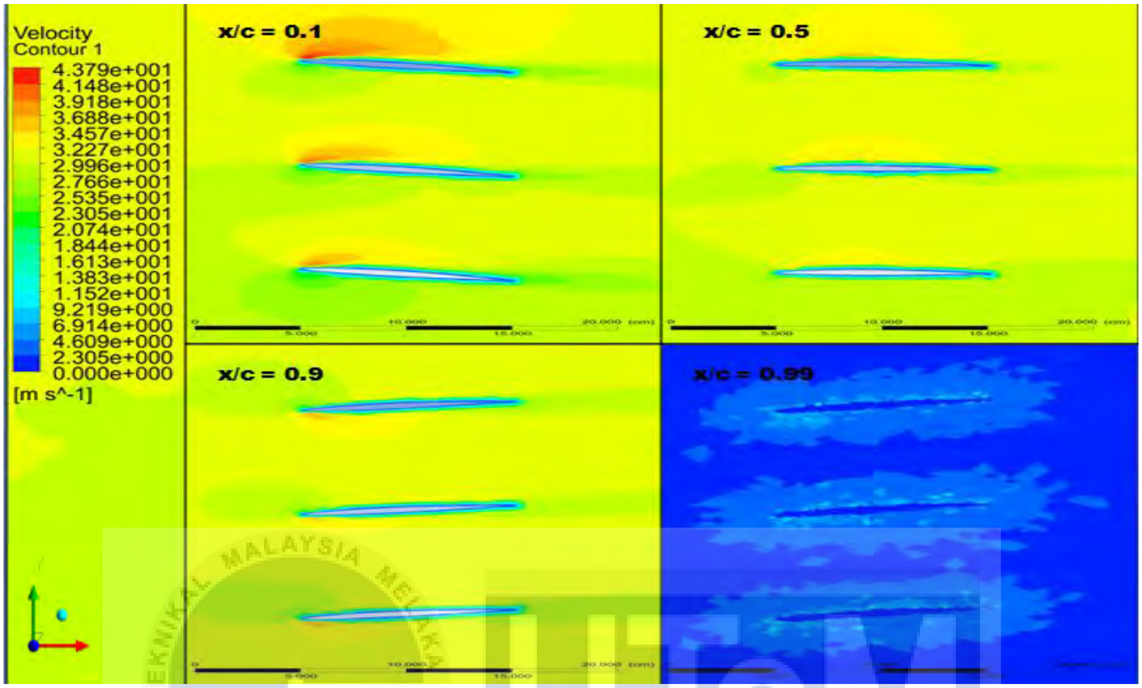

Figure 4.10 Velocity profile of 3 blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

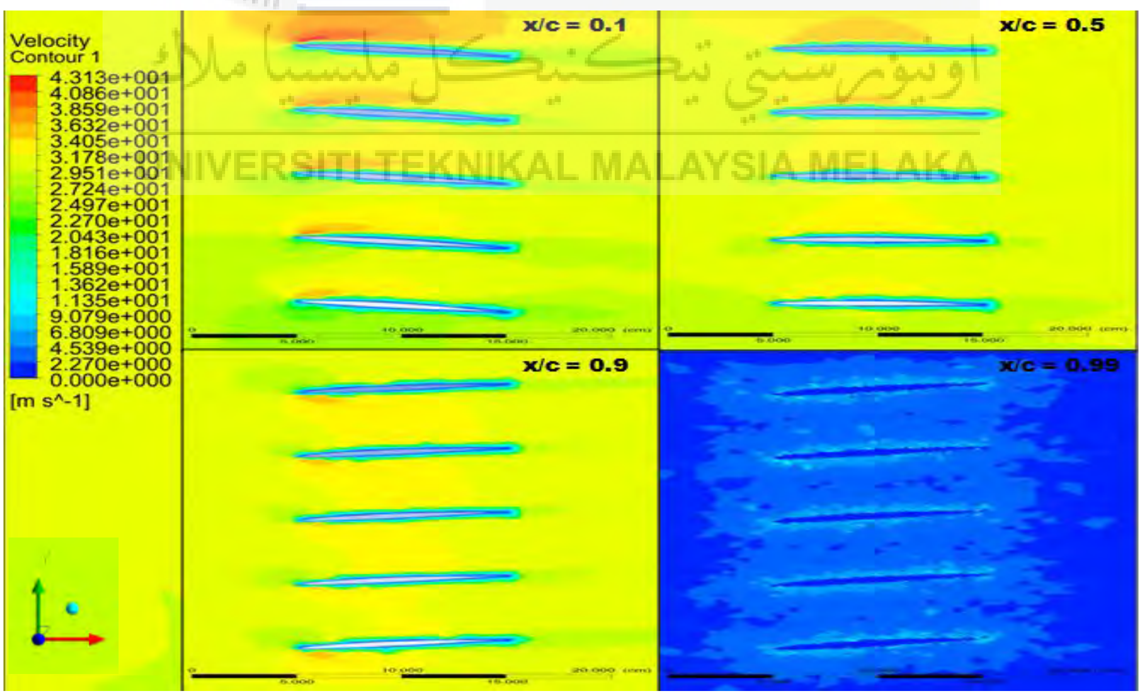

Figure 4.11 Velocity profile of 5 blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

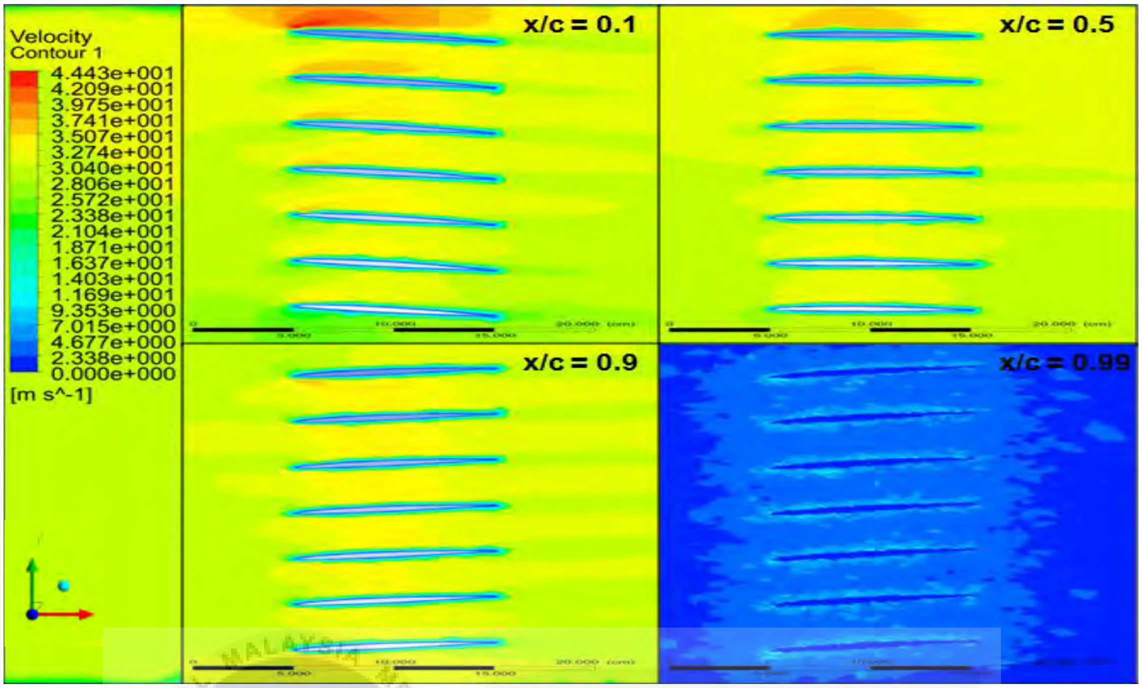

Figure 4.12 Velocity profile of 7 blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

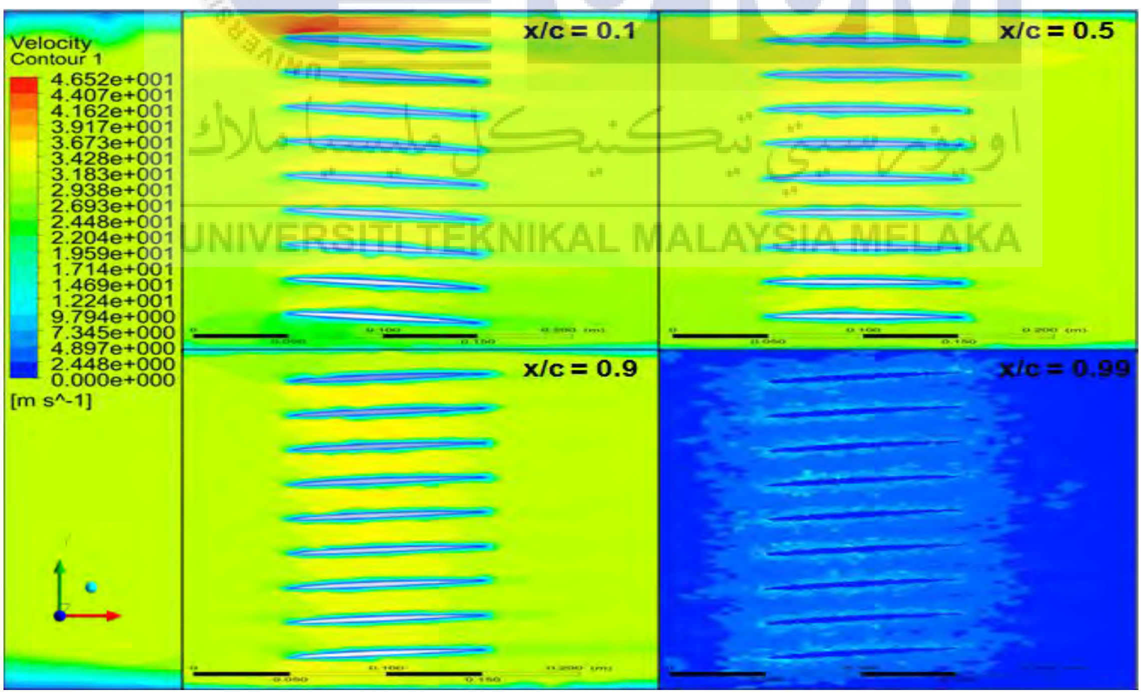

Figure 4.13 Velocity profile of 9 blades
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Working fluid: Air at 15ºC)

Figure 4.10 shows the velocity profile for three number of blade at various ratio position plane. The profile is shown in contour and streamline. The boundary condition of blade is set as no slip wall which considered zero velocity on the surface of blade. Based on Figure 4.10, it is show that the velocity on the suction side of cascade airfoil is higher while it is lower on the pressure side of airfoil. The red region occurred at ratio plane of 0.1 which represented high speed of air on the leading edge of suction side, also it appeared at ratio plane of 0.9 which the small region on the leading edge of pressure side. The speed of air at the inlet before it passes through the cascade blade is decreased from hub to tip of blades. The blue region shows that it is uniform with zero velocity around the cascade.

The pattern of velocity contour shown in Figure 4.11 and Figure 4.12 is similar, where the velocity of upper side is higher at ratio 0.1and 0.5. However, decreasing of angle of attack due to twisted angle result the changing of velocity on the suction side of cascade airfoil at ratio of 0.9. The exit velocity on trailing edge of cascade airfoil for Figure 4.11 has longer lower speed compared to the Figure 4.12 at ratio 0.9.

According to the velocity profile for nine number of blade as shown in Figure 4.13, the maximum speed of air across blades is occurred at the ratio plane of 0.1. As the plane location near to the tip of blade, the velocity on the leading edge of suction side of airfoil is decreased. It is clear when the air passed through the cascade blade, the speed of air is changes otherwise it is uniform along the inlet and outlet. At the ratio of0.99, the velocity is drop since there is no resistance to pass it.

4.1.4 Comparison of Velocity profile

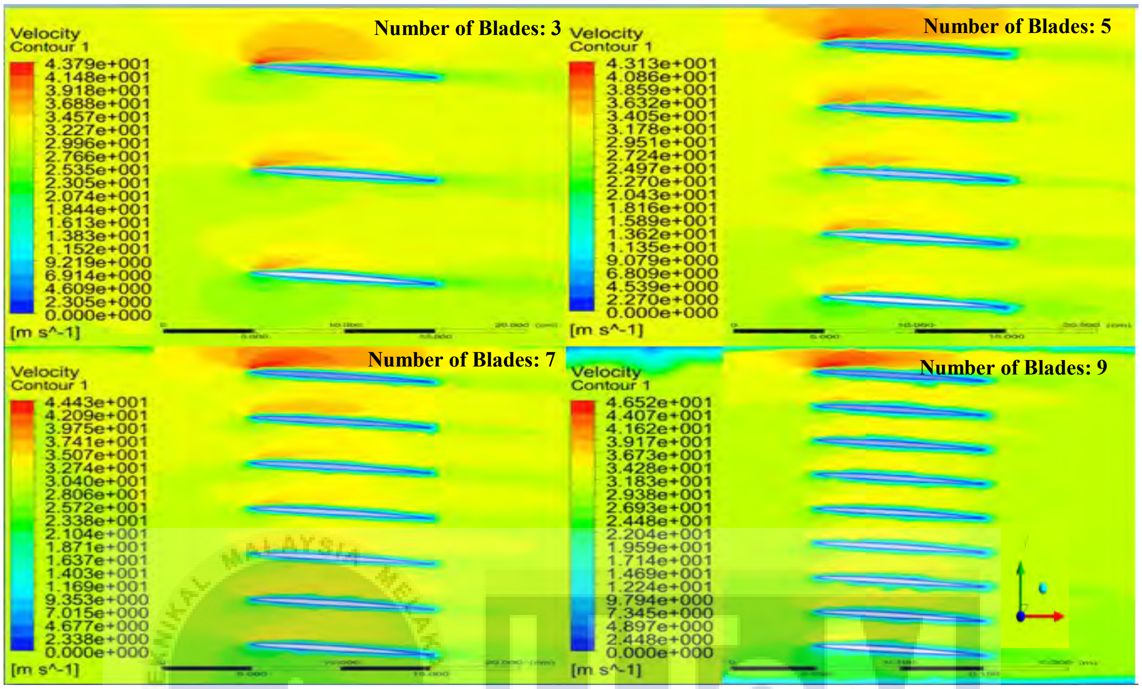

Figure 4.14 Velocity profile of four case study at ratio of 0.1
(AOA: 10°, Angle of twist: 18°, Inlet velocity: 30 m/s, Working fluid: Air at 15°C)

Figure 4.14 show the result of velocity profile at ratio of 0.1. Based on the result, it is obvious that the flow of air is divides into upper and lower flows at the leading edge of cascade airfoil. The separation of flow can be seen by the different colour that hit the leading edge of cascade airfoil, where represented by the red and green colour. In overall, it is clearly show that the maximum velocity of air passed through the suction side blades near the upper wall. Meanwhile, the velocities are slow when it across the pressure side of near wall blade compared to the air speed blade passage. However, the air flow pass the trailing edge are lower. The flow leaves the trailing edge is smoothly, as follow the Kutta-Joukowsky condition.

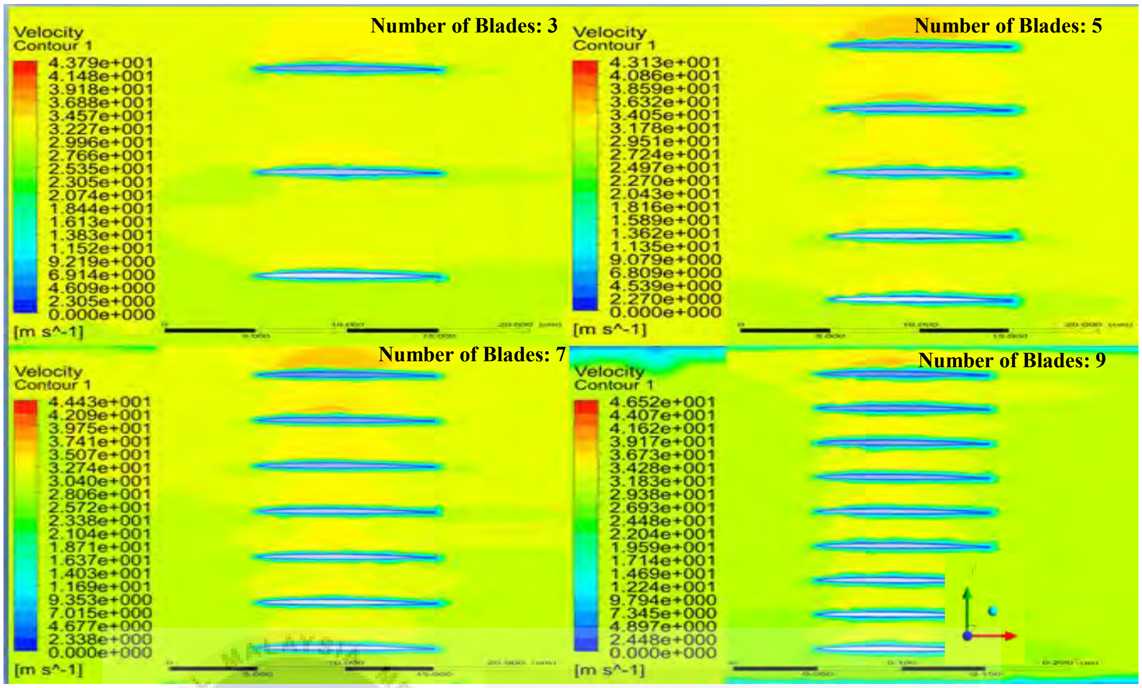

Figure 4.15 Velocity profile of four case study at ratio of 0.5
(AOA: 10º, Angle of twist: 18º, Inlet velocity: 30 m/s, Workingfluid: Air at 15ºC)

The velocity profile of four various number of blade at ratio plane of 0.5 is shown in Figure 4.15. The velocity is higher on the suction side of cascade blade with represented by red and yellow colour region. In comparison of four case study of velocity profile, it is found that the velocity contour for nine blades are the most higher in range of 34.28 m/s to 41.62 m/s. This is because the spacing between the blades are short. The distance between the blades are result the less region of high velocity distributed around the blade passage. There are two colour region which is green and yellow with different value of velocity.

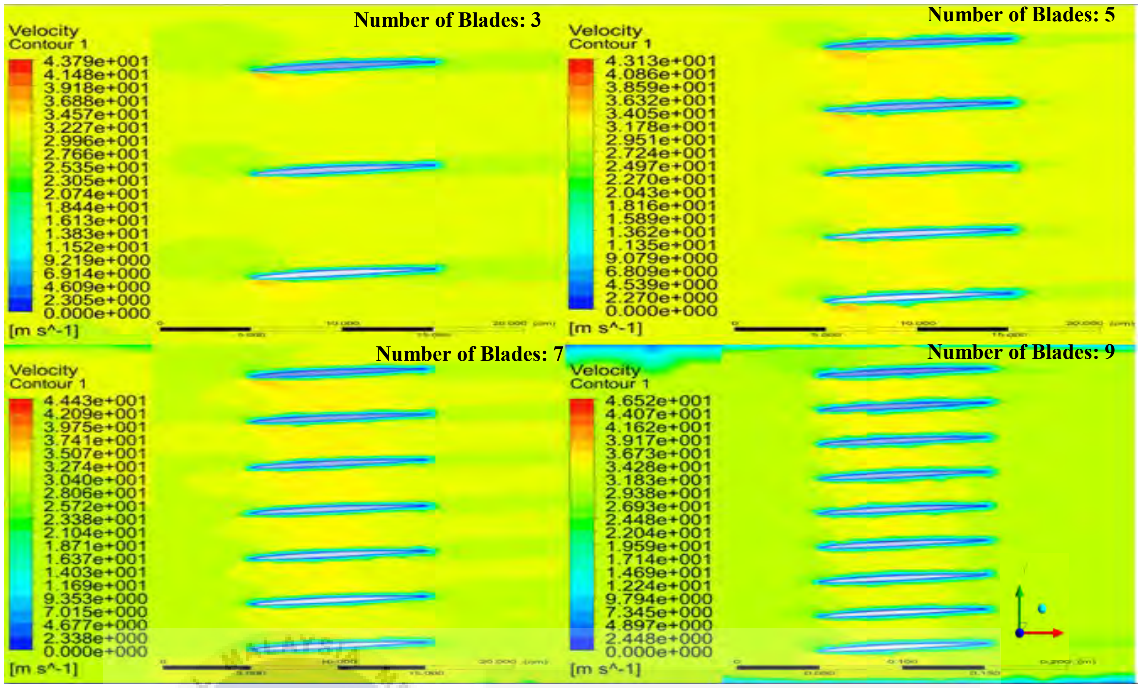

Figure 4.16 Velocity profile of four case study at ratio of 0.9
(AOA: 10^{o}, Angle of twist: 18^{o}, Inlet velocity: 30 m/s, Working fluid: Air at 15^{o}C)

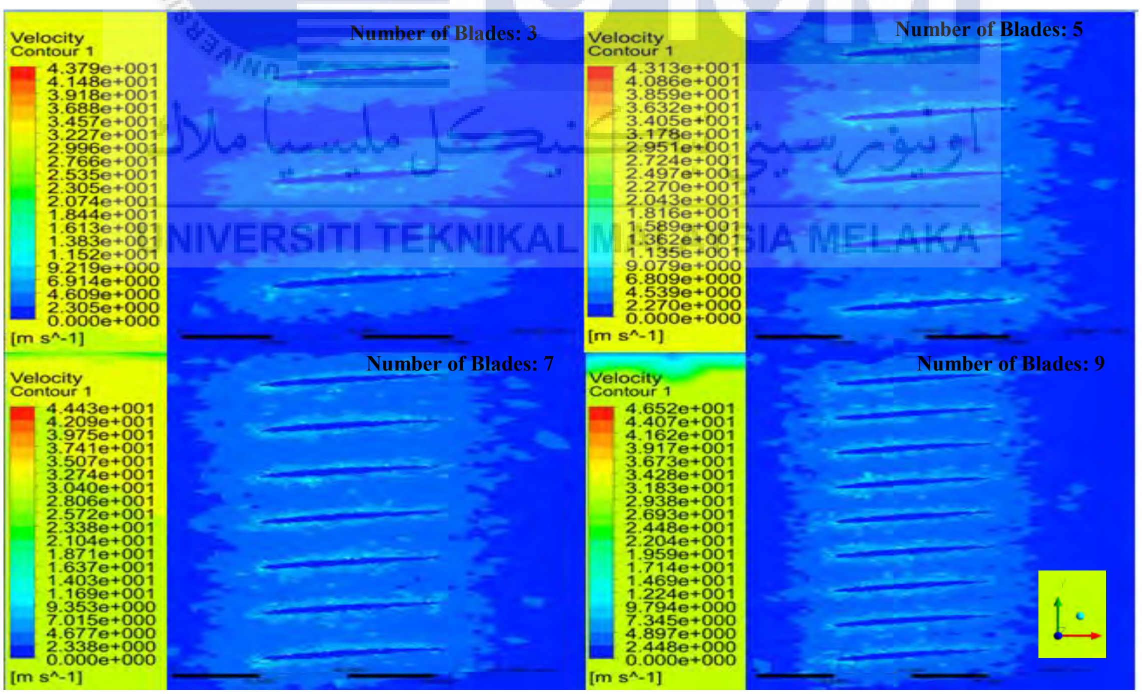

Figure 4.17 Velocity profile of four case study at ratio of 0.99
(AOA: 10^{o}, Angle of twist: 18^{o}, Inlet velocity: 30 m/s, Working fluid: Air at 15^{o}C)

The result of velocity profile obtained in Figure 4.17 is totally different. The low velocity occurred on the leading and trailing edge of cascade airfoil while the higher velocity occurred on the pressure side of airfoil at the leading edge. However, velocity inlet and velocity outlet are constant speed in average of 30.61 m/s, for nine blades. At ratio plane of 0.99, there only have two tone colour of blue. The dark blue as shown in Figure 4.17 has no velocity. So, there is zero velocity of air across the tip clearance. Meanwhile, the velocity increase up to 2.3 m/s. In comparison of four case study, nine blades of compressor has the higher increasing of velocity across the tip clearance followed by seven, five and three number of blades.

4.1.5 Pressure Different

Pressure different is the pressure inlet minus pressure outlet of cascade blades. The point of pressure taken is depicted in Figure 4.18. In order to obtain the accurate result, only four pressure point which located in the middle of cascade blade which is two point at the inlet and two point at the outlet is taken. The average of two pressure point is used to obtain the value of pressure different.

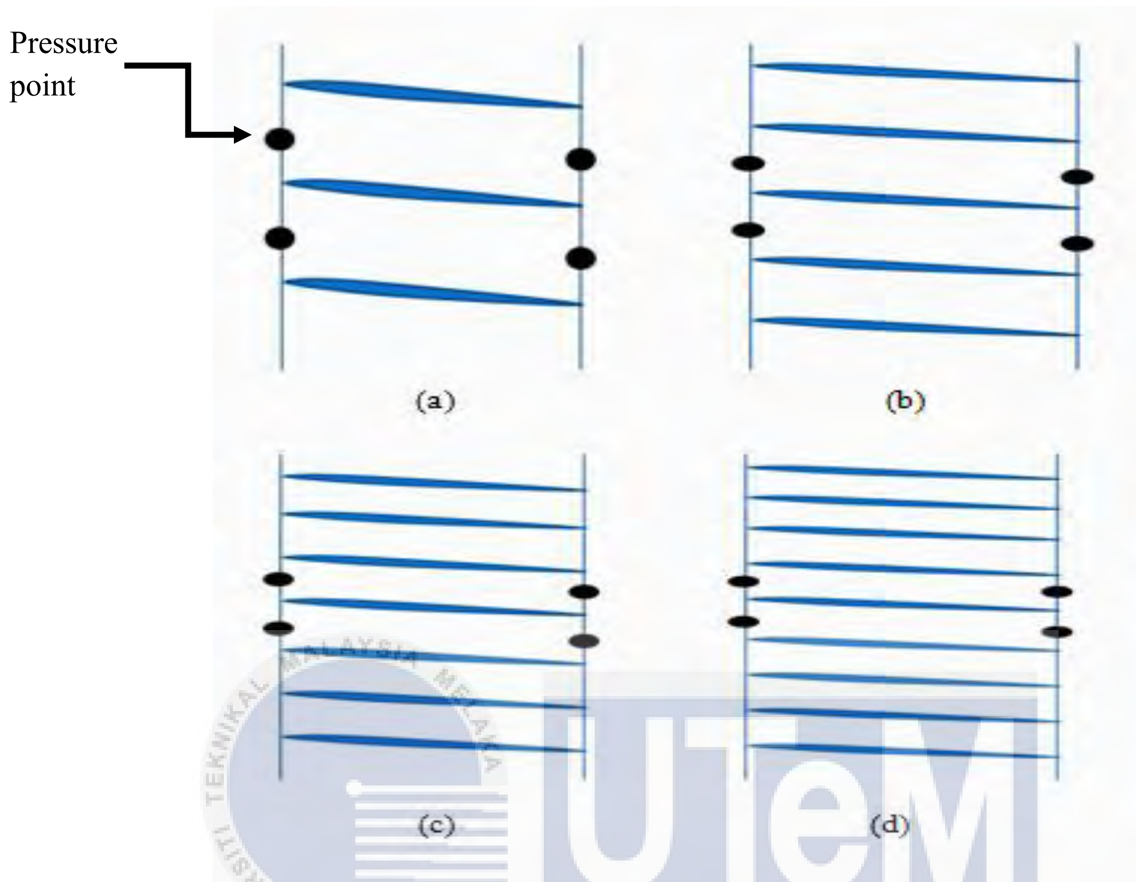

Figure 4.18 Position of pressure point for various numbers of blades

The pressure different is then calculated and plotted in the Figure 4.19. The graph shows that the pressure different for three number of blade is -2.7881 Pa and it is then follow by 1.7485 Pa, 3.6264 Pa and 5.7095 Pa for other number of blades, respectively. In short, pressure different of four case study is increased with addition of number of blades.

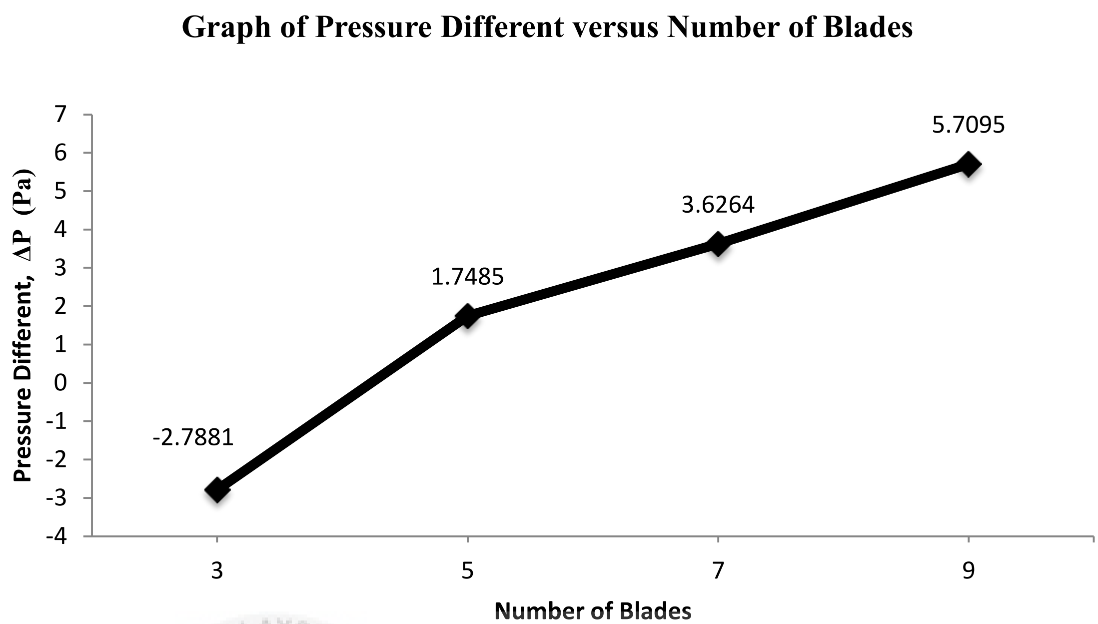

Figure 4.19 Pressure different of four case studies.

4.1.6 Lift and Drag Coefficients

Table 4.1 Lift and Drag coefficient

Number of Blades	Lift Coefficient, C_L	Drag Coefficient, C_D
3	0.147	0.019
5	0.22	0.028
7	0.266	0.037
9	0.3	0.045

Table 4.1 shown the lift and drag coefficient of cascade blade for four number of blades. It is found that the lift coefficient is larger than drag coefficient. The result is obtained directly during simulation using ANSYS Fluent code, where the iteration is fully converged.

Lift and drag coefficient is nondimensionalized form of lift and drag forces on compressor cascade blade. According to the Table 4.1, the lift coefficient is increased when the number of blades is added just as drag coefficient. However, the graph of lift to drag ratio in Figure 4.20 shows that the ratio is increased when more two blades is added from three blades. Then, the ratio are decreased when the number of blades are added to seven and nine blades

Graph of Lift to Drag ratio versus Number of Blades

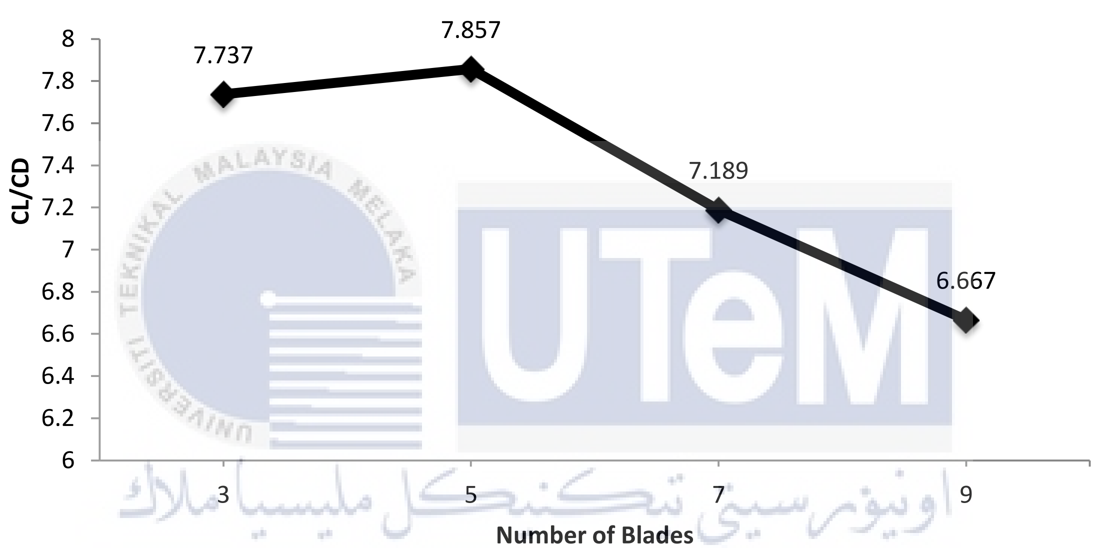

Figure 4.20 Lift to drag coefficient ratio of four case studies

4.1.7 Efficiency of compressor cascade blade

Compressor cascade efficiency obtained in Figure 4.21 is calculated by using the equation following:

$$\pi_{D,max} = 1 - \frac{2C_D}{C_L} \tag{4.1}$$

Equation 4.1 is the maximum efficiency for compressor cascade which assumed constant lift-drag ratio to give the optimum flow angle. The optimum mean angle is 45º. Since the geometry of blade is fixed, the inlet flow angle and the outlet flow angle are approximately similar for four case studies. Furthermore, this study is focused on the different number of blade for each study. Thus, the performance of cascade blade is compared based on the maximum efficiency of cascade obtained from equation 4.1. Based on Figure 4.21, the efficiency of three blades is increased and begins to decrease at five blades till the blade is added to nine.

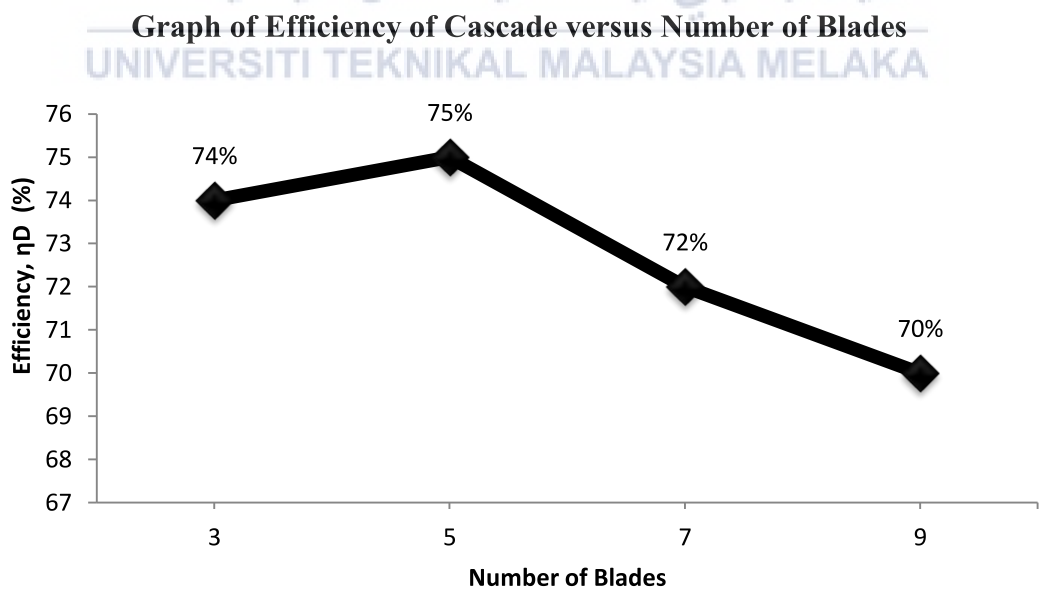

Figure 4.21 Cascade efficiency of four case studies

4.2 DISCUSSION

Difference numbers of blades in cascade to the performance are discussed. Blade profile, geometry and parameter setting in CFD analysis for each case study are fixed. However, the results obtained from the simulation are different. Based on the result of pressure distribution obtained from the simulation, it is shows that the patterns of pressure distributed around the cascade blade are similar for four case studies with different number of blades. It is found that the upper side of cascade airfoil have lower pressure along the airfoil and higher pressure on the lower side of cascade airfoil. The pressure around the cascade airfoil occurred due to the velocity of air that passed over the cascade blade. Thus the pressure distribution is created. According to the Bernoulli's principle, it is stated that high velocity creates the lower pressure system and vice versa. This is proved by the result of velocity profile which the red region represent high speed occurred on the suction side of cascade airfoil. This happened because of airfoil are shaped. It has longer upper surface compared to lower surface. The air enter from the inlet are separated on the leading edge of cascade airfoil. It then crosses over the cascade blade. In order to meet at the trailing edge of airfoil at the same time, the air on the suction side needs to travel faster than pressure side because it has greater distance to travel. However, the pattern of pressure and velocity on the cascade airfoil become different at ratio plane of 0.9, where the higher pressure created on the pressure side of airfoil while lower pressure formed on the suction side. The result is inversely to the velocity profile. The reason might be due to the decreasing of angle of attack to negative. The angle of attack of each ratio plane is different because of compressor cascade blade is twisted at angle of 18 degrees.

In comparison of velocity profile and pressure distribution for four case study with different number of blades, it is show that nine blade has the highest maximum pressure and maximum velocity among the result obtained for other cases. This is because of spacing between the cascade blades. The addition of blades number reduces the gaps between the blades. Theoretically, when the area of velocity inlet is smaller, the velocity of air enter the inlet of cascade blade are increased. Hence, lower pressure is formed. In addition, the pressure between blades passages are constant high pressure compared to the pressure at inlet and outlet of cascade blade. Other than that, the velocity at ratio of 0.99 are the most lower. This is because there is no obstacle as the velocity passes through the clearance gaps between tip of blade and wall. The smaller gap of tip clearance avoid the big losses occurred.

The lift and drag generated by airfoil is depends on the density of the air, the inlet velocity of airflow, shape of airfoil and angle of attack. Lift forces is generated when the lower pressure formed over the lower surface of airfoil due to accelerating of air. Thus, resulting pressure distribution produces a force that act on the airfoil. The results of lift and drag coefficient for compressor twisted cascade blade followed the compressor cascade blade, which required to produce maximum lift with low drag coefficient. In comparison of each case study with different number of blades, it is show that the lift coefficient is increased as number of blade is added. As lift coefficient, the drag coefficient also increased when the number of blade increased. The addition of number of blades is reducing the spacing between blades. Thus, pressure occurred between the cascade blade of each case study are different. The graph of lift to drag ratio shows that the ratio is slightly increased from three blade to five and then start to decrease until number of blade is nine.

According to graph of pressure different versus number of blades, it is indicated that the increasing of pressure different when the number of blades is added. The pressure is taken at two point of inlet and outlet of blade passage. The reason is to know the different of pressure of incoming air and pressure of air when it passed over the cascade blade. Other than that, efficiency of cascade blade is calculated. This is to show the performance of cascade blade by comparing their efficiency.

Table 4.2 shows the summary of CFD result in term of lift to drag ratio, pressure different and cascade blade efficiency. According to the Table 6, it is clearly stated that 5 blades of compressor have the good performance compared to the 3, 7 and 9 blades. It is because it have high efficiency and high ratio of lift-drag.

Table 4.2 Summary of analysis results of compressor cascade blade

Number of Blade	**Lift to drag ratio (C_L/C_D)**	**Pressure Different, ΔP (Pa)**	**Compressor Cascade Efficiency, η_D**
3	7.737	-2.7881	0.74
5	7.857	1.7485	0.75
7	7.189	3.6264	0.72
9	6.667	5.7095	0.70

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

Axial flow compressor has an advantage in term of pressure ratio, efficiency and flow rate, hence this type of flow compressor is chosen for present study. The direction of flow for axial is parallel to the axis of shaft. The blade is twisted for purpose of better airflow. The twist angle along the blade reduces the angle of attack toward the tips. The linear velocity hit the blades is different which it is higher from hub to tip. NACA 6 series is chosen as blade profile because its advantages. The analysis is done in 3D for cascade blade modelling. ANSYS Fluent used to simulate the analysis. The parameter of meshing generation is different according to the model, but the setting is same for all the cases. Similar to the Fluent set up, the parameter is same such as density of air, inlet velocity and so on. Based on the CFD analysis, the effect of number of blades to the performance of the cascade blade is studied. The performance of cascade blade for four different number of blade, namely 3, 5, 7, and 9 shown in pressure distribution, velocity profile, lift to drag ratio, pressure different and cascade efficiency. The pressure distribution around the surface of airfoil for nine blades has a higher value of maximum pressure. The pressure is decreased from hub to tip of blades. The result is shown in different position of plane ratio since the compressor cascade blade is modelled in 3D. Ratio plane of 0.99 shows the pressure on the tip clearance between blade tip and wall. However, reality of axial compressor function of clearance gaps is to prevent the blade tip rubbing to the shroud. Angle of attack influenced the pattern of pressure formed and the separation of air at the leading edge of blades. The result shown in the four different location of plane on cascade blade. The acceleration of velocity produces lower pressure. Thus, the statement is achieved as the result shown in four case of different number of blades that the velocity on the suction side of airfoil is higher than the pressure side. The results of pressure different of inlet and outlet of blade passage between the blades also pointed that nine blades have higher pressure different with value of 5.7095 Pa. It is shows that the outlet pressure is higher than the inlet pressure in average of two point. In contrast to the lift to drag ratio and cascade efficiency, it is shows that five blades have the higher value for both. The ratio and efficiency increased from three blades and decreased after that. Concluded that the best performance of cascade blade in between three blades to five blades since the efficiency of cascade blade keep increasing begins with three to five blades.

5.2 RECOMMENDATION

For future works, it is suggested to run the simulation with several of inlet velocity. The reason is to see the difference of result to the cascade blade when different speed of air passes through the cascade blade. Other than that, various angle of attack of cascade blade is recommended for future study. This is because angle of attack related to the lift coefficient generated due to the pressure distribution around the blade. Thus, the performance of cascade blade could be compared. Other suggestion is the modification could be making on the arrangement of cascade blade. Instead of linear arrangement, annular cascade blade should be considered. This is because the arrangement of blade more alike the real compressor cascade blade. Therefore, the losses such as secondary losses, annular losses and etc. can be calculated.

REFERENCES

Anoop, K.S., and Onkar, S., 2014. Effect of Compressor Inlet Temperature and Relative Humidity on Gas Turbine Cycle Performance. *International Journal of Scientic & Engineering Research* , 5(5), pp.664-671.

Arangi, H.G., Sivaram, P., HariBabu, N., 2015. Analysis of Inlet Air Temperature Effect on Gas Turbine Compressor Performance. *International Research Journal of Engineering and Technology (IRJET), 2* (8), pp.846-853.

Airfoil Tools, (2017) *NACA 65-206.*[online] Available at :http://airfoiltools.com/airfoil/details?airfoil=naca65206-il [Accessed on 20 October 2016]

Boyce, P.M. 2011. Axial Flow Compressor. In *Gas Turbine Engineering Handbook* (Boyce, P.M., 4th ed.) , pp. 163-193. UK: Elsevier Publication.

Back, S.C., Hobson, G.V., Song, S.J., Millsaps, K.T., 2012. Effects of Reynolds Number and Surface Roughness Magnitude and Location on Compressor Cascade Performance. *Journal of Turbomachinery*, 134.

Back, S.C., Sohn, J.H., Song, S.J., 2010. Impact of Surface Roughness on Compressor Cascade Performance. *Journal of Fluids Engineering*, 132.

Carter, A.D.S., Turner, R.C., Sparkes, R.C., Burrows R.A., 1957. *The Design and Testing of an Axial-Flow Compressor having Different Blade Profiles in Each Stage.* London: Aeronautical Research Council Reports and Memoranda.

Chun, Z.S., Qin, M.S., Long, W.Z., 1981. Effect of Blade Stagger Angle on Performance of a Transonic Compressor with Low Hub-Tip Ratio. *The American Society of Mechanical Engineers* , pp.1-10.

Hackert, F., 2014. *Gas turbines: Edison Tech Center*. [Online]. Available at: http://www.edisontechcenter.org/gasturbines.html [Accessed on 4 December 2016].

Jafar, M.H., Assim, H.Y., Omar A.K., 2015. Effect of Surface Roughness Height on the Aerodynamics Performance of Axial Compressor Cascade Blades. *Al-Nahrain University, College of Engineering Journal (NUCEJ)*, 18(1), pp.128-139.

Jingjun, Z., Shaobing, H., Huawei, L., Xiaoxu, K., 2013. Effect of tip geometry and tip clearance on aerodynamic performance of a linear compressor cascade. *Chinese Journal of aeronautics*. 26(3), pp.583-593.

Metin, O., 2014. *Meshing Workshop*.[online] Available at https://www.ozeninc.com/wpcontent/uploads/2014/11/MESHING_WORKSHOP_2014.pdf [Accessed on 4 April 2017].

Kurz, R., 2005. Gas Turbine Performance. *Proceedings of the thirty-fourth turbomachinery symposium*. San Diego, California. Solar Turbines Incorporated.

Katoh, Y., Tsuda, Y., Yanagida, M., Toriya,H., Sasada, T., 1996. *Hitachi, Ltd., Tokyo, Japan Pat. 5,554,000.*

Lebele-Alawa, B.T., and Jo-Appah, V., 2015. Thermodynamics Performance Analysis of a Gas Turbine in an Equatorial Rain Forest Environment. *Journal of Power and Energy Engineering* , 3, pp.11-23.

Mohsinali, B., Vimal, P., 2014. Optimization of Number of Bladein High Pressure Compressor. *International Journal of Emerging Technology and Advanced Engineering*, 4 (6), pp.287-292.

Naeim, F., Sheng, L., Qaisar, H., 2013. Effect of Ambient Temperature on the Performance of Gas Turbine Power Plant. *International Journal of Computer Science Issues(IJCSI)*, 10 (1), pp.439-442.

Pradhapraj, M., Gopinathan. V.T., Gowtham, M., Mohamed, H.F., Agila, S., 2016. Numerical and Experimental Investigation of Compressor Cascade of a Turbo-Machine. *International Conference on Emerging Engineering Trends and Science(ICEETS)* , pp.21-26.

Pandey, K.M., Chakraborty, S., Deb, K., 2006. CFD Analysis of Flow through Compressor Cascade. *International Journal of Soft Computing and Engineering*. 2(1), pp.362-371.

Rhoden, H.G., M.A., Wh. Sc., 1952. *Effect of Reynolds Number on the Flow of Air through a Cascade of Compressor Blades.* London: Aeronautical Research Council Reports and Memoranda.

Rathakrishnan, E., 2013. *Theoretical Aerodynamics*. India: Wiley & Sons Singapore Pte. Ltd.

Salim, B., 2013. Performance of axial cascade. *Open Journal of Fluid Dynamics* , pp.191-197.

Sathiyalingam, K., Renald C.J.T., Manikandan, R., Sivakumar, R., Kumar, T.A., Shanavas, S., Kumar, V.S., 2012. Prediction and Measurement Pressure Distribution over a Compressor Blade through a Cascade Studies. in: Assistant Professor,Sri Ramakrishna Engineering College, *International Conference on Modelling, Optimisation and Computing*, Bengaluru, India, 2012, SciVerse ScienceDirect

Sullivan, A., 2010. *Aerodynamic forces acting on an airfoil*.[online] (May,6) Available at :\http://physics.wooster.edu/JrIS/Files/Sullivan_Web_Article.pdf [accessed on November 2016].

Sheikh G.M., Manivannan, P., 2015. Computational Analysis of Compressor Blade. *International Journal of Innovative Research in Science,Engineering and Technology* , 4(3), pp.1131-1138.

Turek, T., (n.d). *Flow Field in the Compressor Blade Cascade NACA 65-100*.[online]Available at: http://stc.fs.cvut.cz/pdf/TurekTomas-349143.pdf [accessed on October 2016]

Zenouze, R. T., Farzin, G., Majed, E., 2015. Experimental and Numerical Investigations of Flow Incidence Effects on Surface Pressure Distributions of Axial Compressor Blades. *Journal of Material Science and Engineering* , pp.80-86.

APPENDICES

APPENDIX A: Sample of Calculation

Take Three Number of Blade as the sample of calculation for:

The lift to drag ratio:

$$\frac{C_L}{C_D} = \frac{0.147}{0.019}$$

$$C_L/C_D = 7.737$$

The pressure different, ΔP:

$$\Delta P = P_{OUTLET} - P_{INLET}$$

$$\Delta P = -5.1027 - (-2.3146)$$

$$\Delta P = -2.7881 \text{ Pa}$$

Compressor cascade efficiency, η_D:

$$\pi_{D,max} = 1 - \frac{2C_D}{C_L}$$

$$\pi_{D,max} = 1 - \frac{2(0.019)}{0.147}$$

$$\pi_{D,max} = 0.74 \times 100 = 74\%$$

Position ratio of location plane:

$$\frac{x}{c} = \frac{5}{50.1} = 0.1$$