# INVESTIGATION ON TURBINE BLADE'S PERFORMANCE BY USING 3-DIMENSIONAL CFD

**SIDDIIQ BIN AB RAHIM**

**This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Thermal-Fluid) with Honour**



**Faculty of Mechanical Engineering**

**UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

**JUNE 2017**

**DECLARATION**

I declare that this project report entitled "Investigation On Turbine Blade Profile Performance by Using 3-Dimensional CFD" is the result of my own work except as cited in the references

Signature :

Name : SIDDIQ BIN AB RAHIM

Date : 16/6/2017

# 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-Fluid) with Honour.

Signature

Name of Supervisor : DR. Yusmady Mohamed Arifin

Date : 16/6/2017

**DEDICATION**

To my beloved mother and father

**ABSTRACT**

The study of turbine blade profile performance by using 3-Dimensional CFD is very important because it determining the quality of turbine to produce the power output. Basically, there are two methods when study the turbine's blade profile performance, first is by doing the experimental method and second is by using simulation software such as CFD. In this study, using 3- Dimensional CFD simulation method is chosen, since this method is cheaper and save more time compare to experimental method. The geometry used for this analysis are NACA 8412, NACA 8413 and NACA 8414 that are referring to airfoil geometry. All the selected turbine 's blade profiles are conduct the analysis with the same parameter. The parameters are used for the turbine' blade analysis are velocity inlet of 265 m/s, angle of attack of 0 degrees, temperature of 1112 K and twist angle of 36 degrees. After conduct the simulation, the result is compare to determine which turbine 's blade has the best performance. Hence, the performance of turbine is evaluating based on the coefficient of lift, coefficient of drag, lift and drag coefficient, velocity streamline and also pressure contour. As result of simulation obtain, NACA 8414 is chosen to have the best turbine 's blade profile performance since give good result for all the coefficient of lift, coefficient of drag, lift and drag coefficient, velocity streamline and also pressure contour.

## ABSTRAK

***Kajian prestasi profil bilah turbin dengan menggunakan 3-Dimensi CFD adalah sangat penting kerana ia menentukan kualiti turbin untuk menghasilkan output kuasa. Pada asasnya, terdapat dua kaedah ketika kajian prestasi profil bilah turbin, pertama adalah dengan melakukan kaedah eksperimen dan kedua adalah dengan menggunakan perisian simulasi seperti CFD. Dalam kajian ini, dengan menggunakan 3- Dimensional kaedah simulasi CFD dipilih, kerana kaedah ini adalah lebih murah dan menjimatkan lebih banyak masa berbanding dengan kaedah eksperimen. Geometri digunakan untuk analisis ini adalah NACA 8412, NACA 8413 dan NACA 8414 yang merujuk kepada aerofoil geometri. Semua profil bilah turbin 's dipilih adalah menjalankan analisis dengan parameter yang sama. Parameter yang digunakan untuk turbin 'analisis bilah adalah halaju masuk 265 m / s, sudut serang 0 darjah, suhu 1112 K dan sudut twist 36 darjah. Selepas menjalankan simulasi, hasilnya adalah perbandingan untuk menentukan turbin 's bilah mempunyai prestasi yang terbaik. Oleh itu, prestasi turbin sedang menilai berdasarkan pekali daya angkat, pekali seretan, lif dan pekali seretan, halaju arus dan juga tekanan kontur. Sebagai hasil simulasi mendapatkan, NACA 8414 dipilih untuk mempunyai terbaik bilah prestasi profil turbin 'sejak memberikan hasil yang baik untuk semua pekali daya angkat, pekali seretan, lif dan pekali seretan, halaju arus dan juga tekanan kontur.***

# ACKNOWLEDGEMENT

First of all, I would like to convey my gratefulness to my supervisor, Dr. Yusmady Mohamed, for his assistance and his guidance towards the progress of this project. Throughout of this project, Dr Yumasdy has been helping me by sharing his knowledge and skill on how to manage a project. He's also monitoring my progress and guided in the right direction and offering encouragement. Without his guidance and assistance on my progress, obviously my project will be uncertain. Next, special thanks to lecturer of Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka for their advice and help, especially to Dr fudhail , Dr tee for sharing their knowledge about CFD.

My special appreciation and thanks to all my comrades for their invaluable support toward this project. Not to forget, many thanks to my dearest parent Mr Abdul Rahim bin Ismail and Mrs Faridah Binti Tahir for their unfailing encouragement and financial support given to me over the years. Lastly, I would like to express my gratitude to all parties for sharing their advice and knowledge. Thank you, your sincere help will be remembered for my entire life.

# CONTENT

**Table of Contents**

## LIST OF FIGURES

# LIST OF TABLE

## LIST OF ABBEREVATIONS

| | |
|---|---|
| NACA | National Advisory Committee for Aeronautics |
| CFD | Computational Fluid Dynamic |
| BC | Before Century |
| LHV | Lower Heating Value |
| GTCC | Gas-turbine combine-cycle |

## LIST OF SYMBOL

| | |
|---|---|
| $c$ | Chord length, |
| $l$ | length |
| $\rho$- | density of air |
| $Vrel^2$ | relative velocity |
| $F_D$ | Drag Force |
| $C_L$ | Coefficient of lift |
| $C_D$ | Coefficient of lift |
| $F_L$ | Lift Force |

# CHAPTER 1

# INTRODUCTION

## 1.1 Background

The investigation on the turbine blade's profile performance is associated to the concept of turbomachinery. Turbine, compressors and fans are wholly member of the same family of turbomachinery. A turbomachine is a power or head producing machine which employs the dynamic action of a rotating element the rotor; the action of the rotor changes the energy level of the constantly flowing fluid through the turbomachine (Yahya, 2005). The study of turbomachinery is involving in many engineering field such as gas turbine plant, steam turbine plant and many more. In this case, the study of turbine's blade profile is concentrate on gas turbine application.

Gas turbine plant is plant that consists of turbo-compressor, heat exchanger and turbine. The first patent that of gas turbine was introduce is in 1791 by John Barber, his invention was the most important element present in the modern day gas turbine, and his turbine was to design horseless carriage (David Gordon Wilson, 1998). The study of gas turbine continues until the present day. For the study of the turbine's blade profile performance, two methods are considered in this investigation, first is using experimental method or another way is using numerical simulation on turbine's blade performance by using CFD. The second are considered because of performing experiment is require more time and money. (Hadi karrabi, 2011).

Computational fluid dynamic (CFD) is a proven simulation tool which can be used to any field of engineering. The CFD software is extensively use to engineer so that they can perform analyse, and optimize various engineering designs. In this study, the discussion is focused on investigation on various type of standard turbine blade profile in term of its effects on performance. The turbine blade profile just be match to the other blade turbine profile and suggest the one has the best performance. The blade's profiles are used in this study are 3-Dimensional model that have fix turbine blade profile, fix incidence angle, fixed inlet fluid flow velocity but with various case with difference blade profiles. Then the turbine's blade profile is test on the wind tunnel. After that, to perform CFD simulation the data and the information from the journal are use as reference.

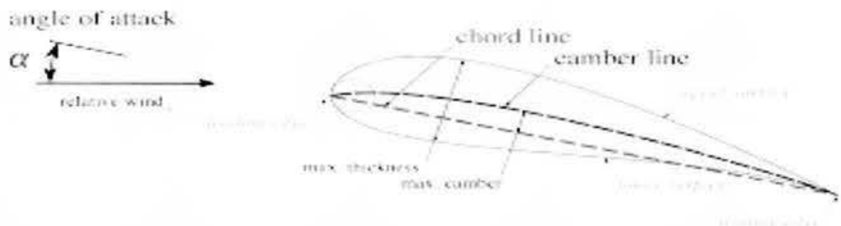



Figure 1.1 Airfoil Geometry (Kaurase2, 2016)

Next the journal and the internet source are used to obtain information for designing the turbine's blade profile. In the journal, most the turbine blade's profiles are referring to the airfoil profile shape. Airfoils are the detailed geometric shapes that are used to produce mechanical forces due to relative motion of the airfoil and a surrounding fluid. Most of turbines are using airfoil shape to improve the mechanical power. The shape of airfoil profile is depends on what application it used. For example the wind turbines normally the airfoil that have been used is NACA 6409 and NACA 4412 and for gas turbine use NACA 8412 (Kaurase2, 2016). Therefore, it is important to know study the previous journal before do the simulation using CFD.

## 1.2 Problem Statements

Study the flow of turbine's blade profile in 2-Dimensional is always play important part when designing a turbine blade. The use of 2-Dimensional turbine's blade profile model can provide much in formation when performs analysis using CFD. For example, the efficiency of turbine's blade, the coefficient of drag, the velocity flow through the turbine blade, the coefficient of lift and the angle of attack of turbine blade. Although the 2-Dimensional turbine blade can provide good analysis and many information, but there a certain analysis in 2-Dimensional could not be done when using CFD analysis. According to the (Hadi karrabi, 2011), There are many methods that can be used to simulate axial turbine such as 0D, 1D, 2D, quasi 3D and 3D. In order to achieve the main objective, the numerical model must enable us to input all the detail geometry. 3D simulation of turbomachinery has the highest accuracy. Therefore, the investigation on turbine's blade profile must be in 3-Dimensional because 3-Dimensional can perform twisted angle shape of model. In this project three turbine blade's profile are choose for the study of turbine's blade performance which are NACA 8412, NACA 8413 and NACA 8414 (Kaurase2, 2016).

## 1.3 Objective

The objectives of this study are as follows:

1. To investigate various type of standard turbine blade profile (twisted) in term of its effect on performance.
2. To suggest a better turbine's blade profile that has better performance compare to others.

## 1.4 Scope of project

The scopes of this project are:

1. The study is using 3-Dimensional CFD software in order to stimulate the fluid flow through a single blade profile.
2. The single turbine blade's profiles are used in this analysis are NACA 8412, NACA 8413 and NACA 8414.
3. The parameter set up for single turbine's blade profile are chord length of 10cm, span length of 50 cm, angle of attack $\alpha$ at 0 degrees and twisted of angle $\beta$ at 36 degrees.
4. The parameter geometry of test section for wind tunnel used for this analysis is in box shape with length of 100 cm, height and width of 50.6 cm.
5. The analysis is ANSYS FLUENT 16.0 as solver preference and the model use for this analysis is K-epsilon turbulence model.
6. Evaluation of the performance of turbine's blade profile are based on coefficient of drag, coefficient of lift and lift to drag ratio at angle of attack 10 degree in order to suggest the best performance of turbine's blade profile.

# CHAPTER 2:

# LITERATURE REVIEW

## 2.1 Introduction of Turbomachinery

The investigation on the turbine blade's profile performance is identified with the idea of turbomachinery. Turbine, compressors and fans are all individual from a similar group of turbomachinery. A head generating machine or power which uses the dynamic action of a rotating component of rotor is known as turbomachine, the energy level of the continuously flowing fluid through the turbomachine change the activity of rotor (Yahya, 2005). This turbomachine can produce change in enthalpy in a stream of fluid through it and exchange work through a rotating shaft; the interaction between the fluid and the machine is essential fluid-dynamic lift. This change in enthalpy also can be brought by the heat transfer. The overall enthalpy change in through the engine is almost negligible when considering the heat transfer (Wilson, 1998).

The first person who encourage of the early development of turbomachinery was Hero of Alexandria in 100 BC. He invented a device known as Aeolipile shown in figure 2.1 This device is operating when the container is heated the simple bladeless radial steam turbine is will spin The steam jets exiting the turbine can produce torque which is same concept as tip jet or a rocket engine (Meher-Homji, 2000). Therefore, the advancement of turbomachinery invented by Hero of Alexandria has encouraged many famous researchers to studies about turbomachinery.

Figure 2.1: Aeolipile Developed by Hero of Alexandria(Meher-Homji, 2000)

In 1500AD Leonardo Da Vinci study the device known as chimney jack as shown in figure 2.2 .This device has turbine which allow hot air passed over a fan like a blades and crude bevel gear use to rotate the spit (Meher-Homji, 2000).

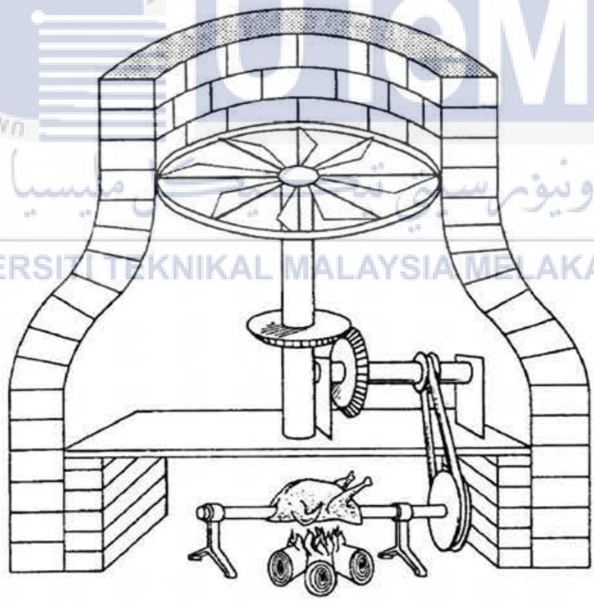




Figure 2.2: Chimney Jack (Meher-Homji, 2000)

In 1687, Sir Isaac Newton uses his formula of law of motion, which was fundamental to the development of all type of turbomachinery. The device name as "Newton Steam Carriage" shown in figure 2.3 and also known as "Steam Wagon". This device can utilize a reaction jet to provide forward movement. In this device there are a spherical boiler with a four-wheel carriage mounted over fire and nozzle that designed to provide a reaction jet (Meher-Homji, 2000).

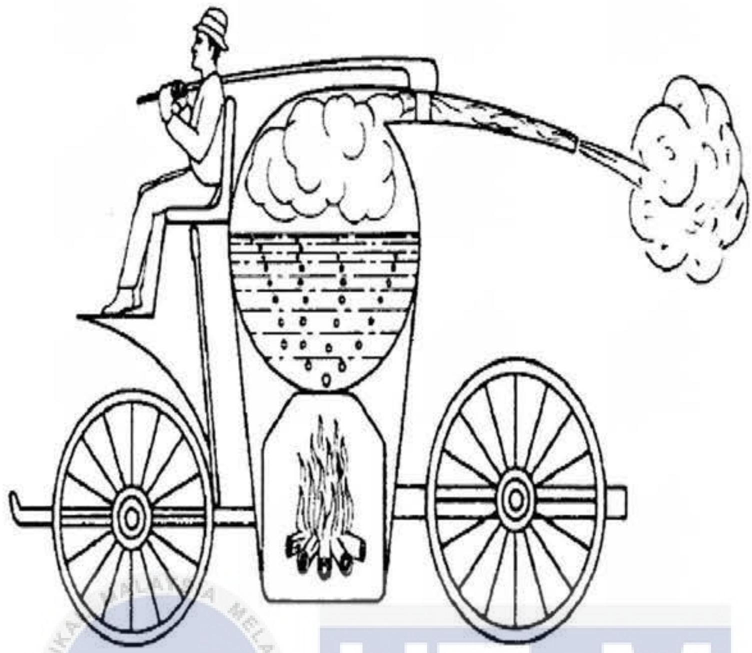


Figure 2.3: Newton's Steam Carriage (Meher-Homji, 2000)

The development of turbomachinery is known through the world, until now many applications that use concept of turbomachinery have been introduced such as Pelton turbine, wind turbine, gas turbine and many more. In this study, the investigation of turbine blade profile is applying on the gas turbine.

## 2.2 Gas Turbine

Gas turbine term is referring as an abbreviation for a gas turbine engine, which define as heat engine that accepts heat, reject heat and produces work. Basically the form of fuel that is burned is the input heat (giving rise the term as "combustion engine"), and other it also come from another process through a heat exchanger. The heat is rejected to atmosphere in the form of hot engine –exhaust flow, and also can be rejected through heat exchanger. Through all the process, the work may be given as output torque in turning shaft or as the velocity and pressure in a jet, which produce in larger size of compressor. The other term "gas turbine" also can be applied more narrowly for just the turbine expander. (Wilson ,1998)

The early design use thermodynamic cycle of the modern gas turbine was introducing by John Barber in 1791. In his design the turbine is equipped with a chain –driven reciprocating type of compressor and had a combustor and turbine. He then proposed the use of gas, charcoal or other suitable fuel to produce inflammable gas as shown in Figure 2.4: Patent Drawing of John Barber of Gas Turbine Cycle in 1791 (Meher-Homji, 2000).

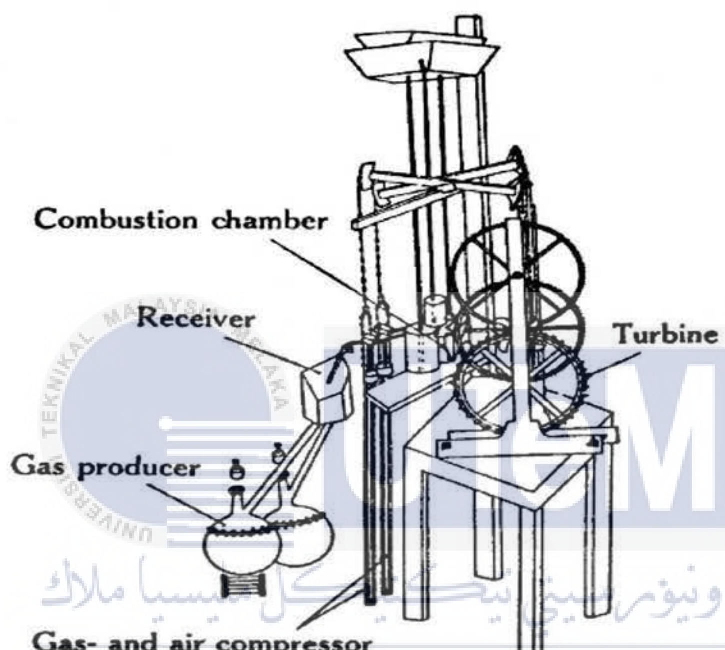





Figure 2.4: Patent Drawing of John Barber of Gas Turbine Cycle in 1791 (Meher-Homji, 2000)

In 1872 J. F(Franz) Stolze developed the first practical gas turbine with trials being made in 1900 and 1904. The design of his gas turbine was a single shaft unit supported by a bearing at each end and had a single silo combustor. The power of both end of the shaft belt drove alternators were rated at 150kW. There axial flow compressor comprised 10 stages (pressure ratio of 2.5:1) and the reaction turbine had 15 stages. The inlet temperature of turbine machine operated is 400°C and the efficiency of compressor is cover by 70 percent for the unit to be self-sustaining. Stolze's gas turbine uses a reheater with combustor being essentially a rudimentary coal gasifies. Some the heated air that come from recuperator was used to volatilize the coal. The recuperator was a U bank tube and linked to the compressor discharge plenum below the gas turbine. Finally, the gas turbine exhaust gas was routed under the machine to pass over 96 U tubes before discharging into main stack (Meher-Homji, 2000).

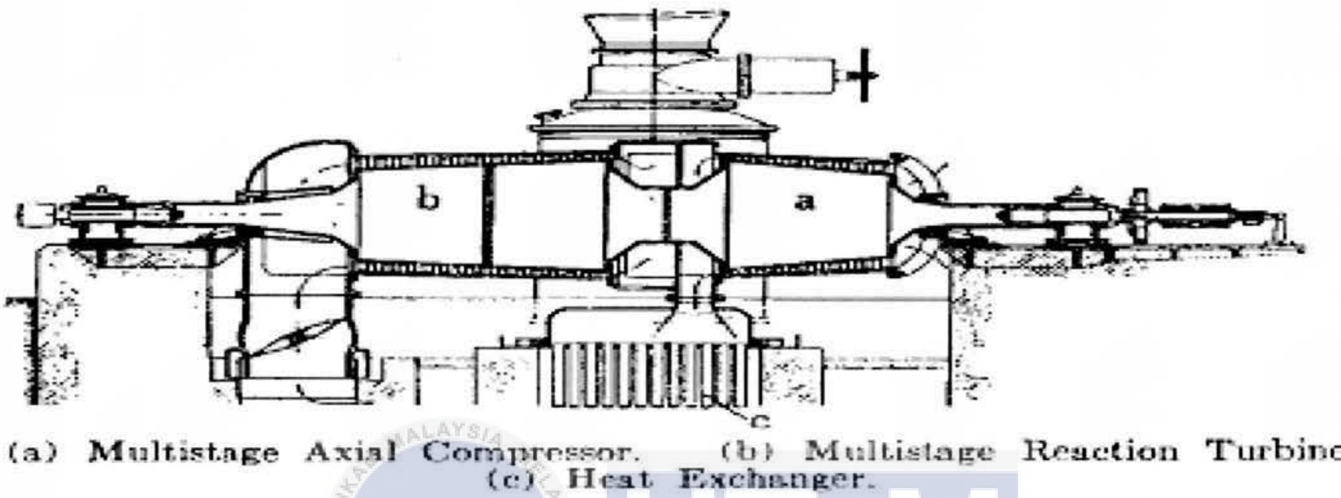



Figure 2.5 : Franz Stolze Gas Turbine Arrangement :(Meher-Homji, 2000)



In addition, the development of gas turbine still continues up until today. In 2011, the modern design of gas turbine by company known as Mitsubishi Heavy Industries Ltd was introduce. The gas turbine model is known as M501J model show in figure 2.6. This gas turbine is known as "Gas-turbine combine-cycle (GTCC) power generation because it is the cleanest and most efficient power generating system of all the fossil-fuel burning process. The developer claims that the GTCC power generations is demand on market because GTCC system can be constructed quickly and provide a stable source of electricity (Hada, 2013).

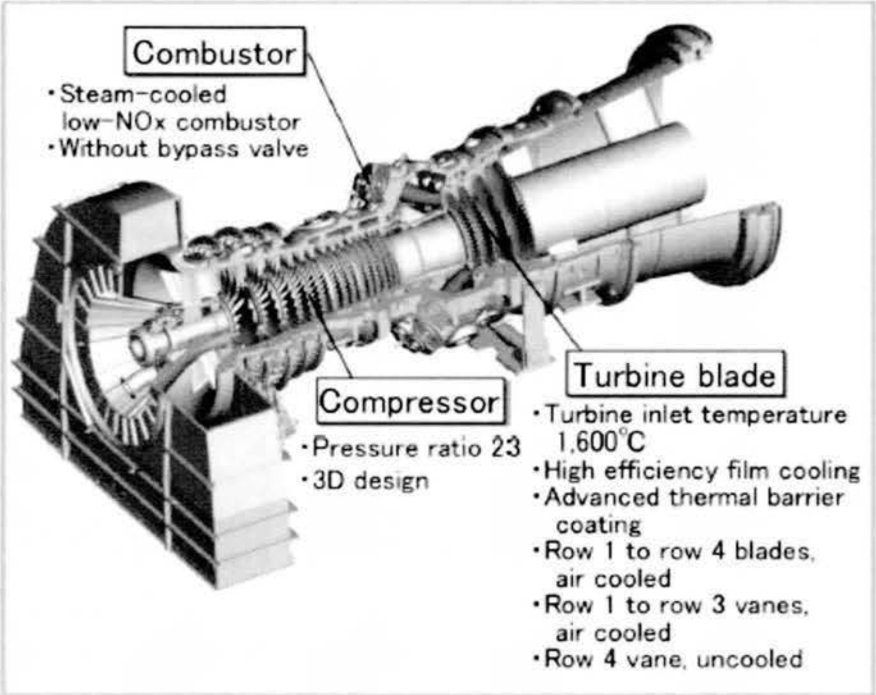



Figure 2.6:M501J Gas Turbine Features (Hada, 2013)

In feature the M501J gas turbine was designed with a turbine inlet temperature at 1600°C by integrating the proven component technologies utilize in the 1400°C F-series and the 1500°C G and H-series turbine. The previous development of gas turbine has given benefit to the M501J gas turbine for the national 1700 °C gas turbine project show in figure 2.7 The adaption of higher inlet temperature and newest component technology have given combine-cycle gross thermal efficiency significantly better than the conventional machine which is 61.5% on LHV (Lower Heating Value) basic shown in figure 2.8. Therefore, carbon dioxide emission approximately 60% can be reducing compare to a conventional coal-burning thermal power plant if use J-series combine-cycle power plant (Hada,2013).

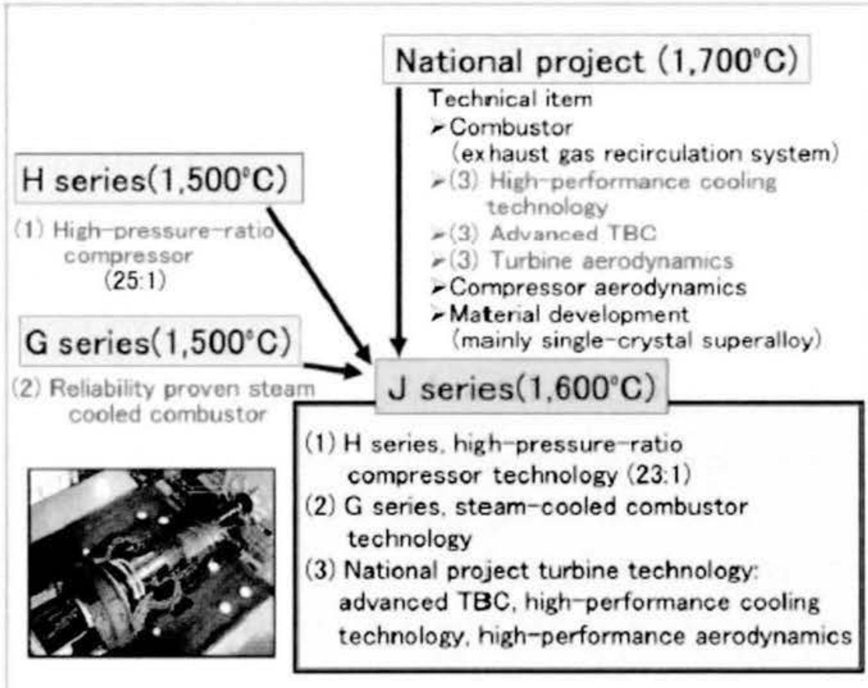



Figure 2.7:M501J Gas Turbine Concept (Hada,2013)



**2.3 Thermodynamic in Gas Turbine**

Gas turbines are being considered to become the first choice for future power generation systems, because of their high fuel conversion efficiency and reduced power generation cost. In recent years' various studies of gas cycle based power plants have shown that increasing turbine inlet temperature and cycle pressure ratio can substantially improve the overall power plant efficiency (S. Kumar, 2008)

George Brayton was the first person who proposed the Brayton cycle used in reciprocating oil burning engine (Yunus Cengel, 2015). He developed the cycle around in 1870. Nowdays; it is used for application of gas turbines only where both of the compression and expansion processes take place in rotating machinery. The process of gas turbine in thermodynamic cycle start when the heat release is convert into mechanical energy by burning the fuel, and it achieve by first compressing air in air compressor, then injecting and burning the fuel at ideal pressure, after that it expanding the hot gas in the turbine (Kurz, 2005) .

The use of turbine is to provide the power for compressor to operate. The power that left will use as mechanical output of the engine. In figure 2.8 shows the thermodynamic in an enthalpy-entropy (h-s) diagram. The process of air compression in engine is occurring at state 1 and state 2. Then in state 2 to state 3 is where the heat is added into the combustor. After that the hot gas is expand in single shaft turbine at for state 3 to state 7, while for two shaft engine the gas is expanded from state 3 to state 5 and after that state 5 to 7 in the power turbine (Kurz, 2005).

The work output describes the difference between line 1-2 and 3 -7. In this cycle most of the work generated by the expansion 3-7 and it used to provide the work to drive the compressor. For the two shaft engine the distance 3 to 5 and 1 to 2 are same because the compressor work is used by the gas generator turbine work output. Finally, the line 5 to 7 is describing work output of the power turbine (Kurz, 2005).

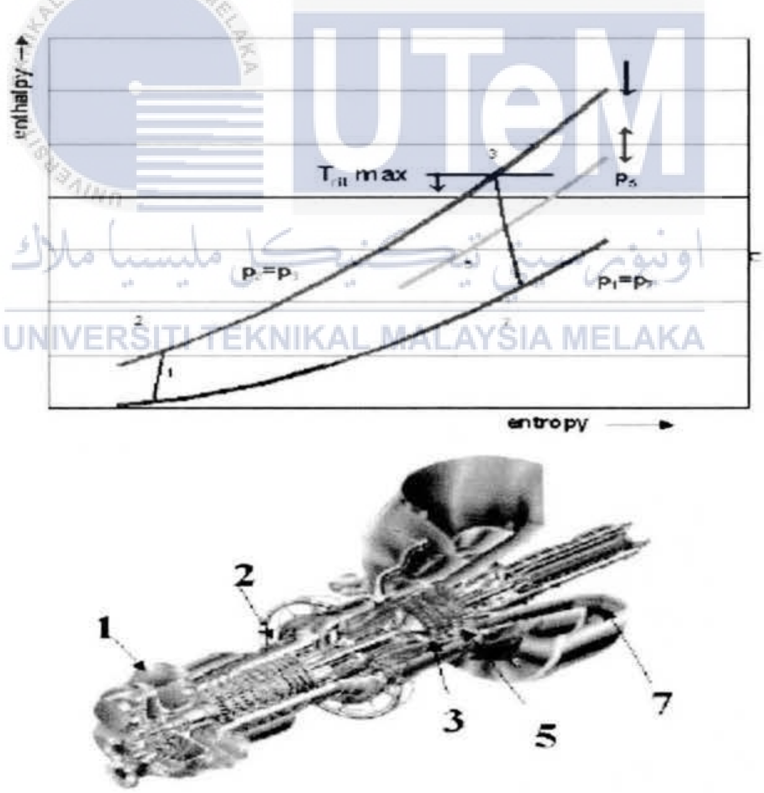





Figure 2.8: Enthalpy-Entropy Diagram for Brayton Cycle (Kurz, 2005).

## 2.4 Axial Flow and Radial Gas Turbine.

One of the applications that used in almost of gas turbine power plant is the axial flow gas turbine. The advancement of axial flow gas turbine was hindered by the need to increase both a sufficiently high flow rate and compression ratio from compressor to maintain the air use for the prerequisite combustion process and subsequent expansion of the exhaust gases. In turbine there are two categories of turbines which are the axial flow type and the radial or centrifugal flow type. Normally in industrial and shipboard application the use the axial flow turbine showed, in figure 2.9 a rotating assembly of the Rolls-Royce Nene engine of single-stage turbine. For this engine, the single-stage turbine is directly linked to main and cooling compressor. While for the more stage of axial flow turbine is located immediately to the rear of engine combustion chamber. In addition, the use of turbine is to extract the kinetic energy from the expanding gases as the gases is come from the burner, which alters the kinetic energy into shaft power to drive the compressor and the engine accessories (Gorla & Khan, 2003)



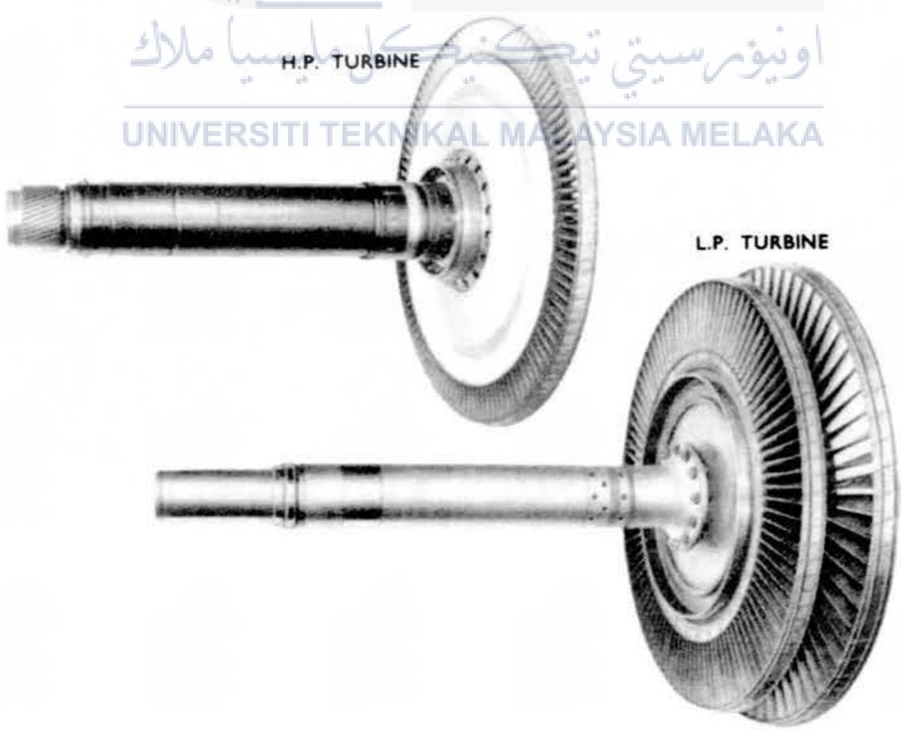



Figure 2.9: Axial Flow Turbine rotors(Gorla & Khan, 2003)

Next, for the radial turbine is appear commonly as centrifugal compressor but the diffuser the radial turbine use ring of nozzle guide vane to replace the diffuser vane. A radial turbine is a turbine which operational fluid enters radially to the shaft. Commonly, in radial flow turbine have two kinds of flow which are inward flow radial turbine and outward flow radial turbines. The flow of gas in radial flow turbine is tangential with high velocity and is directed inwards and leaves the rotor with small whirl velocity near the axis of rotation. Furthermore, when compare radial flow turbine with axial flow turbine, the radial turbine can handle low mass flows more efficiently than axial flow turbine. Due to that reason, the radial flow turbine is suitable used in cryogenic industry as turbo expander and also in turbochargers for reciprocating engine (Sondkar Pratik, 2016)

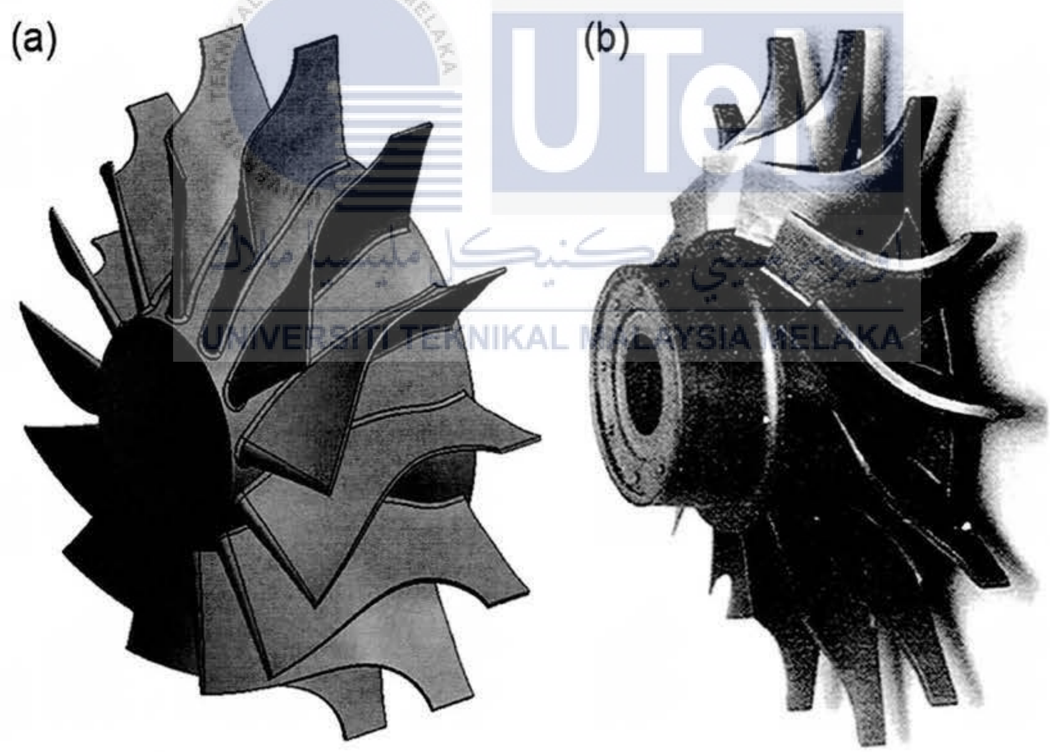





Figure 2.10: Radial Flow Turbine (He P, 2012)

## 2.5 Turbine blade Profile

The analysis of turbine blade exposes facts of various engineering considerations. Basically, turbine is known as combustion rotary engine that extract energy from a flow of combustion gas and it's involved with upstream mechanical device coupled to a downstream rotary engine, and combustion chamber middle. Next, energy is supplementary to the gas within the combustor; any fuel is mixed with air and ignited. The combustion of fuel will increase, within the air mass atmosphere of combustor. The product of the combustion square measured forced into rotary engine section. Then, the high speed and volume of the gas flow is directed through a nozzle over the turbine's blade, spinning the rotary engine that powers the mechanical device and, for some of other turbines, drives the mechanical output. Finally, the energy given up to the rotary engine comes from the reduction within the temperature of the exhaust gas.(Gendcha & Kaurase, 2016)

In addition, the turbine blades of gas are in charge of separating energy from the high temperature, high pressure gases. These blades are worked at elevated temperature in forceful situation and are subjected to huge centrifugal force. Upward of 42% of the failures in gas turbine were just because of blading issues and the failures in these turbine blade can have sensational impact on the safety and performance of the gas turbine engine (Rao et al ,2014)



### 2.5.1 Airfoil Geometry

For this investigation, the turbine's blade profile is referring to geometry of airfoil shape. Airfoils are specific geometric shapes structure that can generate mechanical force due to relative motion of the airfoil and a surrounding fluid. Airfoil also uses in many engineering applications such wing of airplane, blade of wind turbine, and compressor blade (Fazil & Jayakumar, 2011).

In general, airfoil plays important role in performance of the gas turbine as they are important component of the gas turbine. The characteristic design criterion of the airfoil helps to generate the whole body on its profile. The different design parameter such as chamber position, chord length, value of chamber, blade thickness will help to develop a unique blade profile which will perform better for the working condition (Gendcha & Kaurase, 2016)

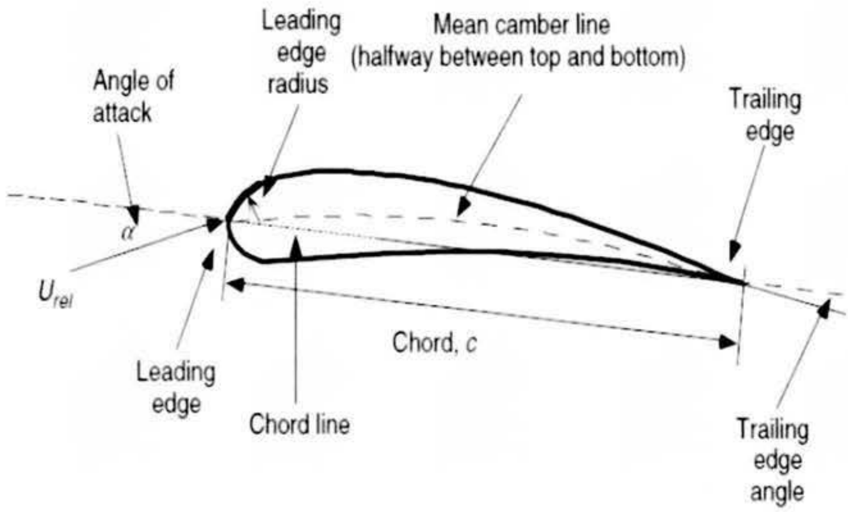



Figure 2.11: Terminology Airfoil(Patel, Patel, & Bhensdadiya, 2015)

For every airfoil contains an outer surface airfoil surface and outer airfoil surface. In geometry shape of airfoil, the inner airfoil and outer airfoil surface are representing a vane airfoil thickness(Castan,2003). Airfoil incorporate a main edge positioned along first outer and inner airfoil surface intersection, a trailing edge positioned along a second outer and inner surface are specially configured to give an airfoil camber line, positioned between the two airfoil profile and reaching out along the length of the airfoil that is significantly length of the camber line (Eppler., 1990).



### 2.5.2 NACA Airfoil

In this study, NACA airfoils are choose as turbine blade's profile. NACA is defining as National Advisory Committee for Aeronautics (NACA), it develops airfoil shapes for aircraft (E. N. Jacobs, 1935). The NACA airfoil shape is described by using the series of digit and following by the word of "NACA", for example NACA2412. The parameter is in the numerical code can be entered into equations and generate the cross section and calculate its properties. The NACA has distinctive series of digit airfoils: -

i. 4 Digit series of airfoil
ii. 5 Digit series of airfoil
iii. 7 Digit series of airfoil
iv. 8 Digit series of airfoil

Furthermore, the NACA airfoil can be generating by using NACA four-digit generator. For instance, NACA 2412, airfoil has maximum camber of 2% located 40 %( 0.4 chords) from the leading edge with a maximum (Fazil & Jayakumar, 2011). Figure 2.12. show NACA 4 Digit Generator (NACA 2412)

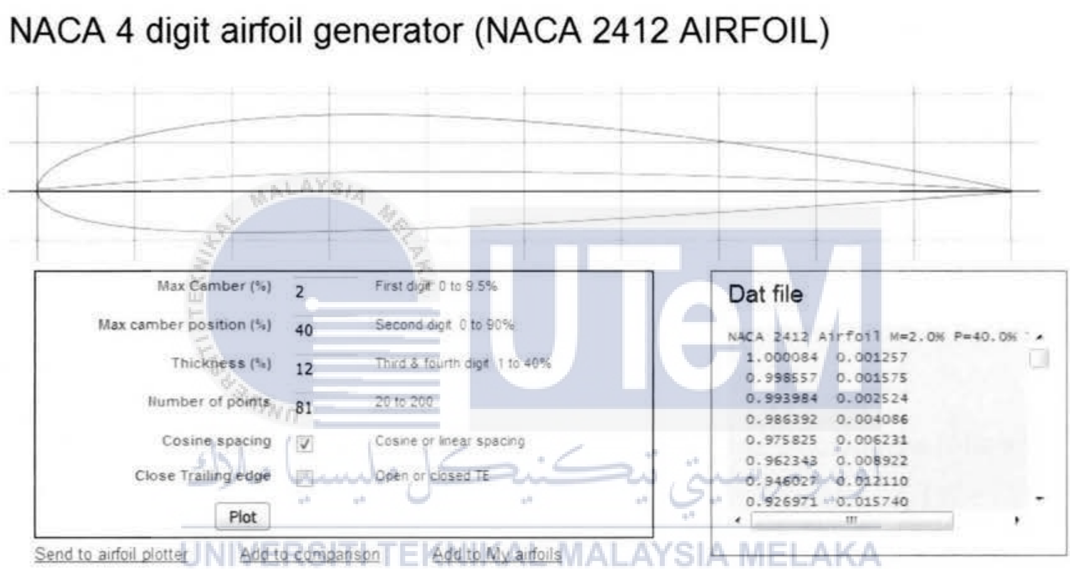



Figure 2.12: NACA 4 Digit Generator (NACA 2412)

### 2.5.3 Lift and Drag on Airfoil

lift and drag are defined with reference to airfoils. When velocity is flow over the over airfoil is increases over the convex surface, it is resulting the average pressure on the suction side of the airfoil lower compare with the concave side of the airfoil. Furthermore, viscous friction, friction between airfoil surface and the air decrease the air flow to some extent on the surface. The lift force on airfoil is well-defined to perpendicular to direction of the oncoming air flow. The lift force is a result of the unequal pressure on the upper and lower airfoil surfaces. While drag force on the airfoil is characterized to be parallel to the direction of the approaching air flow. The drag force is because both to viscous friction at the surface of the airfoil and to unequal on the airfoil surface towards and away from the oncoming flow. Figure 2.13 shows the Lift and Drag Force on Airfoil(Patel et al., 2015).

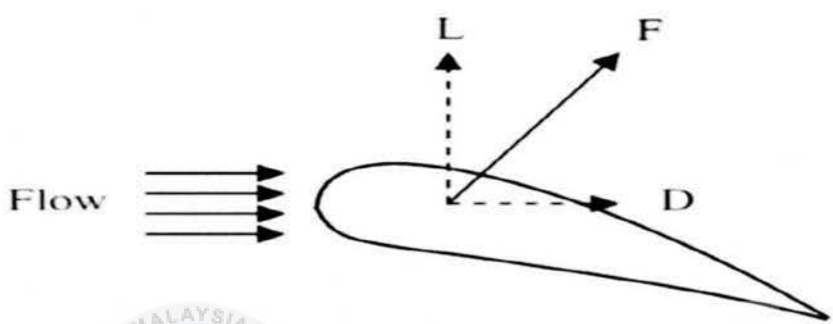



Figure 2.13: Lift and Drag Force on Airfoil (Patel et al., 2015)



Hence the lift and drag force of airfoil can be calculated by using the following formula.

$$F_L = \frac{1}{2} \times \rho \times C_L \times c \times l \times Vrel^2 \qquad (1)$$

$$F_D = \frac{1}{2} \times \rho \times C_D \times c \times l \times Vrel^2 \qquad (2)$$

## 2.6 Computational Fluid Dynamic

Computational Fluid Dynamic (CFD) is computational tool software for various analysis of engineering related application based problems. This software is mostly use by the engineer so that they can perform and analyse and optimize various engineering design. An example of the engineering problem that can solve by using CFD tools are relate to turbomachinery, building performance for different weather condition, storm analysis, heat exchangers, fully developed turbulent flow in pipes, analysis of fluid flow airfoil an air plane and using cooling electronic chips in the processor using force convection (Jathar 2015).

In CFD also, can provide a qualitative and quantitative of fluid flow by mean of mathematical modeling and numerical method. The CFD help the engineer to do analysis to compute the flow numerically in a 'virtual flow laboratory'. The engineer can use CFD to analyses the problem statement, mathematical modeling, mesh generation, space discretization, time discretization, iterative solve, stimulating run, post processing, and verification (Hossain, 2014).

The software that is going to be use in this CFD analysis is ANSYS software. ANSYS is vast computational software that provide researcher to analyse the problem related to different engineering sector. In this software there are program that can help engineer to solve the problem that related to heat transfer, fluid flow, turbulence, industrial machineries, explicit dynamics and structural with assistance of numerical analysis (Chervonenko,1991).



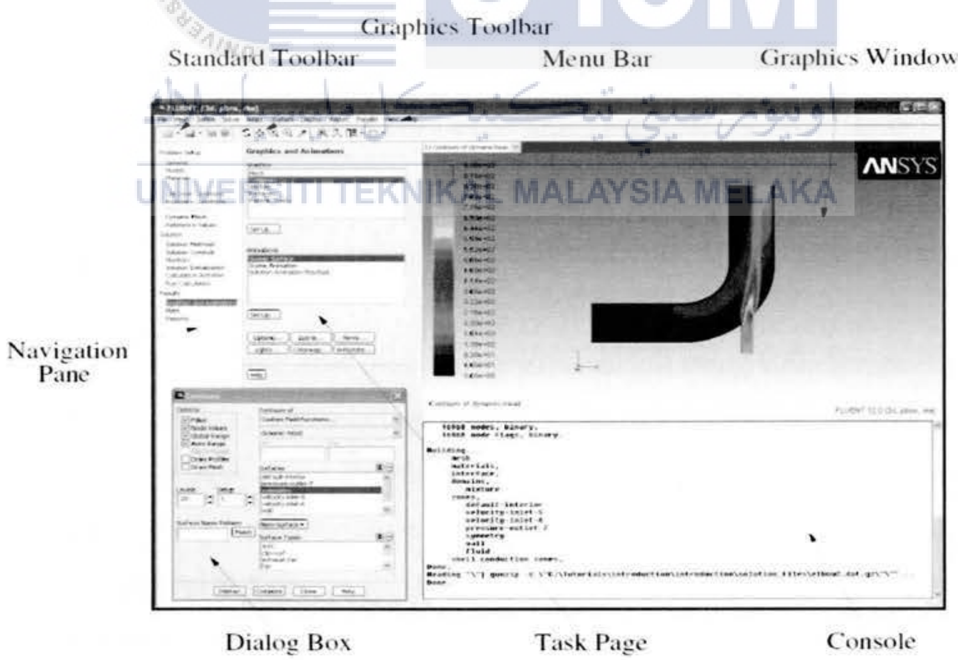



Figure 2.14: ANSYS FLUENT Software (Muthén & Muthén, 2009)

Basically there are two technique consider for investigation turbine's blade profile. First is by using experimental method or another method is by using numerical stimulating software. Because of performing experimental test is consuming more time and money, computer simulation is chosen to investigate the effect of turbine 'blade performance (Hadi karrabi, 2011). Table 2.1 shows the comparison between simulation of CFD and Experiment

**Table 2.1: Comparison Simulation (CFD) and Experiment** (Zuo, 2005)

| | Simulation (CFD) | Experiment |
|---|---|---|
| Cost | Cheap | Expensive |
| Time | Short | Long |
| Scale | Any | Small/Middle |
| Information | All | Measured Point |
| Repeatable | Yes | Some |
| Safety | Yes | Some Dangerous |

Based on the table, the cost of simulation of CFD is cheaper when compare with experiment and takes less time to do. Next the scale used for simulation CFD can be any size. Furthermore, this simulation can be repeated and= very safety. Finally, it provides more information compare with experiment method.



### 2.6.1 Analysis of Transonic Flow over Supercritical Airfoil Using CFD for Gas Turbine Blade

Analysis of transonic flow over supercritical airfoil using CFD for gas turbine is used as a reference for investigation on turbine's play using 3D CFD. In this analysis, the author uses NACA 8-series airfoil because this NACA has been widely used in many researchers for generating the optimum turbine optimum airfoil design for gas turbine blade(Gendcha & Kaurase, 2016). Thus NACA 8412, NACA 8413 and NACA 8414 are choosing for this study

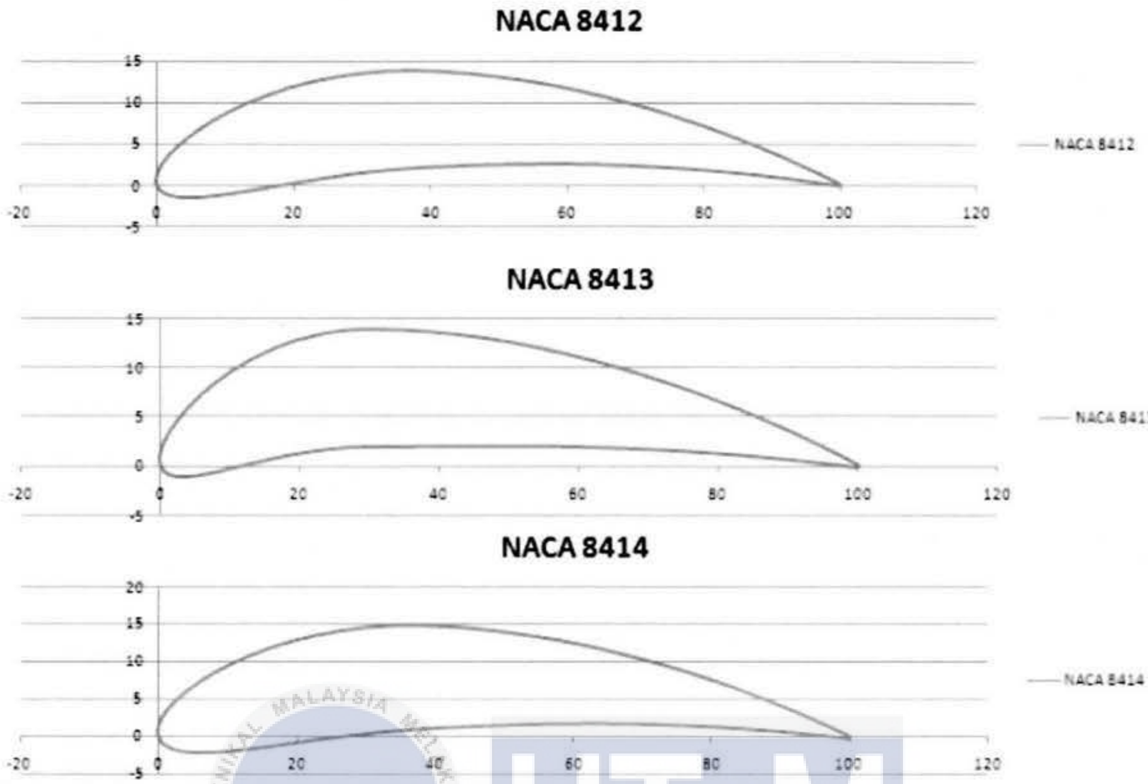



Figure 2.15: NACA 8-Series Supercritical Airfoils (Gandecha & Kaurase, 2016)



The purpose of this analysis is to identify the best airfoil for gas turbine, therefore the author creates a proper methodology is used for 2D simulation for this analysis as shown in Figure 2.16: Flowchart of Methodology for Analysis(Gendcha & Kaurase, 2016)s.

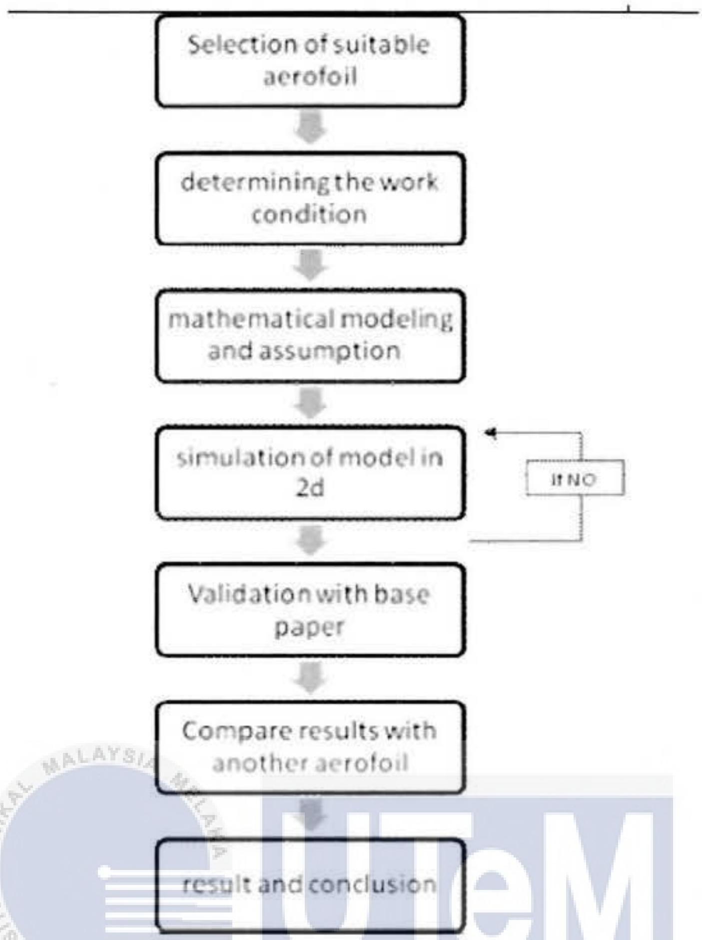



Figure 2.16: Flowchart of Methodology for Analysis

First of all, there are two categories of airfoil which are critical and non-critical airfoil. In this study the authors use REA2282(Kumar, 2015) airfoil as non-critical airfoil. Thus, the author compares the REA 2282 airfoil with NACA 8412, NACA 8413 and NACA 8414. The analysis parameter and boundary condition used is as shown in Table 2.2 (Gendcha & Kaurase, 2016).

. Table 2.2: Analysis of Parameter (Gendcha & Kaurase, 2016)

| **parameters** | **Values** |
|---|---|
| Mach No. | 0.729 |
| AOA (base) | 2.79° |
| Free stream pressure | 1 Atm |
| Free stream temperature | 300 |

In this analysis, the aerodynamic parameter need to be calculated are coefficient of lift and drag also location of shock wave for varying angle of attack. The angles of attack, α used for this analysis are 0° to 30°. Figure 2.17 shows the shock wave location on NACA 8412 airfoil for various angles of attack. The shock wave location for various angles of attack is applied on all selected airfoils.




Figure 2.17: The Shock Wave Location on NACA 8412 Airfoil for Various Angles of Attack (Gendcha & Kaurase, 2016)

After doing the iteration on selected airfoil, the result of selected airfoil is being compare with following factor. first is comparison of coefficient of drag of non-critical airfoil with reference. In this factor, the best suitable airfoil is the airfoil that has minimum coefficient of drag. Based on table 2.3 and figure 2.18, NACA 8414 has the minimum coefficient of drag(Gendcha & Kaurase, 2016).

Table 2.3: Comparison of coefficient of drag of non-critical airfoil with reference(Gendcha & Kaurase, 2016)

| **Cd** | | | | |
|---|---|---|---|---|
| **AOA** | **RAE2282** | **NACA8412** | **NACA8413** | **NACA8414** |
| 0 | 0.0111 | 0.02206 | 0.023315 | 0.02042757 |
| 5 | 0.03818 | 0.02449 | 0.050429 | 0.04807726 |
| 10 | 0.123 | 0.132315 | 0.132833 | 0.0131 |
| 15 | 0.2468 | 0.263543 | 0.3635436 | 0.2625303 |
| 20 | 0.33555 | 0.42249 | 0.4224585 | 0.42255459 |
| 25 | 0.4591 | 0.574338 | 0.57525499 | 0.5740471 |
| 30 | 0.54 | 0.6889468 | 0.68874011 | 0.68978 |

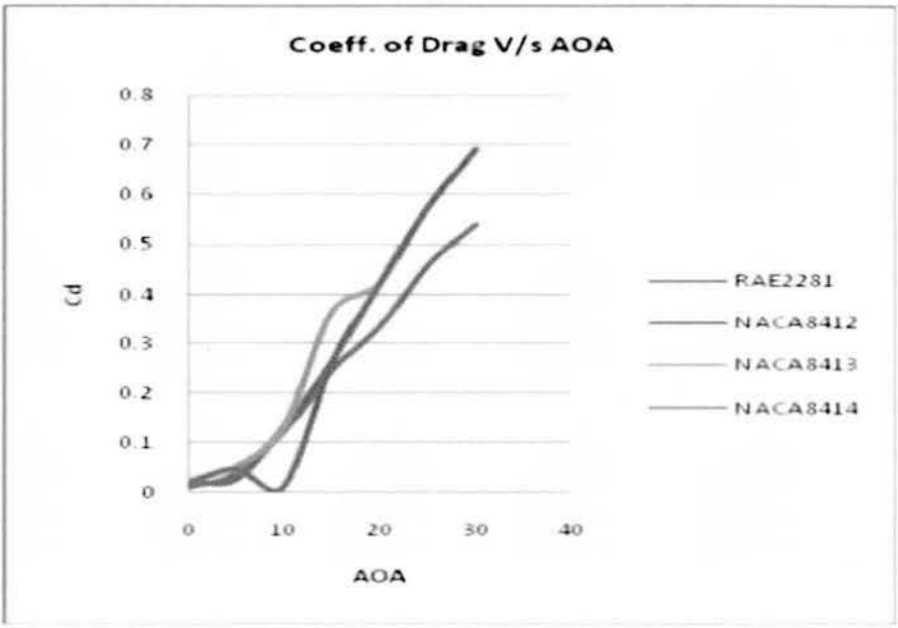



Figure 2.18: Graph of Coefficient of Drag versus angle of attack for all selected airfoil(Gendcha & Kaurase, 2016)

Next is comparison of coefficient of different airfoil lift at various angle of attack. In this factor, the best suitable airfoil is the airfoil that has maximum value of coefficient of lift. Based on table 2.4 and figure 2.19, NACA 8414 has the minimum coefficient of lift (Gendcha & Kaurase, 2016).

Table 2.4: Comparison of coefficient of different airfoil lift coefficient at various angles of attack (Gendcha & Kaurase, 2016)

| Cl | | | | |
|---|---|---|---|---|
| **AOA** | **RAE2282** | **NACA8412** | **NACA8413** | **NACA8414** |
| 0 | 0.00698 | 0.02861 | 0.0281731 | 0.02263889 |
| 5 | 0.3414 | 0.030742 | 0.35555 | 0.353405 |
| 10 | 0.6535 | 0.670527 | 0.66763665 | 0.6685788 |
| 15 | 0.8873 | 0.9502519 | 0.94724414 | 0.95054 |
| 20 | 0.8976 | 1.153737 | 1.1489945 | 1.1560971 |
| 25 | 0.9712 | 1.2324614 | 1.2296566 | 1.2351153 |
| 30 | 0.95388 | 1.191695 | 1.1904706 | 1.1922397 |

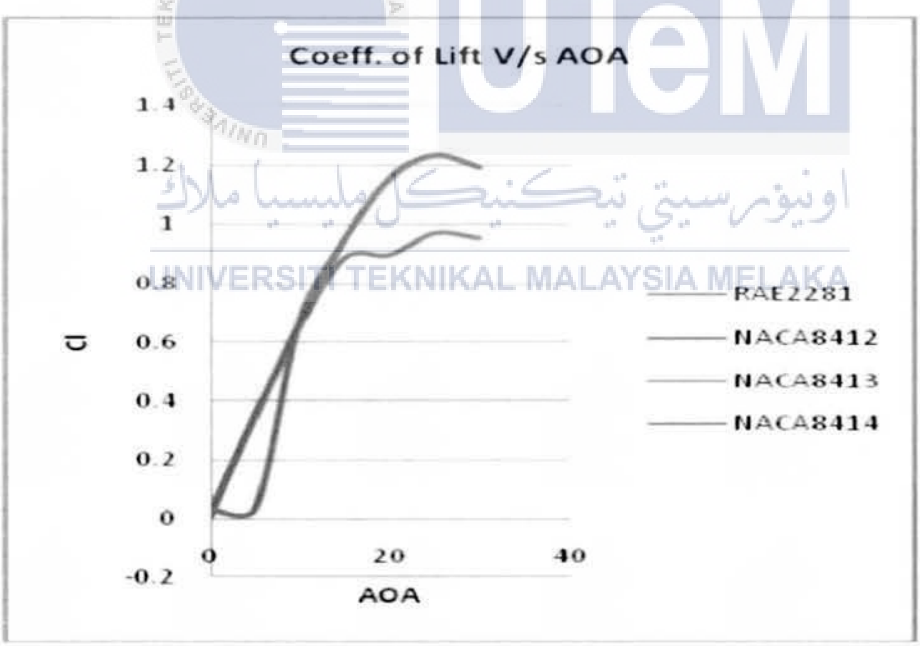





Figure 2.19: Graph of Coefficient of Lift versus Angle of Attack for All Selected Airfoil (Gendcha & Kaurase, 2016)

Lastly , is comparison of lift to drag ratio for various angle of attack of different airfoil this factor, NACA 8414 airfoils shows best characteristcics among all selected aifoil which has the highest lift to drag ratio an aangle of attack α=10°, shown as in table 2.5 and figure 2.20(Gendcha & Kaurase, 2016).

Table 2.5: Comparison of coefficient of lift over drag of non-critical airfoil with reference (Gendcha & Kaurase, 2016)

| Cl/Cd | | | | |
|---|---|---|---|---|
| AOA | RAE2282 | NACA8412 | NACA8413 | NACA8414 |
| 0 | 0.628829 | 1.2969174 | 1.2083680 | 1.1082517 |
| 5 | 8.941854 | 1.2552878 | 7.0505066 | 7.3507724 |
| 10 | 5.313008 | 5.0676567 | 5.0261354 | 51.036549 |
| 15 | 3.595219 | 3.6056806 | 2.6055860 | 3.6206868 |
| 20 | 2.675011 | 2.7308030 | 2.719780 | 2.7359709 |
| 25 | 2.115443 | 2.1458816 | 2.1375852 | 2.1515922 |
| 30 | 1.766444 | 1.7297344 | 1.7284757 | 1.7284347 |



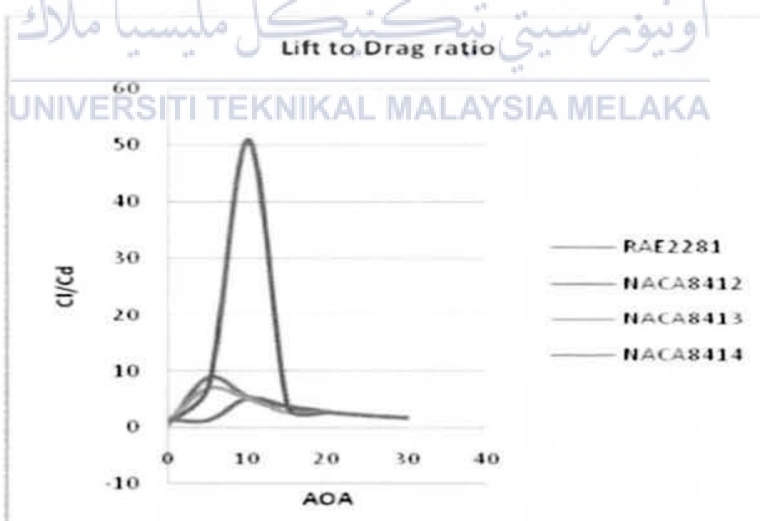



Figure 2.20: Graph of Lift to Drag Ratio versus Angle of Attack(Gendcha & Kaurase, 2016)

In this journal, the author has acquired different parametric outcome identified with coefficient of lift and coefficient of drag, moreover shock position of critical airfoil REA 2282 and other three supercritical airfoils. While considering the drag coefficient individually NACA 8414 was indicating slightest estimation of the drag which is alluring condition for the gas turbine blade profile, also in case of lift coefficient NACA 8414 demonstrate the best outcome but with regards to compare both result in well proportionate manner i.e lift to drag coefficient ratio the NACA 8414 airfoil demonstrate the best outcome which was exceedingly expected while looking at critical and non-critical airfoil in transonic zone condition(Gendcha & Kaurase, 2016). In previous study, the author used 2-Dimensional CFD to obtain the result of performance of gas turbine. But for this study, 3-Dimensional CFD are used for this study since the turbine blade have twisted angle geometry.

# CHAPTER 3

# GENERAL METHODOLOGY

## 3.1 Introduction

This chapter is describing the methodology used in this study and show the data need to use for CFD is obtain. When study the performance of turbine blade profile by using 3-Dimensiona CFD, there are a lot of information need to find out. One of information need to find out is the suitable parameter for construct the turbine blade and how to design it before doing the simulation. Normally, the turbine blade geometry is referring to the geometry shape that obtain form NACA airfoil. Another information need to be considered is to find out the boundary condition that need to use for the simulation such as velocity and the temperature. The information will ensure the objective of this project success. Therefore, Figure 3.1 shows the flow for this study.

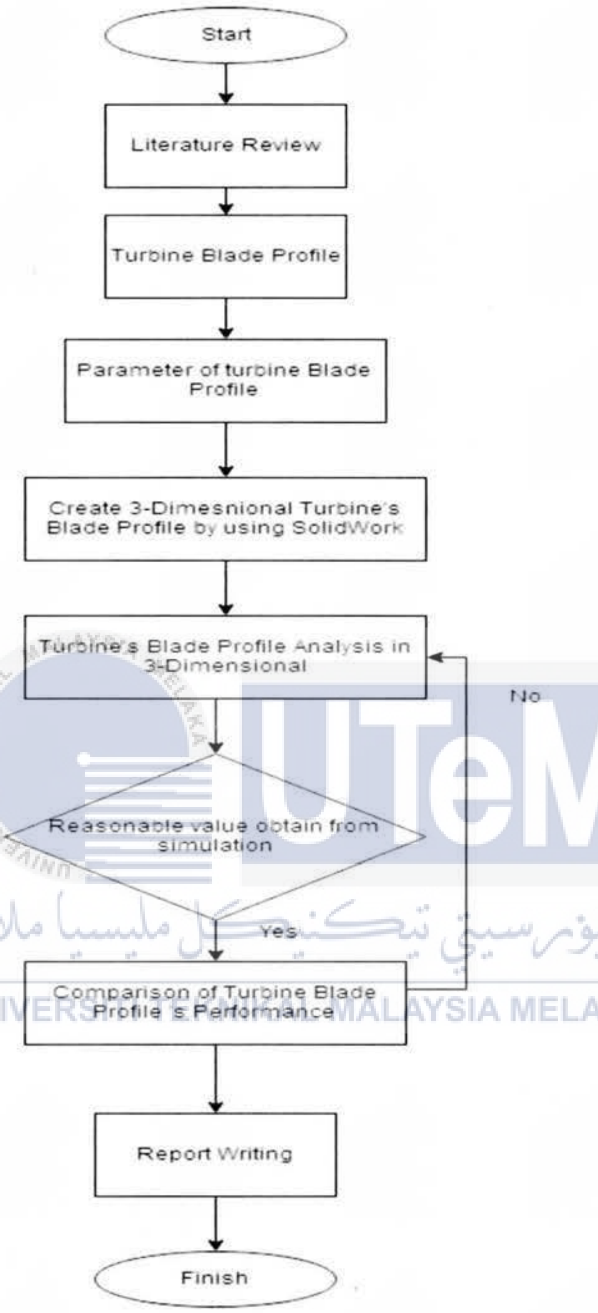
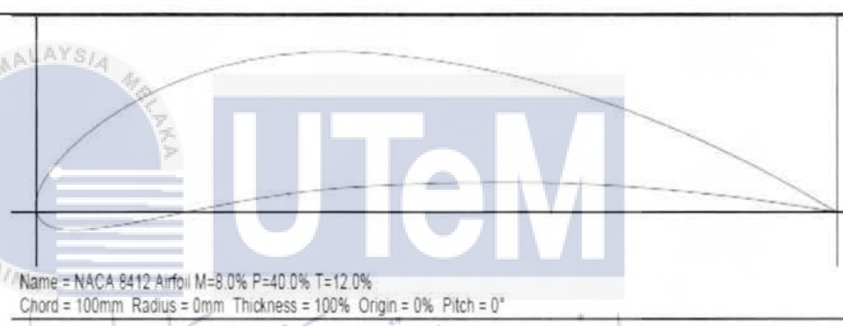
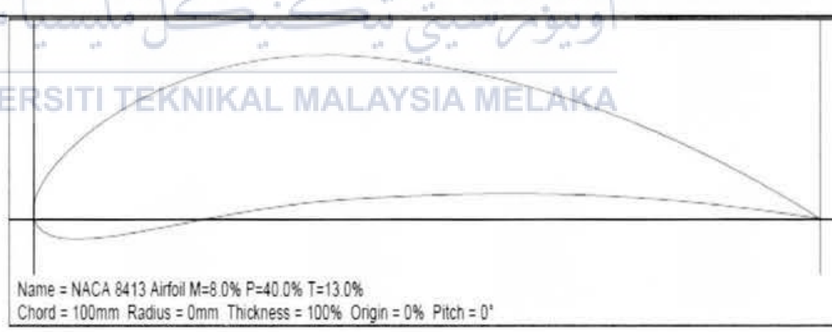
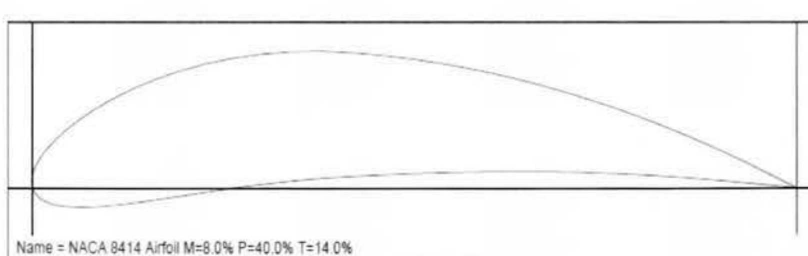



Figure 3.1: Flow chart

## 3.2 Turbine Blade Profile

The turbine's blade profile was choosing based on study from the literature review. In this study, the turbine's blades profiles chosen are NACA 8 series airfoil. This NACA is known as supercritical airfoil and has been widely used by many researchers for generating the optimum airfoil for the gas turbine. Based on (Kaurase2, 2016), the airfoil that have been choose in this investigation is NACA 8412, NACA 8413 and NACA 8414. Based on their study, they used 2D simulation for the analysis of the airfoil to identify best airfoil for the gas turbine blade profile.

Table 3.1: Turbine 's Blade Profile Parameter

| **Turbine Blade Profile** | **PARAMETER** |
|---|---|
| NACA 8412 |  |
| NACA 8413 |  |
| NACA 8414 |  |

### 3.3 Parameter of Turbine Blade profile

The parameter of turbine's blade profile is based on study of journal and website. According to (Jayakumar, 2011) the airfoil parameter can be generating by using website (http://www.airfoiltools.com/index, n.d.). For example, the NACA 8412 airfoil has a maximum chamber of 8% located at 40 %( 0.4 chord) from the leading edge of 12 % . (http://www.airfoiltools.com/index, n.d.).

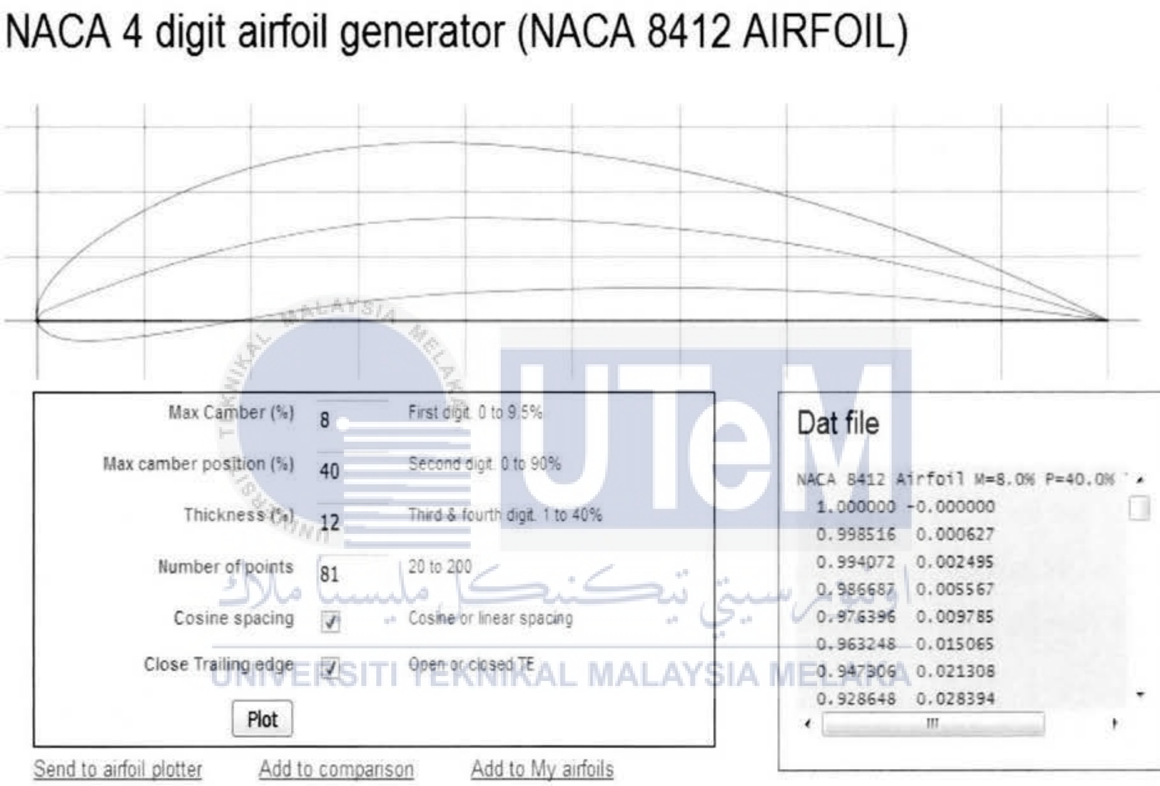





Figure 3.2: NACA 8412 Airfoil is Generate Using Airfoil Generator. (http://www.airfoiltools.com/index, n.d.)

**Table 3.2:** Parameter of all turbine blade profile

| **Parameter** | NACA 8412 | NACA 8413 | NACA 8414 |
|---|---|---|---|
| **Maximum Camber (%)** | 8 | 8 | 8 |
| **Maximum Camber Position** | 40 | 40 | 40 |
| **Thickness (%)** | 12 | 13 | 14 |
| **Number of Points** | 80 | 8 | 80 |
| **Chord length (cm)** | 10 | 10 | 10 |
| **Pitch or Angle of Attack, α(citation)** | 0°(Gandecha & Kaurase, 2016) | 0°(Gandecha & Kaurase, 2016) | 0°(Gandecha & Kaurase, 2016) |

Table 3.1 shows the parameter use for this analysis. Based on table 3.1 all the turbine ‘s blade parameter has the same properties for maximum camber, maximum camber position, number of points, angle of attack and chord length. The only things different are the percentage of thickness of the thickness.

| Airfoil surface | |
|---|---|
| X(mm) | Y(mm) |
| 96.592583 | -25.881905 |
| 96.465467 | -25.782932 |
| 96.084557 | -25.487478 |
| 95.45073 | -24.999608 |
| 94.565866 | -24.32583 |
| 93.432523 | -23.475525 |
| 92.054225 | -22.459889 |
| 90.435399 | -21.292529 |
| 88.581736 | -19.988533 |
| 86.500063 | -18.564535 |
| 84.19859 | -17.038574 |
| 81.686609 | -15.429291 |
| 78.97532 | -13.755939 |
| 76.077026 | -12.038278 |
| 73.005684 | -10.296092 |
| 69.776631 | -8.549022 |
| 66.406794 | -6.816199 |
| 62.914245 | -5.11634 |
| 59.31855 | -3.467213 |
| 55.640792 | -1.885959 |

Figure 3.3: NACA 8412 airfoil at angle of attack of 10°generate using website. (http://www.airfoiltools.com/index, n.d.)



Figure 3.6 show the data file obtain for NACA 8412. This data file is obtaining from NACA airfoil generator. The data file is important, because it will be used in SolidWork to create a 3- Dimensional geometry.

### 3.4: Create 3-Dimensional turbine's blade profile by using SolidWork

The turbine's blade profile of NACA 8412, NACA 8413 and NACA 8414 model is create by using SolidWork. The Data file coordinate that obtain is used in SolidWork to create the geometry of NACA 8-series airfoil. Figure 3.7 shows the data file used to create the parameter of turbine blade profile of NACA 8412.

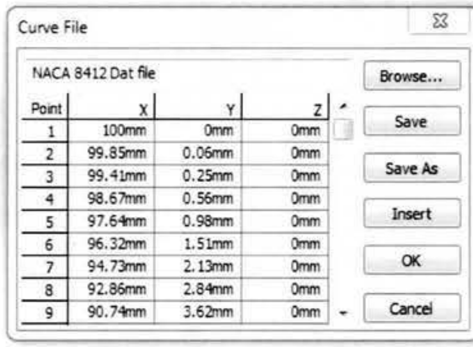



Figure 3.4: Coordinate of NACA 8412 creates by using Curve file in SolidWork.

Next the NACA of 8-series airfoil is convert into 3-Dimensional by using "swept boss" feature in SolidWork. The span length for NACA 8-series airfoil is 50 cm and the angle of twist, $\beta$ is 36° (Ainley & Mathieson, 1951).




Figure 3.5: NACA 8412 in 3-Dimensional with Twisted Angle (create by author)

### 3.5 Turbine Blade Profile Analysis in 3-Dimensional

The 3-Dimensional model of turbine's blade profile that creates by using SolidWork are used in ANSYS FLUENT 16.0 software to perform the analysis. In ANSYS FLUENT 16.0 software there are several procedures before getting the result of performance result. The following are the needed procedure need for analysis of turbine blade profile performance: -

I. Geometry

In geometry, the turbine blade profile that create by using SolidWork is import into ANSYS geometry. Then the geometry of turbine blade profile is generating and appear in ANSYS geometry as shown in figure. Next the wind tunnel is creating by using primitive box feature in ANSYS geometry, the parameter use for are wind tunnel are 100.3 cm for length and 50.6 cm with including tip clearance of 0.6 cm width and 50cm is for height. The turbine blade's profile will locate at the centre of wind tunnel as shown in figure 3.9. Finally, Boolean operation feature is use to subtract the target body (wind tunnel) and tool body (turbine blade profile) as shown in figure 3.10.



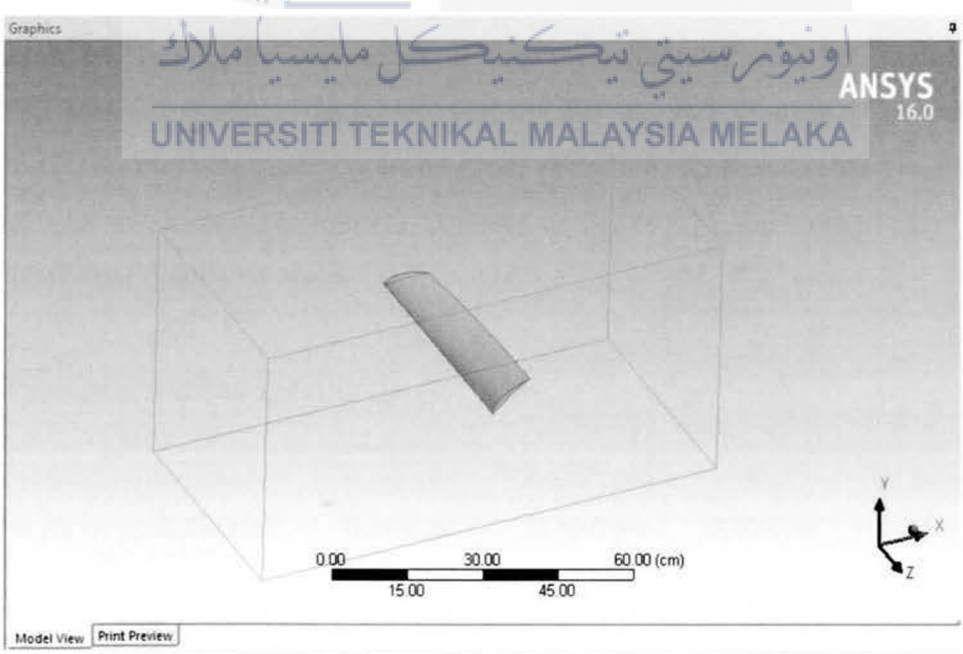



Figure 3.6: Test section of Wind Tunnel creates in ANSYS FLUENT Geometry.

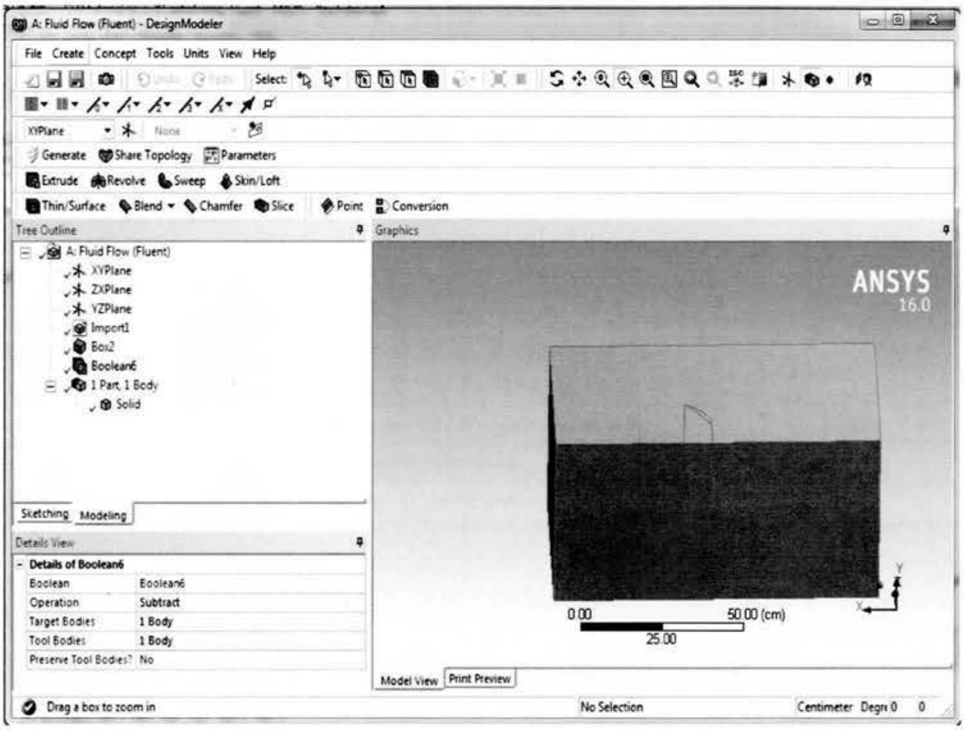



Figure 3.7: Example of Boolean Operation in ANSYS FLUENT.

II. Meshing

In CFD grid is known as mesh, mesh is the collection of all elements or cells. The mesh analysis is the step before setting the parameter used for Fluent setup analysis. Meshing is play on important part in CFD analysis, in order to get the good CFD result, the meshing quality must be good. Therefore, skewness is used to measure the quality of mesh shown in figure 3.11 Based on figure 3.11, the mesh that has lower value has better quality of mesh.

**Skewness mesh metrics spectrum**

| Excellent | Very good | Good | Acceptable | Bad | Unacceptable |
|---|---|---|---|---|---|
| 0-0.25 | 0.25-0.50 | 0.50-0.80 | 0.80-0.94 | 0.95-0.97 | 0.98-1.00 |

Figure 3.8: Skewness mesh metric spectrum(Zuo, 2005)

First step in mesh analysis is to set up the name selection. The name selection is used to determine which part of the body are inlet, outlet, wall, airfoil wall and fluid as shown in figure 3.13. Next step is to determine what option need to set up for detail of meshing as shown in table 3.2. In detail meshing, the option is set up as default, the Physics Preference use is CFD because of CFD is suitable for fluid flow analysis, the Solver Preference use is Fluent s. Next, Used Advanced Size Function is set up on Curvature, and Relevance Center used is coarse shown in table 3.2.

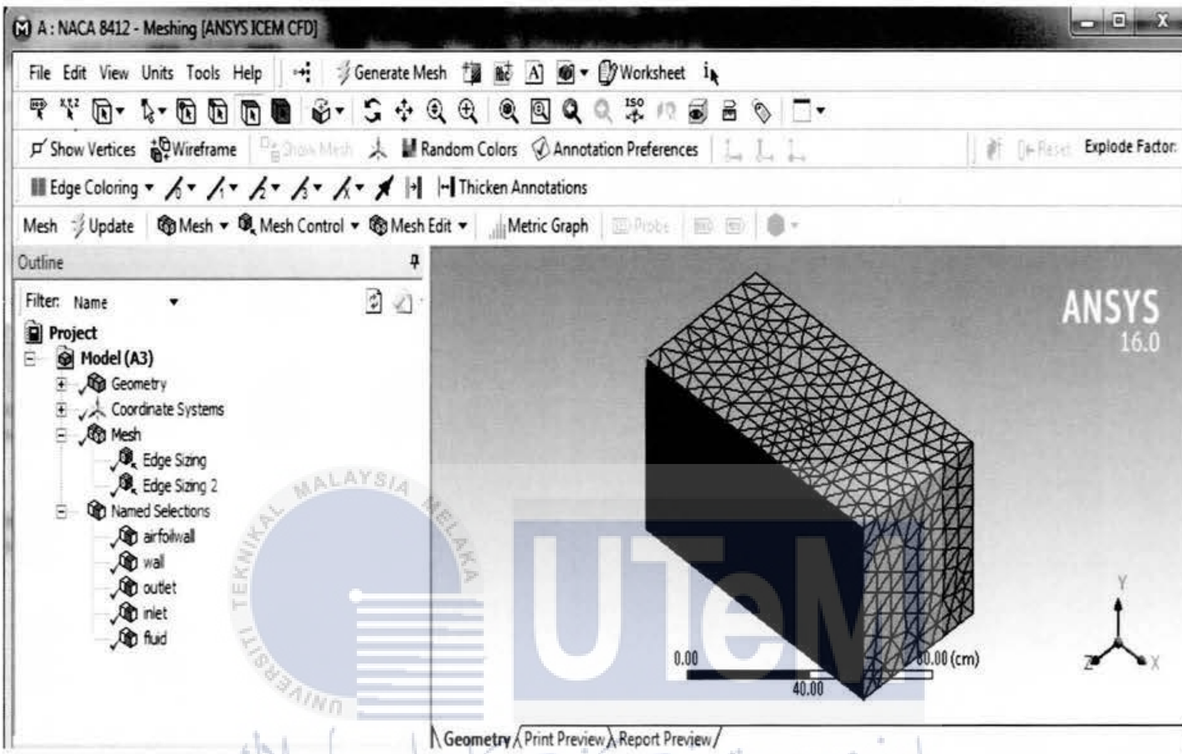



Figure 3.9: Name Selection



Table 3.2: Example of Meshing in ANSYS FLUENT

| | |
|---|---|
| **Physics Preference** | CFD |
| **Solver Preference** | Fluent |
| **Use Advance Size Function** | Curvature |
| **Relevance Center** | Coarse |

Initially the meshing is generating automatically for the first attempt, but it gives high value of skewness. Since the value of skewness is high, the meshing is set up with tetrahedral method with some mesh control to get lower value of skewness. The mesh control used for this analysis is sizing and it apply only at edge sizing of the airfoil wall shown in figure 3.13. Next, the number of divisions of edge sizing need to be set up in order to get the best value of skewness as shown table 3.3. After setting up all the parameter for meshing figure 3.14 how the final meshing used for this analysis.

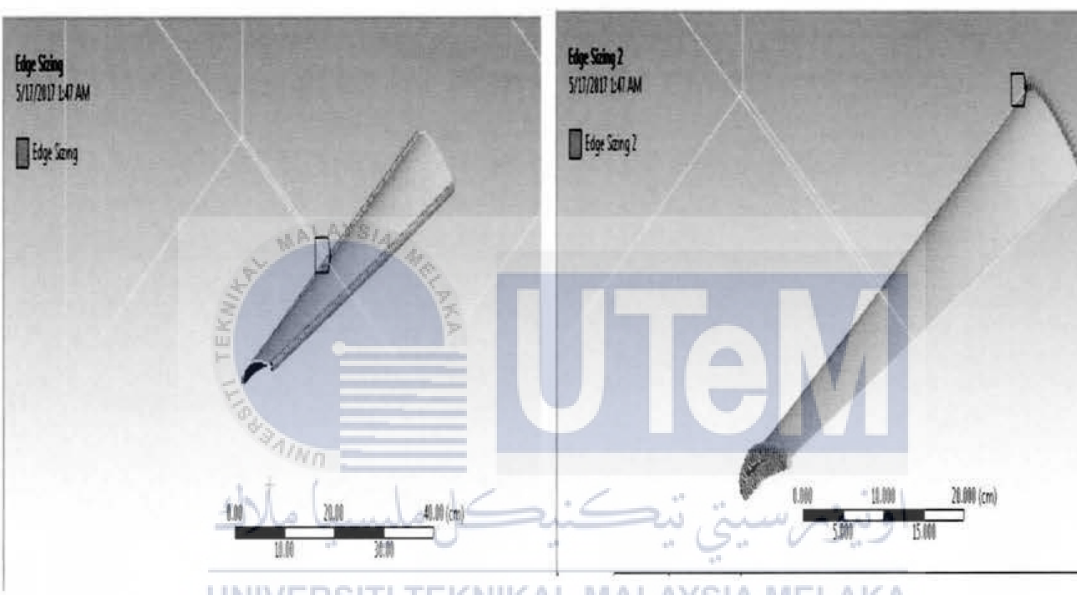





Figure 3.10: Edge sizing

Table 3.3: Parameter of Edge Sizing

| Sizing | Number of divisions |
|---|---|
| Edge Sizing | 80 |
| Edge Sizing 2 | 20 |

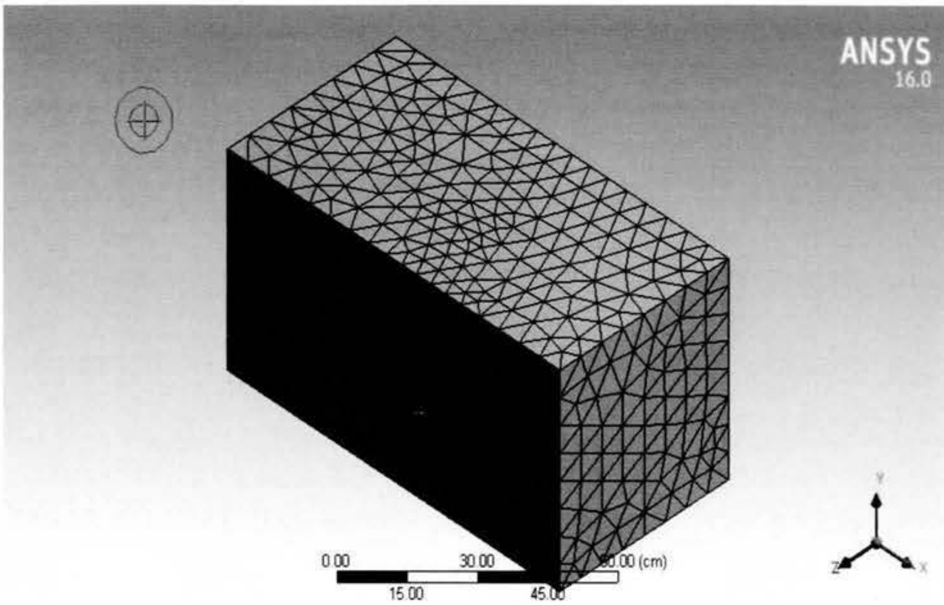



Figure 3.11: Final Meshing



III. Fluent Setup

Fluent setup is where all the important parameter such as temperature of turbine blade, fluid use for turbine blade, velocity inlet and pressure of turbine blade is located. In meshing analysis, the model is used tetrahedral mesh, but in fluent the tetrahedral mesh is converted into polyhedral mesh shown in figure 3.15. The advantage of polyhedral model in this analysis is used to reduce the iteration time and also improve the quality of skewness meshing.

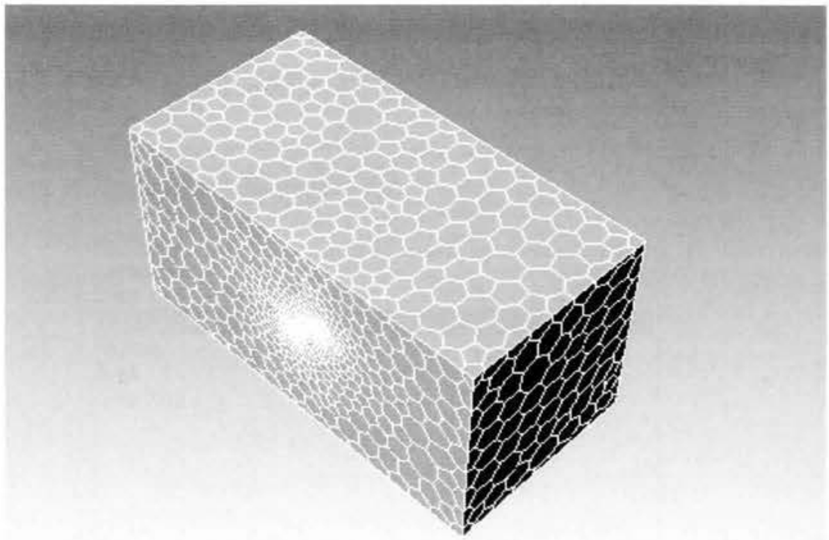


Figure 3.12: Polyhedral Meshing

Next in fluent analysis is to set up the model need to use for fluent analysis. The model use is k-epsilon 2 equation model and the energy model shown in figure 3.16. The energy model need to turn on so that the temperature can be inserted into boundary condition. Then, define the fluid as air for this analysis



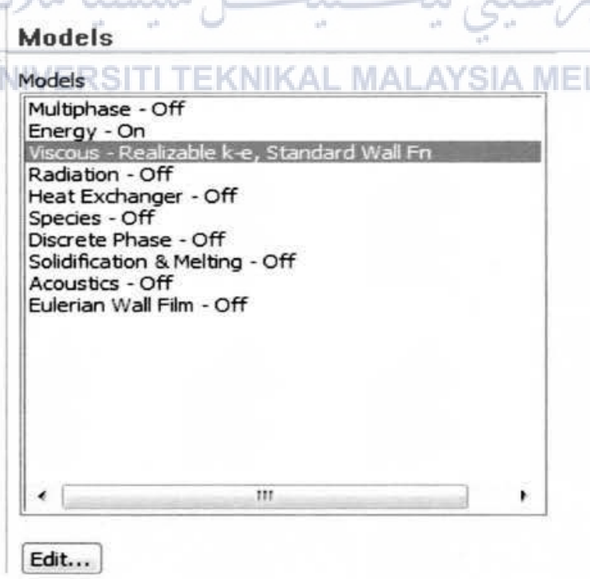



Figure 3.13: Model in Fluent

. After that, the setting up boundary condition used for this analysis as shown in figure 3.17. The boundary condition is where the temperature and the velocity is used to set up for this analysis. The velocity and the temperature used for this analysis are 265 m/s and 1112 k and it set up on the inlet boundary condition.

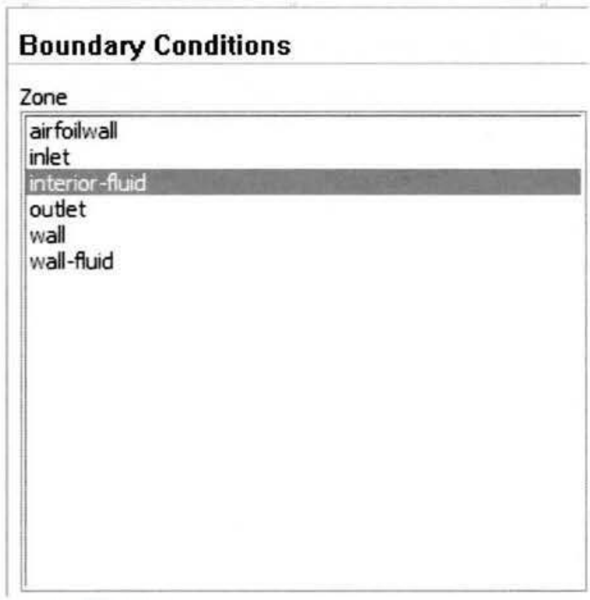



Figure 3.14: Boundary Condition

Then, the solution method used for this analysis need to be determine. In this analysis the solution model used is the second order because it has more accuracy compare with first order. Finally, the number of iteration used for calculate in the simulation. The number of iteration set up for this simulation is 1000 iteration because is the range where solution is fully converged for this analysis.



### 3.6 Performance Turbine's Blade Profile

The performance of turbine blade profile is determining after finish the iteration calculation on the ANSYS FLUENT setup. The performance of turbine 's blade profiles are evaluated based on: -

- Velocity Streamline
- Pressure Distribution
- Coefficient of lift for angle of attack $\alpha=0^{o}$
- Coefficient of drag for angle of attack $\alpha=0^{o}$
- Ratio of coefficient of lift and drag for angle of attack $\alpha=0^{o}$.

$$Ratio\ of\ coefficient\ of\ lift\ and\ drag = \frac{C_L}{C_D}$$

Ratio of coefficient lift and drag is equation used to determine whether the turbine's blade profile has good performance or not. The higher the ratio of coefficient lift and drag the better performance.

## 3.7: Comparison of Turbine Blade Profile Performance

The Result of performance of each turbine's blade profiles are compare in term of their performance and suggest which turbine's blade profile has better performance.

# CHAPTER 4

# RESULT AND DISCUSSION

## 4. 1 Result

For this chapter, the discussion is focused on the result that obtains from the CFD simulation. The simulation that has been carried out is involved with three different cases. For each case has different turbine blade profile. All the three turbine blade profile are carried out using the same parameter, which are velocity of 265 m/s, temperature of 1112 K, angle of attack of 0 degree angle, and twisted angle of 36 degree by using 3-Dimensional CFD .



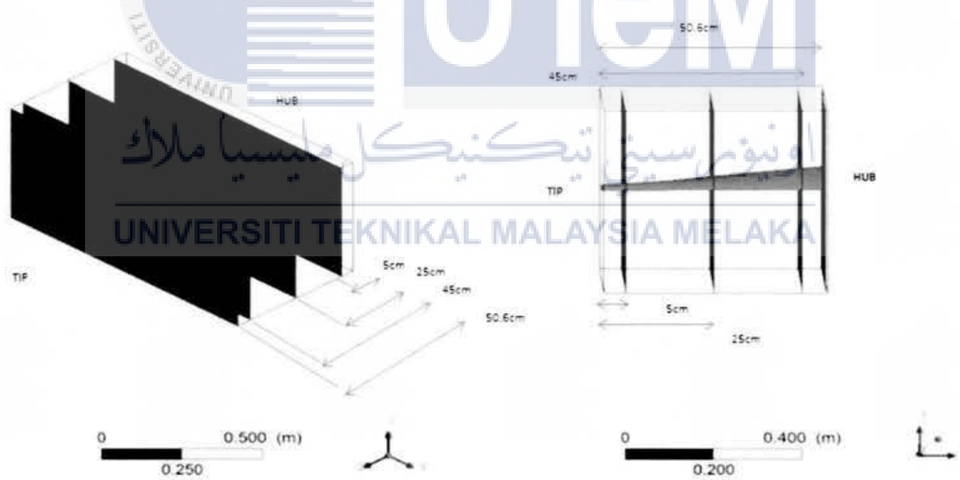



Figure 4.1: 4 Plane Section from Hub to Tip.

Since the turbine blade profile is in 3-Dimesional geometry, Figure 4.1 show how the result of turbine blade profile is presented. Based on figure 4.1 the turbine blade profiles are cut into four plane section from hub to tip of turbine blade profile. The first plane distance from hub is 5 cm, the second plane distance is 25 cm, the third plane distance from hub is 45 cm and the fourth plane distance from hub is 50.3 cm. The fourth plane is known as tip clearance gap. These planes are used to obtain result for velocity streamline and the pressure distribution.

### 4.2 Result of Analysis

There are three cases which are NACA 8412, NACA 8413 and NACA 8414 turbine blade profile need to discuss. The discussion is based on velocity streamline, pressure distribution and coefficient of lift and drag.

#### 4.2.1 Velocity Streamline (NACA 8412)

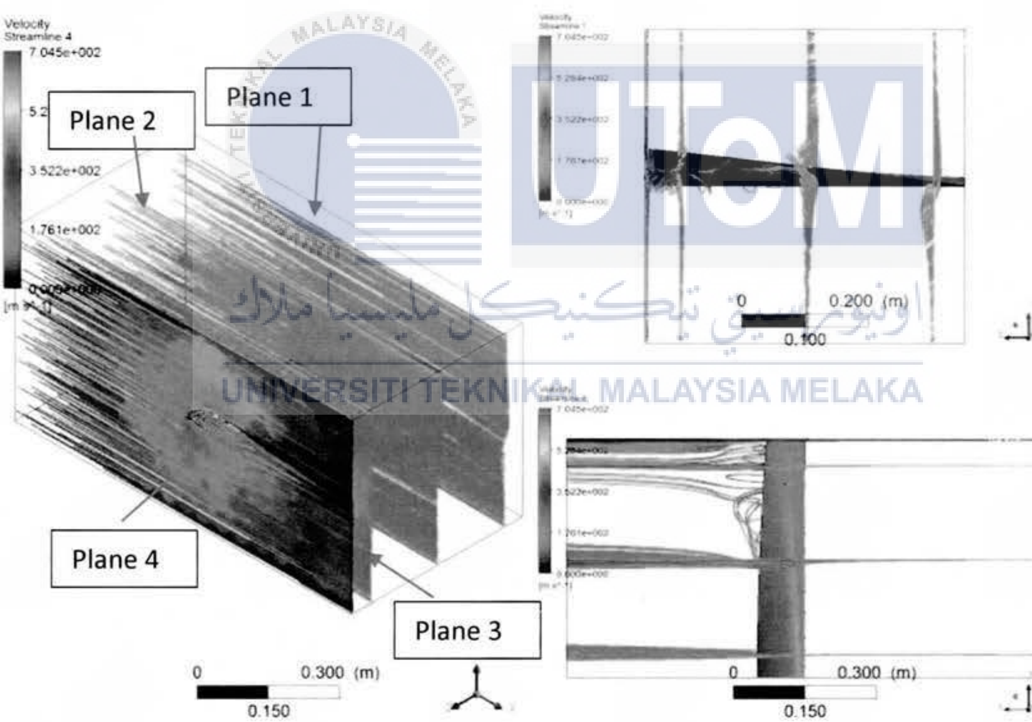





Figure 4.2: Velocity Streamline of NACA 8412 in 3-D View

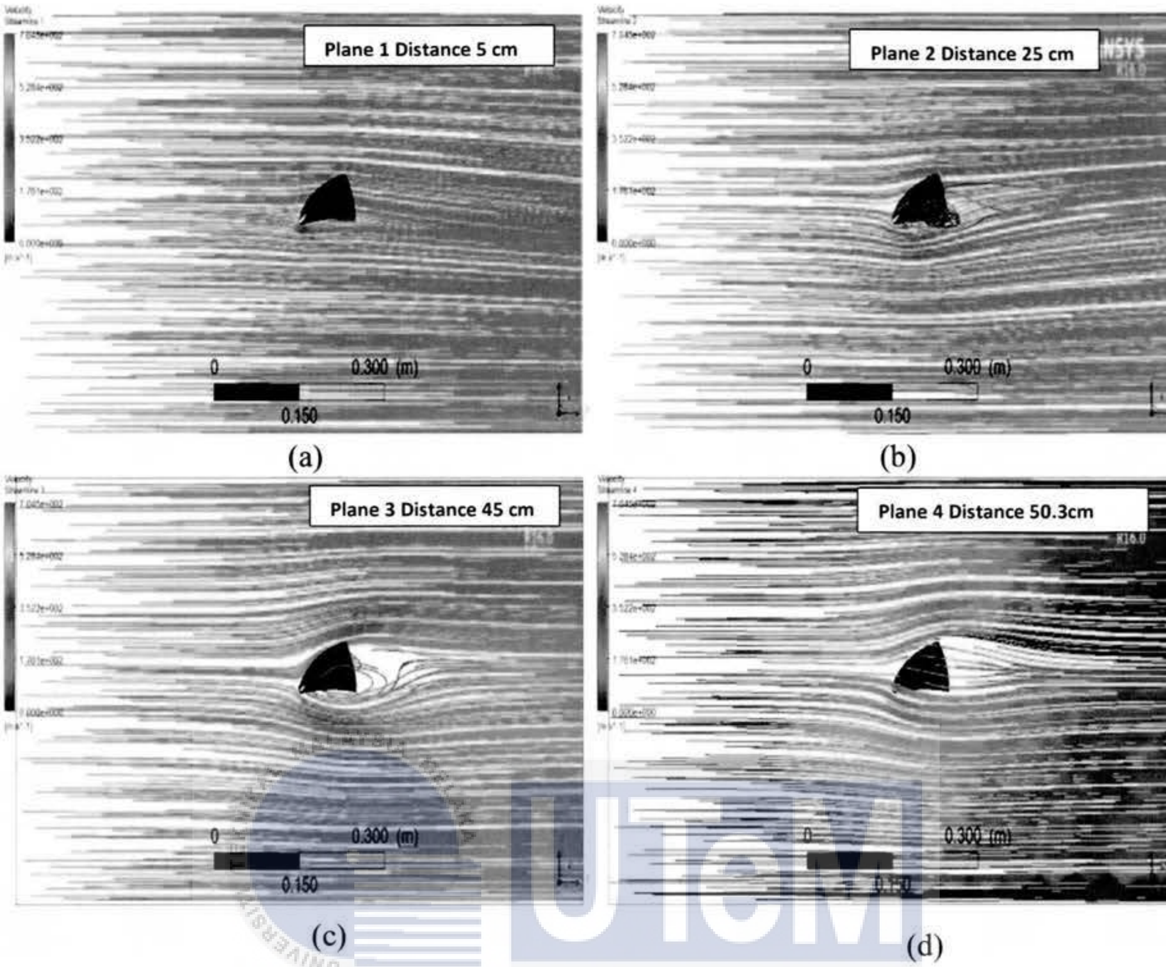



Figure 4.3: Velocity Streamline of NACA 8412 (4 PLANE SECTION)



Figure 4. 3(a) show velocity Streamline 1 of NACA 8412. The velocity streamline 1 with distance of 5cm shows the flow move smoothly without turbulence, the velocity that moves around turbine blade is 177. 1 m/s and has maximum velocity at leading edge with 352 m/s, there is no turbulence occur at velocity streamline because the shape of turbine blade profile does not twist and have angle of attack of 0 degree.

Figure 4.3(b) shows velocity streamline 2 of NACA 8412. At velocity streamline 2 with distance 25 cm shows that the flow starts to turbulence after the flow move past the leading edge. The velocity that occur at turbulence is below 177.1 m/s, this is happen because of the shape of turbine blade profile at 25 cm have twisted angle, and have higher angle of attack cause the turbulence occur.

Figure 4.3(c) shows the velocity streamline 3 of NACA 8412. For velocity streamline 3 with distance 45 cm also shows that the flow moves past the leading edge have turbulence, the reason why the turbulence occur at velocity streamline 3 is because of the shape of turbine blade profile is twisted and have high angle of attack. Based on figure 4.6, velocity streamline 3 is more turbulence compare with the velocity streamline 2 because of velocity streamline 3 have higher angle of attack compare with velocity streamline 2.

Figure 4.3(d) shows the velocity streamline 4 of NACA 8412. For velocity streamlines 4 with distance 50.3 cm (tip clearance) shows that the flow move toward the direction has turbulence but have less velocity compare with other streamline this is because the air flow is not turbine blade profile, and the turbulence that occur at velocity streamline 4 is due to the flow that pass through at the end of tip of turbine blade profile.

### 4.2.2Velocity Streamline (NACA 8413)

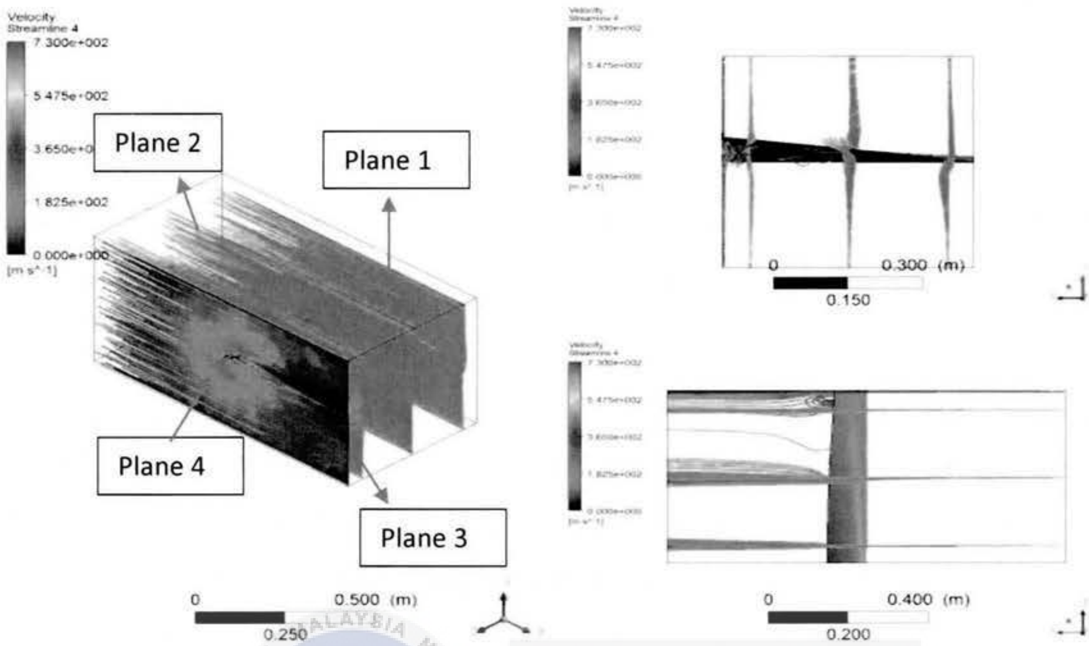



Figure 4.4: Velocity Streamline of NACA 8413 in 3-D View



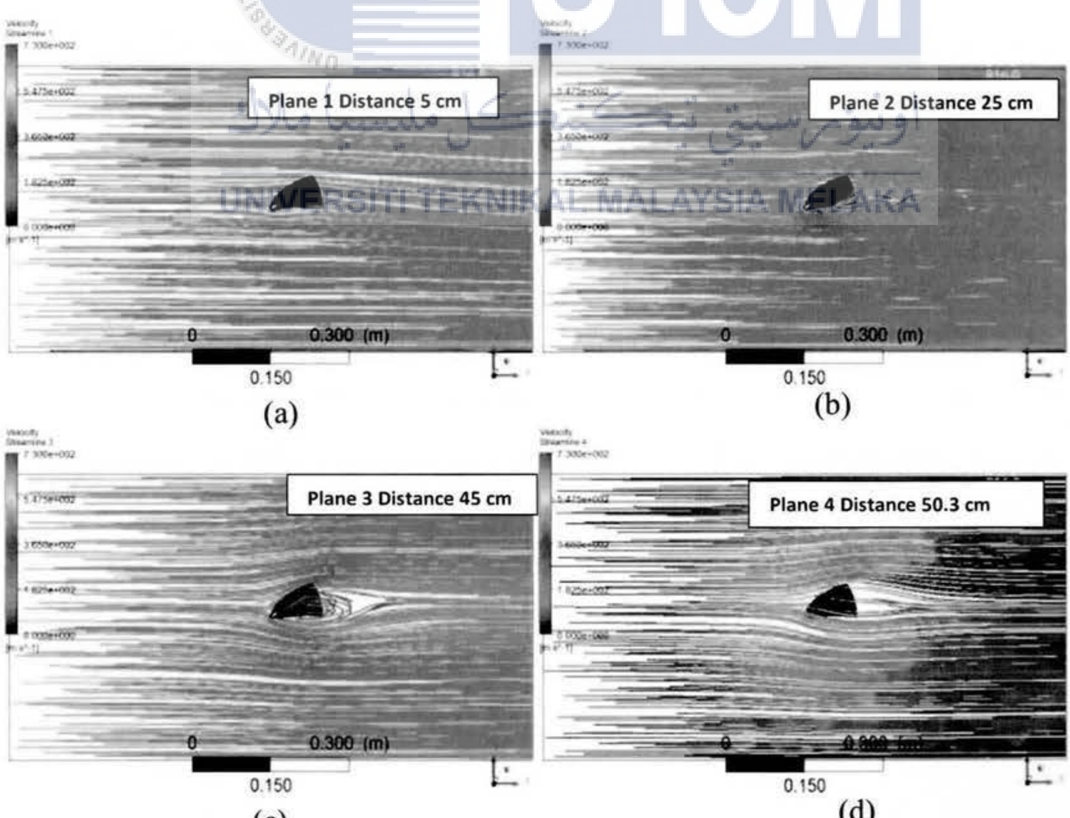



Figure 4.5: Velocity Streamline of NACA 8413 (4 PLANE SECTION)

The velocity streamline 1 shown in figure 4.5 (a) with distance of 5cm shows the flow move smoothly without turbulence and the pattern of velocity streamline 1 for NACA 8412 is almost similarly with velocity streamline 1 that obtain from NACA 8412, the velocity that moves around turbine blade is 182. 5 m/s and has maximum velocity at leading edge with 365m/s. This shows that NACA 8413 has higher velocity around the blade when compare with NACA 8412. Furthermore, there is no turbulence occur at velocity streamline because of the shape of turbine blade profile does not twist and have angle of attack of 0 degree similarly with NACA 8412 shape of turbine blade.

Next, at velocity streamline 2 with distance 25 cm shown in figure 4.5(b) shows that the flow starts to turbulence after the flow move past the leading edge and the turbulence that occur at NACA 8413 is less than NACA 8412. The velocity that occur at turbulence is below 185.1 m/s, this is happen because of the shape of turbine blade profile at 25 cm have twisted angle, and have higher angle of attack cause the turbulence occur similarly with NACA 8412 shape of turbine blade.

For velocity streamline 3 shown in figure 4.5(c) with distance 45 cm also shows that the flow moves past the leading edge have turbulence, the turbulence that occur at velocity streamline 3 is because of the shape of turbine blade profile is twisted and have high angle of attack. Besides, velocity streamline 3 is more turbulence compare with the velocity streamline 2 because of velocity streamline 3 have higher angle of attack compare with velocity streamline 2 similar to case of NACA 8412.



Then for velocity streamlines 4 shown if figure 4.5(d)with distance 50.3 cm (tip clearance) shows that the flow move toward the direction has turbulence but have less velocity compare with other streamline this is because the air flow is not on the turbine blade profile, and the turbulence that occur at velocity streamline 4 is due to the flow that pass through at the end of tip of turbine blade profile similarly with NACA 8412.

### 4.2.3 Velocity Streamline (NACA 8414)

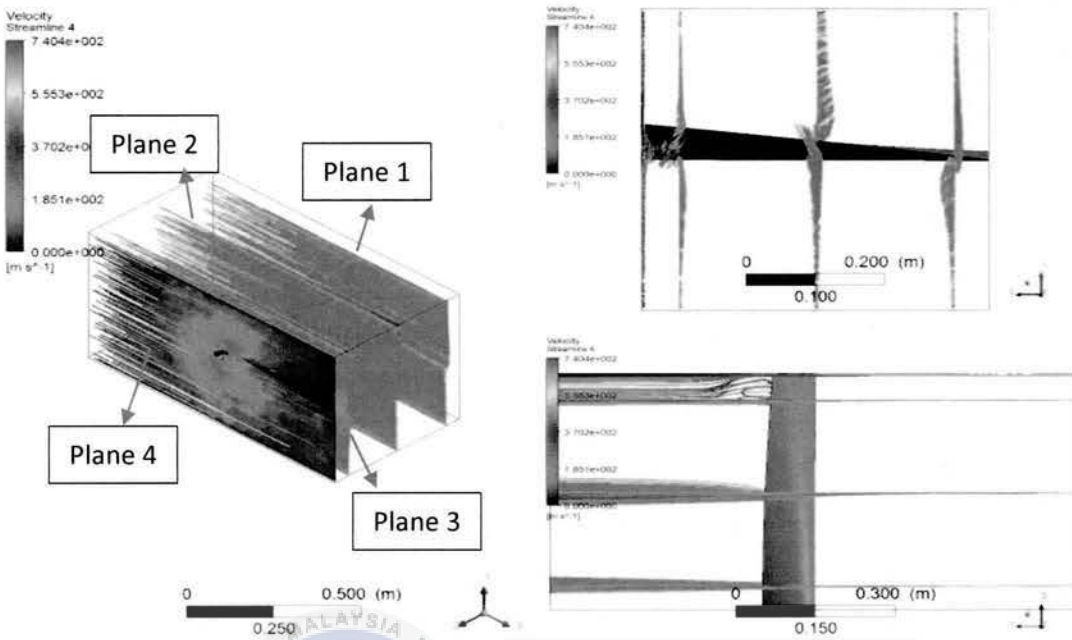



Figure 4.6: Velocity Streamline of NACA 8414 in 3-D View

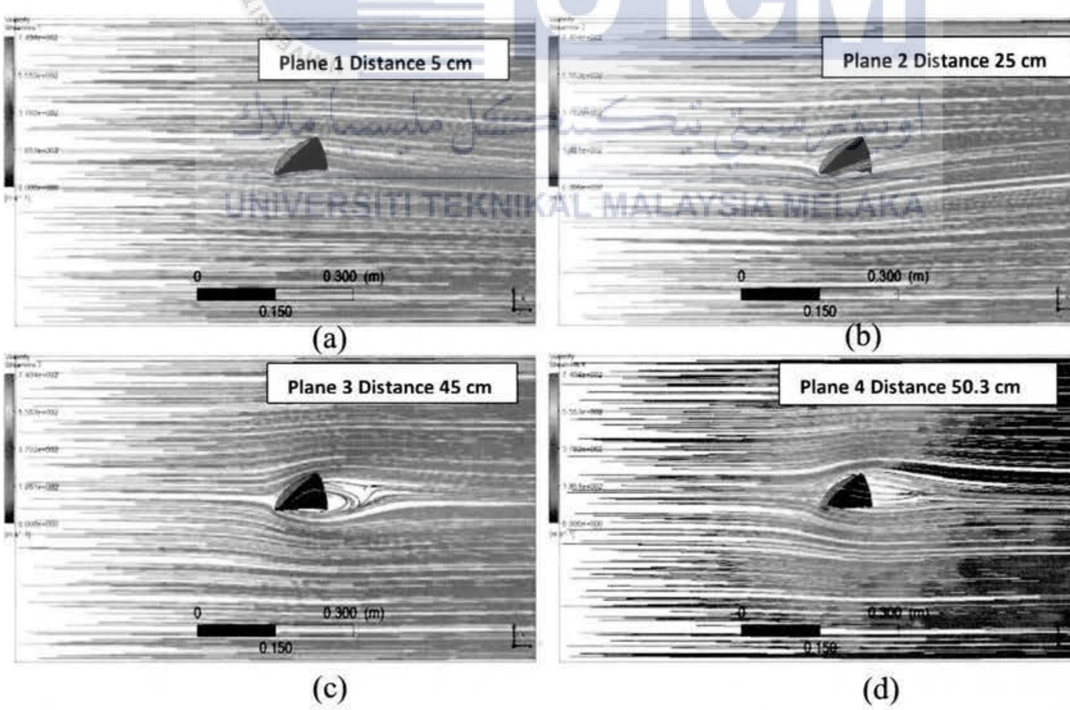



Figure 4.7: Velocity Streamline of NACA 8414 (4 PLANE SECTION)

Figure 4.7(a) shown velocity streamline 1 with distance of 5cm. According to the velocity streamline 1, NACA 8414 has the same pattern of velocity streamline is similarly to the velocity streamline obtain for NACA 8412 and 8413, the velocity that moves around turbine blade is 185. 1 m/s and has maximum velocity at leading edge with 370.2 m/s. NACA 8414 has higher velocity around the blade when compare with NACA 8412 and NACA 8413. For NACA 8414, there is no turbulence occur during the velocity flow past through the trailing edge similarly to velocity streamline obtain for NACA 8412 and NACA 8413. This is proving that, turbine blade can flow move smoothly without turbulence when the shape of turbine of blade does not twist and have zero angle of attack.

Next, at velocity streamline 2 with distance of 25 cm shown in figure 4.7(b), shows that the flow moves smoothly without turbulence when passing through the leading edge of turbine blade. For this turbine blade, the increase of angle of attack cause by twisted angle does not cause the turbulence to occur when hit the leading edge of turbine blade, different with NACA 8412 and NACA 8413. This shows that, NACA 8414 has better shape of turbine blade when comparing with NACA 8413 and NACA 8412. Moreover, the velocity flow around the turbine blade of NACA 8414 is higher than velocity streamline 2 obtain from NACA 8413 and NACA 8414. This prove that, the turbulence can cause the velocity around the turbine blade decrease.



At velocity streamline 3 shown in figure 4.7(c) with distance 45 cm also shows that the flow moves past the leading edge start to turbulence. The turbulence that occurs at velocity streamline 3 is because of the shape of turbine blade profile is more twisted and have higher angle of attack compare with the shape of turbine blade for velocity streamline 2. Furthermore, the turbulence that occur at velocity streamline 3 for NACA 8414 is less than NACA 8412 and NACA 8413. This shows, that NACA 8414 has better shape of turbine blade when comparing with NACA 8413 and NACA 8412.

Lastly, for the velocity streamlines 4 with distance 50.3 cm (tip clearance) shown in figure 4.7(d), shows that the flow move toward the direction has turbulence but have less velocity when compare with other streamline ,this is because the air flow is not on the turbine blade profile, and the turbulence that occur at velocity streamline

4 is due to the flow that pass through at the end of tip of turbine blade profile similarly with case for NACA 8412 and NACA 8413 and only the velocity around the blade.

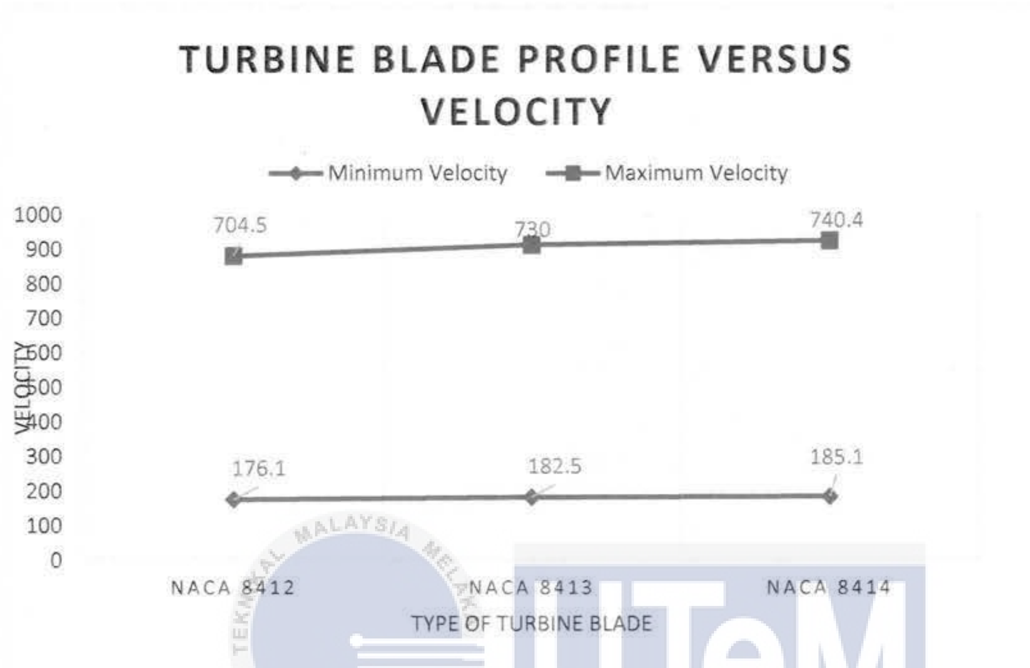



Figure 4.8: of Turbine Blade Profile Versus Velocity



Figure 4.8 shows the graph that been plotted based on value obtain from the simulation for maximum and minimum velocity. According to the figure 8, all the maximum velocity for all the is above 700 m/s and minimum velocity is below 200 m/s. NACA 8414 has the highest maximum and minimum velocity which are 740.4 m/s and 185.1 m/s. Next, NACA 8413 is the second higher maximum and minimum velocity after NACA 8414 which are 730.0 m/s and 182.5 m/s. Then, NACA 8412 has the lowest maximum and minimum velocity with 704.5 m/s and 176.1 m/s. NACA 8414 has the highest velocity is because of NACA 8414 has very less turbulence when compare with NACA 8412 and NACA 8412 for velocity streamline and this prove that, turbulence can affect flow of velocity.

## 4.3 Pressure Contour

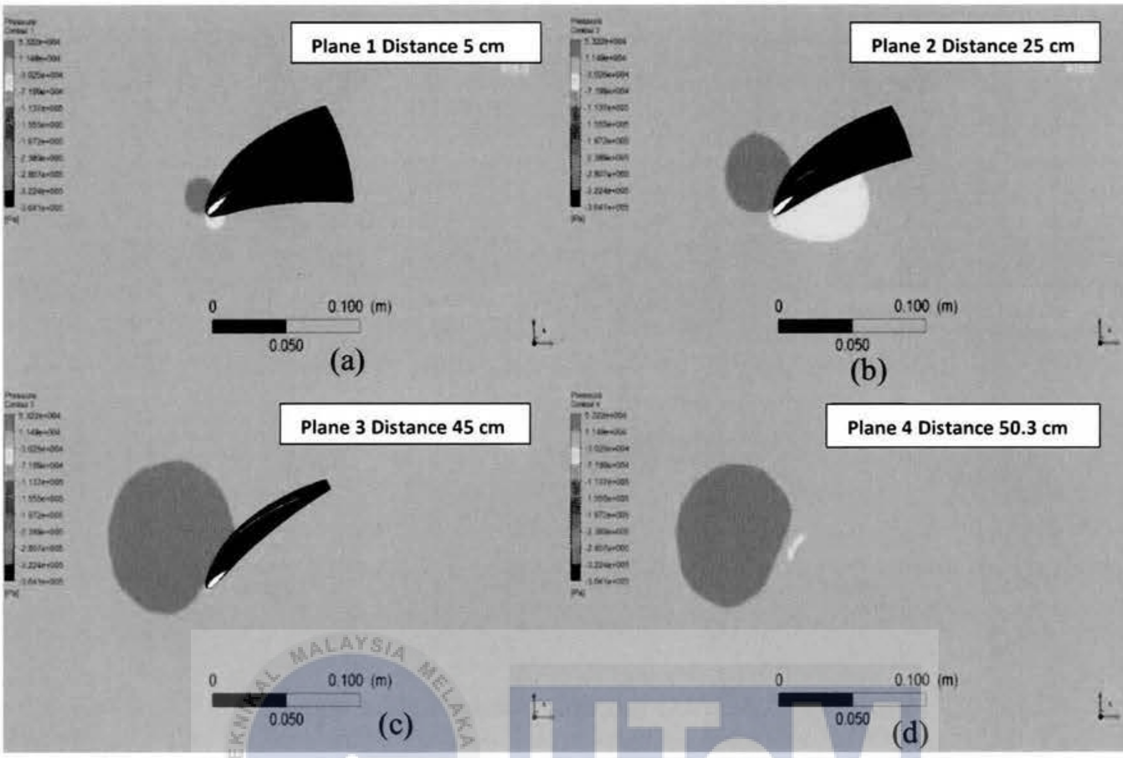



Figure 4.9: Pressure Contour (NACA 8412)



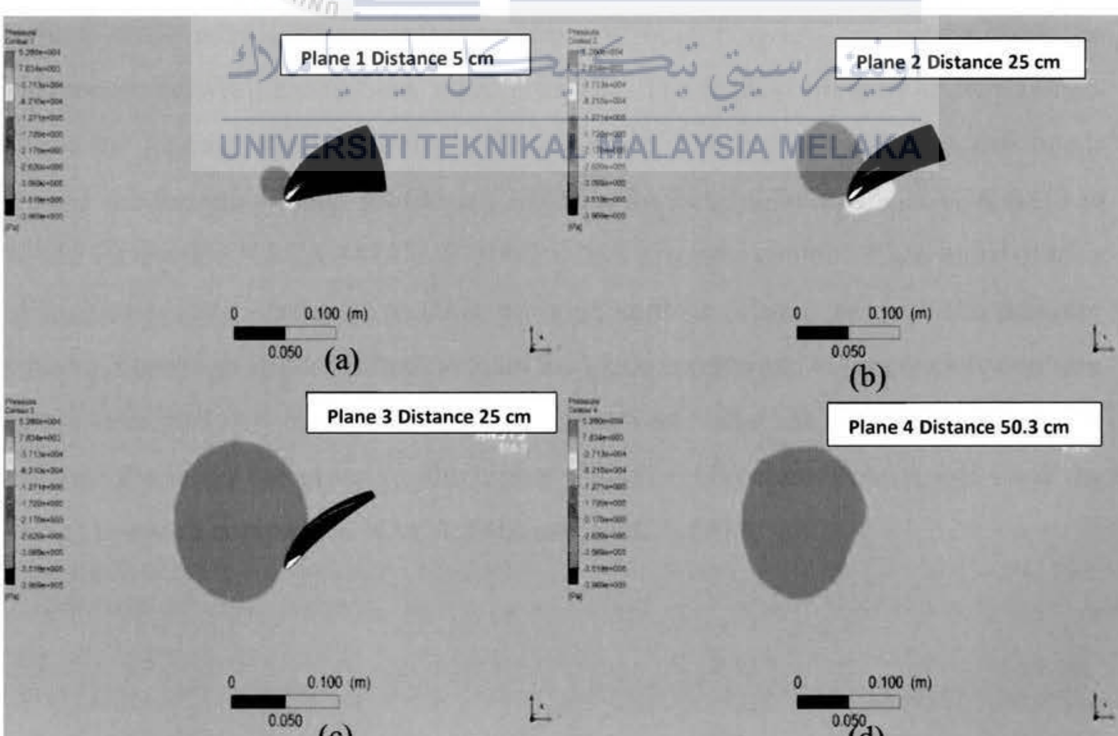



Figure 4.10: Pressure Contour (NACA 8413)

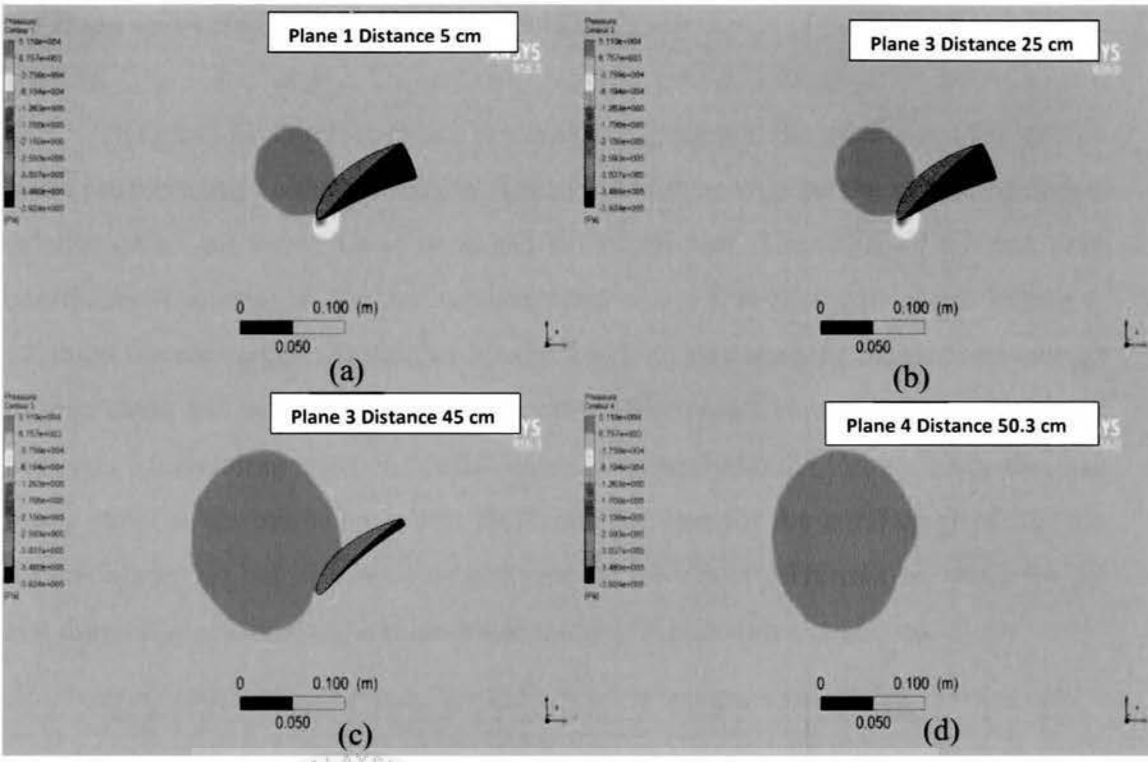



Figure 4.11: Pressure Contour (NACA 8414)

For this analysis, pressure contour is used to figure out which location around the blade has the higher pressure. Based on Figure above, the pressure contour for all turbine blade profile have similar pattern and only pressure are determining the differences between the turbine 's blade profile. The highest pressure mostly occurs above the leading edge of turbine's blade profile. The highest pressures that occur around the turbine's blade profile is 53320 Pa for NACA 8412, for NACA 8413 is 52840 Pa and for NACA 8414 is 51100 Pa. The pressure contour 3 has more of area of highest pressure compare to other pressure contour. This is because the pressure contour 3 has high angle of attack around the blade compare to other pressure contour. In pressure analysis, the pressure which has lowest value has the best performance because the lower the pressure, the higher velocity. Therefore, NACA 8414 has the lowest pressure compare to NACA 8412 and NACA 8414.

## 4.4 Drag and Lift Coefficient (NACA 8412)

Drag and lift coefficient can be obtaining alongside the graph for each turbine blade profile when do the simulation. The simulation need to set the suitable iteration calculation to get the value of drag and lift coefficient. The value of lift and drag coefficient is selected at the last iteration point where it is fully converged. Figure 4. 12 show the converged solution of NACA 8412. In this analysis the performance of turbine blade will be evaluated based on value obtain coefficient of lift, coefficient of drag and lift and drag ratio. In coefficient of drag analysis, the turbine blade that has lower value is chosen to have best performance, then for the coefficient of lift, the turbine blade that has higher value is chosen to have best performance, while for lift and drag ratio analysis, the turbine blade that has higher value is chosen.

| | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 116 | 3.6589e-03 | 6.1530e-05 | 4.1515e-05 | 2.8627e-05 | 1.9396e-16 | 1.7610e-03 | 7.8317e-04 | -1.7996e-02 | 1.1796e-02 | 1:12:01 | 884 |
| 117 | 3.4551e-03 | 5.7921e-05 | 3.8283e-05 | 2.6676e-05 | 1.9256e-16 | 1.6969e-03 | 7.7908e-04 | -1.8017e-02 | 1.1806e-02 | 1:12:16 | 883 |
| 118 | 3.2274e-03 | 5.5818e-05 | 3.4852e-05 | 2.4890e-05 | 1.9156e-16 | 1.6623e-03 | 7.6873e-04 | -1.8038e-02 | 1.1815e-02 | 1:12:27 | 882 |
| 119 | 3.0208e-03 | 5.3997e-05 | 3.1845e-05 | 2.3430e-05 | 1.9298e-16 | 1.6330e-03 | 7.4096e-04 | -1.8053e-02 | 1.1823e-02 | 1:12:34 | 881 |
| 120 | 2.8346e-03 | 5.2181e-05 | 2.9557e-05 | 2.2286e-05 | 1.9033e-16 | 1.5880e-03 | 6.9343e-04 | -1.8061e-02 | 1.1827e-02 | 1:12:40 | 880 |
| 121 | 2.6909e-03 | 5.0243e-05 | 2.8157e-05 | 2.1600e-05 | 1.9289e-16 | 1.5155e-03 | 6.3510e-04 | -1.8062e-02 | 1.1828e-02 | 1:12:43 | 879 |
| iter | continuity | x-velocity | y-velocity | z-velocity | energy | k | epsilon | Cl-1 | Cd-1 | time/iter | |
| 122 | 2.5720e-03 | 4.7691e-05 | 2.7427e-05 | 2.1161e-05 | 1.9251e-16 | 1.4268e-03 | 5.8332e-04 | -1.8057e-02 | 1.1827e-02 | 1:12:44 | 878 |
| 123 | 2.4656e-03 | 4.5095e-05 | 2.7280e-05 | 2.0651e-05 | 1.9325e-16 | 1.3109e-03 | 5.4506e-04 | -1.8045e-02 | 1.1823e-02 | 1:12:44 | 877 |
| 124 | 2.4106e-03 | 4.2282e-05 | 2.6742e-05 | 1.9690e-05 | 1.9465e-16 | 1.1977e-03 | 5.1867e-04 | -1.8028e-02 | 1.1816e-02 | 1:12:44 | 876 |
| 125 | 2.3324e-03 | 3.9448e-05 | 2.5527e-05 | 1.8406e-05 | 1.9247e-16 | 1.1170e-03 | 5.0777e-04 | -1.8009e-02 | 1.1808e-02 | 1:09:47 | 875 |
| 126 | 2.2097e-03 | 3.6780e-05 | 2.3646e-05 | 1.7074e-05 | 1.9241e-16 | 1.0521e-03 | 5.0028e-04 | -1.7989e-02 | 1.1800e-02 | 1:10:20 | 874 |
| 127 | 2.0860e-03 | 3.4932e-05 | 2.1582e-05 | 1.5794e-05 | 1.9344e-16 | 1.0175e-03 | 4.9143e-04 | -1.7972e-02 | 1.1793e-02 | 1:10:45 | 873 |
| 128 | 1.9657e-03 | 3.3378e-05 | 1.9806e-05 | 1.4706e-05 | 1.9809e-16 | 9.8652e-04 | 4.7700e-04 | -1.7959e-02 | 1.1787e-02 | 1:11:04 | 872 |
| 129 | 1.8355e-03 | 3.1822e-05 | 1.8521e-05 | 1.3904e-05 | 1.9121e-16 | 9.6804e-04 | 4.5442e-04 | -1.7951e-02 | 1.1784e-02 | 1:11:18 | 871 |
| 130 | 1.7004e-03 | 3.0488e-05 | 1.7737e-05 | 1.3355e-05 | 1.9281e-16 | 9.4006e-04 | 4.2841e-04 | -1.7948e-02 | 1.1782e-02 | 1:11:29 | 870 |
| 131 | 1.5929e-03 | 2.9224e-05 | 1.7475e-05 | 1.3064e-05 | 1.9243e-16 | 9.1063e-04 | 4.0381e-04 | -1.7950e-02 | 1.1781e-02 | 1:11:36 | 869 |
| 132 | 1.5104e-03 | 2.7809e-05 | 1.7298e-05 | 1.2774e-05 | 1.9534e-16 | 8.7042e-04 | 3.8444e-04 | -1.7955e-02 | 1.1783e-02 | 1:11:41 | 868 |
| iter | continuity | x-velocity | y-velocity | z-velocity | energy | k | epsilon | Cl-1 | Cd-1 | time/iter | |
| 133 | 1.4641e-03 | 2.6421e-05 | 1.6997e-05 | 1.2271e-05 | 1.9241e-16 | 8.2371e-04 | 3.7028e-04 | -1.7963e-02 | 1.1785e-02 | 1:11:44 | 867 |
| 134 | 1.4433e-03 | 2.4824e-05 | 1.6311e-05 | 1.1637e-05 | 1.9205e-16 | 7.7807e-04 | 3.5770e-04 | -1.7973e-02 | 1.1788e-02 | 1:11:45 | 866 |
| 135 | 1.4017e-03 | 2.3283e-05 | 1.5241e-05 | 1.0982e-05 | 1.9418e-16 | 7.4041e-04 | 3.4675e-04 | -1.7984e-02 | 1.1792e-02 | 1:11:45 | 865 |
| 136 | 1.3452e-03 | 2.2281e-05 | 1.4215e-05 | 1.0413e-05 | 1.9476e-16 | 7.0575e-04 | 3.3389e-04 | -1.7993e-02 | 1.1795e-02 | 1:11:44 | 864 |
| 137 | 1.2873e-03 | 2.1511e-05 | 1.3397e-05 | 1.0027e-05 | 1.9425e-16 | 6.7273e-04 | 3.1880e-04 | -1.7999e-02 | 1.1797e-02 | 1:11:42 | 863 |
| 138 | 1.2185e-03 | 2.0574e-05 | 1.2730e-05 | 9.7230e-06 | 1.9130e-16 | 6.3952e-04 | 3.0042e-04 | -1.8003e-02 | 1.1798e-02 | 1:11:40 | 862 |
| 139 | 1.1417e-03 | 1.9543e-05 | 1.2241e-05 | 9.5900e-06 | 1.9319e-16 | 6.0549e-04 | 2.8140e-04 | -1.8004e-02 | 1.1798e-02 | 1:11:37 | 861 |
| 140 | 1.0611e-03 | 1.8479e-05 | 1.1982e-05 | 9.4142e-06 | 1.9468e-16 | 5.7248e-04 | 2.6459e-04 | -1.8001e-02 | 1.1796e-02 | 1:11:33 | 860 |
| ! 141 solution is converged | | | | | | | | | | | |
| 141 | 9.9413e-04 | 1.7509e-05 | 1.1762e-05 | 9.1141e-06 | 1.9310e-16 | 5.4518e-04 | 2.5146e-04 | -1.7996e-02 | 1.1794e-02 | 1:11:30 | 859 |



Figure 4.12: Converged Solution of NACA 8412

Table 4.1: Result of lift coefficient, drag coefficient and Ratio of Lift and Drag

| Type of Turbine Blade Profile | COEFFICIENT OF LIFT | Coefficient OF DRAG | RATIO OF LIFT AND DRAG ,CL/CD |
|---|---|---|---|
| **NACA 8412** | **-0.017996** | **0.011794** | **0.6554** |
| **NACA 8413** | **-0.019049** | **0.011168** | **1.7057** |
| **NACA 8414** | **-0.027324** | **0.005404** | **5.0563** |

Table 4.1 shows the result of lift coefficient, drag coefficient and ratio of lift and drag obtain from simulation. Based on table above, NACA 8412 has the highest value for the coefficient of drag which is 0.011794, then NACA 8413 come for second which is 0.01168 and lastly is NACA 8414 which is 0.005404. Next for the coefficient of lift, NACA 8414 has the highest value which is 0.027324, then NACA 8413 come for second which is 0.019049 and lastly is NACA 8412 which is 0.017996. Lastly, for the ratio of lift and drag, NACA 8414 has the higher value which is 5.0563, then NACA 8413 come for second which us 1.7057 and lastly is NACA 8412 which is 0.6654. In this analysis, the performance of turbine's blade profile is depending on the value of coefficient of lift, coefficient of drag and r lift and drag ratio. The turbine 's blade that has low coefficient of drag has the best turbine's blade, while for analysis based on coefficient of lift and ratio of lift to drag the turbine's blade that has highest value is chosen to have best performance. Following to the result that obtain from this simulation, NACA 8414 is chosen to has the best performance compare with other turbine blade since it has highest value for coefficient of lift, lowest value for coefficient of drag and highest value for lift and drag ratio.

**Discussion**

From the simulation, the results that are obtained are velocity streamline, pressure contour and coefficient of lift and drag. Based on velocity streamline that obtain from simulation is show that why the coefficient of lift is negative value. The negative value of lift coefficient represents that, the turbine blade profile is moving downward force and also because of the shape turbine's blade profile is twisted.

Velocity streamline also shows the velocity around the turbine blade and maximum velocity value can obtained from the simulation, the result from velocity streamline shows that NACA 8414 has the best performance since it has highest maximum velocity and also less turbulence compare with other turbine blade. Next, in analysis of pressure contour, it shows where the location around the turbine blade has higher pressure and also pressure value. According to pressure contour analysis, the maximum pressure is occurring at above the leading edge of the turbine blade profile. Following the pressure contour analysis, NACA 8414 is chosen to have best performance compare with other since it has lowest pressure value compare with other turbine blade.

Lastly, when comparing the result of coefficient of lift, coefficient of drag and lift and drag ratio. NACA 8414 is chosen to have the best performance when compare with NACA 8413 and NACA 8414, because it has highest value of coefficient of lift and lift to drag ratio and also lowest value for coefficient. Therefore, the best turbine blade for this analysis is NACA 8414.

# CHAPTER 5

# CONCLUSION AND RECOMMENDATION

## 5.1 Conclusion and Recommendation

According to the analysis, there are many factor that influence the performance of turbine. One of factor that influence of turbine performance is when selecting the suitable turbine blade geometry. This can be proved from the analysis of velocity streamline. Based on streamline analysis, is shows that the turbine blade that good shape has less turbulence. In velocity streamline comparison, NACA 8412 has most turbulence occur compare with other turbine blade. The influence of turbulence that occur around the turbine cause velocity flow decrease, hence NACA 8412 has the lowest velocity compare to other the turbine's blade. Next factor is pressure, according to the simulation result, shows that the turbine's blade that has lowest value of pressure has high velocity. NACA 8414 has the lowest values pressure compare with other, therefore NACA 8414 has higher value of velocity. Lastly, coefficient of lift, coefficient of drag and lift and drag ratio factor, in this NACA 8414 is chosen to have the best performance since it has highest value of coefficient of lift, lift and drag ratio and lowest value for coefficient of drag compare to other blade.

In conclusion, the investigation of turbine blades by using 3D-CFD performance is evaluate the performance based on coefficient of lift, coefficient of drag and lift and drag ratio value and the best turbine blade is NACA 8414. But there is performance should be evaluated which are the leakage flow loses and energy loses since it has tip clearance. Therefore, for the next recommendation analysis, the investigation should include the leakage flow loses and energy loses.

## REFERENCES


Retrieved from http://www.airfoiltools.com/index.

A.G Chervonenko. (1991). *Effect of attack Angle on the Nonstationary Aerodynamic Characteristics and Flutter Resistance of a Grid of Bent Vibrating Compressor Blades* (Vol. 39). Ukraine: Ukrainian Academy of Sciences, Plenum Publishing Corporation.

Castan., J. P. (2003). *Cambered airfoil use in Turbocharger.*

David Gordon Wilson, T. K. (1998). *The Design of High efficiency Turbomachinery and Gas Turbine.* Upper Saddle River,New Jersey 07459: Prentice Hall.

E. N. Jacobs, K. E. (1935). *The characteristics of 78 related airfoil sections from tests in the variable-density wind tunnel.* NACA Report No. 460.

Eppler., R. (1990). *Airfoil Design and Data.* Springer Science & Business Media, 2012.

Hadi karrabi, M. R. (2011). *The Effect of Blade Lean,Twist and Bow on The Performance of Axial Turbine at Design Point.* International Mechanical Engineering Congress & Exposition.

He P, S. Z. (2012). Aerothermal Investigation of Backface Clearance Flow in Deeply Scalloped Radial Turbines. *ASME. J. Turbomach.*

Jayakumar, J. F. (2011). Investigation of Airfoil Profile Design Using Reverse Engineering Bezier Curve. *Journal of Engineering and Applied Sciences.*

Kaurase, K. G. (2016). Analysis of Transonic Flow Over Supercritical Airfoil Using CFD For Gas Turbine Blades. *INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL*, 19-28.

Krutartha Sudhir Jathar, V. V. (2015). International Research Journal of Engineering and Technology (IRJET). *An Elementary Study of Computational Fluid Dynamics For Various Engineering Applications – A Review.*

MD. Safayet Hossain, M. F. (2014). A COMPARATIVE FLOW ANALYSIS OF NACA 6409 AND NACA 4412 AEROFOIL. *International Journal of Research in Engineering and Technology.*

Sondkar Pratik, S. Q. (2016). THERMODYNAMIC DESIGN OF RADIAL TURBINE FOR TURBO-CHARGER. *INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH*, 634-637.

Yahya, S. (2005). *Turbines Compressorr and Fans.* New Delhi: Mcgraw-Hill Publishing Company.

Yunus Cengel, M. A. (2015). *Thermodynamics An Engineering Approach.* New York: McGraw-Hill Education.

Ainley, D. G., & Mathieson, C. R. (1951). A Method of Performance Estimation for Axial-Flow Turbines. *Aeronautical Research Council Reports And Memoranda, N. 2974, 2974*(2974). Retrieved from http://citeseerx.ist.psu.edu/viewdoc/download?doi=10.1.1.226.7624&rep=rep1&type=pdf

Fazil, J., & Jayakumar, V. (2011). Investigation of airfoil profile design using reverse engineering Bezier curve. *Journal of Engineering and Applied Sciences*, *6*(7), 43–52.

Gandecha, K., & Kaurase, K. P. (2016). ANALYSIS OF TRANSONIC FLOW OVER SUPERCRITICAL AIRFOIL USING CFD FOR GAS TURBINE BLADES, *4*(7), 19–28.

Gendcha, K., & Kaurase, K. P. (2016). Analysis of Transonic Flow over Airfoil using CFD for Gas Turbine Blades, *4*(5), 350–354.

Gorla, R. S. R., & Khan, A. A. (2003). *Turbomachinery*.

Hada, S., Tsukagoshi, K., Masada, J., & Ito, E. (2013). Test Results of the World's First 1,600oC J-series Gas Turbine. *Mitsubishi Heavy Industries Technical*

*Review*, *49*(1), 18–23.

Kumar, K. H., Kumar, C. H. K., & Kumar, T. N. (2015). Cfd Analysis of Rae 2822 Supercritical Airfoil At Transonic Mach Speeds, 256–262.

Kumar, S. (2008). Sanjay Kumar & Onkar Singh. *Film*.

Kurz, R. (2005). Gas turbine performance. *Proceedings of the 34th Turbomachinery Symposium*, 131–146.

Meher-Homji, C. (2000). 'The Historical Evolution of Turbomachinery. *Proceedings of the 29th Turbomachinery …*, 281–322. Retrieved from http://turbo-lab.tamu.edu/proc/turboproc/T29/t29pg281.pdf

Muthén, L. K., & Muthén, B. O. (2009). User ' S Guide, (August).

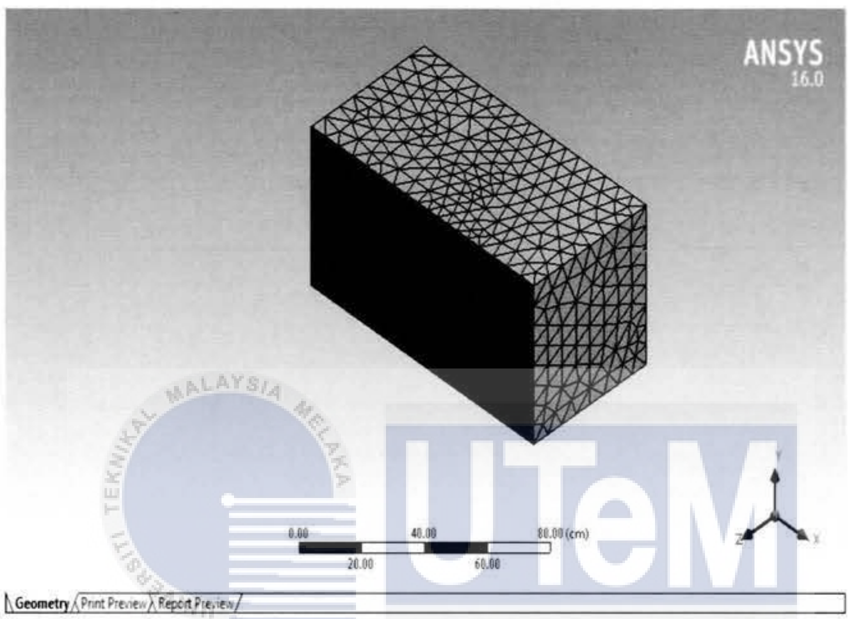
Patel, S. T., Patel, D. T., & Bhensdadiya, M. (2015). An Analysis of Lift and Drag Forces of NACA Airfoils Using Python, *4*(4), 198–206.

Rao, V. N., Kumar, I. N. N., Madhulata, N., & Abhijeet, A. (2014). Mechanical Analysis of 1 st Stage Marine Gas Turbine Blade, *68*(August), 57–64. https://doi.org/10.14257/ijast.2014.68.06

Zuo, W. (2005). Introduction of Computational Fluid Dynamics, 8. Retrieved from http://www14.in.tum.de/konferenzen/Jass05/courses/2/Zuo/Zuo_paper.pdf

# APPENDIX A

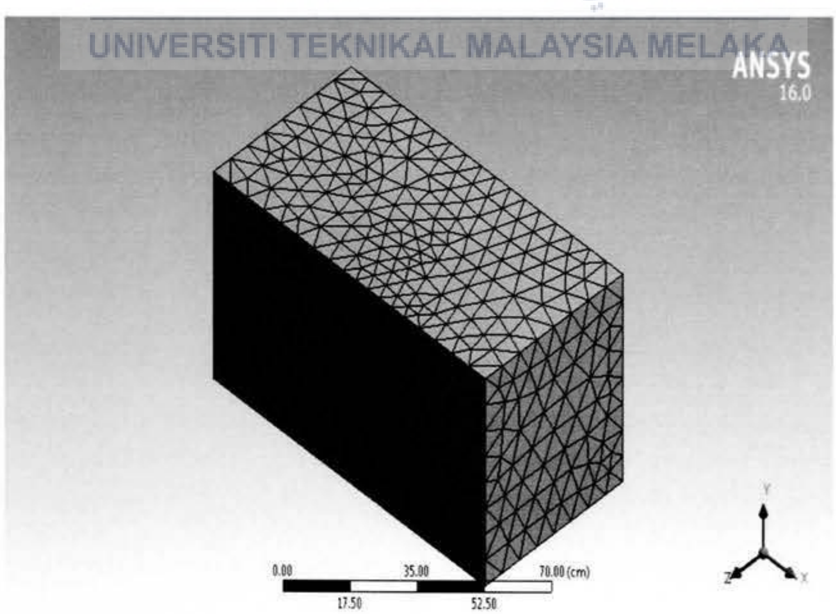
Figure of Meshing



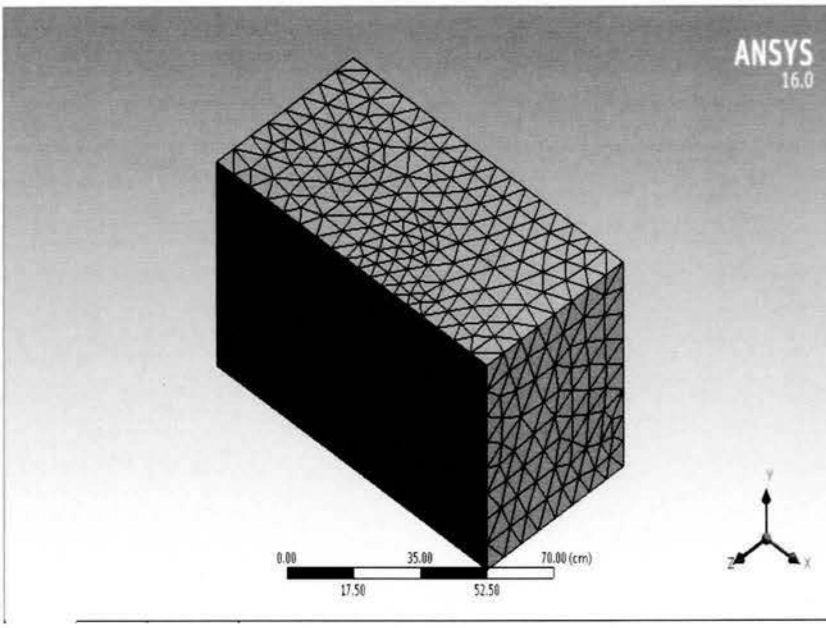



NACA 8412






NACA 8413




NACA 8414