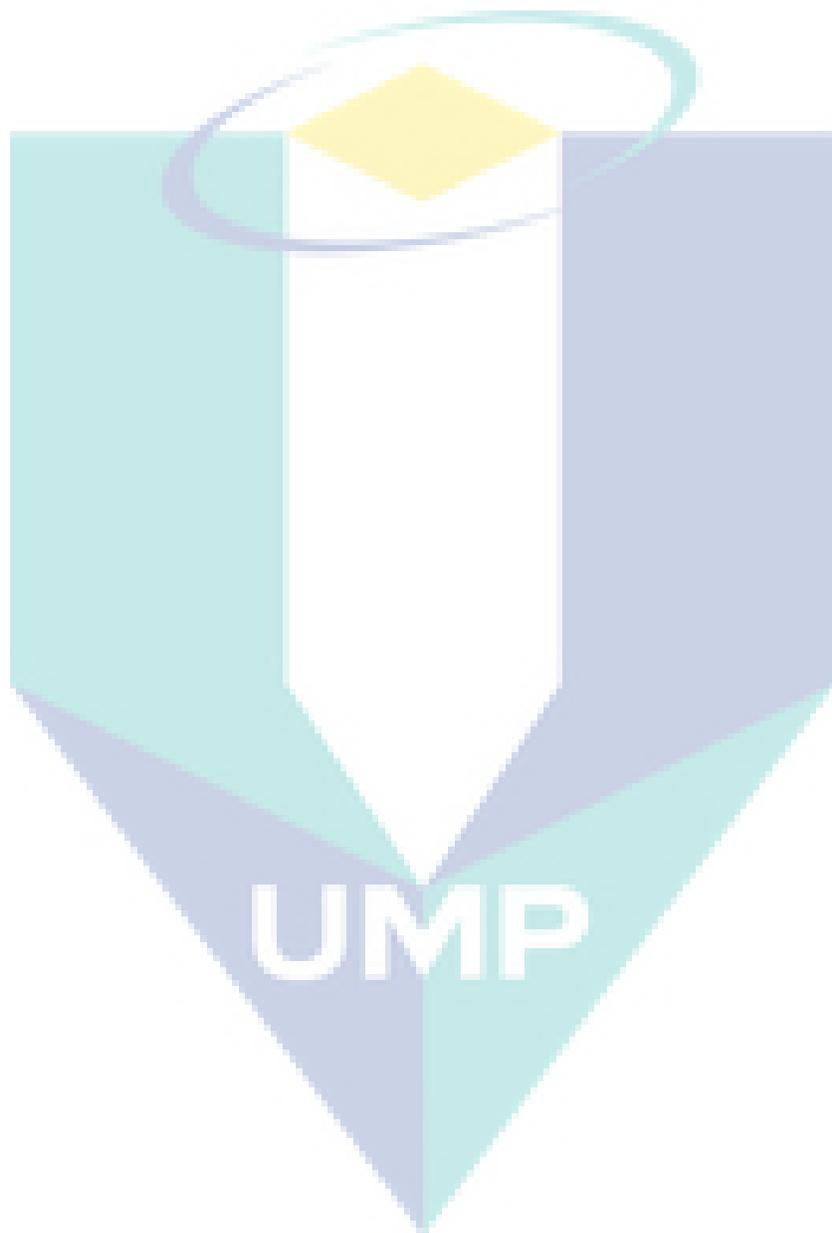


FINAL REPORT

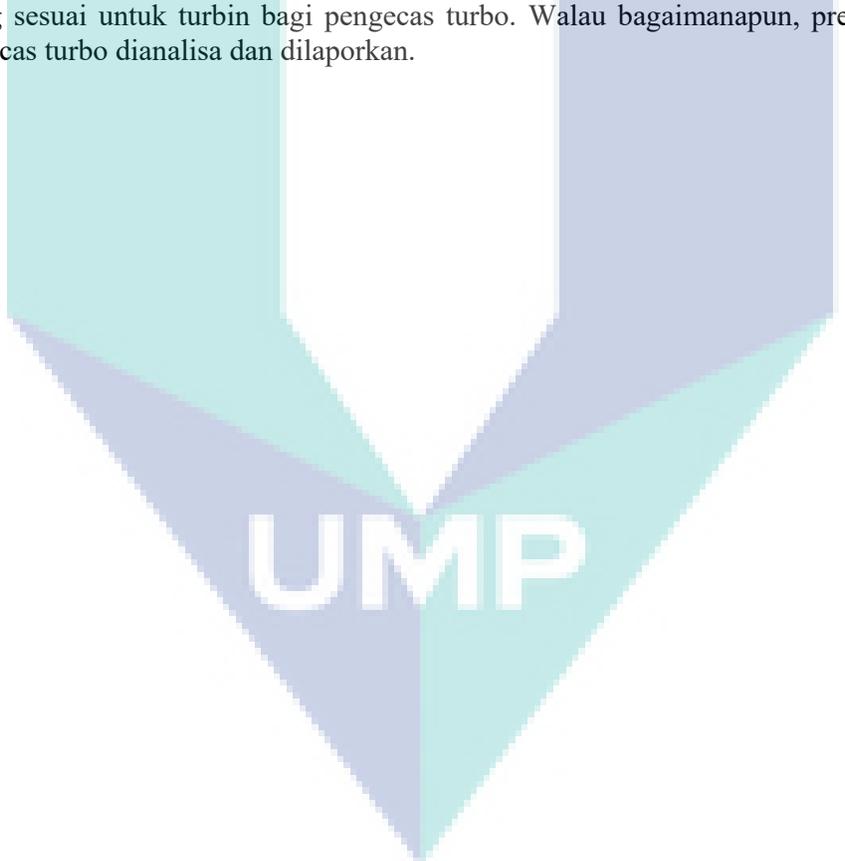
ENHANCING PERFORMANCE AND
EMISSION ON A TURBOCHARGED
GASOLINE ENGINE

UMP



ABSTRAK

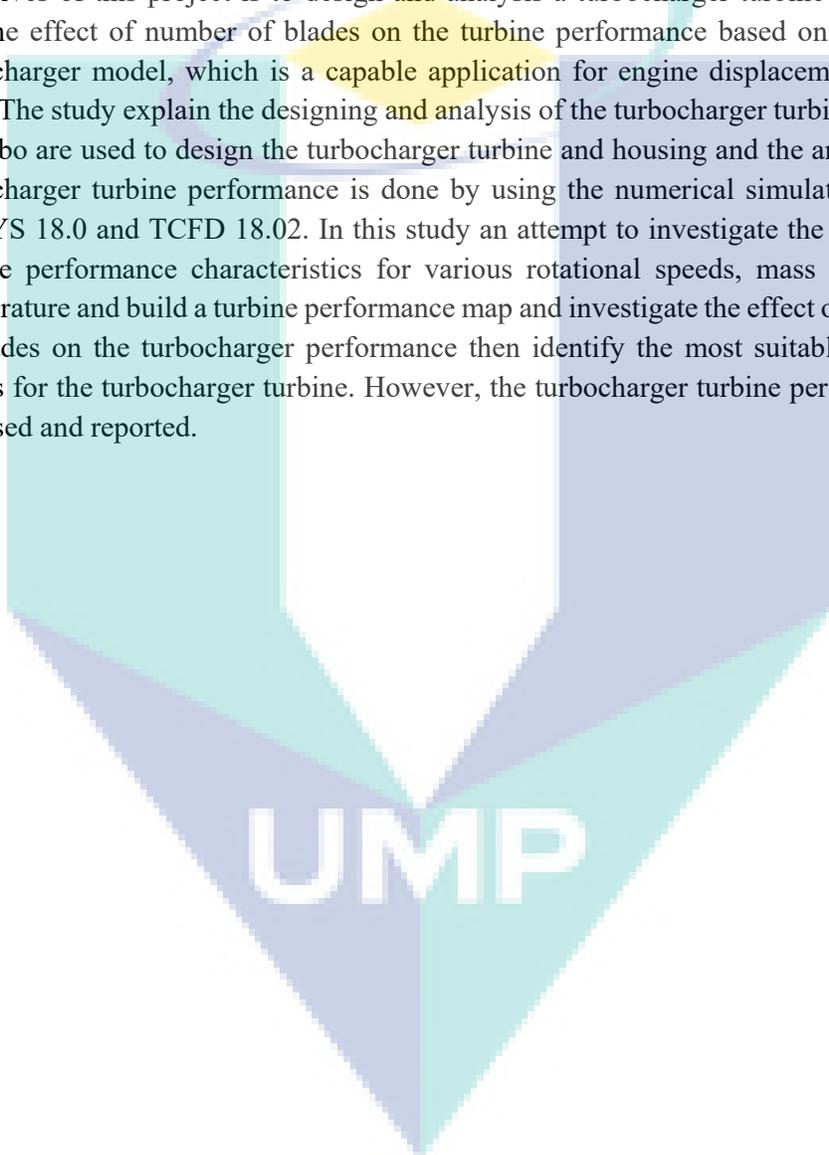
Pengecas turbo adalah menjadi trend terkini didalam dalam perkembangan pembuatan enjin kereta automotif. Pada masa kini, pengecas turbo digunakan untuk membangunkan dan memperbaiki prestasi enjin dan kini digunakan untuk menghasilkan prestasi tinggi dan meningkatkan ekonomi bahan bakar untuk kenderaan dengan saiz enjin kecil dan sederhana. Perkembangan ini membawa kepada pengeluar kereta untuk menggunakan enjin pertengahan saiz dengan sistem pengecas turbo. Objektif utama projek ini adalah untuk merekabentuk dan menilai prestasi pengecas turbo dan kesan bilangan bilah pada prestasi turbin berdasarkan model turbocharger GT2056, yang merupakan aplikasi yang mampu untuk anjakan enjin 1.4 hingga 2.0L. Kajian ini bagi menjelaskan reka bentuk dan analisis pengecas turbo, CAD dan CFTurbo digunakan untuk merekabentuk pengecas turbo dan rumah turbin dan analisis prestasi turbin dilakukan dengan menggunakan perisian simulasi berangka ANSYS 18.0 dan TCFD 18.02. Dalam kajian ini, cubaan untuk menyiasat ciri-ciri prestasi turbincharger turbo untuk pelbagai kelajuan putaran, aliran aliran massa dan suhu dan membina peta prestasi turbin dan menyiasat kesan bilangan bilah pada prestasi pengecas turbo kemudian mengenal pasti bilangan bilah yang paling sesuai untuk turbin bagi pengecas turbo. Walau bagaimanapun, prestasi turbin pengecas turbo dianalisa dan dilaporkan.



UMP

ABSTRACT

The turbocharger became a trend in the automotive cars engine manufacturing developments. Nowadays turbocharger are used to develop and improve the engine performance and, it now applied to produce a high performance and improve fuel economy for a vehicles with small and medium engine size. This development lead the car manufactures to use a mid-sized engine rang with a turbocharger system. The main objectives of this project is to design and analysis a turbocharger turbine performance and the effect of number of blades on the turbine performance based on the GT2056 turbocharger model, which is a capable application for engine displacement of 1.4 to 2.0L. The study explain the designing and analysis of the turbocharger turbine, CAD and Cfturbo are used to design the turbocharger turbine and housing and the analysis of the turbocharger turbine performance is done by using the numerical simulation software ANSYS 18.0 and TCFD 18.02. In this study an attempt to investigate the turbocharger turbine performance characteristics for various rotational speeds, mass flowrate and temperature and build a turbine performance map and investigate the effect of the number of blades on the turbocharger performance then identify the most suitable number of blades for the turbocharger turbine. However, the turbocharger turbine performance are analysed and reported.

The logo for UMP (Universitas Muhammadiyah Purwokerto) is a large, stylized letter 'U' shape. The top part of the 'U' is a light blue horizontal bar. The two vertical sides of the 'U' are composed of two overlapping shapes: a light blue one on the left and a light purple one on the right. The bottom part of the 'U' is a light purple shape. The letters 'UMP' are written in white, bold, sans-serif font across the bottom of the 'U' shape.

UMP

TABLE OF CONTENT

DECLARATION

TITLE PAGE

ACKNOWLEDGEMENTS

Error! Bookmark not defined.

ABSTRAK

iv

ABSTRACT

v

TABLE OF CONTENT

vi

LIST OF TABLES

ix

LIST OF FIGURES

x

LIST OF SYMBOLS

xii

LIST OF ABBREVIATIONS

xiii

CHAPTER 1 INTRODUCTION

1

1.1 Background

1

1.2 Problem Statement

2

1.3 Objectives

2

1.4 Scopes

3

CHAPTER 2 LITERATURE RIVIEW

4

2.1 Introduction

4

2.2 Computational Fluid Dynamics

4

2.2.1 Navier-Stokes Equations

5

2.3 Turbocharger Fundamental

6

2.3.1 Turbocharger versus Supercharger

6

2.4 Turbocharger System

7

2.4.1 Turbocharger System Operation

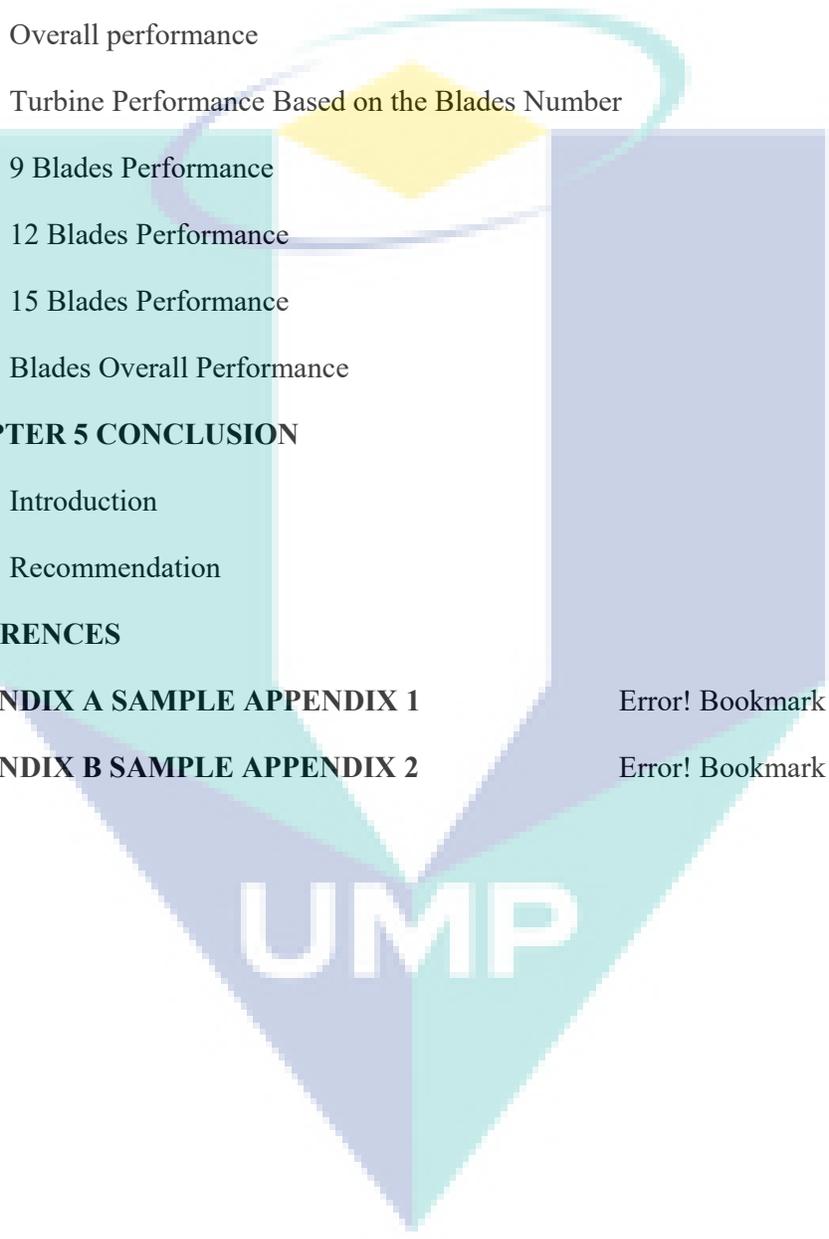
7

2.4.2 Turbocharger System Analysis

8

2.5	Turbocharger Radial Turbine	9
2.5.1	Turbine Blade Angle	9
2.5.2	Turbine Rotor	10
2.5.3	Radial Turbine Volute	11
2.5.4	Turbine Map	12
2.6	Effect of Blade Number on Turbocharger Turbine Performance	13
2.7	Summery	14
CHAPTER 3 METHODOLOGY		15
3.1	Introduction	15
3.2	Model Description	15
3.2.1	Geometry Design	16
3.3	Mesh Design	18
3.3.1	Mesh Using ANSYS Software	19
3.3.2	Mesh Using TCFD Software	20
3.4	Boundary Condition and Simulation Setup	22
3.4.1	Boundary Conditions	22
3.4.2	Numerical Method	22
3.5	Flow Chart	24
CHAPTER 4 RESULTS AND DISCUSSION		25
4.1	Introduction	25
4.2	Model Validation	26
4.2.1	Numerical Validation setup	26
4.2.2	Validation Result	27
4.3	TCFD and ANSYS Fluent Result Performance Comparison	29
4.3.1	ANSYS Fluent Result	29
4.3.2	TCFD Result	30

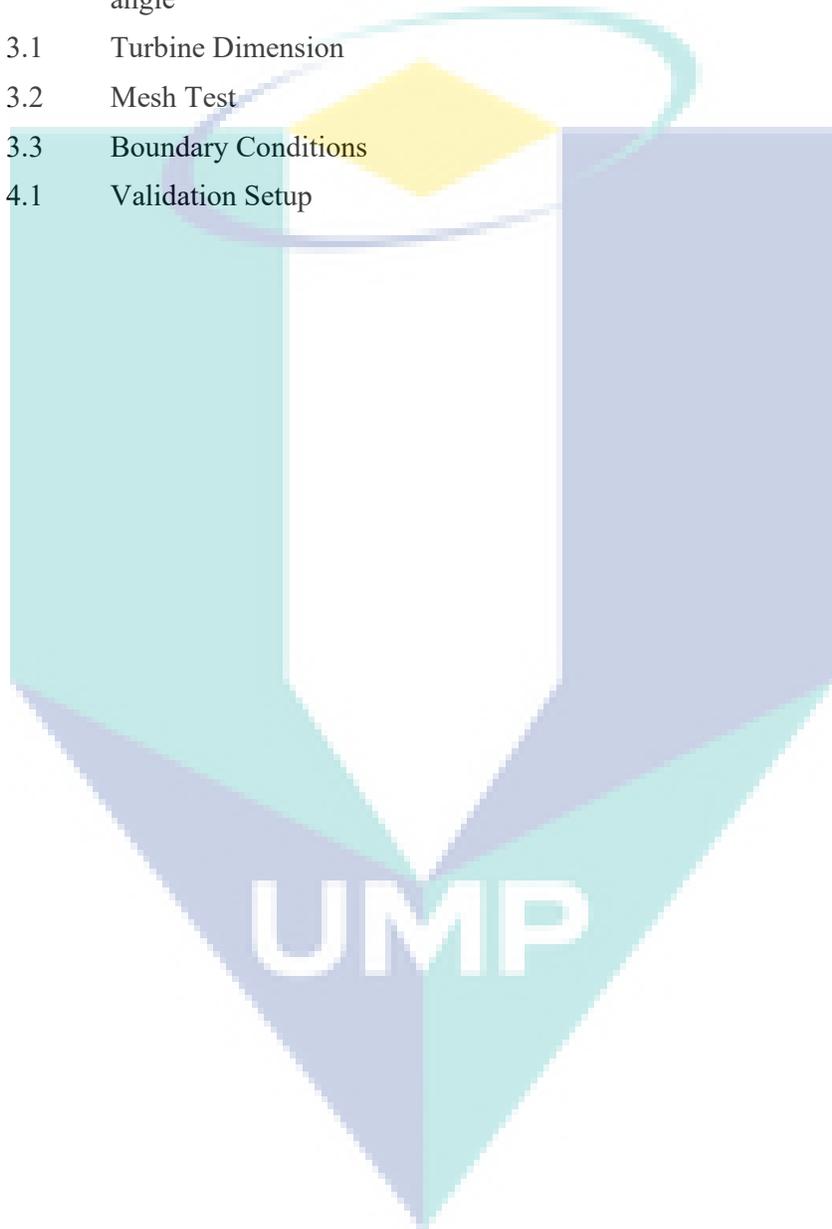
4.4	Turbine Performance Characteristics	32
4.4.1	120000RPM Result	33
4.4.2	150000RPM Result	34
4.4.3	180000RPM Result	35
4.4.4	Overall performance	36
4.5	Turbine Performance Based on the Blades Number	38
4.5.1	9 Blades Performance	39
4.5.2	12 Blades Performance	40
4.5.3	15 Blades Performance	41
4.5.4	Blades Overall Performance	42
CHAPTER 5 CONCLUSION		43
5.1	Introduction	43
5.2	Recommendation	44
REFERENCES		45
APPENDIX A SAMPLE APPENDIX 1		Error! Bookmark not defined.
APPENDIX B SAMPLE APPENDIX 2		Error! Bookmark not defined.



UMP

LIST OF TABLES

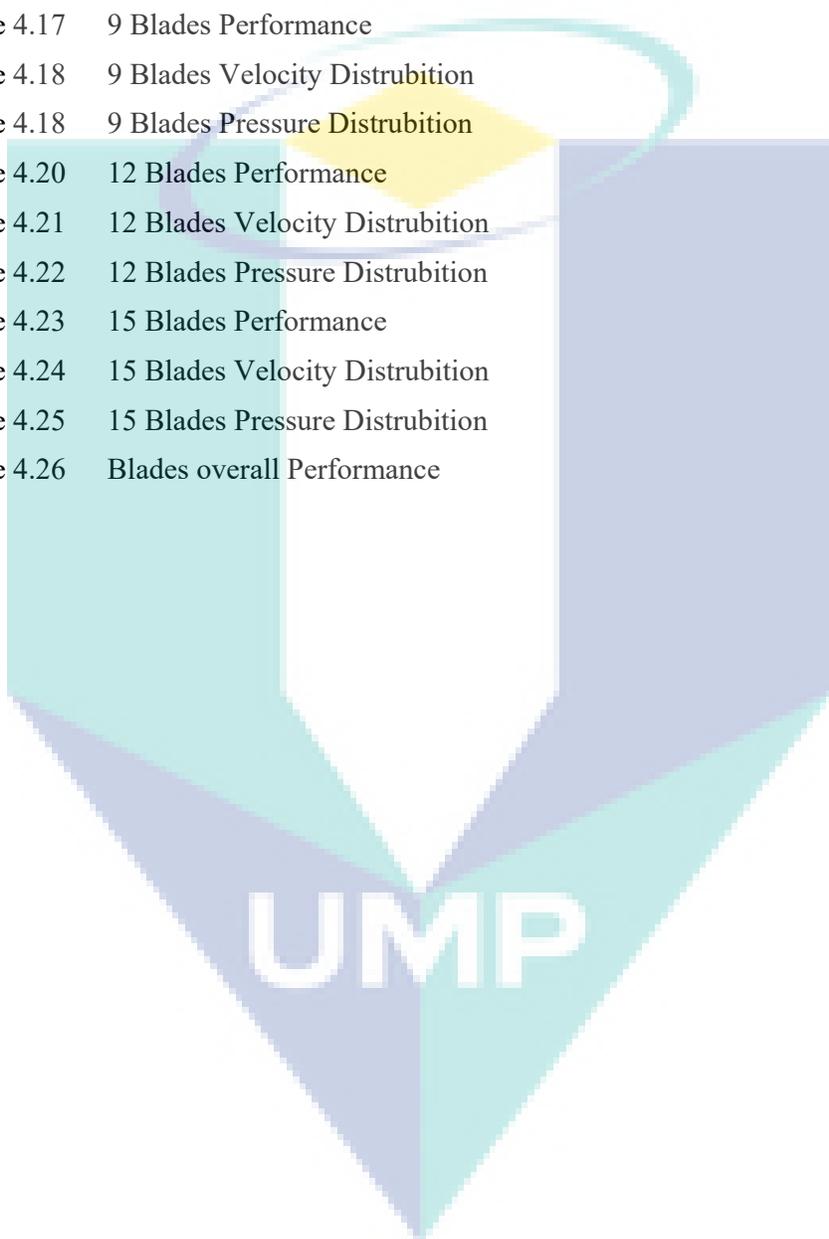
Table 2.1	CFD Simulation And Experiment Comparision	5
Table 2.2	Turbocharger versus supercharger	6
Table 2.3	Result of previous study efficiency versus Cone and inlet blade angle	10
Table 3.1	Turbine Dimension	15
Table 3.2	Mesh Test	19
Table 3.3	Boundary Conditions	22
Table 4.1	Validation Setup	27



LIST OF FIGURES

Figure 2.1	Turbocharger	7
Figure 2.2	Turbocharger Function Diagram	8
Figure 2.3	Turbocharger System Layout	8
Figure 2.4	Radial Inflow Turbine	9
Figure 2.5	The Blade View	10
Figure 2.6	Blade View For Different Cone Angle And 3D Geometric Shape	11
Figure 2.7	Geometric of Two Volute	11
Figure 2.8	Turbine Performance Comparison	12
Figure 2.9	Turbine Characteristic Map	12
Figure 2.10	Total Static Efficiency, Pressure Ratio and Torque vs Blades Number	13
Figure 3.1	GT2056 Turbine Wheels Hosing Diagram	16
Figure 3.2	Blades and Housing Design View	17
Figure 3.3	Turbine Design in 2D View (meridian)	18
Figure 3.4	Radial Turbine 3D Design Views	18
Figure 3.5	Turbine in Design Modeler	19
Figure 3.6	Fluent Flow Turbine Meshing	20
Figure 3.7	SnappyHexMesh workflow	21
Figure 3.8	Mesh View	21
Figure 3.9	Fluent and TCFD controal setup	23
Figure 3.10	Fluent and TCFD Simulation Flow Chart	23
Figure 3.11	Flow Chart	24
Figure 4.1	ANSYS Fluent Validation	27
Figure 4.2	TCFD Validation	28
Figure 4.3	Model Validation	28
Figure 4.4	Fleunt Flow Results	30
Figure 4.5	TCFD Results	31
Figure 4.6	Residuals graph	32
Figure 4.7	variables iteration graphs	33
Figure 4.8	Total Pressure Difference Performance for both Tempretures	33
Figure 4.9	Velocity Performance for both Temperatures	34
Figure 4.10	Total Pressure Difference Performance for both Tempretures	34
Figure 4.11	Velocity Performance for both Temperatures	35

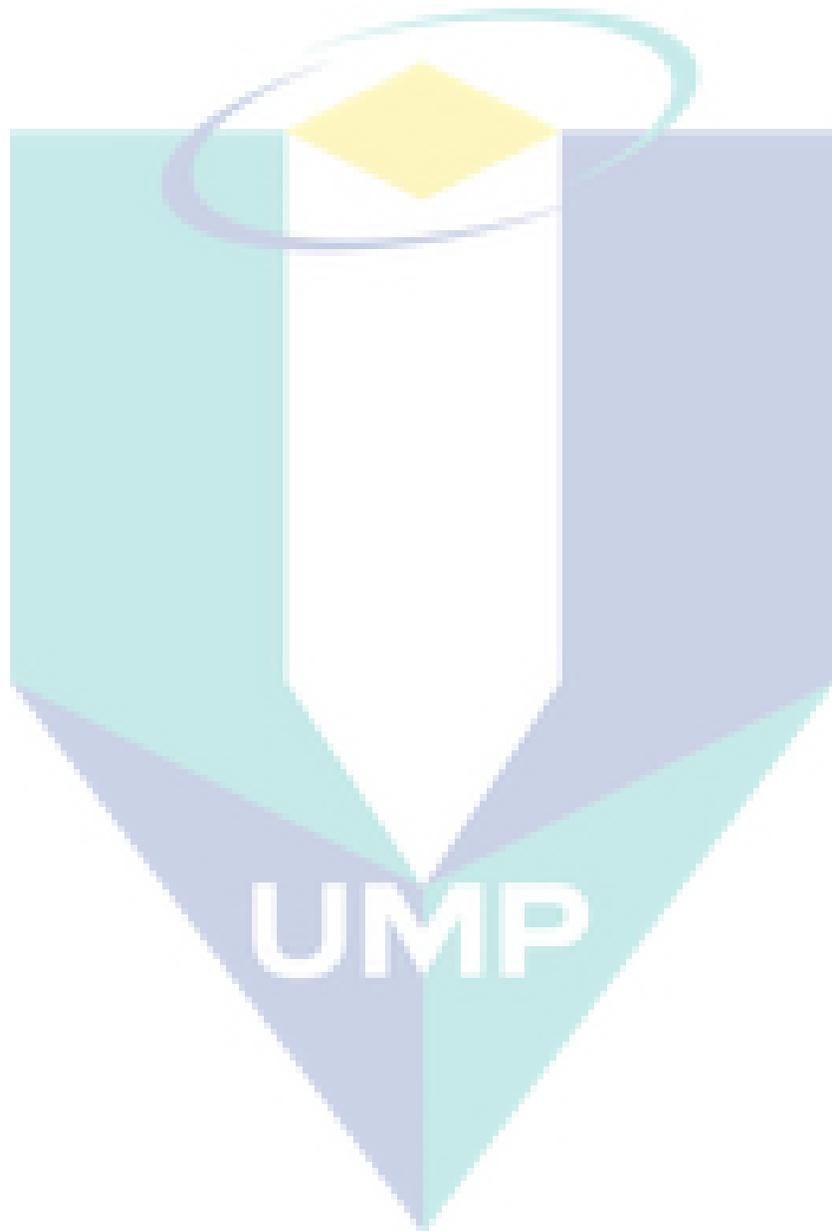
Figure 4.12	Total Pressure Difference Performance for both Temperatures	35
Figure 4.13	Velocity Performance for both Temperatures	36
Figure 4.14	Performance All RPMs Using Temperature 1	37
Figure 4.15	Performance All RPMs Using Temperature 2	37
Figure 4.16	180000rpm Performance for both Temperatures	38
Figure 4.17	9 Blades Performance	39
Figure 4.18	9 Blades Velocity Distribution	39
Figure 4.18	9 Blades Pressure Distribution	39
Figure 4.20	12 Blades Performance	40
Figure 4.21	12 Blades Velocity Distribution	40
Figure 4.22	12 Blades Pressure Distribution	40
Figure 4.23	15 Blades Performance	41
Figure 4.24	15 Blades Velocity Distribution	41
Figure 4.25	15 Blades Pressure Distribution	41
Figure 4.26	Blades overall Performance	42

The logo for UMP (Universitas Muhammadiyah Purwokerto) is a large, downward-pointing arrow shape. It is composed of four quadrants: the top-left and bottom-right are light blue, the top-right and bottom-left are light purple, and the center is white. The letters 'UMP' are written in a bold, white, sans-serif font across the white center.

UMP

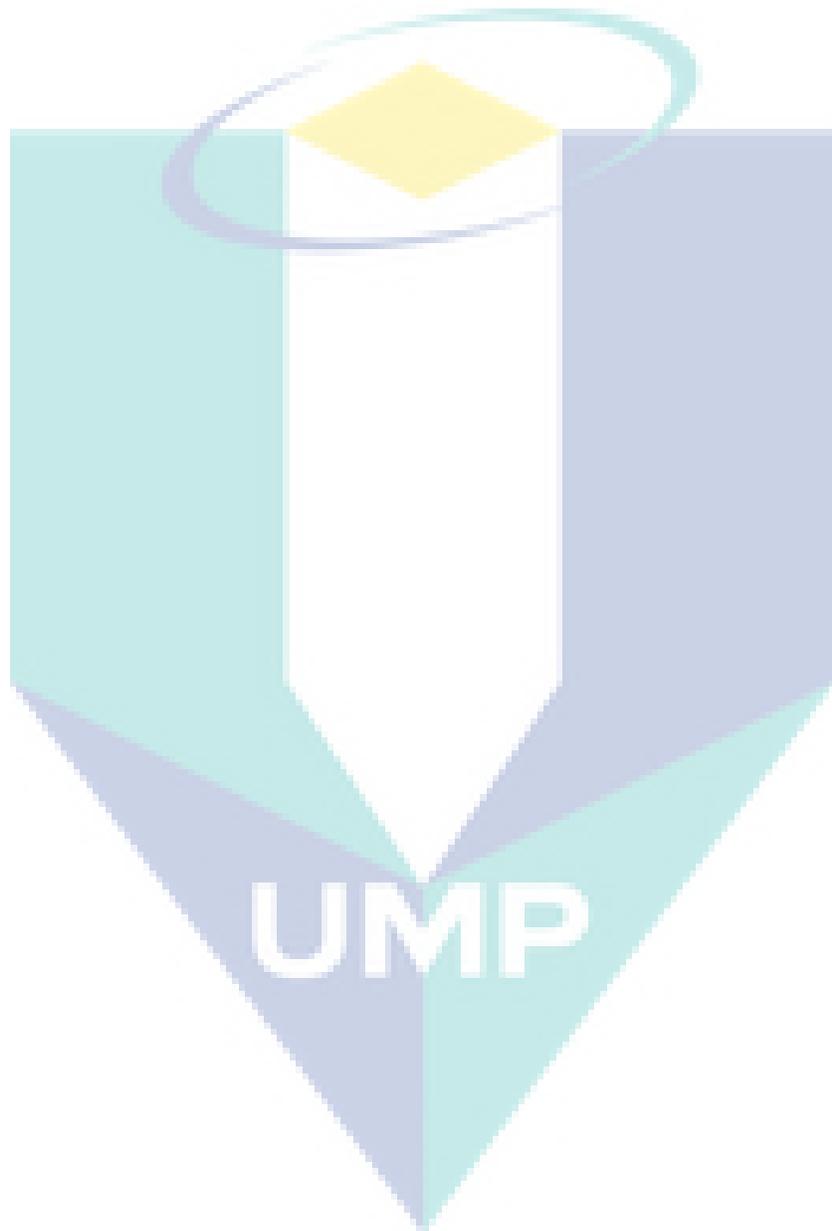
LIST OF SYMBOLS

SBPWM	Simple Boost Pulse Width Modulation
ZSI	Z source inverter



LIST OF ABBREVIATIONS

SBPWM	Simple Boost Pulse Width Modulation
ZSI	Z source inverter



CHAPTER 1

INTRODUCTION

1.1 Background

Normally the exhaust gas pressure is used to generate the turbochargers. During this process turbocharges energy is recovers from remain gases in the exhaust which may otherwise be lost, but due to the breakdown of the turbo, there would be a postponement among the boost pressure building up and when the throttle being opened. This lateness in the response of the turbo can be noticed at the low speed engines only and it is reduce with automatic transition which let the engine speed and flow rate to increase rapidly. Additional way to mitigate the lag of the turbo is to operate the engine at high rpm speed. Even if it will case increase in the fuel consumption. The suppliers of the turbochargers are continuously working to improve the turbocharger system efficiency and reduce the lag of the turbo.

Nowadays improving the efficiency and fuel consumption of the automobiles is becoming much important and that is because of the universal move to decrease CO₂ emissions. The engine supposed to achieve concurrently a high power intensity at lower fuel consumption. The turbocharger at higher pressure is used to improve the fuel consumption of an engine via allowing the engine to downsize. Thus, it is necessary to develop an optimum system for turbo charging to determine the engine specifications. The computational fluid dynamics (CFD) is a simulation software use to analysis the system that involve fluid flow and heat transfer through computer-based simulations. Therefore, Instead of fabricating a turbocharger and test it to determine the optimum engine specifications it better to use CFD simulation software to obtain the characteristics of turbocharger.

1.2 Problem Statement

Nowadays, the fuel energy is considered as one of the most important issues in the worldwide. Due to its high consumption and it's also considered as a huge reason to increase the CO₂. The fuel energy in the automotive cars is divided into three main parts, 1/3 is useful power goes to the crankshaft and 1/3 goes as a heat loss to cooling water and the last 1/3 of the fuel energy is considered as a heat loss through the exhaust. Check the actual.. 30% goes to the power, 30% goes to the heat loss etc... please check this

Turbochargers play an important role in the global move to reduce the CO₂ emission by using the remaining gas energy in the exhaust which may otherwise be lost and allow the engines to downsize for economy fuel. Because the turbochargers improve the automobiles efficiency, drivability and the performance of engines. Turbochargers can help to achieve the global target for fuel economy and CO₂ emission reduction and remain the dominant of the internal combustion. However, the researches on the turbocharger turbine are mostly conducted on using the experimental method. Furthermore, the researches that were conducted using the numerical simulation are more focused on the turbine blades, while a few researches were conducted for the turbine blades and housing. Whatever, in this study both the blades and housing will be considered for the turbine numerical simulation

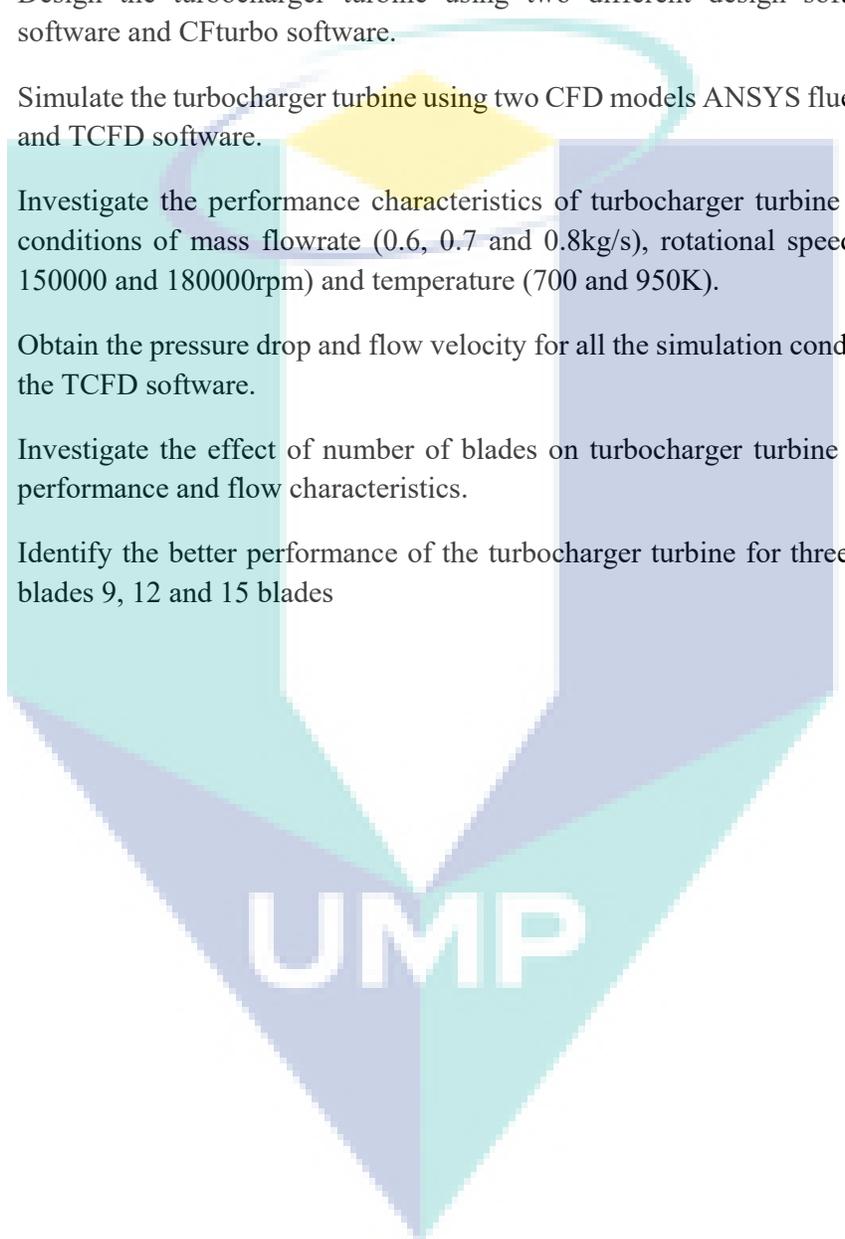
1.3 Objectives

1. To study the performance characteristics of the turbocharger turbine based on the GT2056 model.
2. To analyze the pressure drop and flow characteristics of the turbocharger turbine using numerical simulation.
3. To study the effect of different number of blades on turbocharger turbine performance.

1.4 Scopes

In order to achieve the project objectives, the project scopes must be set to complete the project based on it. The project scopes listed as follow.

- i. Design the turbocharger turbine using two different design software CAD software and CFturbo software.
- ii. Simulate the turbocharger turbine using two CFD models ANSYS fluent software and TCFD software.
- iii. Investigate the performance characteristics of turbocharger turbine for various conditions of mass flowrate (0.6, 0.7 and 0.8kg/s), rotational speeds (120000, 150000 and 180000rpm) and temperature (700 and 950K).
- iv. Obtain the pressure drop and flow velocity for all the simulation conditions using the TCFD software.
- v. Investigate the effect of number of blades on turbocharger turbine in terms of performance and flow characteristics.
- vi. Identify the better performance of the turbocharger turbine for three number of blades 9, 12 and 15 blades

A large, semi-transparent watermark of the UMP logo is centered on the page. The logo consists of a shield-like shape divided into four quadrants by a white 'V' shape. The top-left quadrant is light blue, the top-right is light purple, the bottom-left is light purple, and the bottom-right is light blue. The letters 'UMP' are written in white, bold, sans-serif font across the bottom of the shield.

UMP

CHAPTER 2

LITERATURE RIVIEW

2.1 Introduction

In internal combustion engine (ICE), the chemical energy is converted into thermal energy then to mechanical energy. One third of the energy fuel is consider as heat loss through the exhaust and that is a waste of energy. The exhaust turbocharging is the most general method for boosting pressure in IC engine. Nowadays it is mostly used in almost all advanced IC engines, to utilize the wasted gases through the exhaust using turbocharger which improve the engine performance and help to reduce the CO₂ emission.

2.2 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is simulation and analysis for any system that involve fluid flow and heat transfer using modelling which is the mathematical physical problem formulation and numerical methods such as discretization and numerical parameters, etc. [1]

For fluid study there are two methods theory analysis, experiment and simulation (CFD). CFD consider as a new method and it has many advantages when compared to the experiment method as shown in the table below.[2][3]

Table 2.1 CFD Simulation And Experiment Comparision

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

Source: Rahman (2007).

2.2.1 Navier-Stokes Equations

The conservation laws of mass, momentum and energy is known as governing equation of fluid dynamics. In computational fluid dynamics the conservation laws known as Navier-Stokes Equations[4][1]

I. Continuity equation

The conservation of the mass is described by the continuity equation.

$$D\rho/DT + \rho (\partial U_i/\partial x_i) = 0 \quad 2.1$$

II. Momentum equation

The conservation of momentum is described by momentum equation.

$$\rho (\partial U_j/\partial t) + \rho U_i \partial U_j/\partial x_i = \partial P/\partial x_i - (\partial \tau_{ij}/\partial x_i) + \rho g_i \quad 2.2$$

III. Energy equation

The conservation of energy is described by the energy equation

$$\rho c_\mu (\partial T/\partial t) + \rho c_\mu U_i \partial T/\partial x_i = -P (\partial U_i/\partial x_i) + \lambda (\partial^2 T/\partial x_i^2) - \tau_{ij} (\partial U_i/\partial x_j) \quad 2.3$$

2.3 Turbocharger Fundamental

Recently charging the engine is consider as common strategy to increase the engine performance. Historically, it has been done using the compressor before the intake air to increase the air pressure which provide higher mass in the cylinder. However. This allowed to have a higher power with same engine displacement. [5] [6]

$$Pe = V . mep . (n/\epsilon)$$

2.4

Based on the above equation to increase the engine power with same displacement is depend on the effective pressure or the rotational speed.

2.3.1 Turbocharger versus Supercharger

There are different method to increase the engine power such as supercharging or turbocharging, both methods has different way to increase the engine power.[7]

Table 2.2 Turbocharger versus supercharger

Turbocharger	Supercharger
Used to increase the engine performance	Also used to increase the engine performance
Use the exhaust gases to drive the turbine which rotate the compressor	Use the engine crank shaft to rotate the compressor
Drive better boost at high RPM	Drive their boots at low RPM
Compressed the air with high temperature	Compressed the air with less temperature

Source: Rahman (2007).

2.4 Turbocharger System

A turbocharger is a device that allow the engine to produce more power. However, turbocharger consist of two main parts turbine (hot side) and compressor (cold side). A turbocharged engines are more powerful and efficient than the natural engines and that is due to the more intake air which forced by the turbine, the fuel in combustion chamber is more than using atmosphere pressure alone. It is use to increase the combustion chamber volumetric efficiency. New technologies are found to help the turbocharging of IC engines such as the inter-cooling for the charged air which will increase the mass flow rate and that is happened before it goes to the combustion chamber, these technologies might improve the efficiency more.[8][9][10]

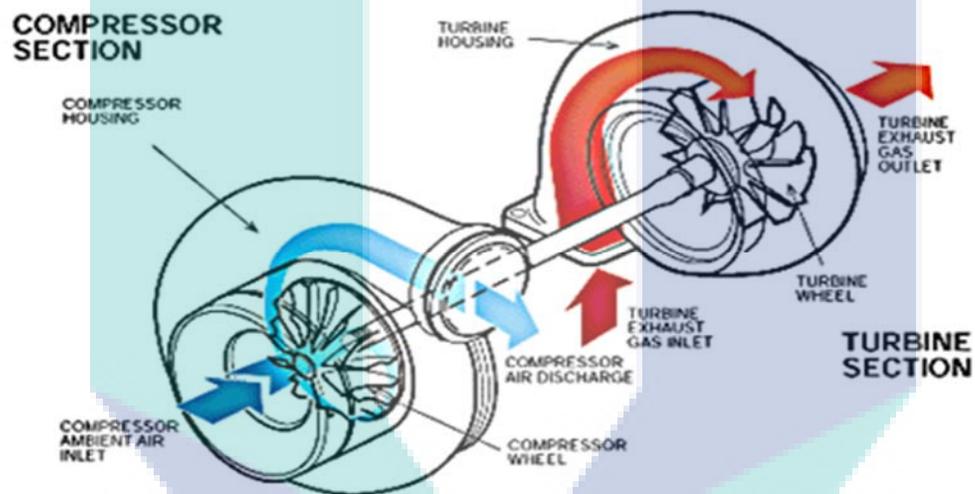


Figure 2.1 Turbocharger

Source: www.thextremexperience.com

2.4.1 Turbocharger System Operation

A turbocharger has two main parts that operates on a single shaft which are turbine (hot side) and compressor (cold side). These two parts are designed to be used on automobile IC engines. When it is installed, the hot gases of the exhaust exit the cylinders to pass into the turbine of the turbocharger, to rotate the blades of the turbine, and so the shaft will rotate. Then the shaft will transfer power to turn the compressor. As the compressor rotates, the pressure of the air that come from the intake air will raise. The high pressure air regularly goes through the inter-cooler to increase the air density. Then

the air is transferred through duct to the cylinder, and thus combustion happens as usual. At the same volume, the air with large mass would allow more fuel to be burned, thus the piston will extract more power during the power stroke, and thus more power will transmit to the crankshaft and then to the wheels.[11][12][13]

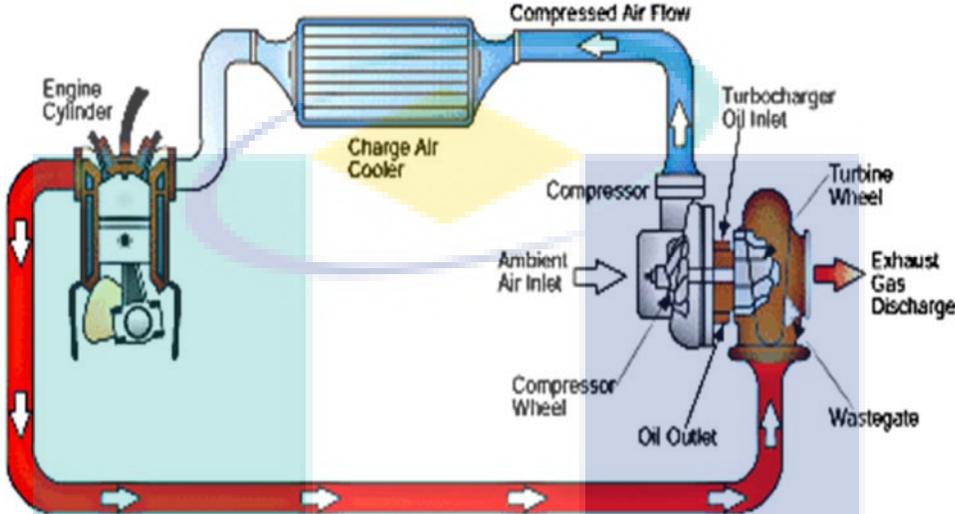


Figure 2.2 Turbocharger Function Diagram

Source: www.arediesel.com

2.4.2 Turbocharger System Analysis

System parametric analysis is based on the turbocharger data of operating conditions and engine requirement. The data needed are flow rate that required to charge the engine and air fuel ratio, the temperature, the pressure required for charging, the temperature and velocity of the exhaust, the turbine presser ratio, assist power percentage the turbine and compressor impellers area ratio.[10][14]

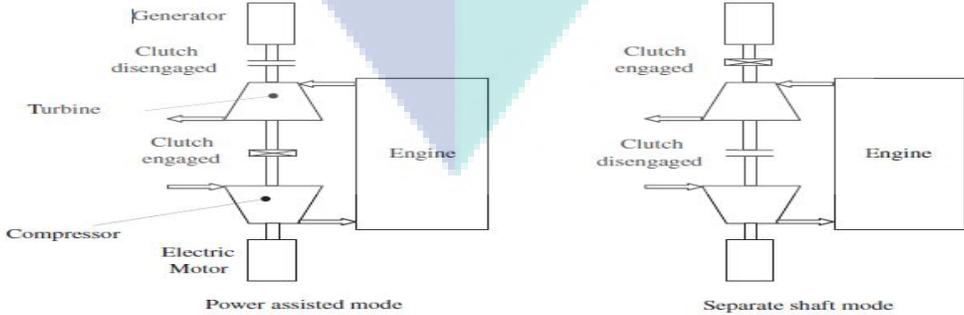


Figure 2.3 Turbocharger System Layout

Source: Energy Conversion and Management 58 (2012) 59–65

2.5 Turbocharger Radial Turbine

Turbocharger turbine is an important part in the turbocharger which regain the energy from the exhaust gas of the engine. There are two type of turbine radial and axial turbine. In the radial turbine the flow enters the turbine radially but the flow drop out axially. The radial turbine consists of four components the rotor, stator, volute and the inlet pipe.[15][16]

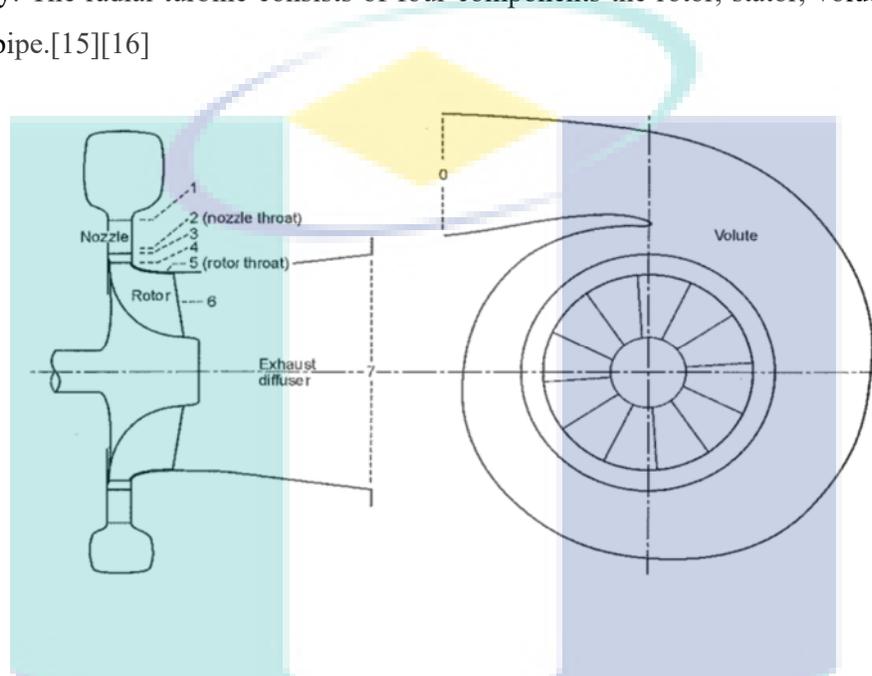


Figure 2.4 Radial Inflow Turbine

Source: Conceptual design tool for radial turbines

2.5.1 Turbine Blade Angle

Automobile that use turbocharging diesel engine and work for different range of loads and speed. The turbine geometry of the mixed inflow is designed using radially bending which is the leading edge of the radial turbine. Its advantage is to reduce the flow path curvature and reduce the secondary flow of the formation which normally found in the radial turbine. By changing the cone of the angle, then a non-zero blade angle will be possible to archive. The force acting on the blades is useful, when the rotor blades shape are suitable and oriented properly. A lot of experimental and numerical studies has been done to study the performance of the mixed inflow turbine that used in internal combustion engines to understand the phenomena of their operation.[17]

Table 2.3 Result of previous study efficiency versus Cone and inlet blade angle

Authors	δ_2 [°]	β_{2b} [°]	η
Baines et al. [1] (experimentally)	40	30	0.77
Abidat et al. [2],[3] (experimentally)	40	20	0.84
		10	0.73
		30	0.72
		30	0.71
		10	0.72
Leonard et al. [4] (Numerically)	45	20	0.71
		30	0.68
		10	0.69
		20	0.66
		30	0.65

Source: Conceptual design tool for radial turbines

2.5.2 Turbine Rotor

The figure below show the mixed inflow turbine with the rotor defined with its constant blades angle.

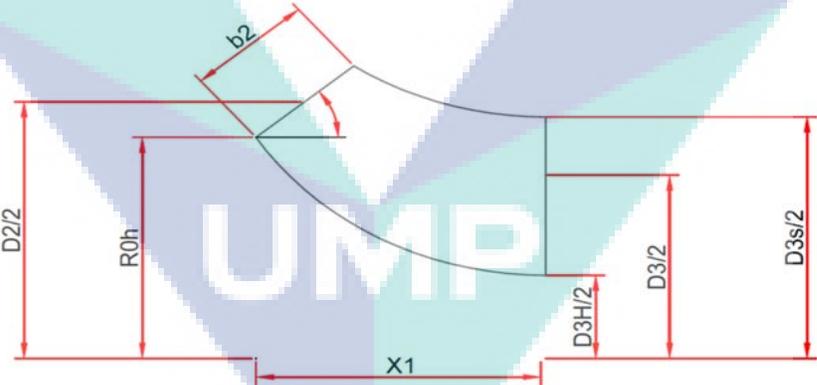


Figure 2.5 The Blade View

Source: M. A. Chelabi, M. K. Hamidou, M. Hamel

In order to obtain new rotor shapes the cone angle values must change with keeping the same geometrical parameter to conserve the same casing. The blade shape for different cone angle shown in the figure below. [18][19]

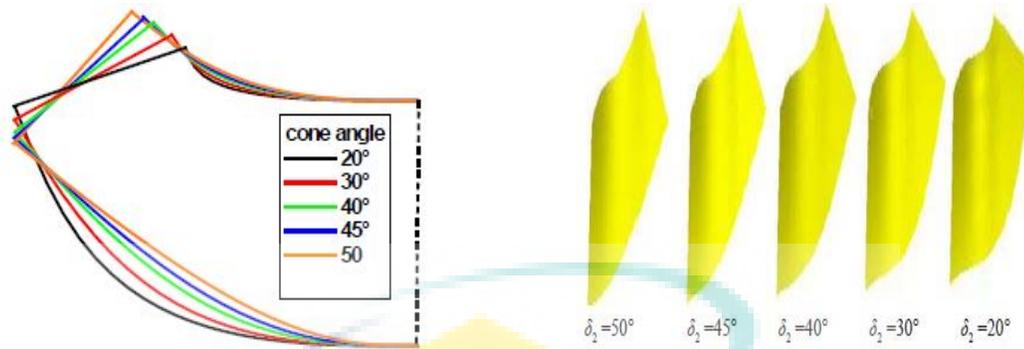


Figure 2.6 Blade View For Different Cone Angle And 3D Geometric Shape
 Source: M. A. Chelabi, M. K. Hamidou, M. Hamel

2.5.3 Radial Turbine Volute

The main function of the volute in the radial is to pass the uniform flow to the rotor at the desired angle to optimize the turbine performance. The turbine volute design is based on steady condition. There are many completed research on the improvement performance of the turbocharger turbine, some research are still ongoing. The turbocharger turbine is subjected to pulsating flow. Numerical and experiment study run for two different volute at cold flow turbocharger.[20][21]

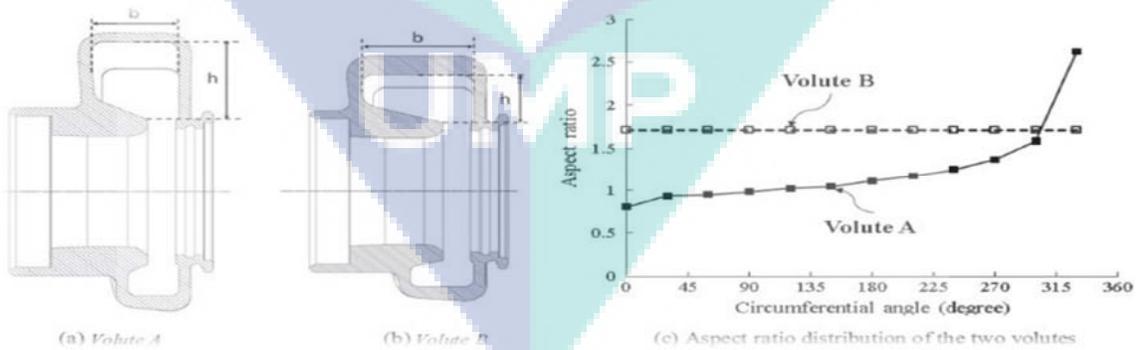


Figure 2.7 Geometric of Two Volute

Source: M. Yang et al. / Energy Conversion and Management 105 (2015) 167–177

The figure 2.13 (a) show a comparison of the two volute configurations at two speeds. The mass flow parameters difference of the two volute is small and that is probably due to the cross sectional shape. The MFP for volute B slightly drop by average percentage of 2% for bot speed compare to volute A. Figure 2.13 (b) shown the turbine efficiency of the two volute which is normalized by volute A peak efficiency at 40k MFP.[22]

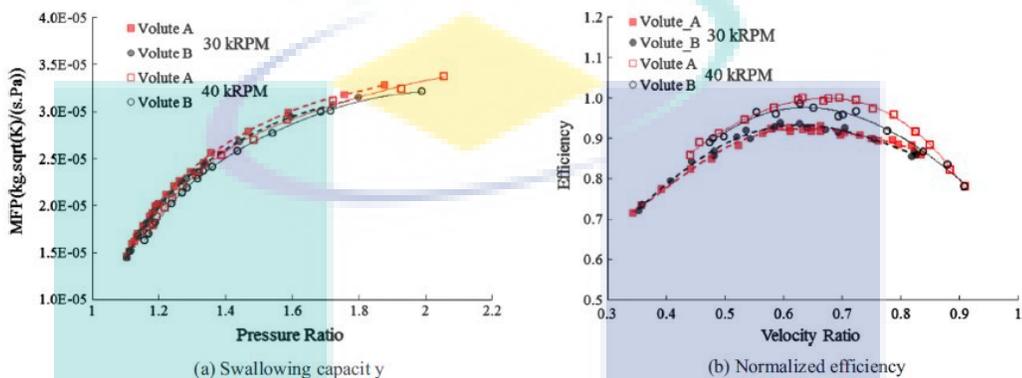


Figure 2.8 Turbine Performance Comparison

Source: M. Yang et al. / Energy Conversion and Management 105 (2015) 167–177

2.5.4 Turbine Map

The turbocharger turbine is consider as machinery machine which it is operating range is shown using curves, this is known as turbine characteristic map. The map characteristic is calculated by testing the turbine components using either the laboratory experiments or the simulation analysis. The test is run with various condition of mass flow rate, pressure ratio and rotational speed, then the characteristic map is drawn with the mass flow in the y axis and the pressure ratio in the x axis.[5][23]

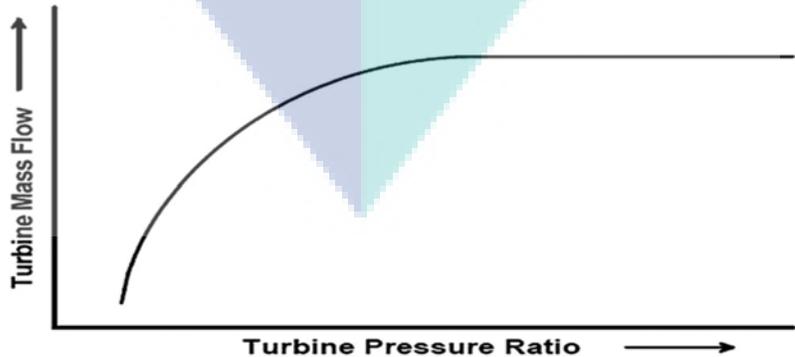


Figure 2.9 Turbine Characteristic Map

Source: <http://dx.doi.org/10.1007/978-1-84882-375-4>

2.6 Effect of Blade Number on Turbocharger Turbine Performance

In order to get better efficiency for any condition of design, the mixed inflow turbine geometry must be optimized. A theoretical and experimental studies show that the characteristics of the geometrical such as the blade angle, axial length and blades number which has a large impact on the turbine efficiency and flow passage. Many simulations are ongoing for the rotor based on various number of blades from 8 blades up to 20 blades, there blades are having the same geometry shape. The result obtained in term of total to static efficiency shows that rotor with 14 blades has the higher efficiency which is about 84%. As shown in the figure 2.12 (a). Another result is obtained in term of variation of total pressure ratio, the result in figure 2.12 (b), shows that poor expansion occur when a less blade number is used. In figure 2.12 (c), the result shows that maximum torque is obtained when using 14 rotor blades.[17][14][24]

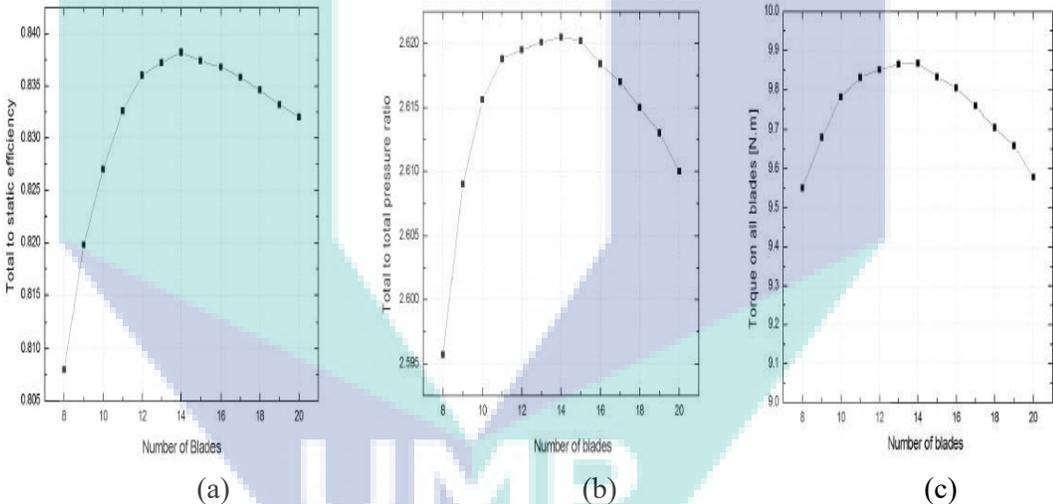
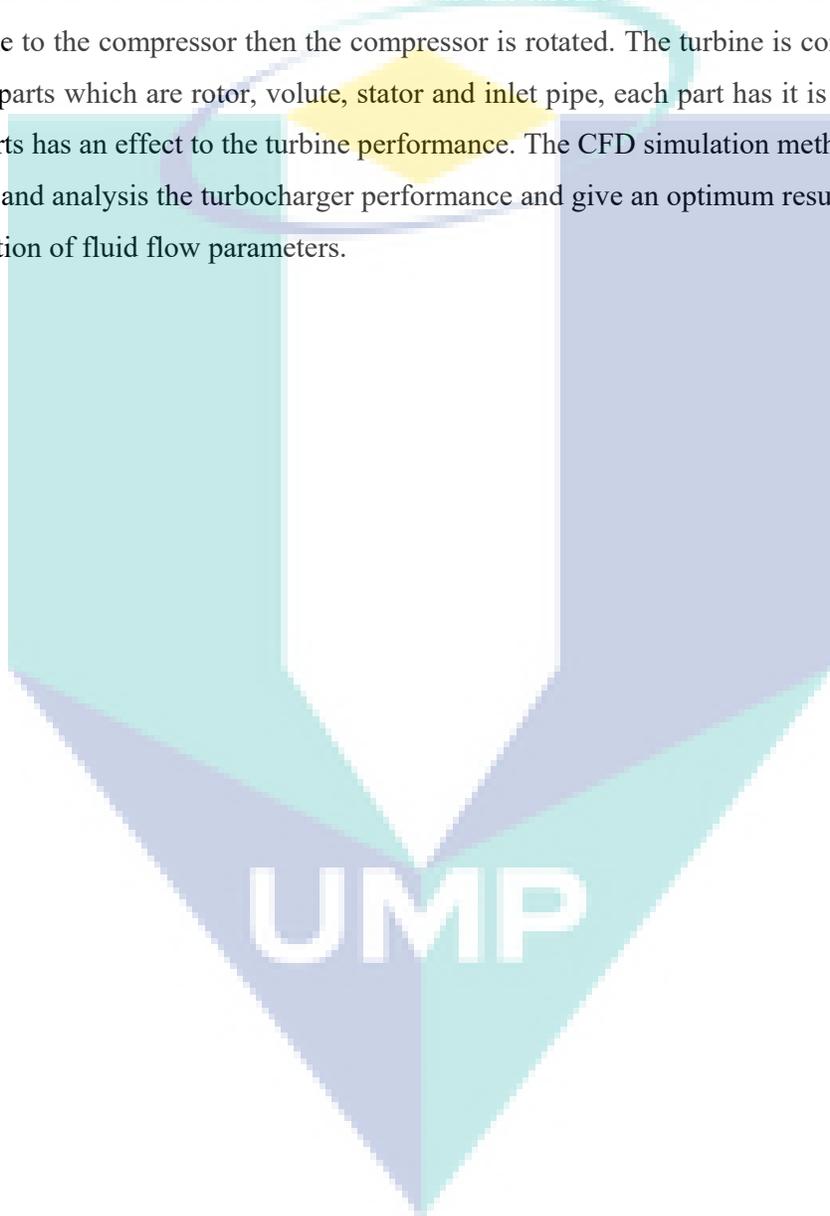


Figure 2.10 Total Static Efficiency, Pressure Ratio and Torque vs Blades Number
Source: Engineering Review, Vol. 37, Issue 3, 349-360, 2017.

2.7 Summery

The turbocharger is used in internal combustion engine using the exhaust wastage gases which counted as one third of the total energy. However, the turbocharger consists of two main parts turbine and compressor and its work based on the gases that comes from the exhaust which rotate the turbine and the turbine drive the shaft that connect the turbine to the compressor then the compressor is rotated. The turbine is consists of four main parts which are rotor, volute, stator and inlet pipe, each part has it is function and all parts has an effect to the turbine performance. The CFD simulation method is used to study and analysis the turbocharger performance and give an optimum result for various condition of fluid flow parameters.



CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter describes the turbocharger turbine geometry design including the blade shape and number, mesh design and the parameters that needed to run the simulation for ANSYS and TCFD software. Besides that, this chapter will also describes the procedure of designing and simulating the turbocharger turbine using numerical simulation. Furthermore, it will also explain the mathematical moulding and boundary conditions that must be set in order to run the simulation smoothly.

3.2 Model Description

The turbocharger GT2056 has be selected as a model for this study. All the dimensions for the wheels and hosing are given by Garrett, the wheel diameters and the tip heights and the Area/Radius which use to describe the turbine housing characteristics, it is the inlet cross section area divided by the turbo centreline radius to the centroid of the area. The wheel dimension and hosing A/R are as follow:

Table 3.1 Turbine Dimension

Variable	Value
Diameter A/B	40mm/47mm
Tip height C	6.1mm
Blades	12
A/R	0.46

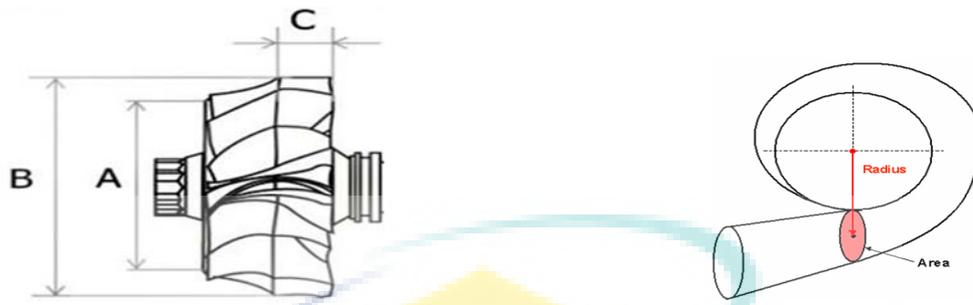


Figure 3.1 GT2056 Turbine Wheels Hosing Diagram

3.2.1 Geometry Design

The geometry design is the most important step in the CFD simulation, so the design must be professional and suitable for the simulation. The radial turbine parts were designed using two different design software solidworks and cfturbo. The turbine wheels were designed with three different blade number 9, 12 and 15, but the housing design same for all the blades designs.

3.2.1.1 Solidworks Geometry Design

Solidworks design software is one of the professional design software and it is the most suitable for CFD ANSYS simulation software, because it is ease to modify and improve the design in the ANSYS geometry design. The radial turbine wheel were design with 40mm inner diameter, 47mm outer diameter, 6.1mm tip height and 12 blades only. The housing was designed with 0.46 A/R. The following figures show the geometry design for the wheels, housing and assembly views.

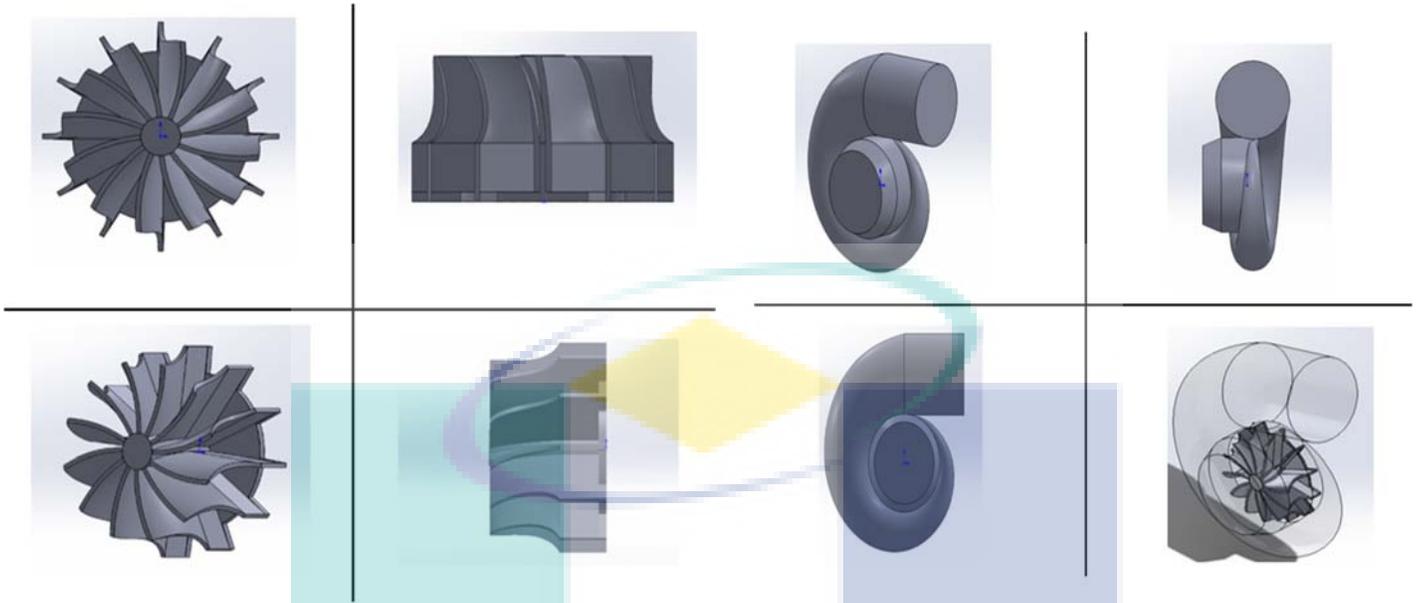


Figure 3.2 Blades and Housing Design View

3.2.1.2 Cfturbo Geometry Design

Cfturbo is a modern and professional design software use to design turbomachinery models such as turbine and compressor. In cfturbo each component of the turbine was design separately the design steps start with designing the rotor and blades then the stator after that the volute and end with the outlet pipe. Using cfturbo there is option for all turbomachinery machine by selecting the turbine option all parameter for turbine is automatically selected just need to change some of the parameters such as the rotor diameter, blade number, inlet and outlet pipe length and the volute type.

Design steps as follow:

- i. **Blade Design:** the first step in designing the blade is the blade dimension it is use to define the impeller dimensions and set all the parameters and it is also show the velocity triangle. The second step is the blade properties which include the blade number and blade shape options. The last step is the blade mainline which shows the 3D view of the blade and the angle of the blade.
- ii. **Volute Design:** to design the volute there are there main steps which are cross section, spiral development and inlet geometry. Step one to define the cross

section area and step two is to set the radius and show the 3D view. Third step is to design the inlet pipe and set its length.

- iii. **Stator design:** the rotor and the outlet pipe are called stator and both have common design steps the most important step is meridional contour.

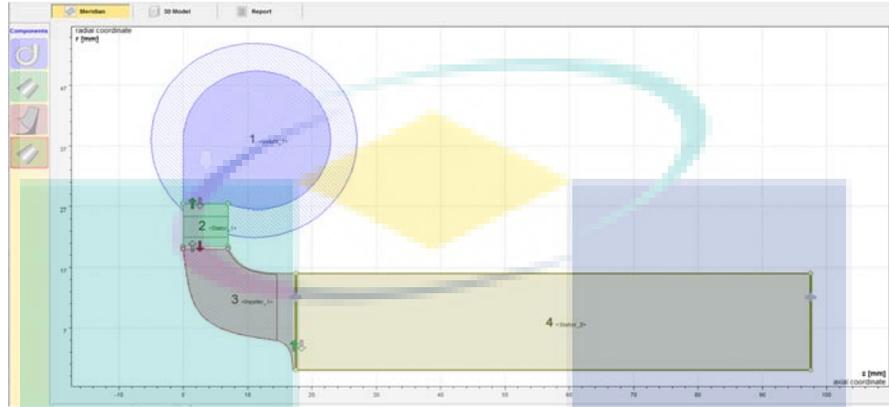


Figure 3.3 Turbine Design in 2D View (meridian)

After completing all the radial turbine design steps successfully and based on the model required dimensions the 3D model is generated as shown below.

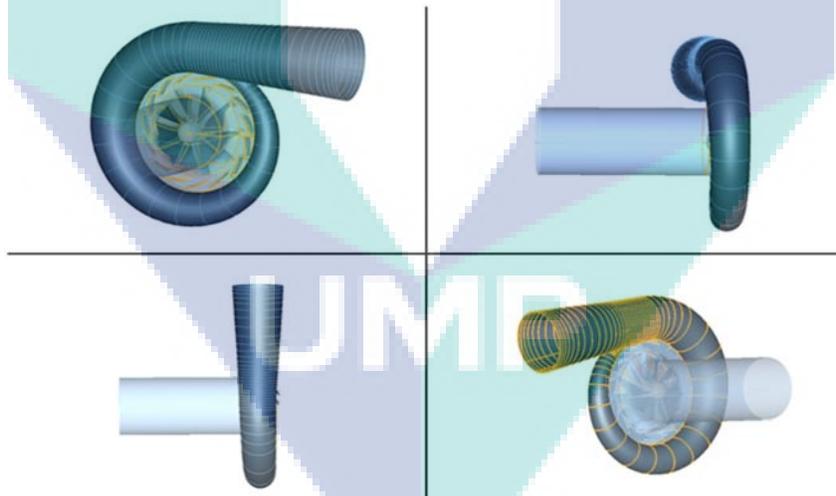


Figure 3.4 Radial Turbine 3D Design Views

3.3 Mesh Design

Since there are two simulation software the mesh preparation and mesh test will be describe for the two software.

3.3.1 Mesh Using ANSYS Software

As the geometry design is finished and imported to ANSYS Fluent flow Design Modeler to modify the design and create the blade cavity in the housing by using the Boolean option and create the rotating domain to allow the blade to rotate. In the figure below it shows the turbine design in the Design Modeler and the blades are show in green colour which represent the cavity of the blades and the housing is set as fluid as shown below.

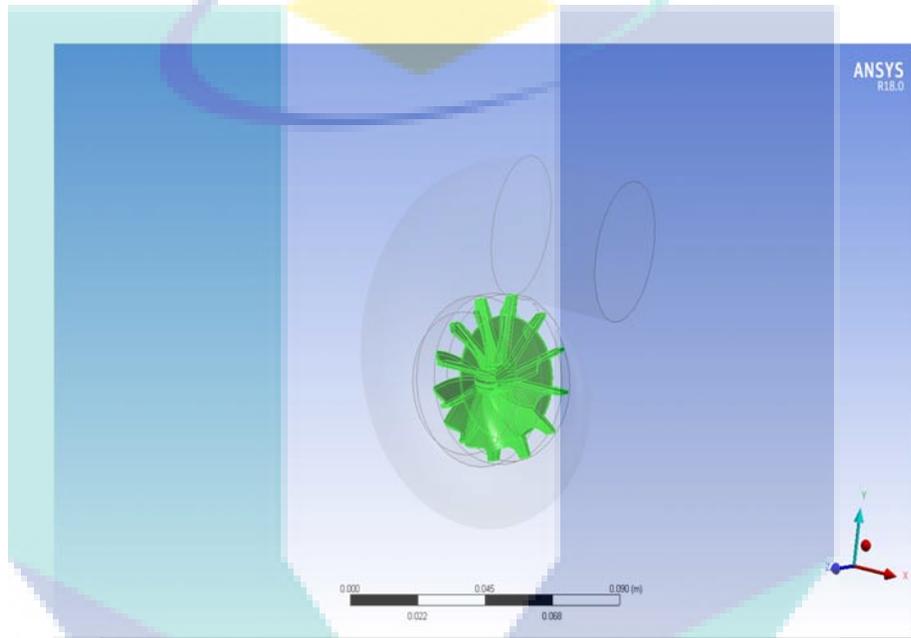


Figure 3.5 Turbine in Design Modeler

The mesh was carried out using ANSYS fluent. The first mesh were generated randomly for all the three mesh type course, medium and fine the mesh quality is often bad and need to be improved. The mesh can be improved by using mesh methods such as automatic method and change the mesh sizing and make mesh test to select the best mesh. The mesh test was done for three different mesh as shown in the table below.

Table 3.2 Mesh Test

Mesh No	Total mesh size	η
1	1 million	0.489
2	1.4 million	0.645
3	1.6 million	0.701

The second mesh was selected to be proceed for the simulation, because it has the closest efficiency to the actual model. The figure below shows the mesh on the housing and the blade.

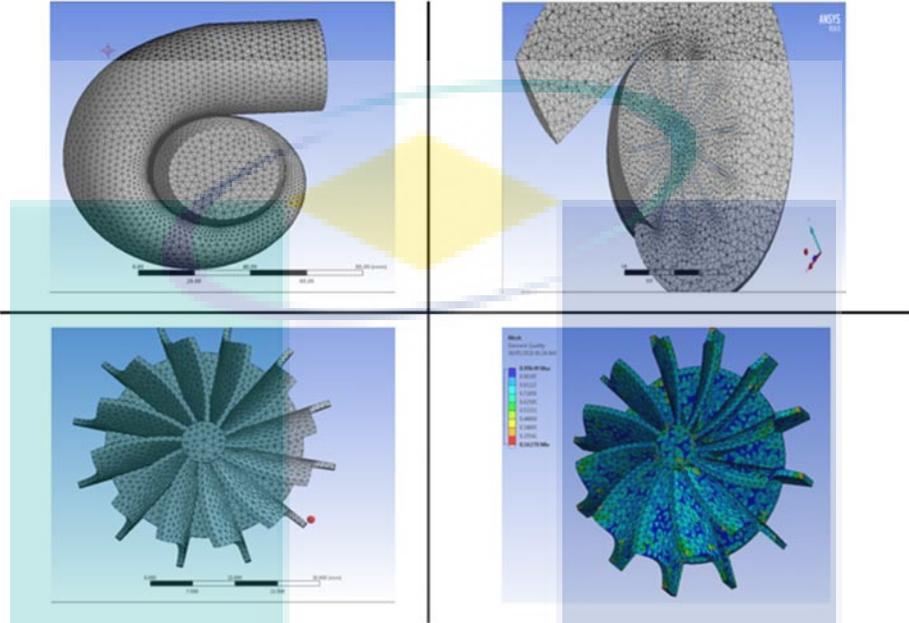


Figure 3.6 Fluent Flow Turbine Meshing

3.3.2 Mesh Using TCFD Software

The mesh has been done using TCFD software which use the OpenFOAM package. The OpenFOAM is using SnappyHexMesh which is a fully automatic mesh and it works based on four steps which are background mesh, castellated mesh, snapped mesh and layered mesh. The quality is automatically after the third step. The SnappyHexMesh consider as the best mesh when using OpenFOAM. The SnappyHexMesh workflow and mesh result are shown in the following figures.

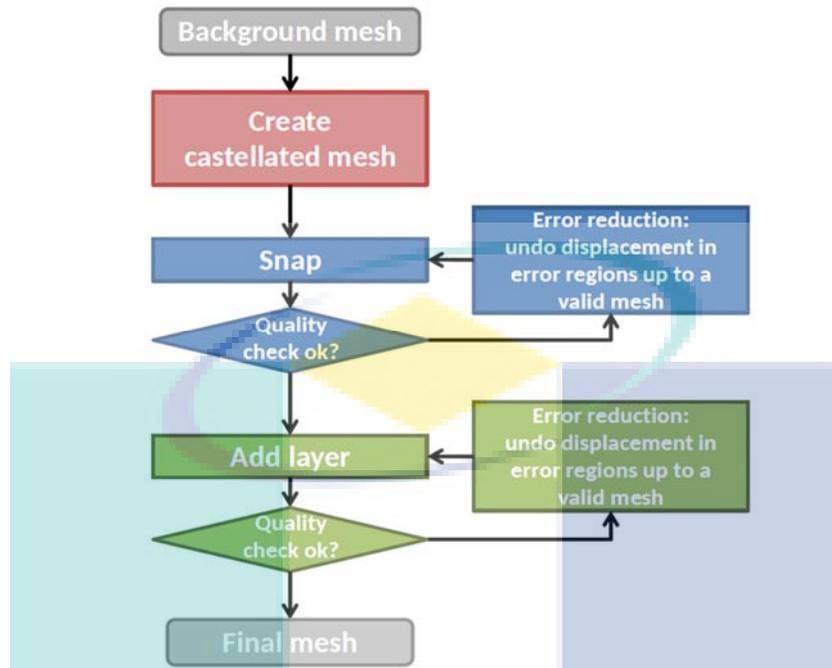


Figure 3.7 SnappyHexMesh workflow

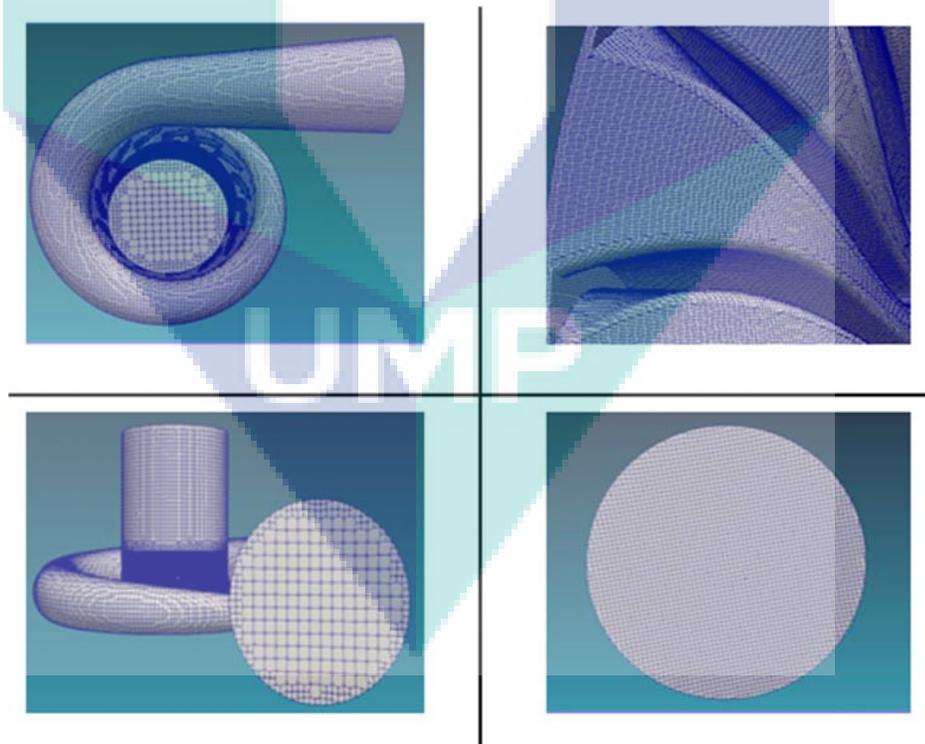


Figure 3.8 Mesh View

3.4 Boundary Condition and Simulation Setup

The two simulation software are using the same boundary conditions and numerical method. The boundary conditions are one of the most important step in numerical simulation, it is necessary to be defined and apply to all the bounding sections of the domain to solve the calculations. The simulation method must be defined as well and select the suitable method for the chosen study case.

3.4.1 Boundary Conditions

In order to calculate the turbine characteristics numerically the boundary conditions must be set. The boundary conditions are divides into two main sections which are inlet boundary at the turbine inlet which use the total pressure, total temperature, rotational speed and mass flow rate and outlet boundary at the turbine exist and use the static pressure. The boundary condition selected for the two simulation software are as follow.

Table 3.3 Boundary Conditions

Parameter	Value
Temperate, k	700 and 950
Mass flow rate, kg/s	0.6, 0.7 and 0.8
Pressure, pa	101325
Rotational speed, rpm	90000, 120000, 150000 and 180000

3.4.2 Numerical Method

The flow characteristics in the radial turbine are obtained by solving the three conservation equations mass (Eq 3.1), momentum (Eq 3.2) and energy (Eq 3.3) also the equation of state (Eq 3.4) as well.

$$D\rho/DT + \rho (\partial U_i/\partial x_i) = 0 \quad 3.1$$

$$\rho (\partial U_j/\partial t) + \rho U_i + \partial U_j/\partial x_i = \partial P/\partial x_i - (\partial \tau_{ij}/\partial x_i) + \rho g_i \quad 3.2$$

$$\rho c_\mu (\partial T/\partial t) + \rho c_\mu U_i = -P (\partial U_i/\partial x_i) + \lambda (\partial^2 y/\partial x_i^2) - \tau_{ij} (\partial U_i/\partial x_i) \quad 3.3$$

$$P = \rho RT$$

3.4

In the turbomachinery the flow is turbulent so, the turbulence model $k - \epsilon$ equation is selected for the radial turbine simulation. The figure below shows the simulation method and control for both software.

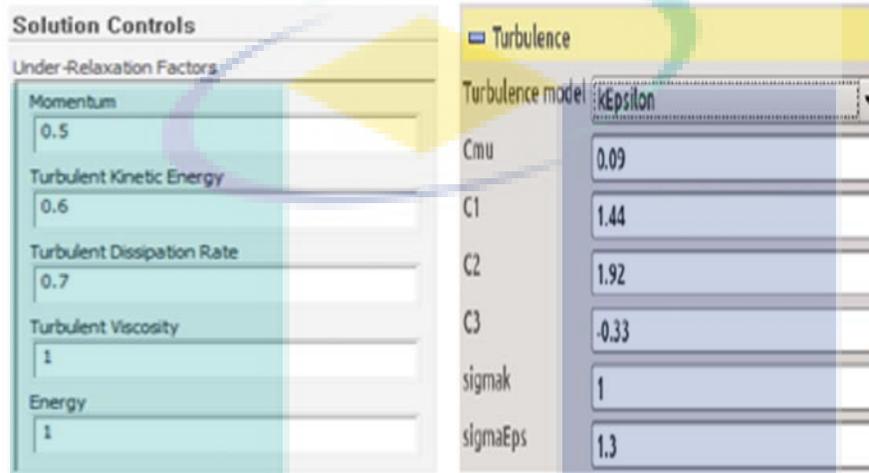


Figure 3.9 Fluent and TCFD control setup

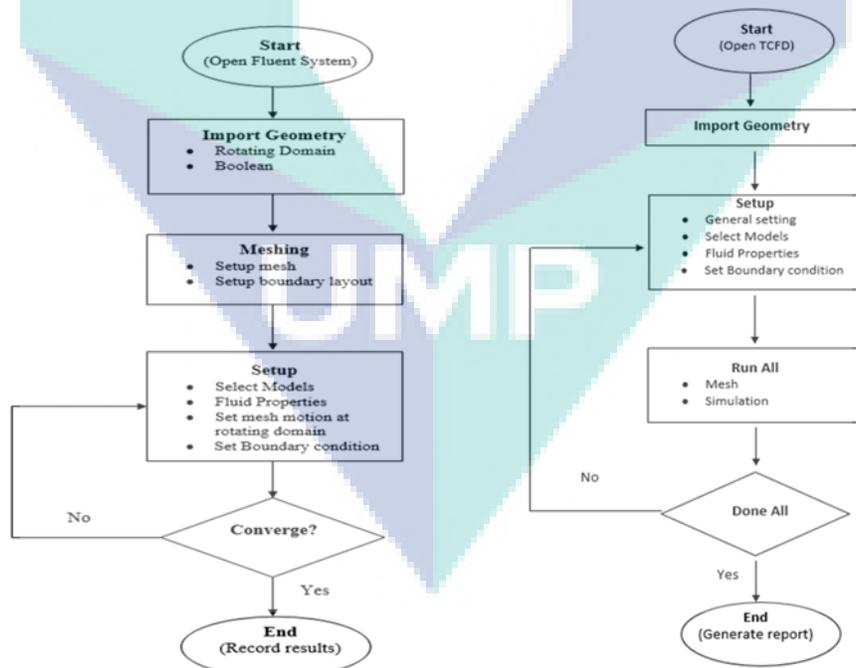


Figure 3.10 Fluent and TCFD Simulation Flow Chart

3.5 Flow Chart

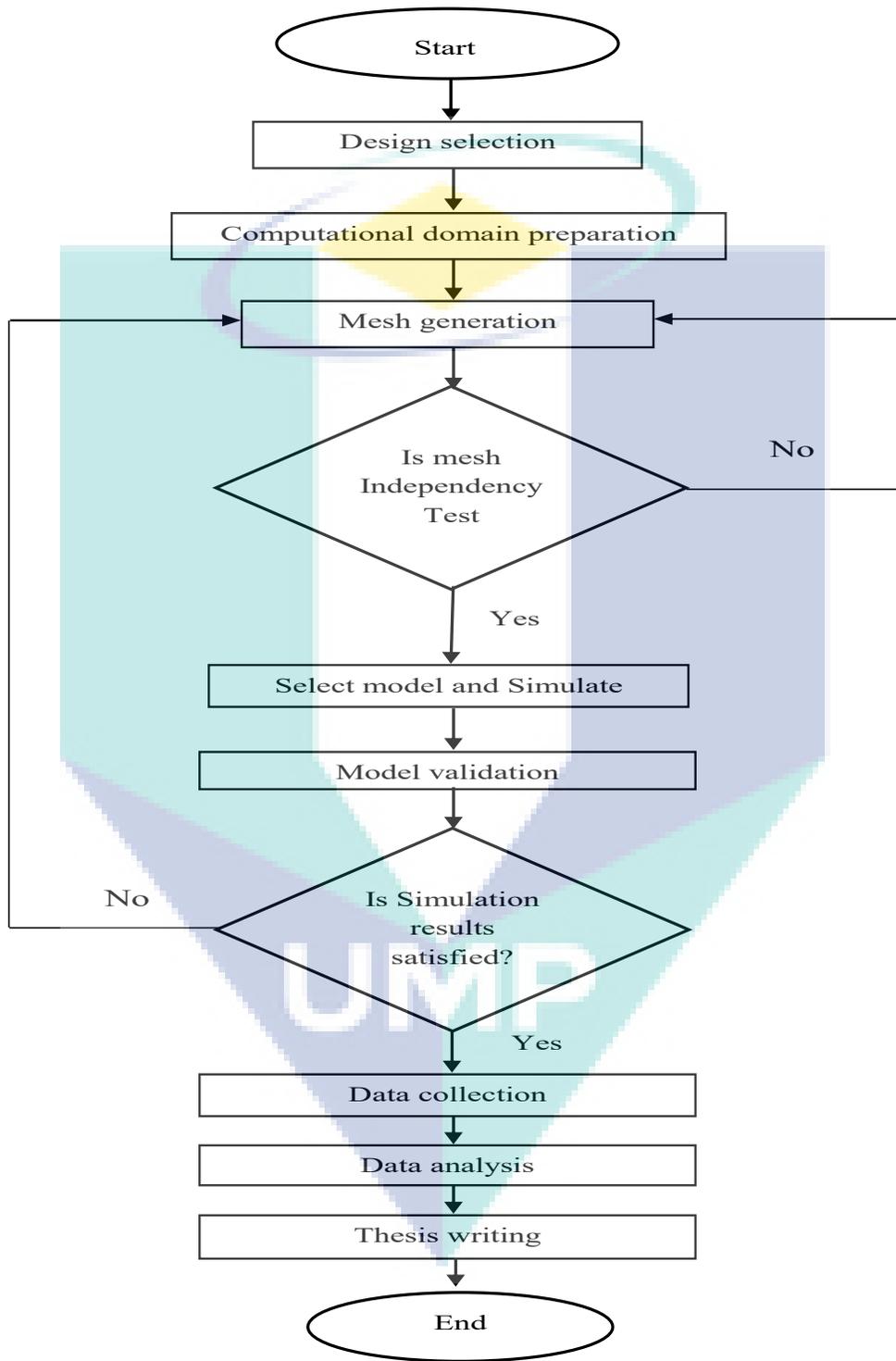


Figure 3.11 Flow Chart

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

This chapter will present the turbocharger radial turbine performance characteristics. The first point in this chapter is the validation model which is an important step to get the accurate CFD simulation setup based on actual model or another study data. Beside that this chapter include the ANSYS Fluent and TCFD simulation results and a brief comparison between the two simulation software results will be considered to identify the best simulation software for the radial turbine to proceed the simulation and to validate the result with the actual model data.

The second interest point in this chapter is to analysis the radial turbine performance characteristics for different mass flowrate and rotational speed using the same geometry design and fluid properties to investigate the flow characteristics such as the pressure drop, temperature difference, torque and efficacy. However, final performance of the radial turbine will be evaluated according to three main parameters which are rotational speed, mass flowrate and pressure ratio to be compared with the actual model turbine map.

The last point is to analysis the effect of the number of blades on the turbine performance. Three number of blades will be used 9, 12 and 15 blades in this study. The simulation will be run with the same geometry design and inlet boundary conditions and one rotation speed for all the three blades number.

4.2 Model Validation

The model validation was carried out to investigate the accurate and correct simulation setup. The numerical validation tested by the actual GT2056 turbocharger turbine model. To consider the numerical validation result the geometry design and inlet boundary conditions must be the same with the actual turbine model. The design dimensions are from the Garrett GT2056 and the inlet boundary conditions are given by the GT- power working point.

The numerical simulation were performed using two different simulation software. ANSYS fluent and TCFD, thus the numerical validation were conducted using the two software. However the geometry design and inlet condition are same for both software to compare the result for the two software and identify the closest result to the actual turbine model performance.

4.2.1 Numerical Validation setup

The numerical validation simulation method k- ϵ was selected to be the same for the both simulation software and the air as the working fluid. The inlet boundary condition such as the mass flowrate, temperature and rotational speed are 0.045 kg/s, 1000 k and 90000 RPM respectively. The number of blades us is the same with actual model which is 12 blades.

For the mesh it could not be the same, because the two software are using different type of mesh. However, the mesh has been tested for various cases for example, using ANSYS fluent software the mesh can be generated with different type and mesh size to develop and modify mesh. Beside that the mesh has been tested as dissuade in chapter 3. In other hand, the TCFD provide special type of mesh for turbomachinery machines such as turbine which is SnappyHexMesh it is a fully automatic mesh and it is quality is automatically checked and improved this type of mesh consider as the best mesh to be used for the turbocharger turbine. The table below summarize the validation inlet boundary conditions and numerical method for the simulation.

Table 4.1 Validation Setup

Fluid	Air
Model	k- ω
Mesh	SnappyHexMesh/fine
Temperature, k	1000
Mass flow rate, kg/s	0.6
Rotational speed, rpm	90000

4.2.2 Validation Result

The validation result will be show separately for each software then it will be compare to the actual turbine model performance and plotted in one graph to analyse the closest performance result to the actual model. The result will be plotted using the mass flow rate and the pressure ratio.

The first run was using the ANSYS fluent and the figure below show the result obtained from fluent flow software and compared to the actual model data.

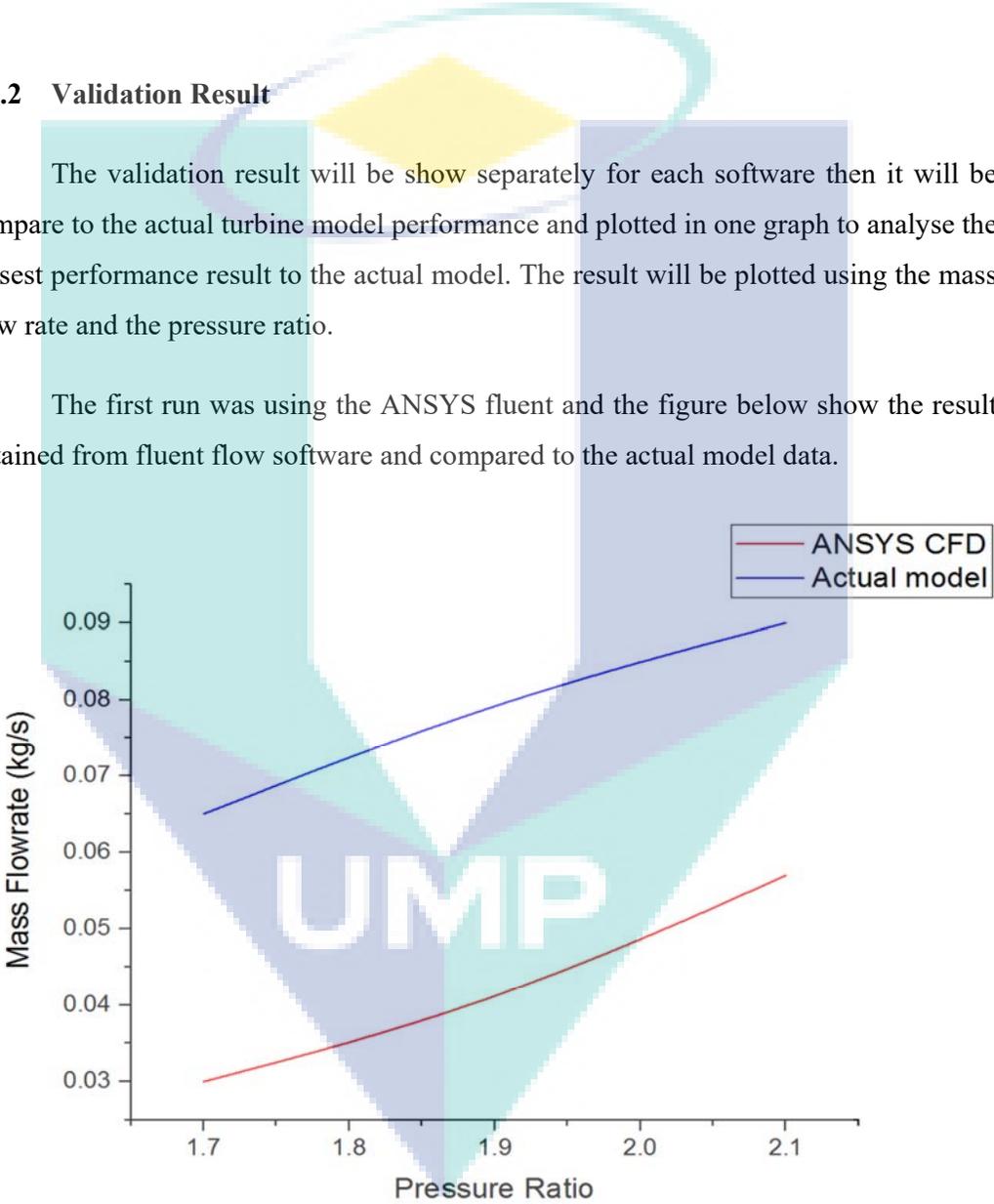


Figure 4.1 ANSYS Fluent Validation

The second run was using the TCFD software the figure below shows the result obtained from the TCFD software and compared to the actual model data.

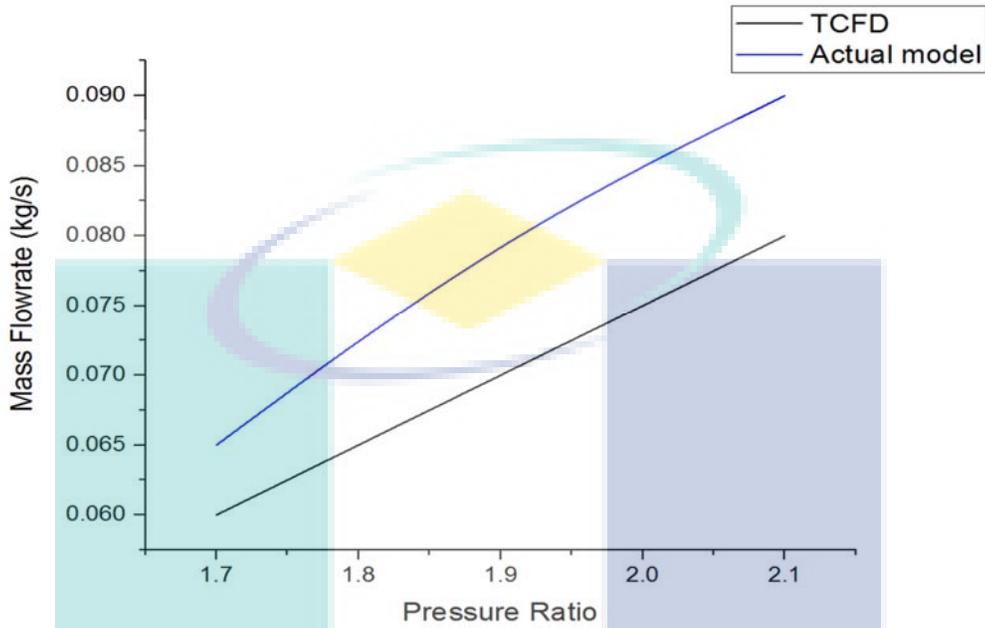


Figure 4.2 TCFD Validation

Lastly both the ANSYS fluent and TCFD validation result plotted in on graph to be compared with the actual model as shown below.

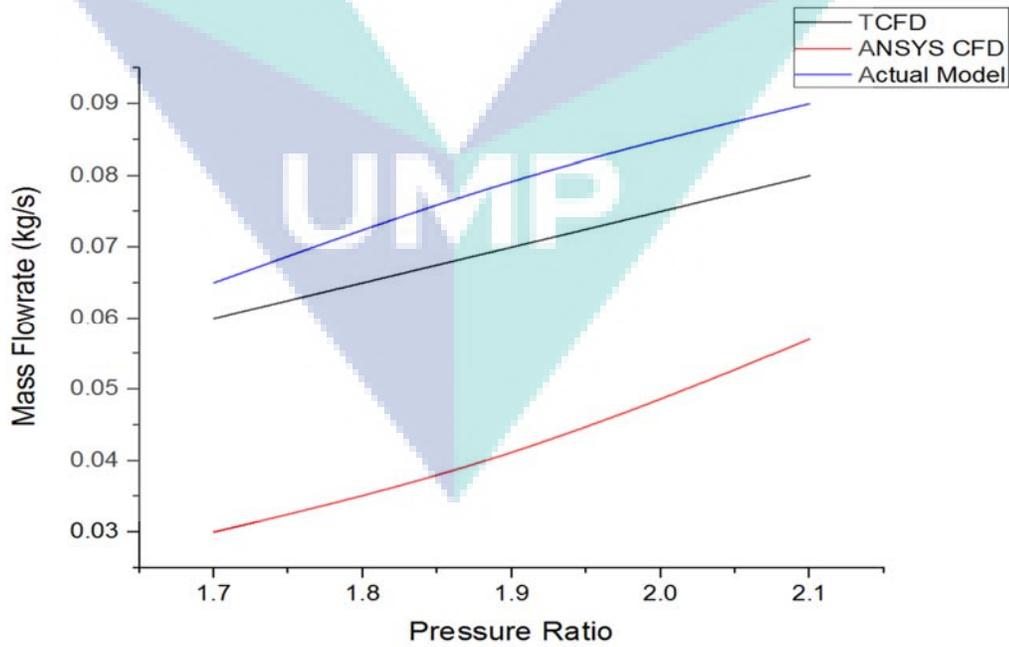


Figure 4.3 Model Validation

4.3 TCFD and ANSYS Fluent Result Performance Comparison

As previously discussed in the validation section. The overall performance of the ANSYS fluent result and the TCFD result when it compared to the actual model performance. It obvious that the TCFD result is better and much closer to the actual model performance. Consequently, the TCFD software has been selected for the radial turbine numerical simulations. However, the ANSYS fluent has be used for a few simulations including the validation simulation.

This section discuss the difference between the two simulation result and illustrate the main features of both software to evaluate and select the most suitable software. Wherefore, the result and the simulation of each software will be compared to summarize the advantages and disadvantages of each software. It is consider as a second validation, but it is only between the two software results.

4.3.1 ANSYS Fluent Result

The ANSYS fluent is a simulation software that use for fluid flow the analysis and investigate the flow characteristics. The ANSYS family has a special workbench for turbomacheniry machine simulations, but it depends on the machine type, for example the pump machine impeller and volute can be design using the Vista CPD option then setup the simulation using the CFX or the throughflow. However, for the turbocharger compressor and turbine only the impellers can be design or import the design using another design software. In the other hand, the simulation step using ANSYS software is quite complicated and takes long time to simulate and collect the result.

In this study a few simulation were conducted using the ANSYS fluent. The simulation was only for 12 blades, one rotational speed and one case of inlet boundary condition. Nevertheless, the result is not much accurate and not closer enough to the actual model. Mass flowrate and pressure ratio is consider as the most layout adopted to evaluate the turbocharger main component performance, so for the fluent analysis the mass flowrate and pressure ratio are selected to evaluate the turbine overall performance. In additional, the efficiency and the total pressure drop of the turbine will be discussed as well.

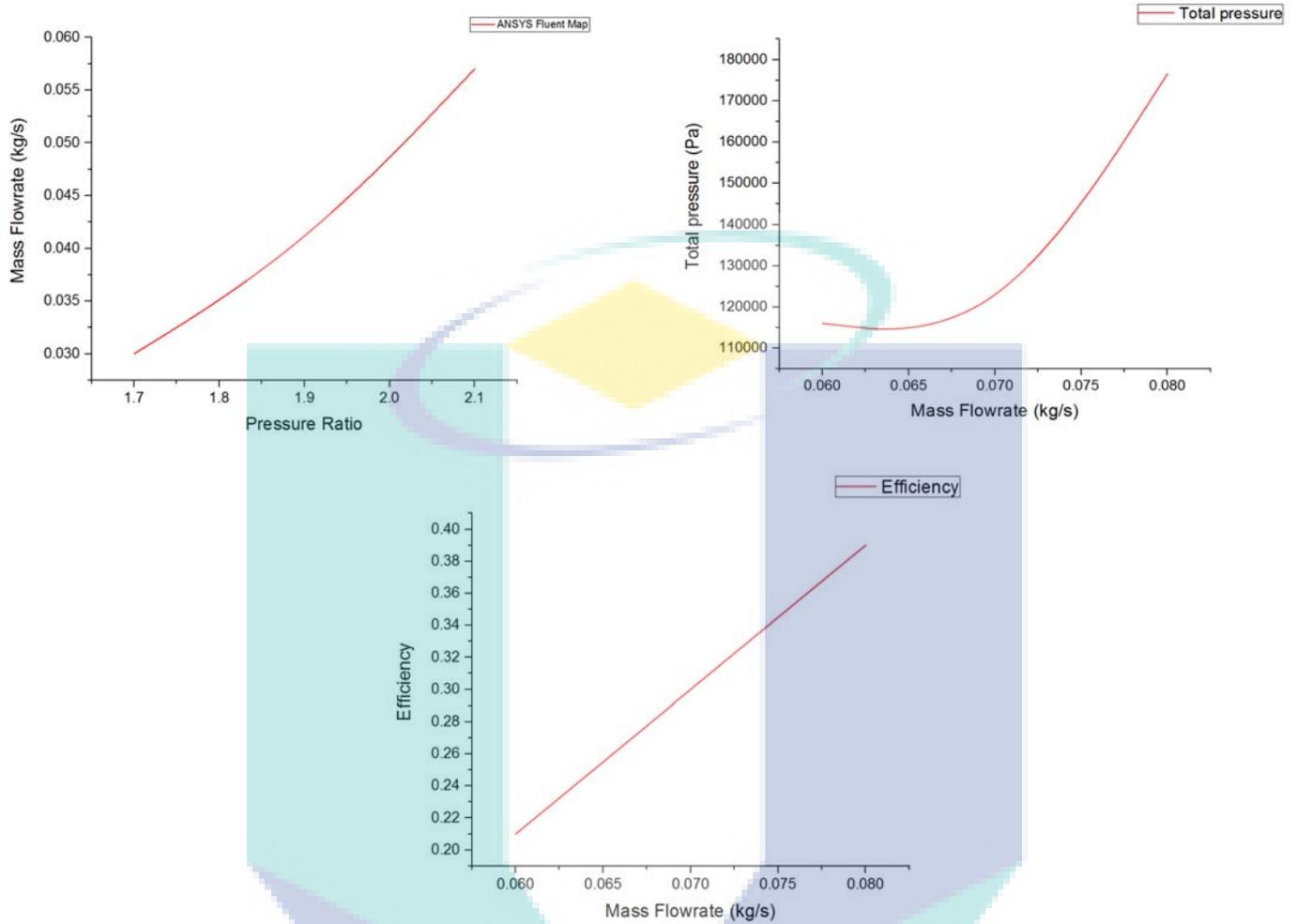


Figure 4.4 ANSYS Fluent Flow Results

4.3.2 TCFD Result

The turbomachinery CFD is a numerical simulation software that covers all the simulation processes such as Pre-processing, Processing and Post-processing for the turbomachinery machines. However the TCFD can run the simulation for various mass flowrate, rotational speed and temperature at the same time. In addition, it also can work for a specific simulation points which can be set before run the simulation. Besides that, TCFD software generate a full report for the simulation result, the report include the graphs, simulation details and images for the components result. These feature are consider as a better advantages for the TCFD to be used for the turbomachinery

simulation. In other hand, using this software can save more time and improve the result quality.

The result report provide a various type of graph results such as torque, efficiency, total pressure and total temperature etc. Conversely, some of them will be shown in this section and the rest will be presented in the following sections in this chapter. The figure below show some of the TCFD results.

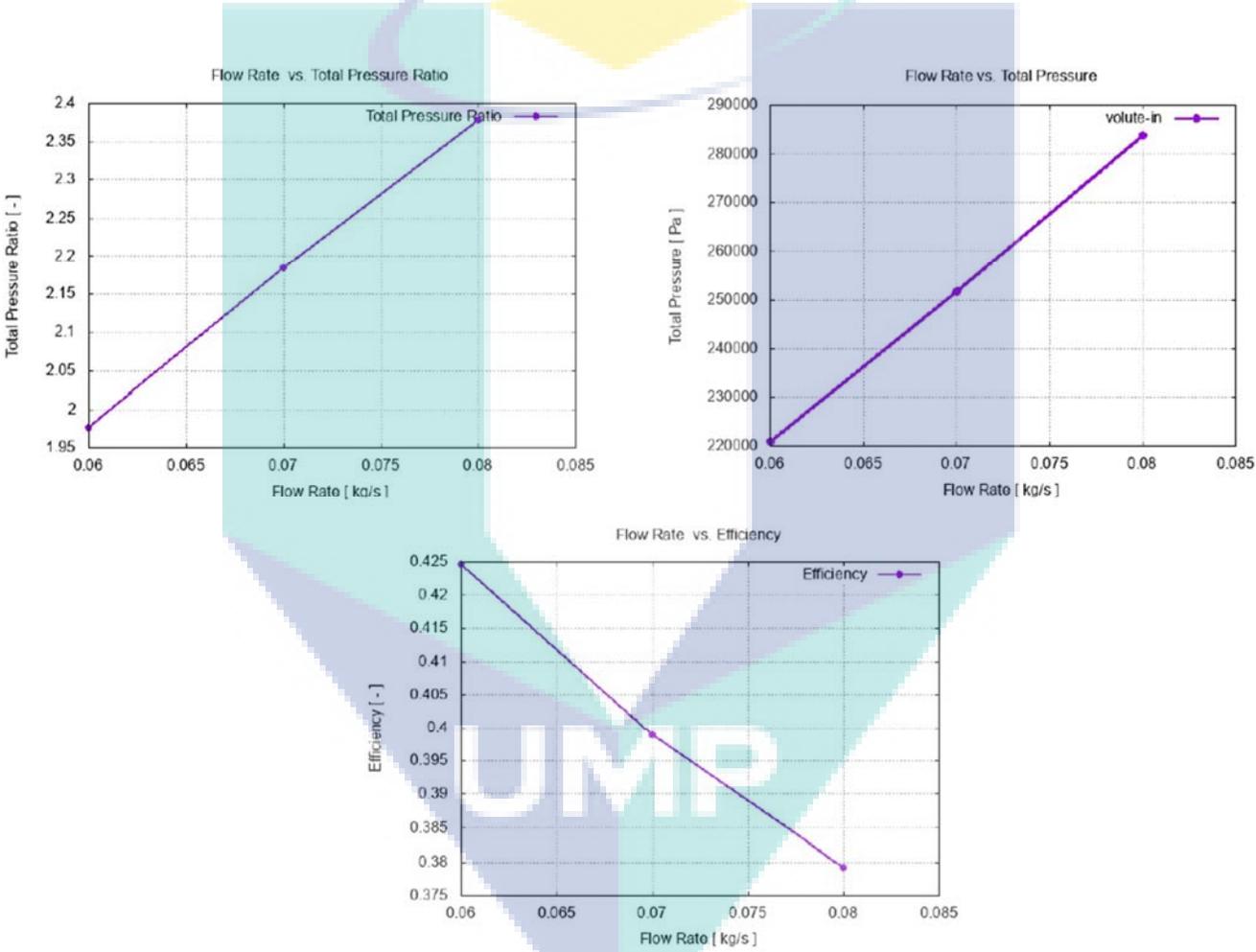


Figure 4.5 TCFD Results

Previously the overall result of the two software were compared to the actual model performance. In this sub-chapter both software result were compared, it can be conclude that the TCFD software is better to be used for the turbocharger turbine.

4.4 Turbine Performance Characteristics

The turbocharger turbine performance has been analysed for various rotational speed and mass flow rate and for two different temperature values 700k and 950k. The rotational speed selected (RPM) for the simulation are 120000, 150000 and 180000. However, the mass flowrate (kg/s) are 0.6, 0.7 and 0.9.

The numerical simulation was performed at 2 processes for 3 points which was set before run the simulation. The numerical convergence was shown in for the residuals verse the iterations which explain the stabilization of the variables as shown in the following figure.

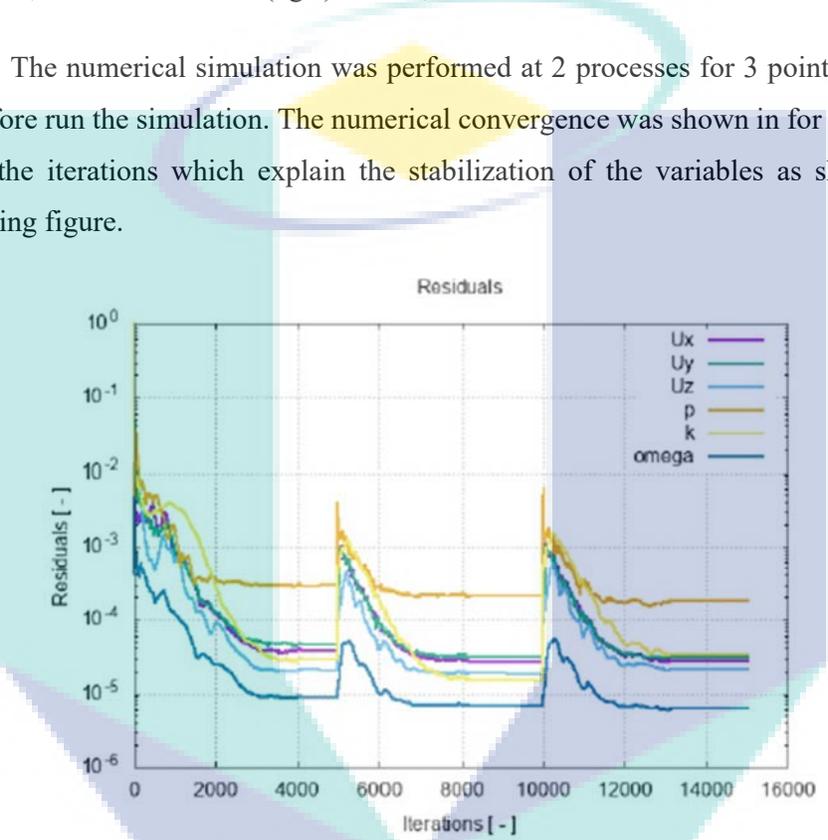


Figure 4.6 Residuals graph

The simulation iteration processes can be shown for all the desired variables. However, the last value for the flow variables in the last iteration the selected are the selected solution for the inlet boundary conditions. The simulation process is depend on the variable iterations to obtain the correct variable value for the solution. The figures below shows some of simulation variable iteration.

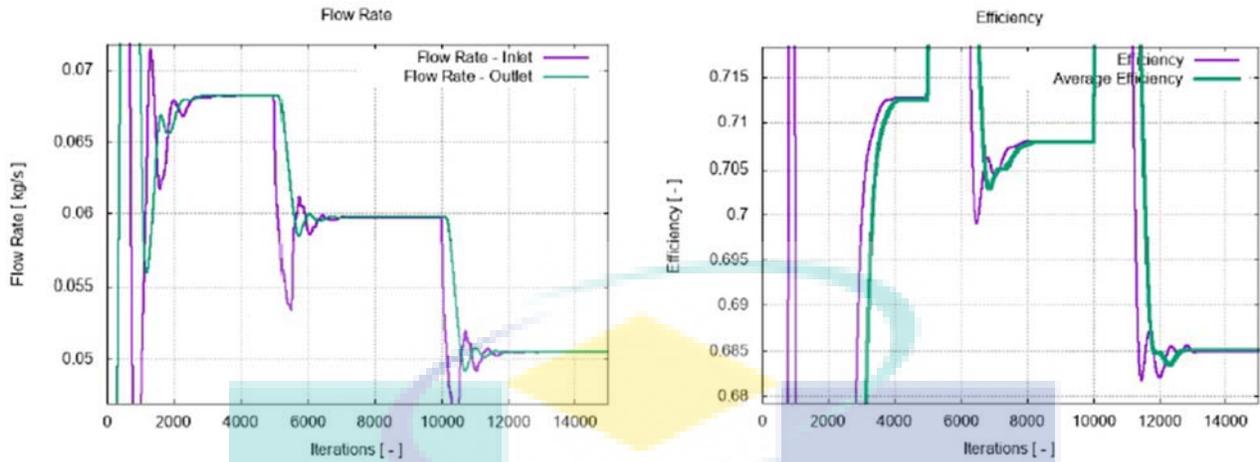


Figure 4.7 variables iteration graphs

The result will be presented for each rotational speed using the two temperature values. Moreover, all the flow variables will be analysed separately then it will combine together for each temperature value in term of pressure ration verse mass flowrate to investigate the overall performance of the turbine. However, the result combined for each temperature will be compared to evaluate and identify the most proper temperature value.

4.4.1 120000RPM Result

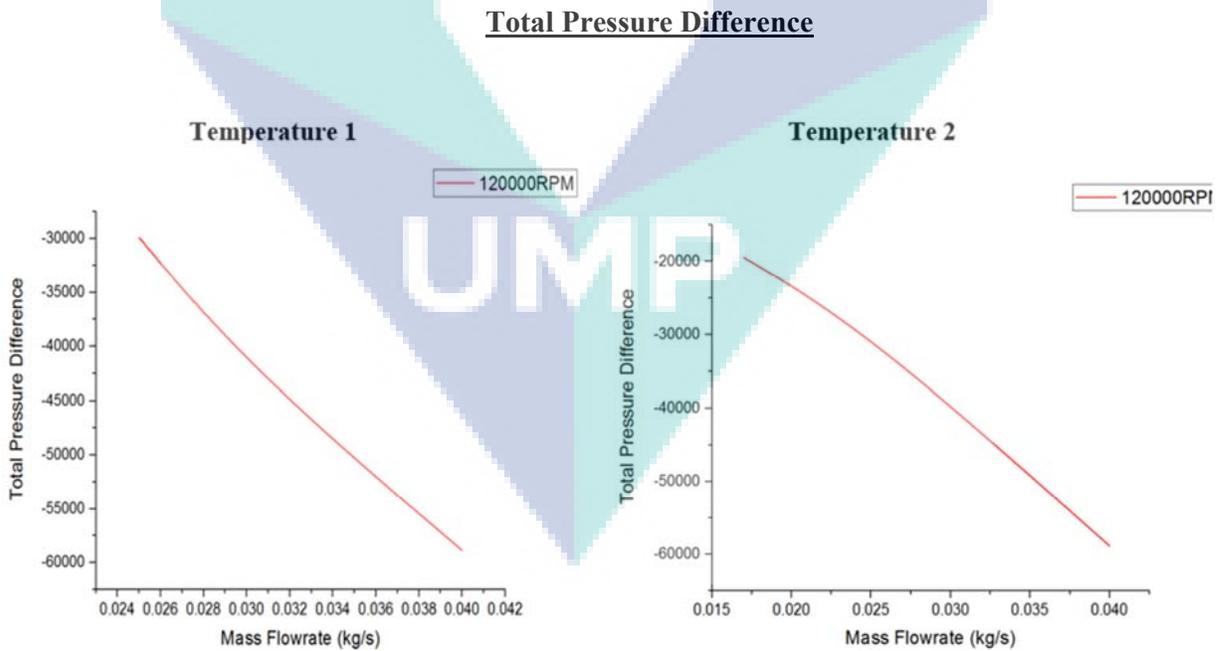


Figure 4.8 Total Pressure Difference Performance for both Temperatures

Velocity

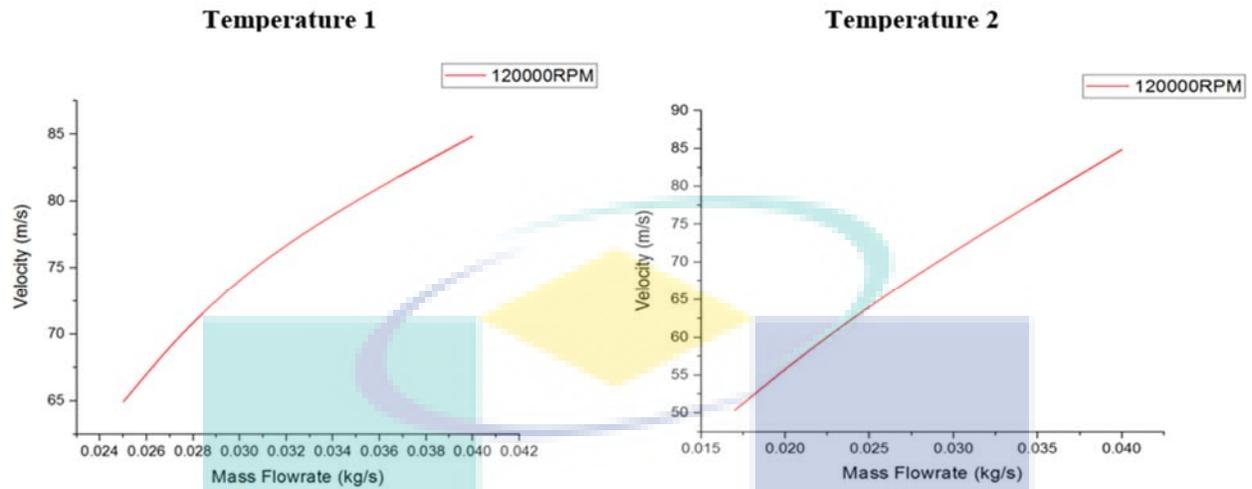


Figure 4.9 Velocity Performance for both Temperatures

The velocity is increasing when the mass flow rate increase, but the total pressure difference is decrease. The result obtained from temperature one are slightly similar to the result obtained from temperature two with a small difference in the values of each variable.

4.4.2 150000RPM Result

Total Pressure Difference

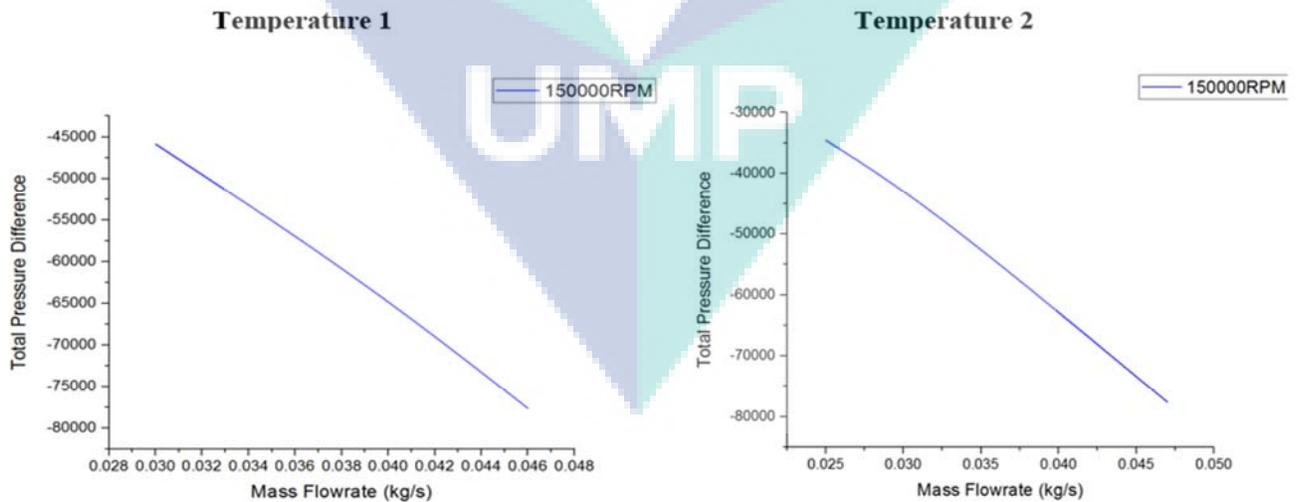


Figure 4.10 Total Pressure Difference Performance for both Temperatures

Velocity

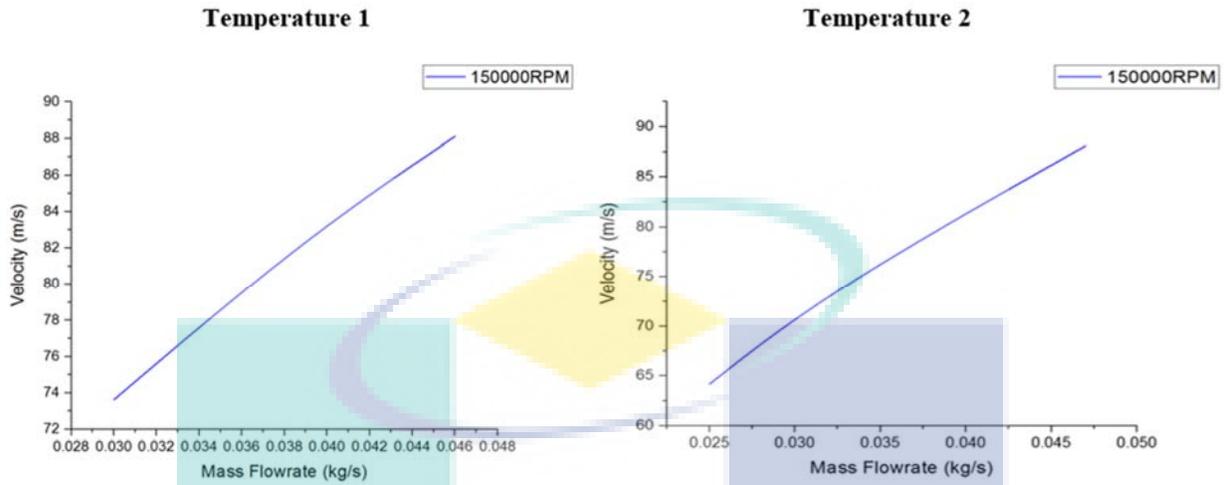


Figure 4.11 Velocity Performance for both Temperatures

It is obvious that the result obtained from using the two temperature are alike, but there are a quite difference in the values. Whatever, the result will be clarified in detail in the following sections in this chapter.

4.4.3 180000RPM Result

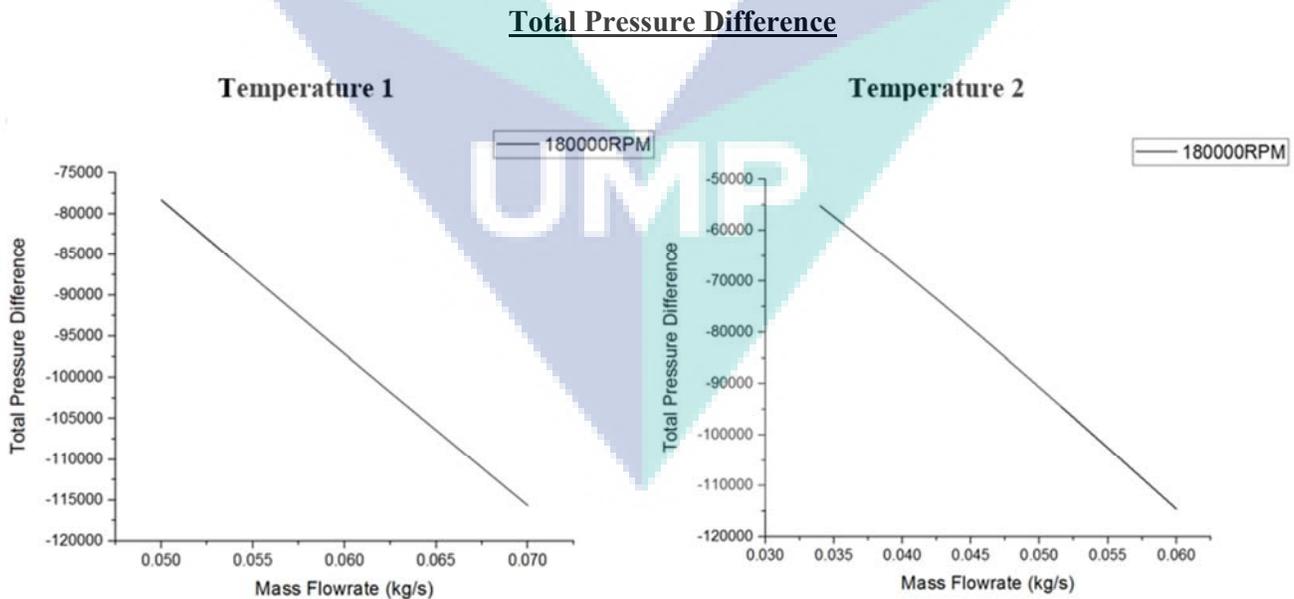


Figure 4.12 Total Pressure Difference Performance for both Temperatures

Velocity

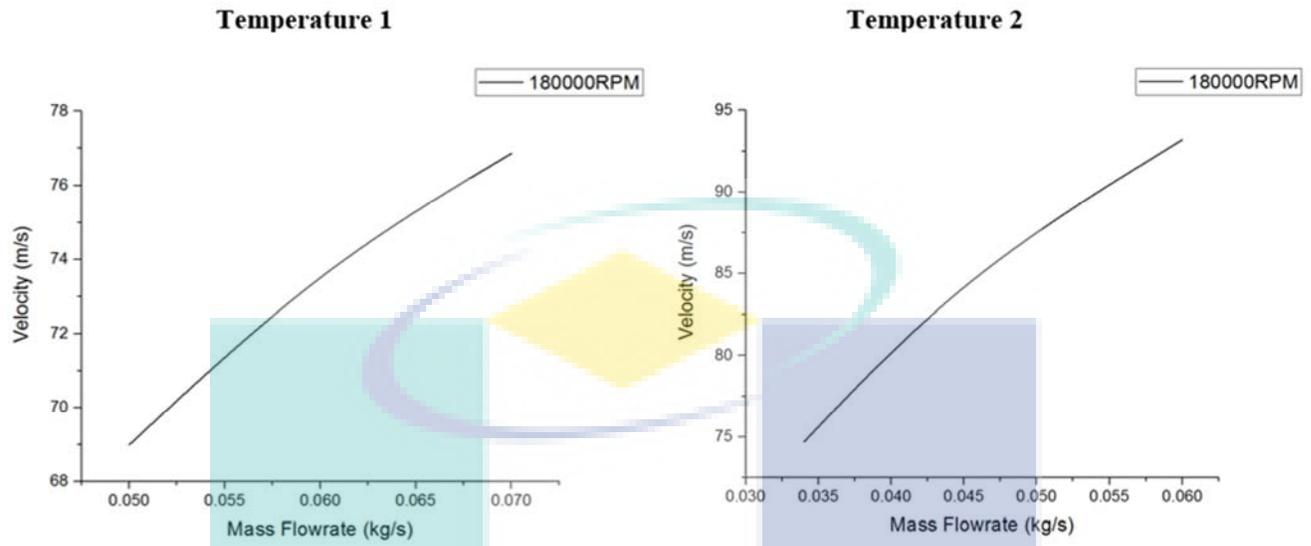


Figure 4.13 Velocity Performance for both Temperatures

As expected, when the mass flowrate increase the velocity increase as well, but the total different pressure is decrease.

4.4.4 Overall performance

As shown in the previous sections. The result for all the rotational speeds using two different temperature were analysed based on the total pressure difference and the flow velocity. Moreover, the total pressure difference is decreasing when the mass flowrate increase, opposite to the velocity which has a proportional relation with the mass flowrate. However, there is slight difference in the values of each variable for all the rotational speeds, taking an example the total pressure difference of the 120000rpm using temperature one is larger than the total pressure difference using temperature two for the same rotational speed and the velocity is larger as well. In additional, the total pressure difference and the velocity values for 150000rpm and 180000rpm using temperature one are greater than the values using temperature two.

Since the mass flowrate and the pressure ratio are the main variables when evaluating the turbocharger turbine overall performance. The overall performance for all the rotation speeds and the two temperature will be identified in term of mass flow rate and pressure ratio.

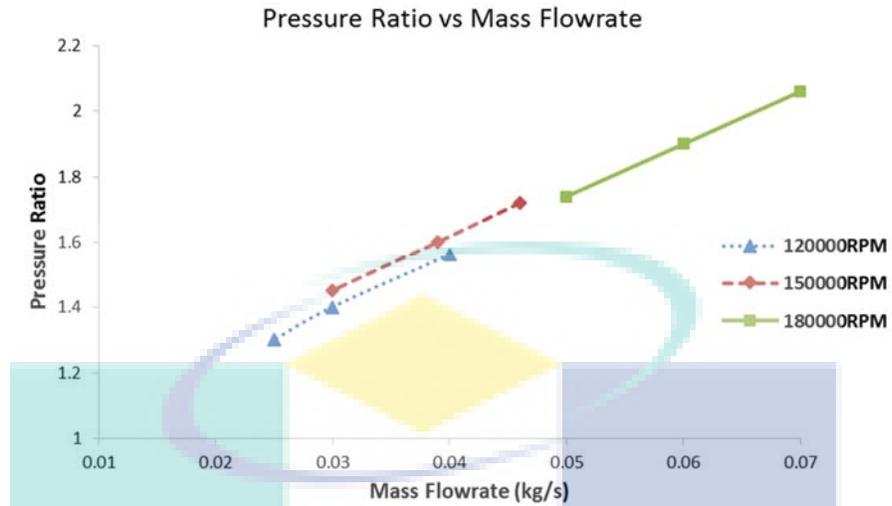


Figure 4.14 Performance All RPMs Using Temperature 1

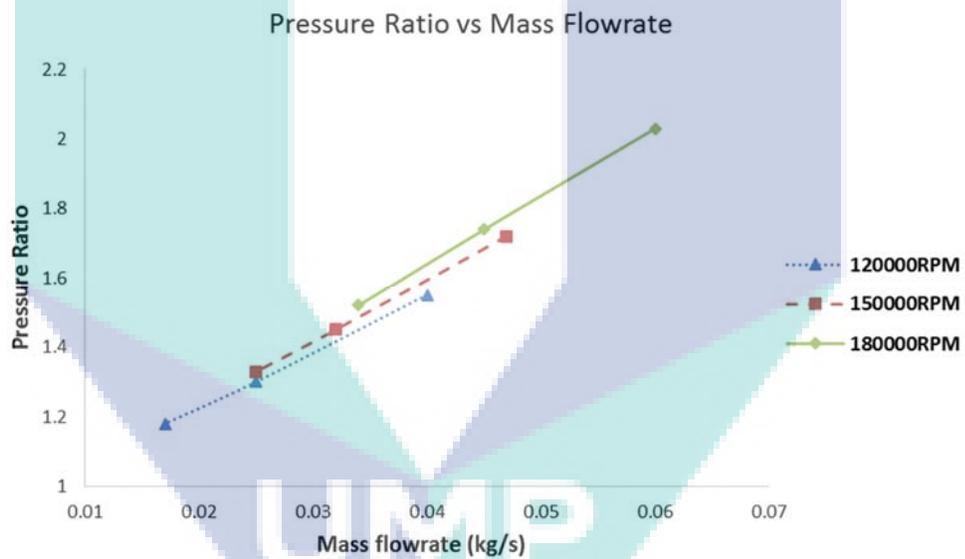


Figure 4.15 Performance All RPMs Using Temperature 2

It obvious that the 180000rpm performance is the greatest among all the rotational speeds for both temperature values. However, to compare the performance of both temperatures and since the 180000rpm performance is the best between all the rotational speeds it has been selected to evaluate the temperature performance based on it.

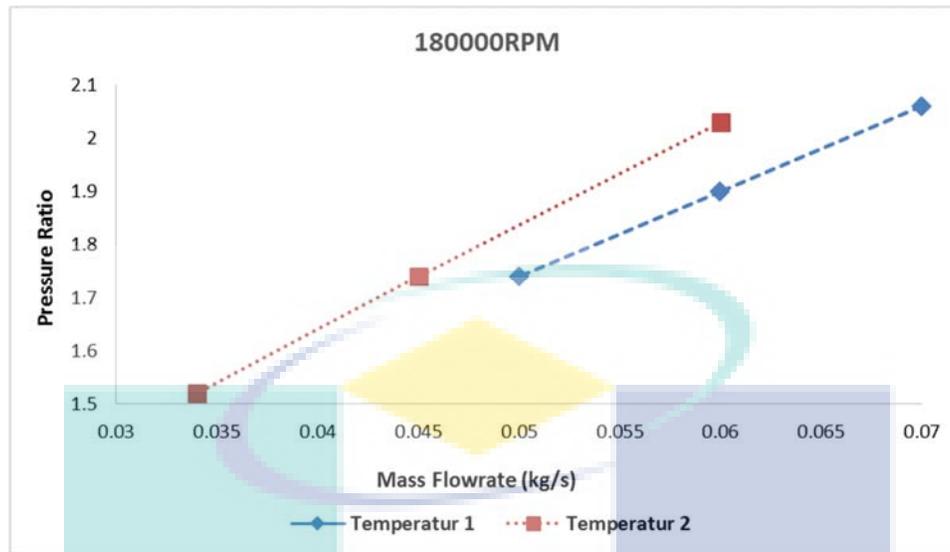


Figure 4.16 180000rpm Performance for both Temperatures

The red line represent the performance of the turbocharger turbine when the rotation speed is 180000rpm and the temperature is 950k, while the blue line represent the performance of the turbocharger turbine performance at the same rotational speed and different temperature which is 700k. Furthermore, in all the simulation points temperature 1 values are greater than those for temperature 2 except in one point which the value of temperature two is larger than that in temperature one. However, the overall performance of the turbocharge turbine is better when using the first temperature. For the turbine flow characteristics pictures refer to **Appendix**

4.5 Turbine Performance Based on the Blades Number

The second main point in this study is to study the effect of the blades number on the turbocharger performance. The GT2056 is the actual model for this study, it has a 12 blades and based on that the blades selected for this project are 9, 12 and 15. However, the geometry design is similar to the previous except the blade number, the inlet boundary conditions and numerical method are also similar. Furthermore, the simulation for all the blades number were conducted using the same rotational speed, the rotational speed has been chosen according to the actual model validation, since the model has 12 blades, so to study the performance for another blades number the rotational speed should be the same, the selected rotational speed is 90000rpm. The performance will be analysed in term of pressure ratio and mass flowrate.

4.5.1 9 Blades Performance

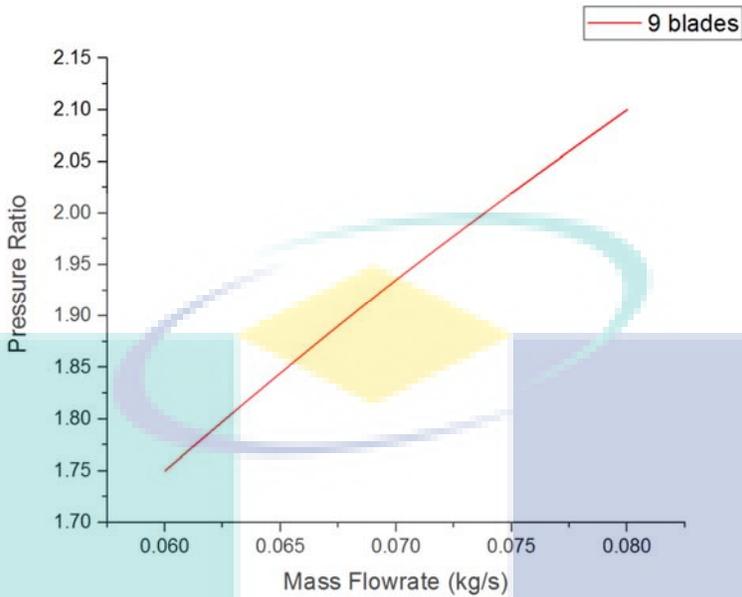


Figure 4.17 9 Blades Performance

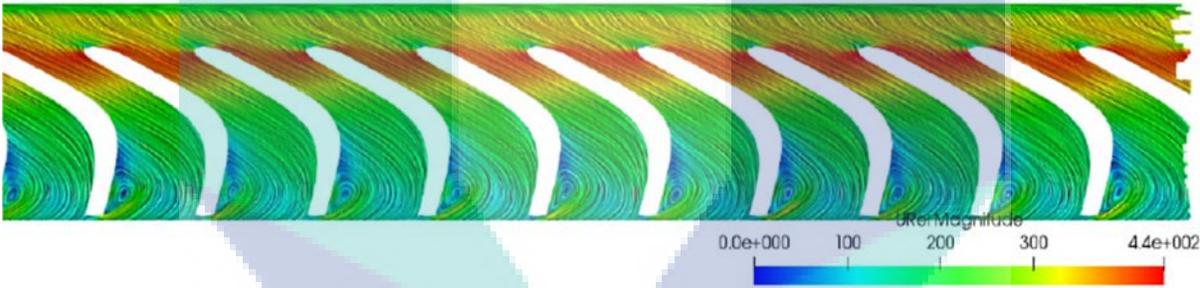


Figure 4.18 9 Blades Velocity Distribution

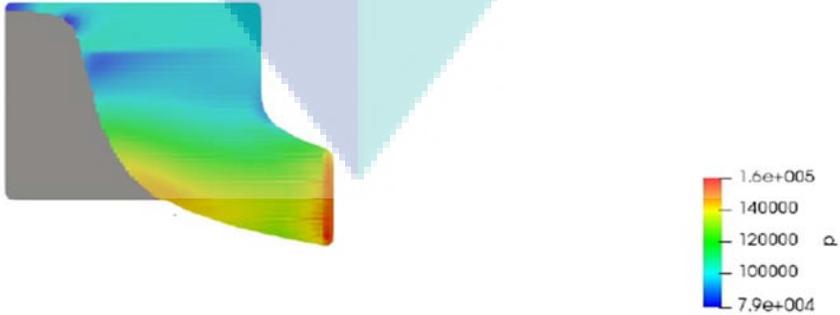


Figure 4.19 9 Blades Pressure Distribution

4.5.2 12 Blades Performance

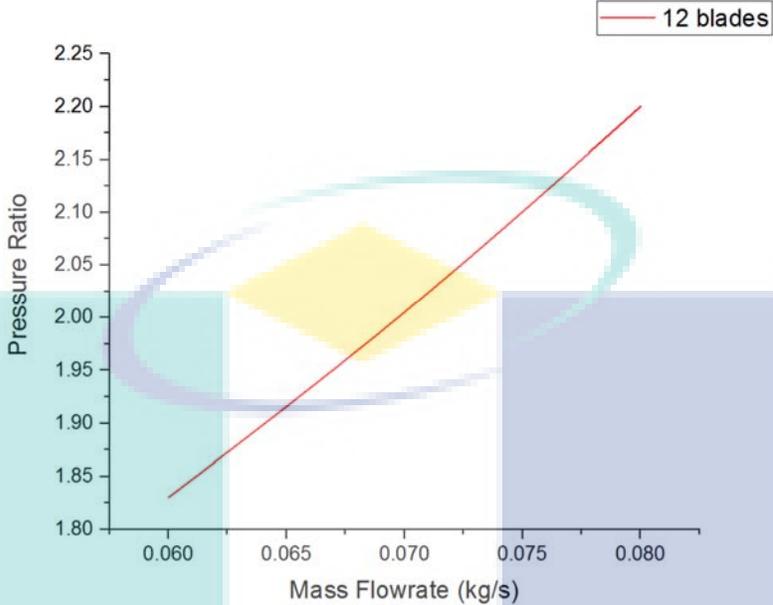


Figure 4.20 12 Blades Performance

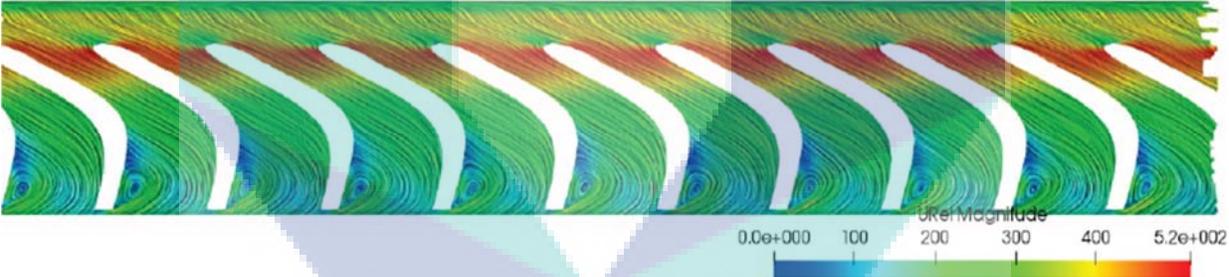


Figure 4.21 12 Blades Velocity Distribution

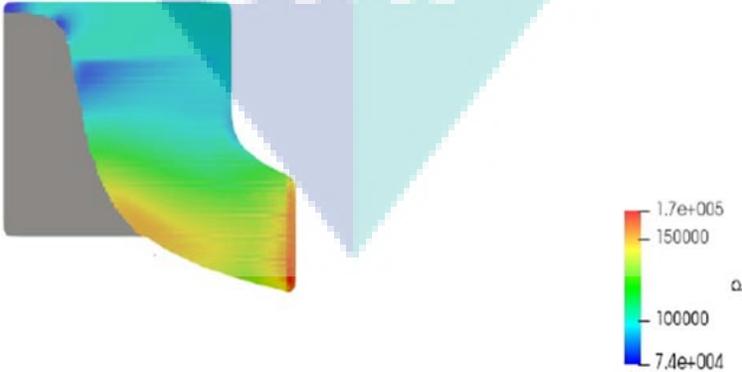


Figure 4.22 12 Blades Pressure Distribution

4.5.3 15 Blades Performance

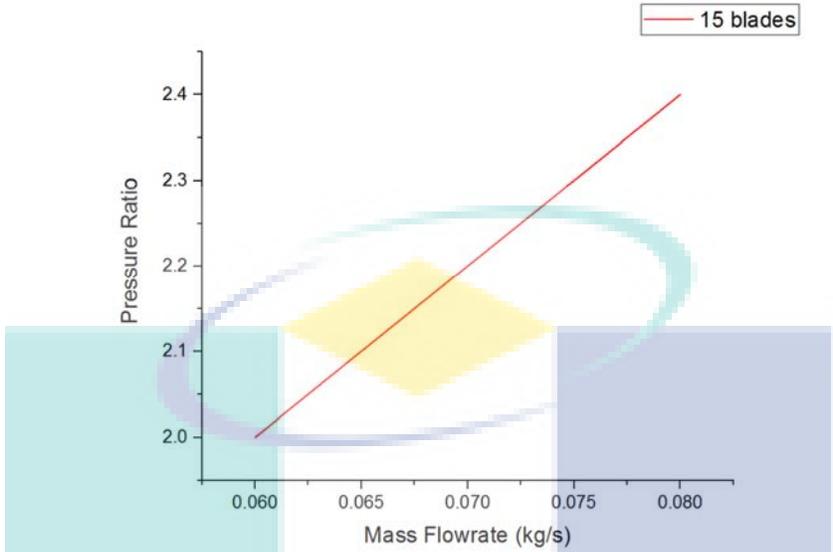


Figure 4.23 15 Blades Performance

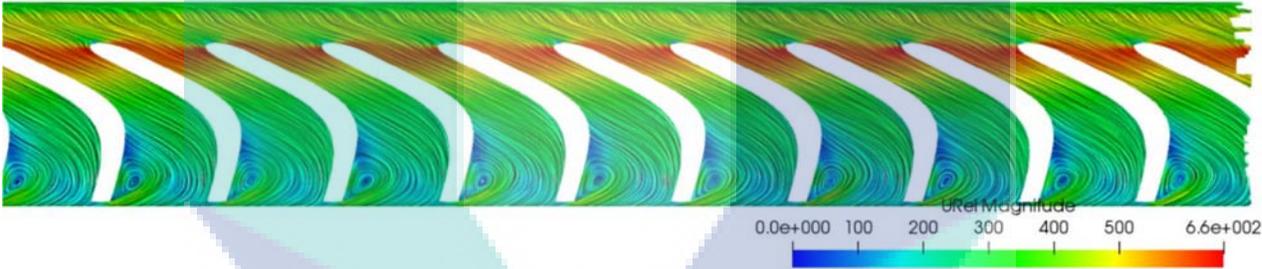


Figure 4.24 15 Blades Velocity Distribution

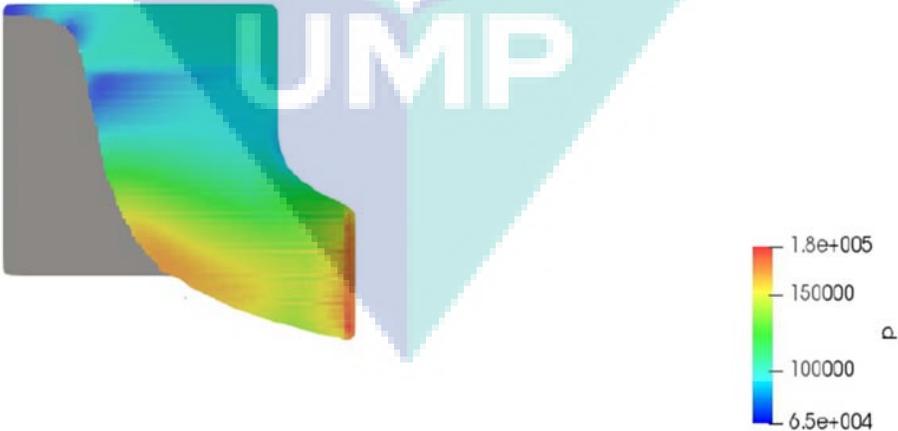


Figure 4.25 15 Blades Pressure Distribution

4.5.4 Blades Overall Performance

As shown in the previous sections. The three number of blades performance were analysed in term of pressure ratio and mass flowrate, it obvious that the pressure ratio is increasing when the mass flowrate increase. Additionally, the velocity and pressure distribution on the blades were presented, the velocity and pressure distribution in the blade tip are higher than the other blades parts. The flow enters the turbine and cross the blade at the blade tip, that is the reason behind the high values for the velocity and pressure distributions. Furthermore, the overall performance of the blades will be combine to identify the most proper number of blades among the three number of blades.

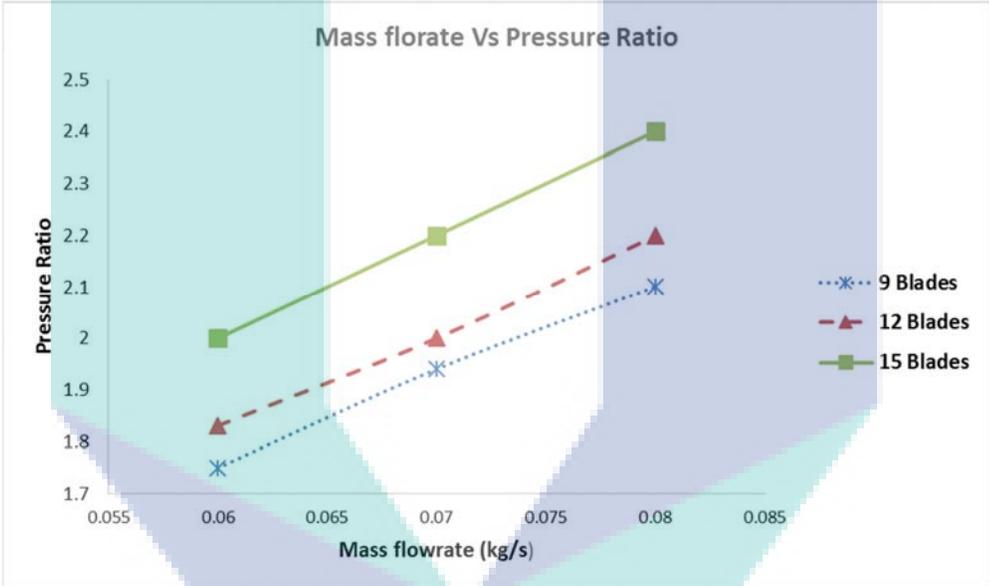


Figure 4.26 Blades overall Performance

As shown in the figure above the overall performance of turbocharger turbine for all the number of blades. Moreover, the turbine performance presented in term of pressure ratio and mass flowrate, it shows that the rotor with 15 blades has the higher performance.

CHAPTER 5

CONCLUSION

5.1 Introduction

The objectives of this thesis are to study and analysis the turbocharger turbine performance for various rotational speeds and mass flowrates and to study the effect of different blades number on the turbocharger turbine performance. The geometry design were performed using CAD and CFTurbo design software taking the GT2056 as a target for this work. Besides that, the numerical simulation for the turbocharger were carried out using ANSYS fluent and TCFD simulations software. The model simulated with a steady state condition, using the GT Power as inlet boundary conditions. The reason behind using two simulation software for the turbine simulation is that the result obtained from the ANSYS fluent is not accurate and not closer to the actual model and the TCFD has been developed for the turbomachinery machines. However, all the objectives of this work has been achieved successfully.

The turbocharger turbine were simulated for many simulation, in order to investigate a better performance for the turbine. The result were presented and shows the turbocharger turbine has a better performance for the 180000rpm for all the mass flowrates. Besides that, the temperature of 700k show a higher performance for the turbine when compared to the second temperature that has been used in this study. Moreover, the effect of number of blades has been studied as well, the result show that the turbine performance is higher when using 15 blades.

It can concluded that when increasing the rotational speed the turbine performance improve and the overall performance is better. Moreover, the most suitable blades for the turbocharge turbine are 15 blades based on the result obtained in this study.

5.2 Recommendation

Numerical simulation on the turbocharger performance has been signed as a major filed that use computational fluid dynamics in their study to analysis the performance of the turbocharger components which are turbine and compressor, in order to investigate the most accurate performance when compared to actual model or experimental data. There are a few impossible recommendations for further work on turbocharger, the recommendations as follow.

1. Using the TCFD simulation software that has been design to simulate the turbomachinery machines such as turbine and compressor and it provide a better and accurate result when compare to other simulation softwares.
2. Further study on turbocharger turbine should be use different gases to evaluate the best performance based on different gases, because only one gas considered in this current study.
3. Further study on turbocharger performance should be include the turbocharger compressor in order to investigate the compressor performance based on the result obtained from the turbocharger turbine.
4. Conduct an experiment of the turbocharger turbine for the same model to compare with the simulation result.

REFERENCES

- [1] J. H. Ferziger and M. Peric, "Computational Methods for Fluid Dynamics," in *Vasa*, 2002, p. 423.
- [2] W. Zuo, "Introduction of Computational Fluid Dynamics," 2005, p. 8.
- [3] C. F. Dynamics, and C. F. Dynamics, "Computational Fluid Dynamics What is Computational Fluid Dynamics (CFD)? ! Beginning of CFD," 2013.
- [4] J. D. Anderson and J. A. Jr, "Governing equations of fluid dynamics," in *Comput. fluid Dyn.*, 2009, pp. 15–51.
- [5] C. Capiluppi, "Three Dimensional CFD Simulation of a Turbocharger Turbine for Motorsport Applications," 2012.
- [6] S. L. Dixon, "Basic Thermodynamics, Fluid Mechanics: Definitions of Efficiency," *Fluid Dyn. Thermodyn. Turbomach.*, 1998.
- [7] B. Beach, P. Napier, B. Robie, and Z. Wilson, "Fsaec Turbo-System Design 2010," 2010.
- [8] N. N. Bayomi and R. M. Abd El-Maksoud, "Two operating modes for turbocharger system," in *Energy Conversion and Management*, 2012, vol. 58, pp. 59–65.
- [9] J. Kech, R. Hegner, and T. Männle, "Turbocharging : Key technology for high-performance engines - MTU," 2011.
- [10] B. AN and T. SHIRAISHI, "Development of Variable Two-stage Turbocharger for Passenger Car Diesel Engines," in *Mitsubishi Heavy Industries Technical Review*, 2010, vol. 47, no. 4, pp. 1–6.
- [11] A. Romagnoli and R. Martinez-Botas, "Heat transfer analysis in a turbocharger turbine: An experimental and computational evaluation," in *Elsevier Journal of Applied Thermal Engineering*, 2012, vol. 38, no. 0, pp. 58–77.
- [12] N. Basavarajappa, S. Educational, and N. Basavarajappa, "Cfd analysis of turbocharger turbine," 2016, no. January.
- [13] J. K. Thoma, D. Shehan, B. Naravage, and J. Melvin, "by Design , Fabrication , and Testing of an Automotive Turbocharger-Based Gas Turbine Engine," 2010.

- [14] M. S. A. I. Vastav, "AUTOMOTIVE DESIGN AND ANALYSIS OF TURBO CHARGER WITH ELEVEN AND TWELVE BLADES," 2015, no. 11, pp. 120–125.
- [15] Elias Mikael Vagn Siggeirsson, "Conceptual design tool for radial turbines," p. 94, 2015.
- [16] E. López, "Study on a Radial Turbine Stage with Inlet Guide Vanes for an ORC Process with an Electrical Output of 3.5 kW," p. 83, 2013.
- [17] L. S. Ali, H. Mohammed, and H. M. Kamel, "the Number of Blade Effects on the Performance of a Mixed Turbine Rotor," 2017, vol. 37, no. 3, pp. 349–360.
- [18] A. Isenstadt, J. German, M. Dorobantu, D. Boggs, and T. Watson, "Downsized , boosted gasoline engines," in *The ICCT Internaltional Council of Clean Transportation*, 2016, no. 21, p. 23.
- [19] M. A. Chelabi, M. K. Hamidou, and M. Hamel, "Effects of cone angle and inlet blade angle on mixed inflow turbine performances," in *Periodica Polytechnica Mechanical Engineering*, 2017, vol. 61, no. 3, pp. 225–233.
- [20] M. Yang, R. Martinez-Botas, S. Rajoo, T. Yokoyama, and S. Ibaraki, "An investigation of volute cross-sectional shape on turbocharger turbine under pulsating conditions in internal combustion engine," in *Energy Conversion and Management*, 2015, vol. 105, pp. 167–177.
- [21] A. Kusztelan, Y. F. Yao, D. R. Marchant, and Y. Wang, "A Review of Novel Turbocharger Concepts for Enhancements in Energy Efficiency," in *International Journal of Thermal and Environmental Engineering*, 2010, vol. 2, no. 2, pp. 75–82.
- [22] J. Galindo, P. Fajardo, R. Navarro, and L. M. García-Cuevas, "Characterization of a radial turbocharger turbine in pulsating flow by means of CFD and its application to engine modeling," in *Applied Energy*, 2013, vol. 103, pp. 116–127.
- [23] A. J. Feneley, A. Pesiridis, and A. M. Andwari, "Variable Geometry Turbocharger Technologies for Exhaust Energy Recovery and Boosting-A Review," in *Renewable and Sustainable Energy Reviews*, 2017, vol. 71, no. December 2016, pp. 959–975.
- [24] D. de Carvalho and J. Tomita, "3D Turbulent Flow Analysis in a Turbocharger Radial Turbine Operating At Design-Point," no. Cobem, pp. 9688–9698, 2013.