# ANALYSIS OF THE CENTRIFUGAL BLOWER PERFORMANCE WITH DIFFERENT DIMENSION CONFIGURATION BY USING ANSYS SOFTWARE

MOHAMAD ASYRAF BIN NAIM

اونيونرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# ANALYSIS OF THE CENTRIFUGAL BLOWER PERFORMANCE WITH DIFFERENT DIMENSION CONFIGURATION BY USING ANSYS SOFTWARE

**MOHAMAD ASYRAF BIN NAIM**

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

**Faculty of Mechanical Engineering**

**UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

**JUNE 2017**

**DECLARATION**

I declare that this project report entitled "Analysis of The Centrifugal Blower Performance With Different Parameter Configuration by Using ANSYS Software" is the result of my own work except as cited in the references

Signature : ..........................................

Name : Mohamad Asyraf bin Naim

Date : .......................................

اونيۏرسيتي تيکنيکل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

**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).

Signature :. ..................................

Name of Supervisor: Prof. Madya Dr. Ir Abdul Talib bin Din

Date : ..................................

اونيۆرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# DEDICATION

To my beloved mother and father

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# ABSTRACT

The centrifugal blower is a device for move the air of other gases in difference condition. The rotating impeller in the centrifugal blower function as to increase the speed and volume of an air stream. This device used kinetic energy system to increase the volume of the air or gas stream. Dimension of the centrifugal blower also play a role for the performances of the process. The project was carried out to investigate the research problem centrifugal blower operation process due to that now is difficult to study. Usually to done the experiment of centrifugal blower performances need to fabricate and get the data. However, this method is not consistent because the accuracy of the data and the results produce are not sure. Therefore, this project was conducted to investigate or carry out experiments on the centrifugal blower performance analysis and simulation using ANSYS software with difference dimension. Methodology to do this project are by using SolidWorks Software for design the centrifugal blower and ANSYS Software for the simulation. Findings of this project is the volume air flow generated from each type dimension of centrifugal blower.

اونيۏرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# ABSTRAK

*Kipas empar merupakan alat yang beroperasi untuk membawa angin atau gas dalam kondisi yang berbeza. Putaran pendesak didalam kipas empar berfungsi untuk meningkatkan kelajuan aliran udara yang dilalui. Sistem untuk proses kipas empar ialah tenaga kinetik iaitu dengan meningkatkan ketumpatan udara atau aliran gas. Saiz dimensi kipas empar juga memain peranan dari segi aliran udara yang dihasilkan. Projek ini dijalankan untuk menyiasat masalah kajian operasi proses aliran kipas empar yang kini sukar untuk dikaji. Kebiasaannya untuk mengkaji prestasi kipas empar, perlu membuat model dan melakukan eksperimen. Namun dengan kaedah ini, kejituan data dan keputusan yang dihasilkan tidak pasti dan proses melakukakn juga akan melibatkan kos yang tinggi. Oleh itu, projek ini dilaksanakan untuk menyiasat atau melaksanakan eksperimen terhadap prestasi kipas empar dengan menggunakan analisis dan simulasi perisian ANSYS berasakan dimensi yang berbeza. Kaedah untuk melaksanakan projek ini ialah dengan menggunakan perisian Solidworks untuk rekabentuk kipas empar dan perisian ANSYS untuk simulasi. Penemuan penting dalam projek ini ialah ketumpatan aliran udara yang dihasilkan berasaskan jenis dimensi kipas empar yang berbeza.*

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# ACKNOWLEDGEMENT

I would like to express my deepest appreciation to my supervisor Prof. Dr. Ir Abdul Talib bin Din for giving me this opportunity to do final year project with him. He never hesitated to give me advice and guidance whenever I confronted problems. I am thankful for his patience and advice while leading me in this project.

Secondly, I would like to thanks to my family who are always give a motivation, support, patience and encouragement to me finish this project. Finally, I would like to thank my course mates for giving their help and teach about CFD simulation

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## CONTENT



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## LIST OF FIGURES



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

اونيۈرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

اونيۏرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## LIST OF TABLES



اونيؤرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## LIST OF ABBEREVATIONS

| | |
|---|---|
| BC | Backward-curved |
| CFD | Computer Fluid Dynamics |
| FC | Forward-curved |
| FD | Forced Draft |
| HVAC | Heating, Ventilating and Air Conditioning |
| ID | Induced Draft |
| PA | Primary air |
| PE | Potential Energy |

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## LIST OF SYMBOLS

| | |
|---|---|
| $V$ | Velocity of the air in the duct |
| $D_1$ | Diameter of middle impeller |
| $D_2$ | Diameter of impeller |
| $F$ | Force |
| $q$ | Quality |
| $\Delta p$ | Average pressure |
| $\gamma$ | Density of air |
| $C_p$ | Coefficient |

UTeM
اونيورسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

# CHAPTER 1

# INTRODUCTION

## 1.1 Background

A centrifugal blower is a device for move the air or other gases in difference condition. The rotating impeller function as to increase the speed and volume of an air stream. Kinetic energy system is used in centrifugal blower impeller processes to increase the volume of the air or gas stream. Centrifugal blower displace air radially, the airflow direction is changing by 90. They are study, quiet, reliable and capable of operating over a wide range of conditions.

This mechanical device will completely functional when the impeller is in good condition. The pressure and flow of a fluid will increase after an impeller rotate. An impeller is a rotating component of a centrifugal blower, usually made of steel, bronze, brass, and aluminium. An impeller transfers energy from the motor that drives the fan or blower to the air or gas being transfer by accelerating the air or gas outwards from the centre of rotation.

There are a few basic types of impellers or fin ribs used in industrial. An example shrouded radial blade, open radial blade, open paddle wheel, backward inclined, backward curved, air foil blade, forward curved multi-vane and backward curved radial. Each type of impeller has their own characteristic based on the design of impeller. An example, backward inclined. A flat blade design is available in blower wheels. These wheels' types are commonly used for general ventilation, forced cooling at higher pressure and in dust collection system where the fan is on the clean air side of the dust collector.

The centrifugal blower is one of the most mechanical device widely used fans. Nowadays, majority HVAC industry commonly used centrifugal blower. This mechanical device is simpler in construction and quite cheaper than axial fans. Industry are used in transporting gas and in ventilation system for buildings. They are also well suited for industrial processes and air pollution control systems. The centrifugal blower produces by the centrifugal force generated in a rotating column of air producing potential energy (PE) and by the rotational (tangential) velocity imparted to the air as it leaves the tip of the blades producing kinetic energy.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 1.2 Problem Statement

In centrifugal blower operation flow process is highly complex. It is have to design and develop the sketching or dimension of impeller and casing. Also, determine to reduce the flow losses significantly. Nowadays, in the centrifugal blower experiment it is difficult to obtain the data of characterization of centrifugal blower performance for different impeller configuration. In this project will examine a simple method for obtaining data for these problems. Next, small factories cannot afford to buy equipment to conduct experiment on the search for centrifugal blower performance. In this project, also will reveal the simple method to carry out experiment for small industries.

## 1.2 Objective

The objectives of this project are as follows:

1. To find the difference process of centrifugal blower between difference dimension.
2. To get the skill of using ANSYS software to conduct simulation of Centrifugal Blower.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 1.3 Scope of Project

The scopes of this project are:

1. The study or theory are presented in this report. The study is about the performances of centrifugal blower for different impeller.

2. Blower operating variables for performances enhancement were analysed using Computational Fluid Dynamics (CFD) and ANSYS.

## 1.4 General Methodology

The method that need to be carried out to achieve the objectives in this project are listed below.

### 1.4.1 Literature review

Journals, articles or any materials regarding the project will be reviewed and analyze the experiment conduct.

### 1.4.2 Study

Study about the centrifugal blower performance for different impeller in theory the relationship of impeller design and the blower performance

### 1.4.3 Experiment

Run the experiment to find the characterization of centrifugal blower performance for different impeller.

### 1.4.4 Simulation

Centrifugal blower operating variables for performances for different impeller enhancement were analyzed using SolidWork Simulation or Computational Fluid Dynamics (CFD) or ANSYS software.

### 1.4.5 Report writing

A report on this study will be written at the end of the project.

The methodology of this study is summarized in the flow chart as shown on Figure 1.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

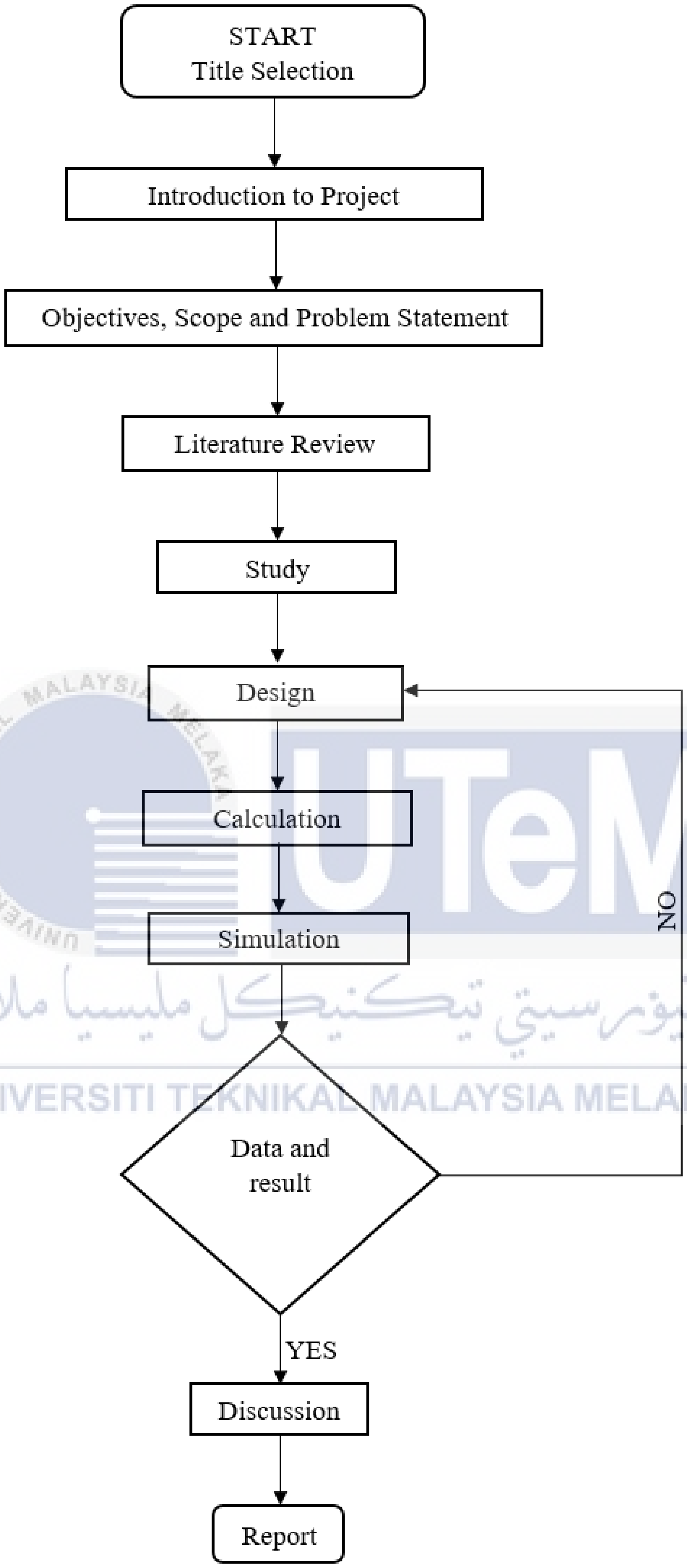



Figure 1: Flowchart

# CHAPTER 2

# LITERATURE REVIEW

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 2.0 Introduction

This chapter reviews the researches that related to the study of characterization of centrifugal blower performance for different dimension. The literatures reseach mainly focus on the performance of centrifugal blower and simulation using ANSYS software.

## 2.1 Centrifugal Blower

### 2.1.1 Introduction

In industry, they widely used different application of centrifugal blower which are proficient of as long as restrained to rise high-pressure and flow rates. There are two main part in centrifugal blower, which are the casing and the impeller. Design dimension of impeller one of the way to improve performance of centrifugal blower.

Energy efficiency of centrifugal blower will increase just by changing some geometric characteristics of the centrifugal blower impeller. (Karthik V and Rajeshkannah T, 2014). To keep manufacturing processes working are the aim for the system are used. Most of industry used this mechanical device that need an air flow for ventilation and industrial processes. The systems consist of an electric motor, impeller or fan, a drive system. Type of material handled, pressure, volume flow rate, space limitations and efficiency are commonly characteristic of fan and blower selection. Efficiency of fan depends on design and types. Two general types fans are centrifugal flow and axial flow. Radial forward curved and backward curved are the major types of centrifugal impeller. Most in industry, radial fans have used because of the high static pressure which is above to 1400 mm WC and ability to handle heavily contaminated air. Radial fan is well suitable for higher temperatures and medium blade tip speed because of the simple design. While, forward-curved fans are used in clean environment and operate at lower temperature but backward-inclined fans are used in clean environment and operate at lower temperature but backward-inclined fans are more efficient. This is because of the peak power consumption and power demand drops of well within their useable airflow range. (Dinesh *et al*., 2016)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

### 2.1.2 Geometrical model of centrifugal blower

Based on the figure 2, shows the projection view. The figure shows the two main components which is the external fan volute, and the fan ribs. The figure also shows the direction of the air flows in the external fan volute. The rotating fan should produce enough energy to supply the air inwards, so that to pump the air from outside (atmospheric region) into the connecting part between the fan and burner.

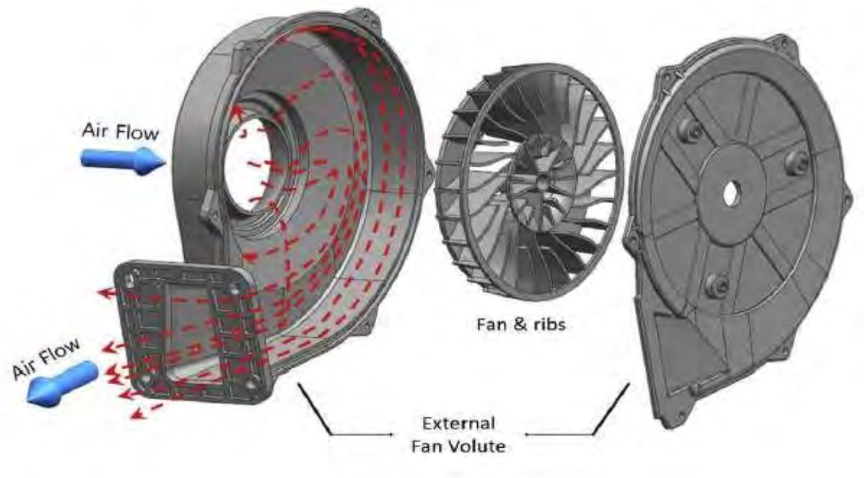



Figure 2: Projection view of the centrifugal blower (Tae Lee et al., 2016)

Two different design of the external volute corresponding. Case 1 (33 degree) and case 2 (22 degree) are shown in the figure 3. The design can effect of the external volute on the fan performances. Referring to figure 3, both cases were fully characterized by a reasonable number of mesh shape cells (approximately $6 \times 10^5$) using an unstructured mesh shape that allowed adjustments of the complicated surface of the centrifugal fan.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

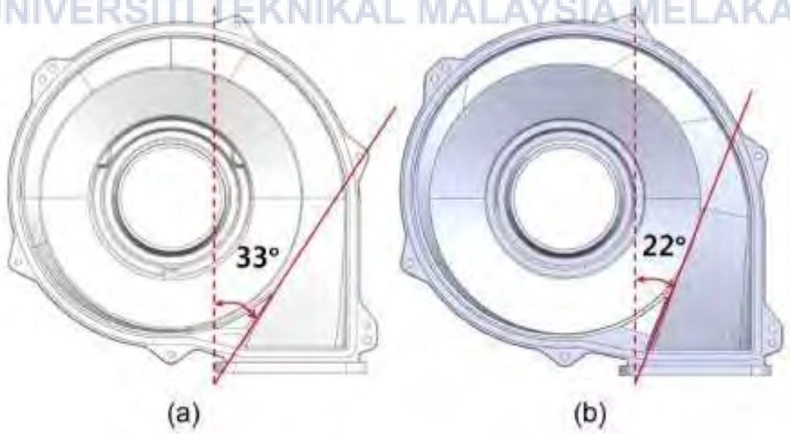



Figure 3: Two volute shapes with different tounge(Tae Lee et al., 2016)

The journal investigated to optimize the design of the fan ribs that have a substantial effect on the performances, focused on the shape and number of fan ribs. The four typical shapes of fan ribs used and the figure shown in figure 4. The typical shapes are the S-type, the straight, the FC and the BC fan ribs. The S-type fan ribs were used to assess the effect of the streamlined shape as well as the external volutes on the fan performances. The straight and backward-curved fan ribs type aim to guide the flow in a proper direction so that maximum output is obtained.

Also, in this journal investigate the effect of the number of fan ribs, the shapes. The conceptual view of the fan ribs with different numbers and with the same shape as that case 4 are shown in figure 5. The design used 10, 12, 14 and 16 ribs. (Tae Lee *et al.*, 2016)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

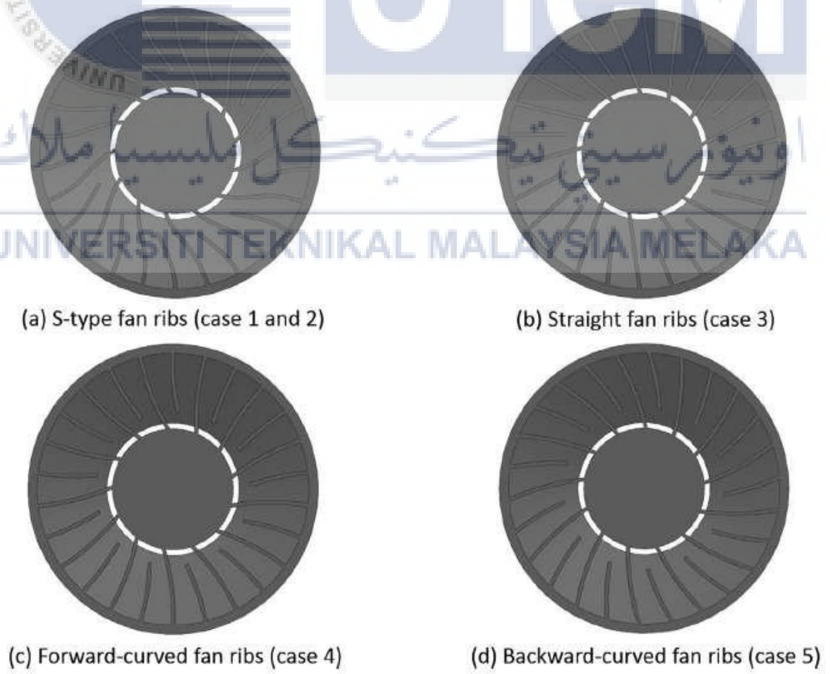



Figure 4: Conceptual view of a few types of ribs shapes (Tae Lee et al., 2016)

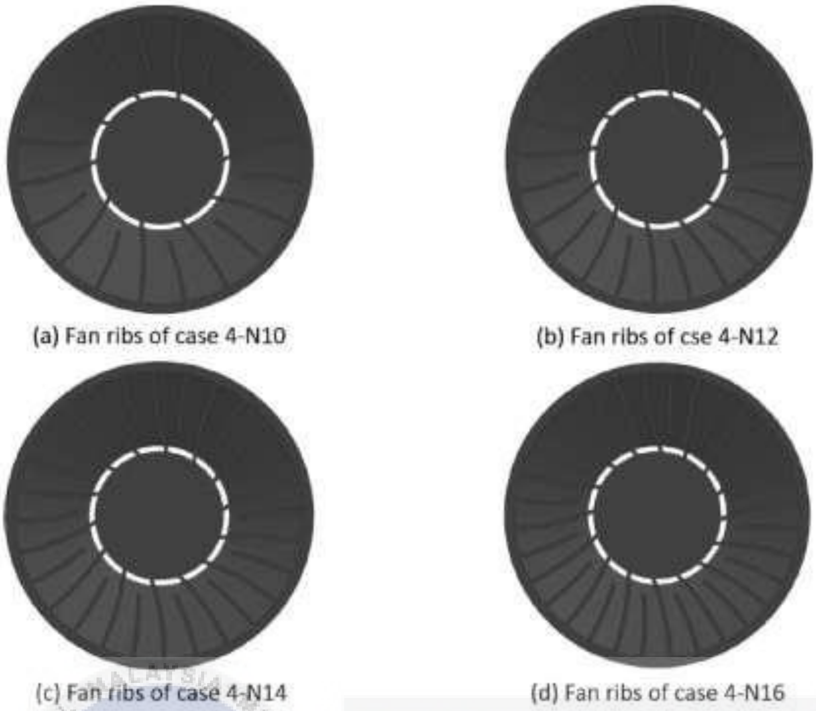



Figure 5: Conceptual view of the fan (Tae Lee et al., 2016)

### 2.1.3 Centrifugal blower analysis

The G4-73 and Y4-73 type centrifugal fans are special designed for middle sized coal- fired power plant as FD, ID and PA. Mine ventilation and usual ventilation usually used G4-73 type fan. The fan impeller consists of 12 backward aero foil shape blades. The impeller are welded between arc cone shape fore plate and flat back plate with the outlet angle of the blade is 45 degree. 800 mm is the original impeller outlet diameter investigated while the 100 mm is the gap between the impeller and volute tongue. The volute is the shape of logarithmic law. An electric motor run with the constant speed of 1450 rpm to rotate the fan. Figure 6 show the schematic diagram of the fan structure. (Chunxi *et al.*, 2011)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

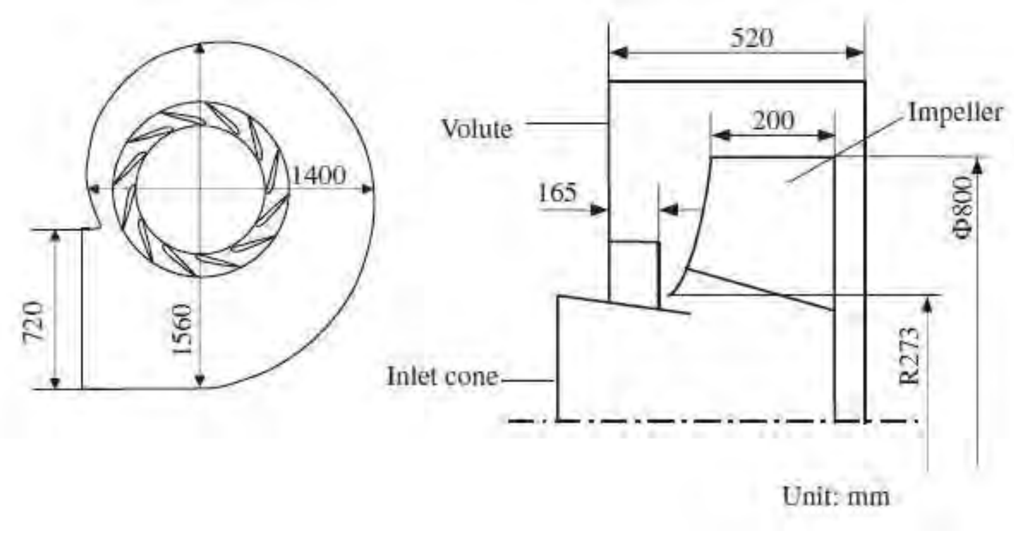



Figure 6: Schematic diagram of the fan structure (Chunxi et al., 2011)

### 2.1.4 The performances of centrifugal blower

The investigation about the type centrifugal fan performances by enlarger the impeller in unchanged volute on G4-73. Comparisons are conducted between the fan with original impeller and two larger impellers. The comparisons are increasing in impeller outlet diameter of 5% and 10% respectively, for numerical and experimental investigations. The internal characteristics are obtained by the numerical simulation, which indicate that there is more volute loss in fan with larger impeller. Experimental results show that the flow rate, total pressure rise, shaft power and sound pressure level have increased. Meanwhile the fan operates with larger impeller, the efficiency has decreased. Variation equations on the performances of the operation points for the fan with enlarged impellers are suggested. The performances of the enlarged fan impeller with less error for higher flow rate can be predict based on the comparison between experimental results and the trimming laws. The reduced impeller-volute gap make the higher noise level with larger impeller fan in noise frequency analysis. Figure 7 shows the comparison of blade after extended. (Chunxi *et al.*, 2011). To change the performances of a fan is by tipping or trimming an impeller. Impeller tipping is to increase the impeller's effective outlet diameter through the addition of blade tip extensions. This method is to increased performances of fan plus it is economical method. (Chunxi *et al.*, 2011).

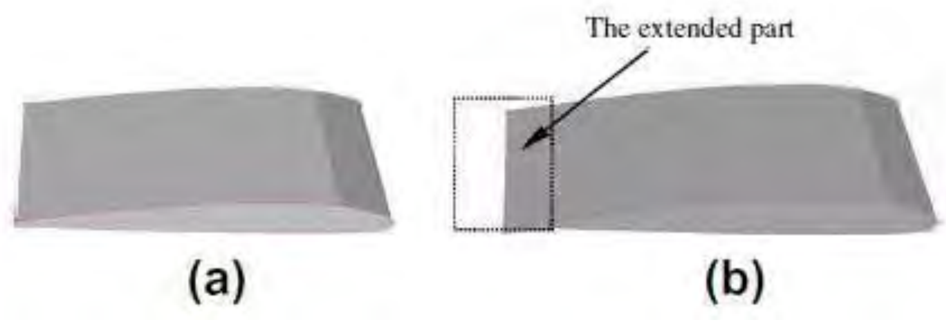



Figure 7: Comparison of blade after extended (Chunxi et al., 2011)

The performances of centrifugal blower also can be improving by increase in trend setting a higher ratio between volute width ($B5$) and impeller width ($B2$) and partial pressure recovery. This strategy has been proven by Hariharan and Govardhan. In centrifugal machine consists one the key component which is volute. This component is to collect the fluid from impeller and transfer to the delivery pipe. Tongue clearance, shape and cross-sectional area of the volute are the main parameters affecting the volute performance. Cross sections shape commonly used in industry are parallel wall, rectangular and circular. (C. HariHaran *et al.,* 2016)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 2.2 CFD analysis

This journal investigates and show the results of the actual standard impeller model and the designed model were analysed virtually by Solidworks Flow Simulation, a CFD Software package. The optimized combination of parameters was obtained and implemented in a model. The combination was analysed virtually to validate the results. The impeller modelled with the standard dimensions was analyzed using the Flow Simulation software. The aim of the simulation to find the relative error percentage and deviation of the CFD results from the standard impeller. The 9 models based on L9 orthogonal array were carried out to predict the performance of the impeller and to determine the flow rate for the given data. The flow rate, pressure distribution, peripheral velocity distribution and torque of the impeller are the objective of this flow simulation.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

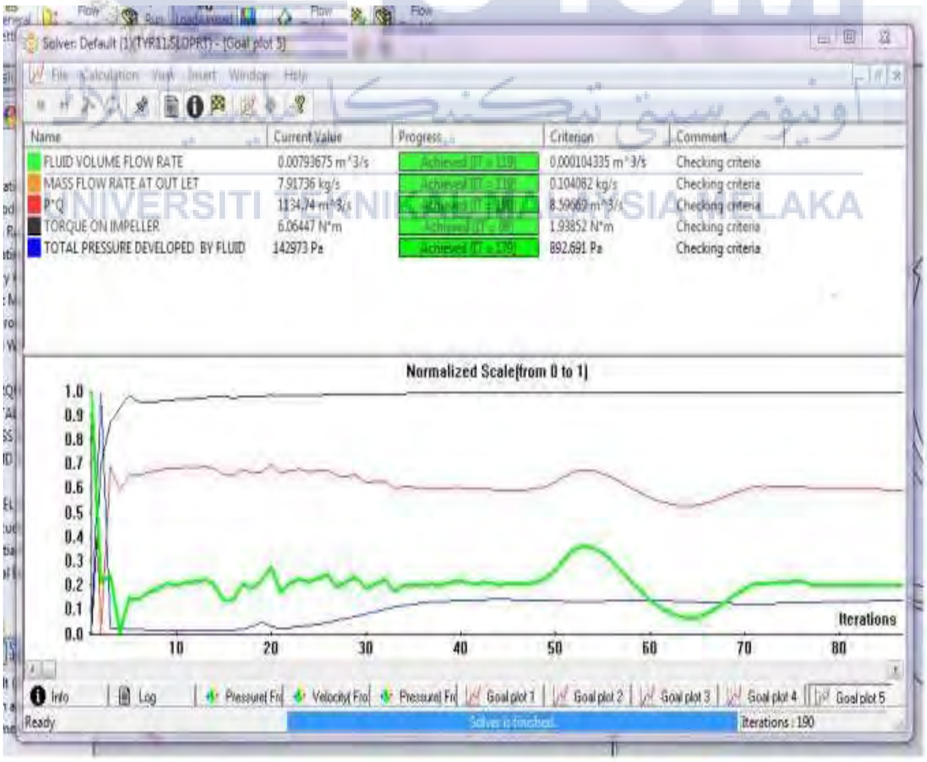



Figure 8: CFD solver result for impeller model (Hudson et al., 2013)

Based on the result, 0.000793 $m^3/s$ or 7.93 lps value of volume flow rate of the actual model of the impeller obtained from the CFD analysis. While, the volume flow rate the impeller as given by the pump manufacturing company is 0.008 $m^3/s$ or 8 lps. This result are obtained the result from from CFD analysis and the pump manufacturing company is quite closer. Then the 9 experiment models carried out with CFD analysis. The results for the L9 orthogonal array are given in the figure below.

| Expt . No | No. of blades | Outer dia $D_2$(mm) | Inlet blade angle(deg) | Impeller trim | VOLUME FLOW RATE$m^3/s$ |
|---|---|---|---|---|---|
| 1 | 6 | 163.4 | 25 | Over filing | 0.008842 |
| 2 | 6 | 159.1 | 30 | Normal | 0.008512 |
| 3 | 6 | 154.8 | 35 | Underfiling | 0.008311 |
| 4 | 7 | 163.4 | 30 | Underfiling | 0.008362 |
| 5 | 7 | 159.1 | 35 | Over filing | 0.008841 |
| 6 | 7 | 154.8 | 25 | Normal | 0.008411 |
| 7 | 8 | 163.4 | 35 | Normal | 0.01005 |
| 8 | 8 | 159.1 | 25 | Underfiling | 0.0080 |
| 9 | 8 | 154.8 | 30 | Over filing | 0.01004 |

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Table 1: L9 orthogonal results (Hudson et al., 2013)

The model is the analysed using Solidworks Flow Simulation software. This simulation to obtain the flow rate to validate the results of L9 Orthogonal array by Taguchi method. Figure 10 and 11 show the pressure contours and velocity distribution contours for the optimized combination model. (Hudson *et al.*, 2013)

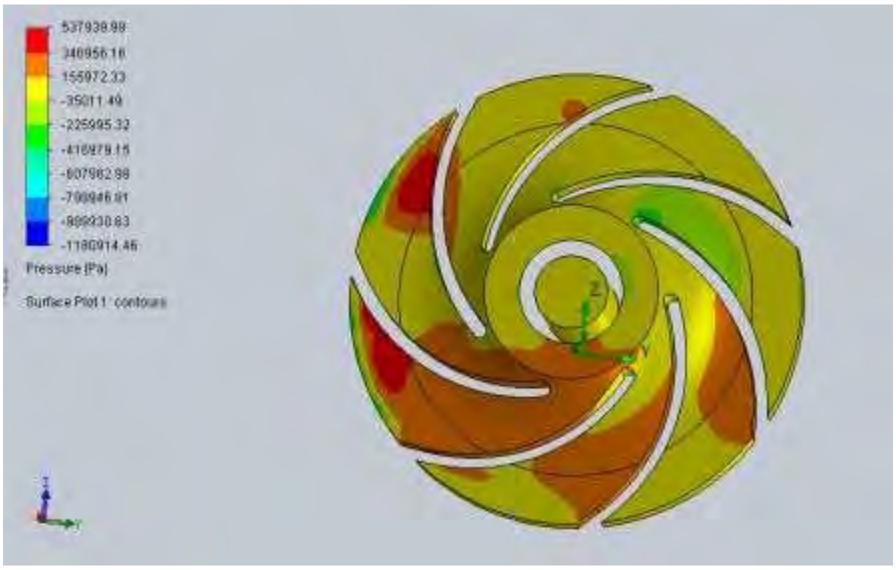



Figure 9: Pressure distribution for optimized model (Hudson et al., 2013)


UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Figure 10: Velocity distribution for optimized model (Hudson

# CHAPTER 3

# METHODOLOGY

UTeM
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 3.0 Introduction

This chapter will elaborate the details explanation of methodology that is being used to make this project complete and working well. Many methodology or findings from this field mainly generated into journal for others to take advantages and improves as upcoming studies. The method is used to achieve the objective of the project that will accomplish a perfect result. The flow chart of the project is shown in figure 12. This project starts by studying, finding and understanding of centrifugal blower process. Also in this project to find the performances of centrifugal blower by ANSYS software and design by using Solidworks Software

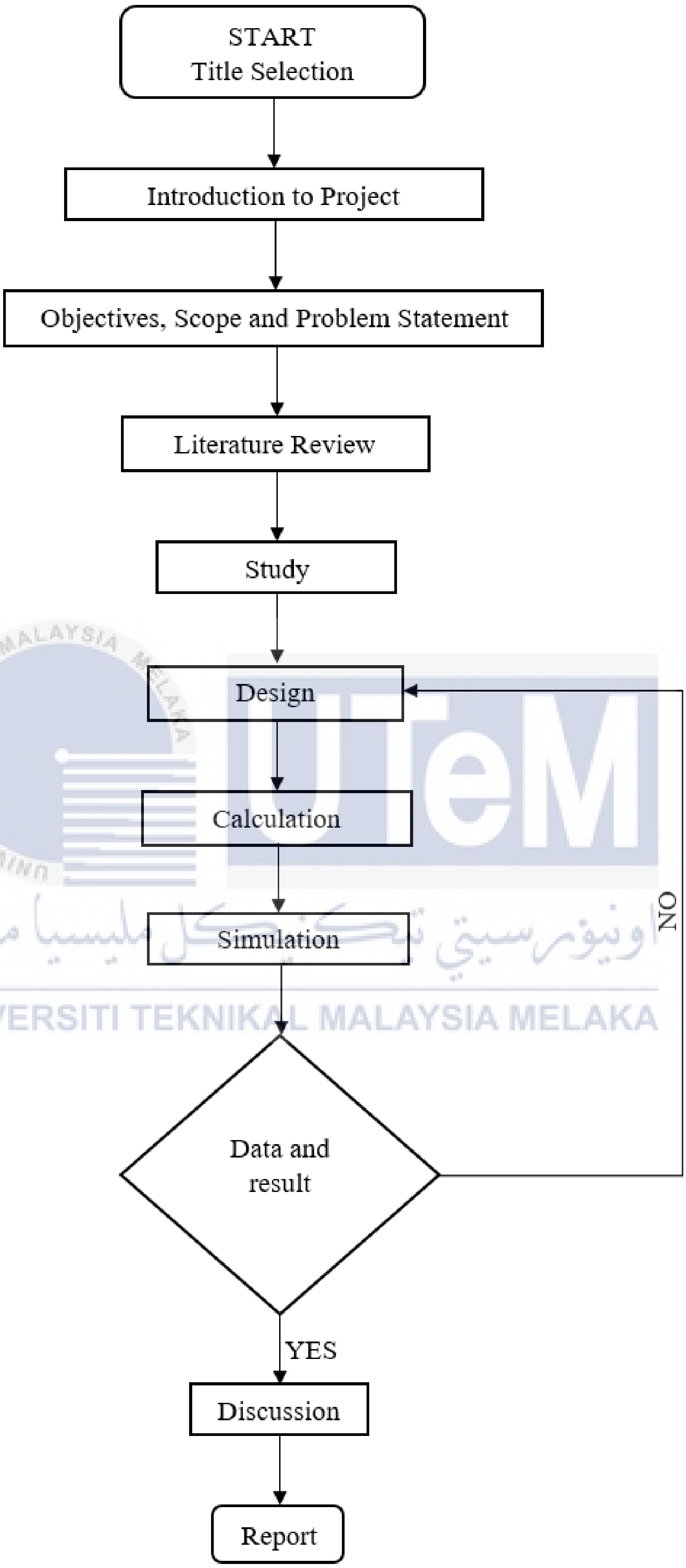



Figure 11: Flowchart of the methodology

## 3.1 Design

The design or dimension of centrifugal blower are selected based on information that have from literature review. Since the simulation are based on different dimension of centrifugal blower, 3 types dimension of centrifugal blower have been chose. The dimension is measure for the impeller diameter. The diameter are 20 cm, 40 cm and 60 cm. The design is drawing using SolidWorks Software. These designs have been design through SolidWorks Software compared to ANSYS Software because SolidWorks is easy to do drawing especially with complicated design. All types of impeller have a same total of blades which is 15 blades. Below are the schematic of impeller dimension.


UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Figure 12: Radius of impeller

| Type of impeller | Radius, cm | Diameter, cm | Num. of blades |
|---|---|---|---|
| Type A | 10 | 20 | 15 |
| Type B | 20 | 40 | 15 |
| Type C | 30 | 60 | 15 |

Table 2: Type of dimension

Since the diameter are different, the centrifugal blower casing will be different as well. The dimension of the casing also has been referred to journal. The figure show the isometric view and the dimension of centrifugal blower casing.

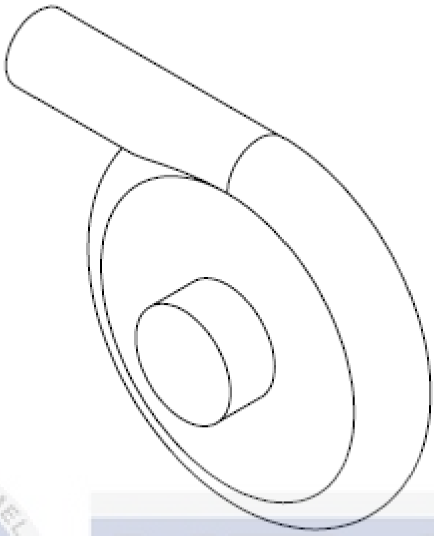


Figure 13: Isometric view of casing


Figure 14: Dimension of the casing

اونيورسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

| Type of Centrifugal Blower | Dimension | | | | |
|---|---|---|---|---|---|
| | A | B | C | D | E |
| A | 5cm | 17cm | 12.5cm | 5cm | 20cm |
| B | 10cm | 35cm | 25cm | 15cm | 40cm |
| C | 15cm | 50cm | 37.5cm | 25cm | 60cm |

Table 3: Dimension of the casing

After draw all the centrifugal blower casing and impeller, save the part as IGES format. This is because to import into the ANSYS software.


Figure 15: Save the part as IGES format

UTeM
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 3.2 Calculation

After study and findings on the journal or books, do a calculation to find the volume flow, pressure, velocity, efficiency etc. This is to determine the performance of centrifugal blower compared to using ANSYS simulation. Centrifugal blowers can achieve much more pressures than centrifugal fan, as high as 1.20 $kg/cm^2$. Centrifugal blowers quite same as centrifugal pumps. The rotating of impeller is typically gear-driven and rotates as fast between 15 000 and 16 000 rpm. Centrifugal blower commonly operates against of 0.35 to 0.70 $kg/cm^2$. The type of impeller has been chosen is backward curved blades. The characteristic of this impeller is having high pressure, high flow and high efficiency. Below shows the formula used to find the velocity.

$$Velocity\ v, \frac{m}{s} = \frac{C_p \times \sqrt{2 \times 9.81 \times \Delta p \times \gamma}}{\gamma}$$

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

$C_p$ is the constant value which is 0.85, while $\Delta p$ is the average differential pressure measured and $\gamma$ is the density of air. Below show the formula use to find the volume flow rate.

$$volume\ flow\ rate\ (Q), \frac{m^3}{sec} = Velocity, V\ \left(\frac{m}{sec}\right) \times Area\ (m^2)$$

For the efficiency, commonly manufacturers use two ways to find the efficiency which are mechanical efficiency and static efficiency

$$Mechanical\ Efficiency\ ,\% = \frac{Volume\ in \frac{m^3}{s} \times \Delta p\ (total\ pressure)\ in\ mmWC}{102 \times power\ input\ to\ fan\ shaft\ in\ kW} \times 100$$

$$Static\ Efficiency\ ,\% = \frac{Volume\ in \frac{m^3}{s} \times \Delta p\ (total\ pressure)\ in\ mmWC}{102 \times power\ input\ to\ fan\ shaft\ in\ kW} \times 100$$

Drive motor kW can be measured by a load analyser.

$$Shaft\ power = drive\ motor\ in\ kW \times motor\ efficiency$$

Based on the both efficiency formula, the calculation need the value of total pressure to get the efficiency of centrifugal blower. Below shows the formula need to be use to find the pressure total.

$$Pressure\ total = Pressure\ Static + Pressure\ velocity$$

## 3.3 Simulation

After all the drawing using Solidworks, do the simulation using ANSYS. Certain data or input value are taken from the journal such the rotation of impeller. Generally, industry used a motor that produce the rotation of impeller about 16 000 rpm, hence the value of rotation input for the simulation is 16 000 rpm. Before the do the simulation, should be import the part or centrifugal blower that have been drawn and save as IGES format.

### 3.3.1 Import part from SolidWorks

Before start for simulation, need to import the part from solidworks. Since this is centrifugal blower, the suitable ANSYS CFD type use for this simulation is Fluid Flow CFX. Below are the figure show the step to start import the part.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

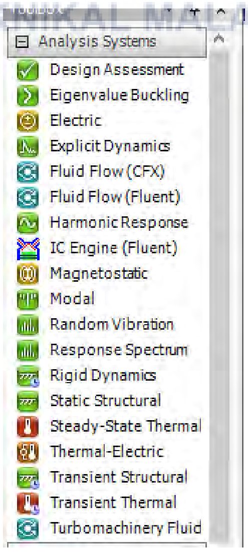



Figure 16: Drag out the Fluid Flow (CFX)

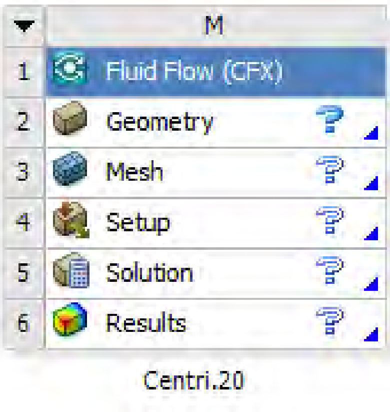



Figure 17: Right click on column Geometry and import the geometry

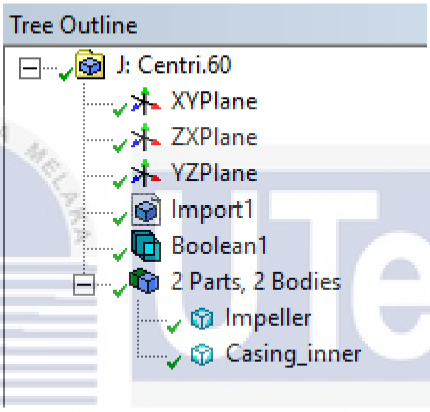



Figure 18: Rename the part at tree outline

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

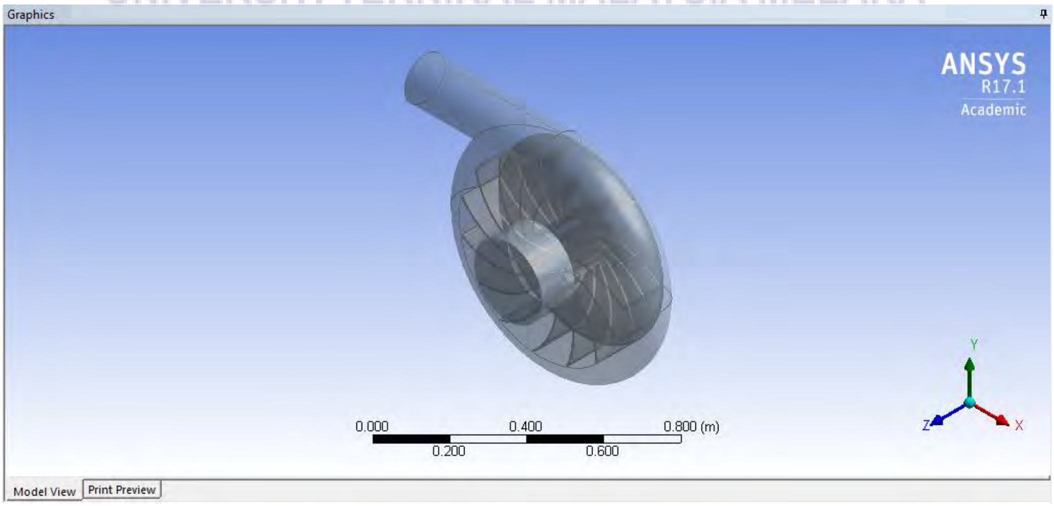



Figure 19: The part show in the geometry

After all done, click generate and close the geometry. Next step is to do the meshing of the body.

### 3.3.2 Meshing

After import the geometry, the next step is to do the meshing body of the centrifugal blower casing and impeller. Below shows the figure with step to do the meshing procedure.

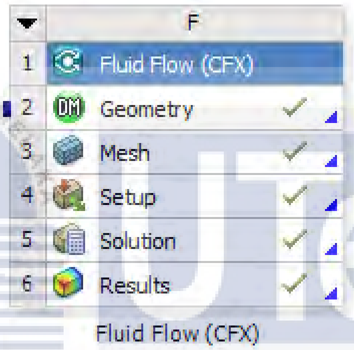



Figure 20: At the Mesh column right click and click edit

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

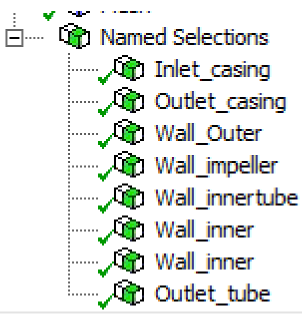



Figure 21: Name selections

Based on the figure 20, show the name selections. For this step need to name the face correctly. Example at the inlet of air at centrifugal blower, need to name the face as inlet if the face as outlet, the face need to name as outlet. After name all the selections for inlet, outlet, wall, impeller etc, at tree outline right click on "Mesh" and click "Generate Mesh". Figure below show the tree outline.

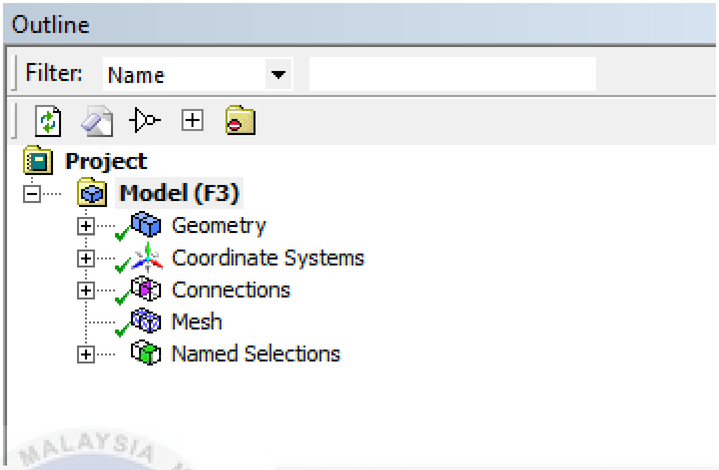



Figure 22: Generate Mesh

### 3.3.3 Setup

After done the meshing step, next for the setup. Setup should be insert the value exactly for the centrifugal blower operate. Each type of centrifugal blower has a same value for this setup. Example like the rotating of the impeller. The rotating is set as generally manufactured done at the industry which is 16 000 rpm. Below are show the step with figure to complete this step.

UTeM
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

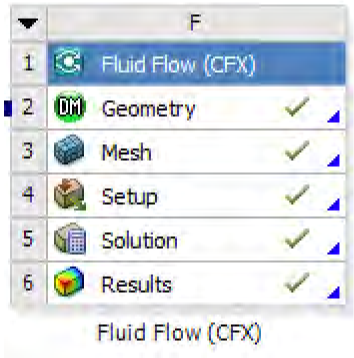



Figure 23: Double click at setup

After double click the setup, pop up will show and need to choose type of run either double precision and large problem. Based on my study abou ANSYS software, usually use double precision to run the simulation. Below are show the figure about type of run. Below are the meshing view of the casing and the impeller after meshing.

UTeM
UNIVERSITI TEKNIKAL MALAYSIA MELAKA
اونيۇرسيتي تيكنيكل مليسيا ملاك

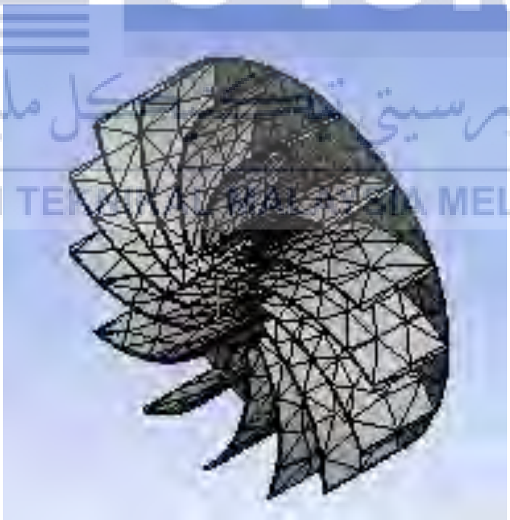


Figure 24 : Meshing (Impeller)

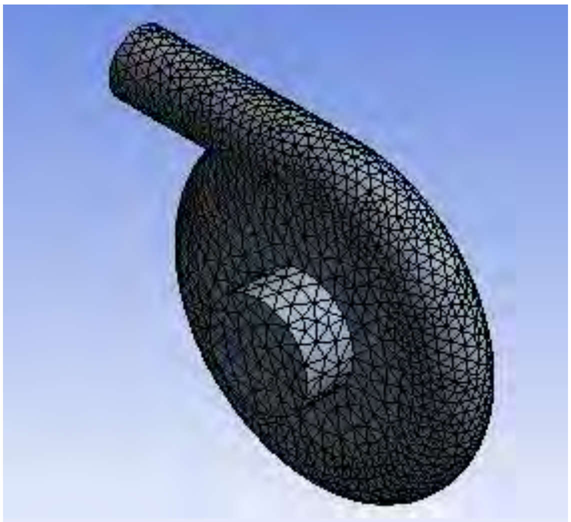


Figure 25 : Meshing (Casing)

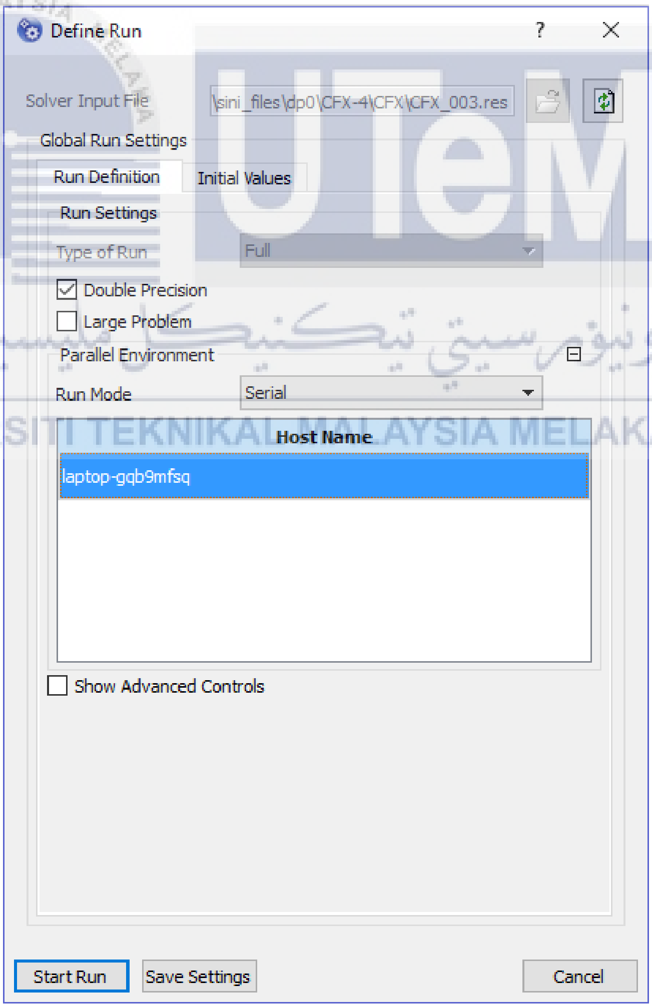



Figure 26: Type of run and after choose click start run

After start run, the outline will come out. Then setup the value for inlet, outlet. Set which part of the centrifugal blower will be as a wall. This step are important to get the result. Below shows the figure outline of the setup.

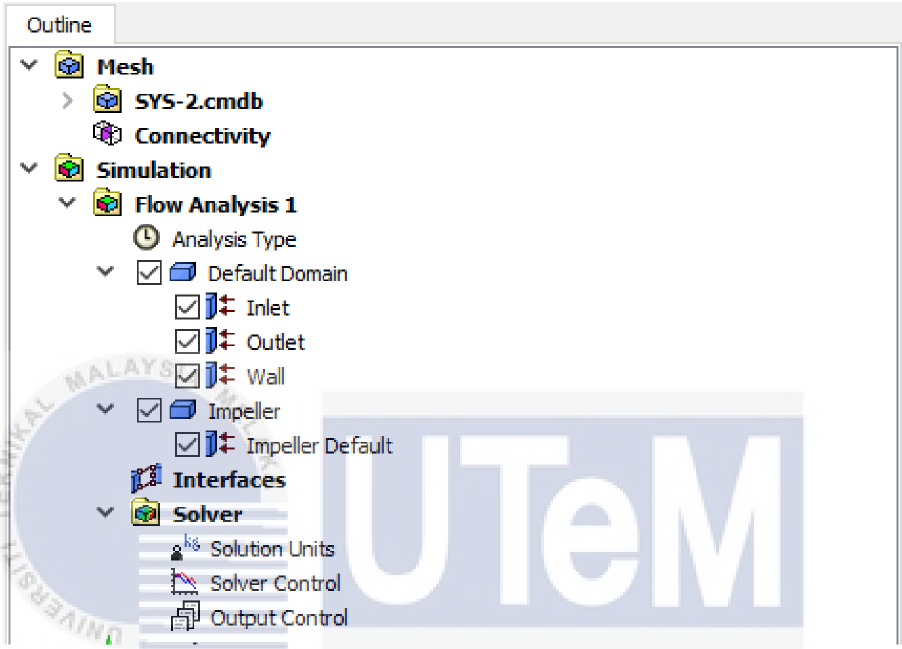



Figure 27: Outline of the setup

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

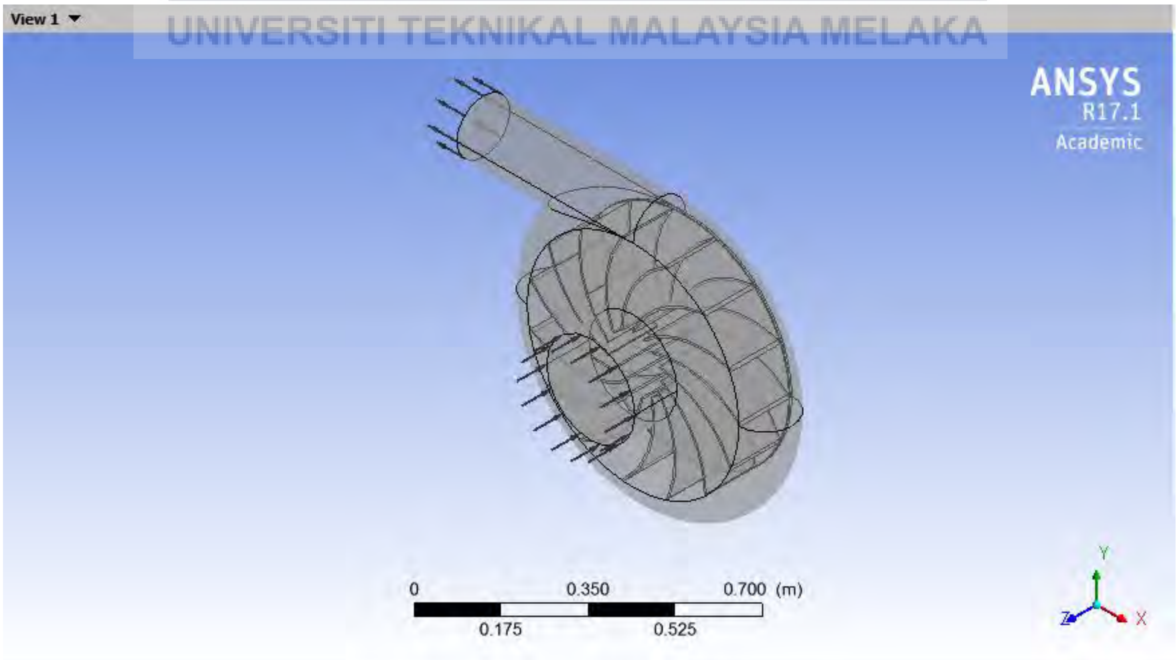



Figure 28: View after setting all the setup

### 3.3.4 Solution

After setting all the setup, close the CFX-pre. The next step is to figure out the solution of the simulation. As usual double click on the solution. Below shows the figure.

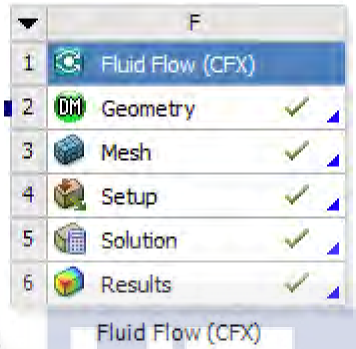



Figure 29: Double click on solution

After double click the solution, the simulation will operate and run itself. The solution will automatic operate and show the result and graph. Below shows the example of result shown.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

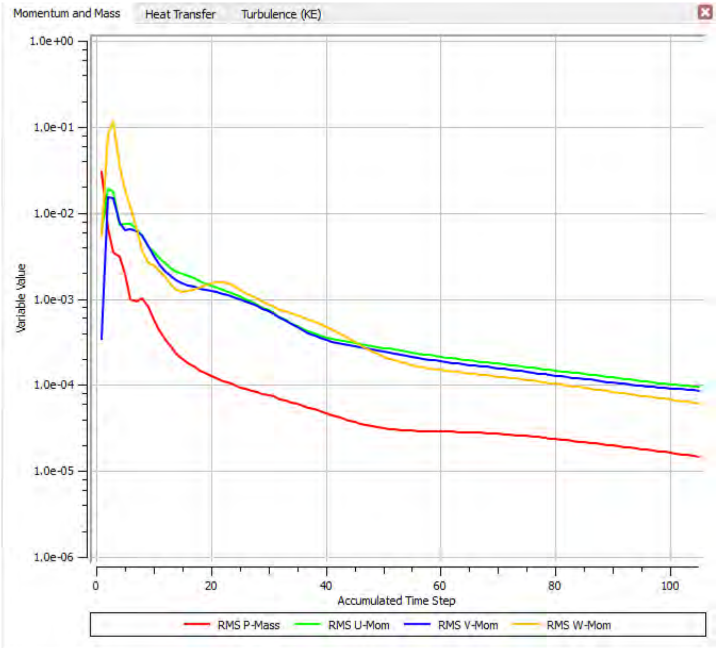



Figure 30: Momentum and mass graph

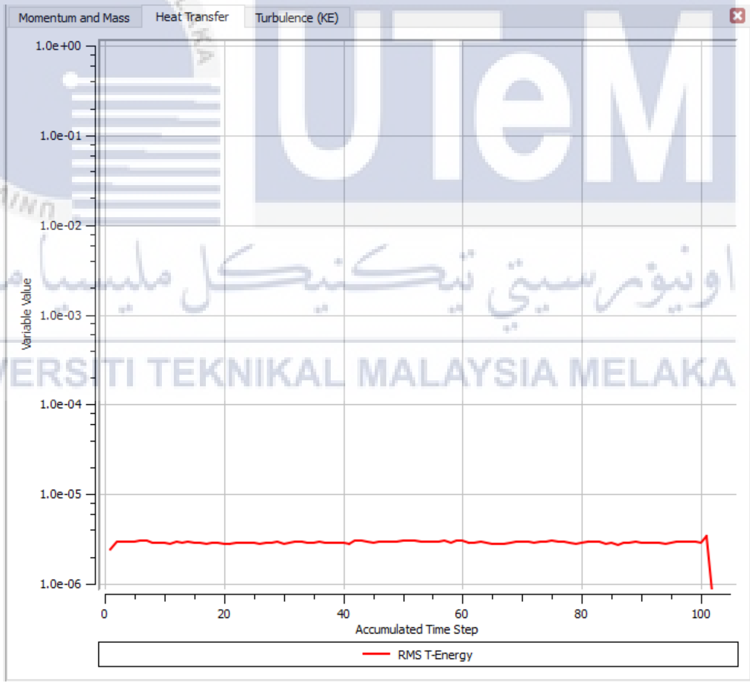



Figure 31: Heat Transfer graph

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

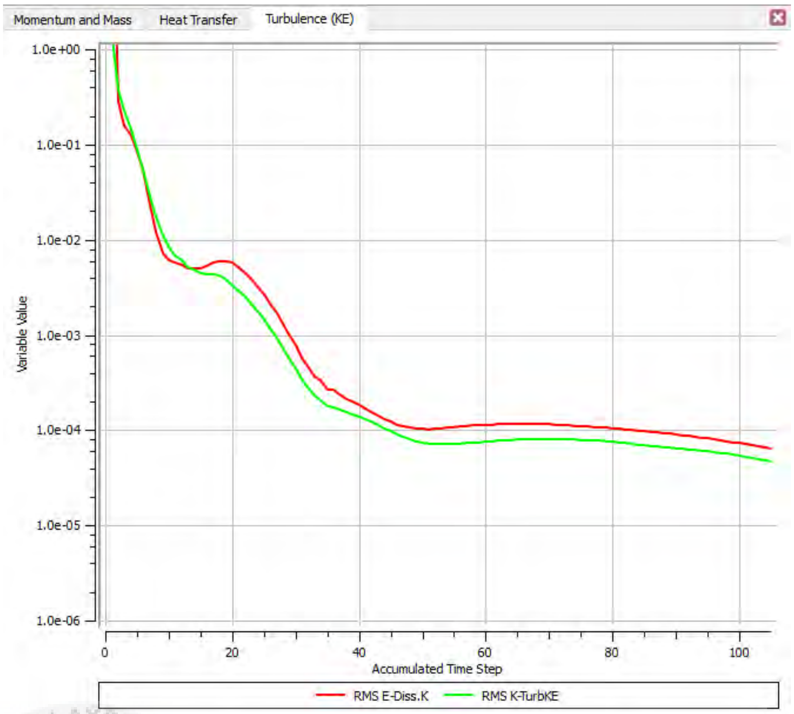



Figure 32: Turbulance graph

```
+--------------------------------------------------------------------+
|                    Variable Range Information                      |
+--------------------------------------------------------------------+

Domain Name : Default Domain
+--------------------------------------------------------------------+
|       Variable Name                         |    min    |    max    |
+--------------------------------------------------------------------+
| Density                                     |  1.19E+00 |  1.19E+00 |
| Specific Heat Capacity at Constant Pressure|  1.00E+03 |  1.00E+03 |
| Dynamic Viscosity                           |  1.83E-05 |  1.83E-05 |
| Thermal Conductivity                        |  2.61E-02 |  2.61E-02 |
| Static Entropy                              |  0.00E+00 |  0.00E+00 |
| Velocity u                                  | -1.57E+03 |  9.10E+02 |
| Velocity v                                  | -5.21E+02 |  1.18E+03 |
| Velocity w                                  | -4.62E+02 |  7.03E+02 |
| Pressure                                    | -7.44E+05 |  1.26E+06 |
| Turbulence Kinetic Energy                   |  1.40E+00 |  4.27E+04 |
| Turbulence Eddy Dissipation                 |  1.39E+04 |  4.03E+08 |
| Eddy Viscosity                              |  3.57E-06 |  4.83E-01 |
| Temperature                                 |  2.98E+02 |  2.98E+02 |
+--------------------------------------------------------------------+

Domain Name : Impeller
+--------------------------------------------------------------------+
|       Variable Name                         |    min    |    max    |
+--------------------------------------------------------------------+
| Density                                     |  7.85E+03 |  7.85E+03 |
| Specific Heat Capacity at Constant Pressure|  4.34E+02 |  4.34E+02 |
| Thermal Conductivity                        |  6.05E+01 |  6.05E+01 |
| Static Entropy                              | -1.03E-03 |  6.69E-04 |
| Temperature                                 |  2.98E+02 |  2.98E+02 |
| Static Enthalpy                             | -3.08E-01 |  2.00E-01 |
+--------------------------------------------------------------------+

+--------------------------------------------------------------------+
|              CPU Requirements of Numerical Solution                |
+--------------------------------------------------------------------+
```

Figure 33: The data of the simulation

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

All the data are recorded either to generate the graph using Microsoft Excel. The simulation should be done for one type of centrifugal blower only. The next dimension should be done by using another Fluid Flow (CFX) simulation.

### 3.3.5 Result

After done all the solution, this step to figure out the air flow in the centrifugal blower in animation flow. As usual after done the solution, close CFX- solver manager. Double click on the result from outline. Below shows the figure of the outline.

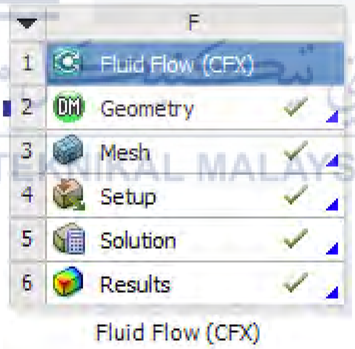



Figure 34: Double click on the result

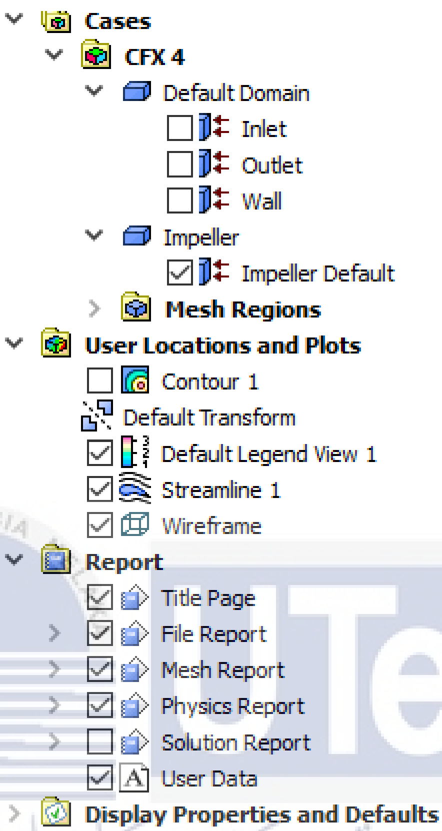



Figure 35: Outline

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Set up either to show the streamline of air flow in the centrifugal blower. In this step also can find out the situation of temperature in centrifugal blower. Below show the example result shown.

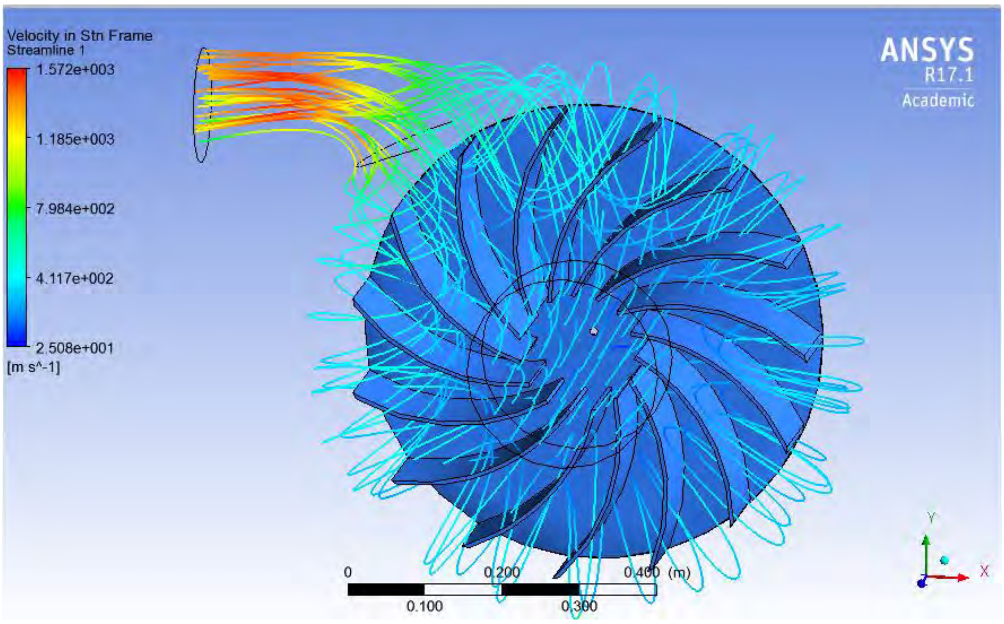
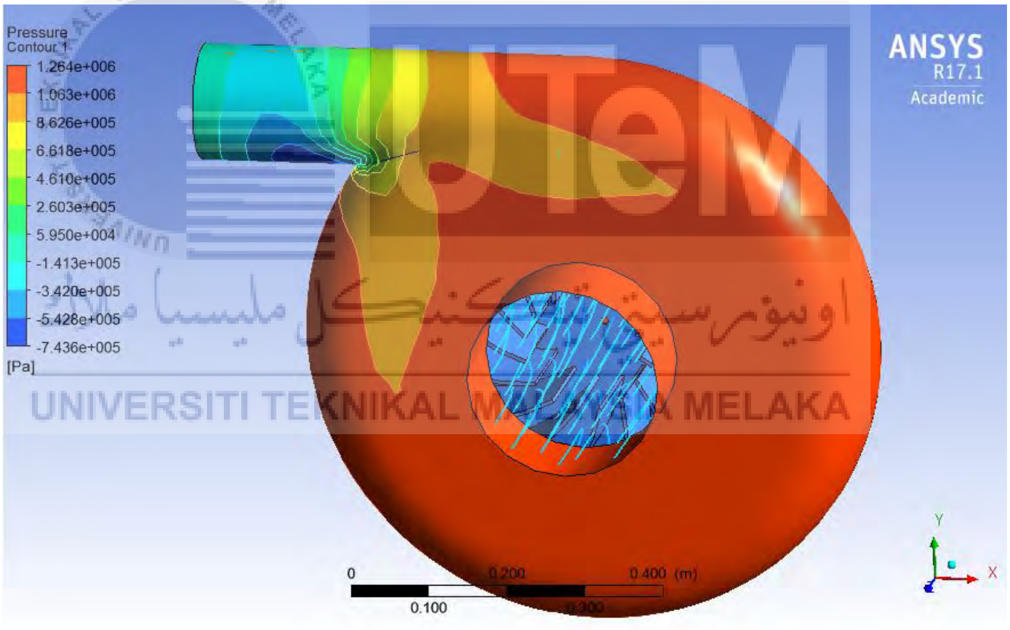



Figure 36: The velocity streamline




UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Figure 37: Pressure view contour

**CHAPTER 4**

# DATA AND ANALYSIS

## 4.0 Introduction

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

This chapter presents the data and analysis of the collected data based on three type dimensions of centrifugal blower. All the data collected from ANSYS Software and from the calculation. The formula used are stated at chapter 3, to find the velocity, mass flow rate etc. Three type of centrifugal blower are type A, B and C. The difference dimension is measure for the impeller diameter. The diameter are 20 cm, 40 cm and 60 cm. The design is drawing using SolidWorks Software and each type of centrifugal blower simulated in ANSYS Software in Fluid Flow (CFX).

### 4.1 Data from calculation and simulation

This is to determine the performance of centrifugal blower compared to using ANSYS simulation. Centrifugal blowers can achieve much more pressures than centrifugal fan, as high as 1.20 $kg/cm^2$. Centrifugal blowers quite same as centrifugal pumps. The rotating of impeller is typically gear-driven and rotates as fast between 15 000 and 16 000 rpm. Centrifugal blower commonly operates against of 0.35 to 0.70 $kg/cm^2$. The type of impeller has been chosen is backward curved blades. The characteristic of this impeller is having high pressure, high flow and high efficiency. Below shows the formula used to find the velocity.

$$Velocity\ v, \frac{m}{s} = \frac{C_p \times \sqrt{2 \times 9.81 \times \Delta p \times \gamma}}{\gamma}$$

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

$C_p$ is the constant value which is 0.85, while $\Delta p$ is the average differential pressure measured and $\gamma$ is the density of air. Below show the formula use to find the volume flow rate.

$$volume\ flow\ rate\ (Q), \frac{m^3}{sec} = Velocity, V\ \left(\frac{m}{sec}\right) \times Area\ (m^2)$$

For the efficiency, commonly manufacturers use two ways to find the efficiency which are mechanical efficiency and static efficiency

$$Mechanical\ Efficiency\ ,\% = \frac{Volume\ in \frac{m^3}{s} \times \Delta p\ (total\ pressure)\ in\ mmWC}{102 \times power\ input\ to\ fan\ shaft\ in\ kW} \times 100$$

$$Static\ Efficiency\ ,\% = \frac{Volume\ in \frac{m^3}{s} \times \Delta p\ (total\ pressure)\ in\ mmWC}{102 \times power\ input\ to\ fan\ shaft\ in\ kW} \times 100$$

Drive motor kW can be measured by a load analyser.

$$Shaft\ power = drive\ motor\ in\ kW \times motor\ efficiency$$

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Based on the both efficiency formula, the calculation need the value of total pressure to get the efficiency of centrifugal blower. Below shows the formula need to be use to find the pressure total.

$$Pressure\ total = Pressure\ Static + Pressure\ velocity$$

Hence, below the dimension of impeller each type of centrifugal blower

| Type of impeller | Radius, cm | Diameter, cm | Num. of blades |
|---|---|---|---|
| Type A | 10 | 20 | 15 |
| Type B | 20 | 40 | 15 |
| Type C | 30 | 60 | 15 |

### 4.1.1 Centrifugal blower Type A

Below are the data collected from ANSYS software.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

| Type | Volume, $m^3$ | Surface area, $m^2$ |
|---|---|---|
| A | 0.0027131 | 0.14849 |

Table 4: Volume and surface area of Type A

| Statistics | |
|---|---|
| Nodes | 7854 |
| Elements | 33158 |
| Mesh Metric | Orthogonal Quality |
| Min | 1.9198e-002 |
| Max | 0.99089 |
| Average | 0.80156 |
| Standard Deviation | 0.1599 |

Table 5: Statistic of Type A

Based on Table 4, show that the volume of centrifugal blower type A is 0.0027131 $m^3$ and the surface area is 0.14849 $m^2$. Meanwhile, table 5 show the node and elements of centrifugal blower type A. This data are collected after the centrifugal blower done with meshing at ANSYS software. Also referred from table 5, it is state that the minimum mesh for centrifugal blower type A is 1.9198e-002 while the maximum is 0.99089. Hence the average is 0.80156. Below show the data collected from CFX simulation.

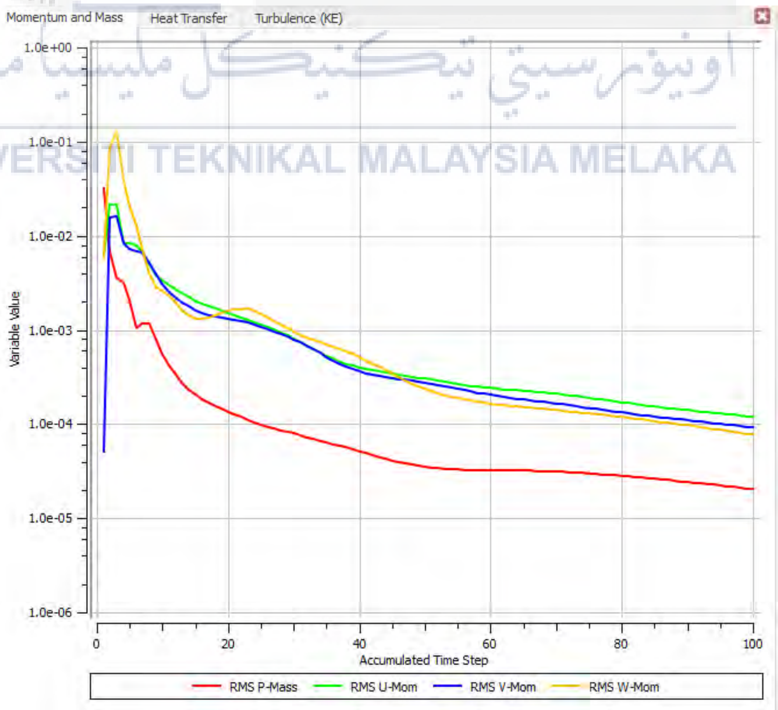



Figure 38: Momentum and Mass (Type A)

UTeM
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

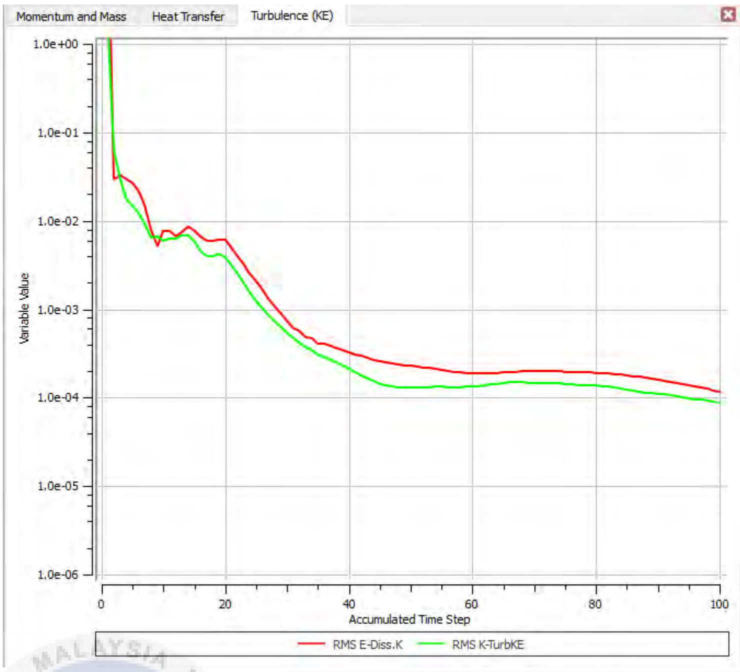



Figure 39: Turbulance (Type A)

Domain Name : Default Domain

| Variable Name | min | max |
|---|---|---|
| Density | 1.19E+00 | 1.19E+00 |
| Specific Heat Capacity at Constant Pressure | 1.00E+03 | 1.00E+03 |
| Dynamic Viscosity | 1.83E-05 | 1.83E-05 |
| Thermal Conductivity | 2.61E-02 | 2.61E-02 |
| Static Entropy | 0.00E+00 | 0.00E+00 |
| Velocity u | -1.44E+04 | 8.21E+03 |
| Velocity v | -4.93E+03 | 1.09E+04 |
| Velocity w | -4.22E+03 | 6.47E+03 |
| Pressure | -5.84E+07 | 1.05E+08 |
| Turbulence Kinetic Energy | 2.68E+01 | 3.60E+06 |
| Turbulence Eddy Dissipation | 6.13E+07 | 9.56E+11 |
| Eddy Viscosity | 5.36E-07 | 1.47E+00 |
| Temperature | 2.98E+02 | 2.98E+02 |

Domain Name : Impeller

| Variable Name | min | max |
|---|---|---|
| Density | 7.85E+03 | 7.85E+03 |
| Specific Heat Capacity at Constant Pressure | 4.34E+02 | 4.34E+02 |
| Thermal Conductivity | 6.05E+01 | 6.05E+01 |
| Static Entropy | -4.55E-13 | 4.55E-13 |
| Temperature | 2.98E+02 | 2.98E+02 |
| Static Enthalpy | -1.46E-10 | 1.75E-10 |

Figure 40: Data recorded (Type A)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

```
Domain Name : Default Domain
    Global Length                                    = 1.3916E-01
    Minimum Extent                                   = 1.0000E-01
    Maximum Extent                                   = 2.9500E-01
    Density                                          = 1.1850E+00
    Dynamic Viscosity                                = 1.8310E-05
    Velocity                                         = 5.2834E+03
    Advection Time                                   = 2.6339E-05
    Reynolds Number                                  = 4.7585E+07

Domain Name : Impeller
    Global Length                                    = 6.3765E-02
    Minimum Extent                                   = 4.3000E-02
    Maximum Extent                                   = 1.9997E-01
    Density                                          = 7.8540E+03
    Thermal Conductivity                             = 6.0500E+01
    Specific Heat Capacity at Constant Pressure      = 4.3400E+02
    Thermal Diffusivity                              = 1.7749E-05
    Average Diffusion Timescale                      = 2.2908E+02
    Minimum Diffusion Timescale                      = 1.0417E+02
    Maximum Diffusion Timescale                      = 2.2531E+03
    Temperature Range                                = 7.3896E-13
```

Figure 41: Average data (Type A)

اونيورسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

### 4.1.2 Centrifugal blower Type B

Below are the data collected from ANSYS software

| Type | Volume, $m^3$ | Surface area, $m^2$ |
|---|---|---|
| B | 0.02101 | 0.57984 |

Table 6: Volume and surface area of Type B

| Statistics | |
|---|---|
| Nodes | 7068 |
| Elements | 29801 |
| Mesh Metric | Orthogonal Quality |
| Min | 2.5418e-002 |
| Max | 0.99275 |
| Average | 0.78066 |
| Standard Deviation | 0.20304 |

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Table 7: Statistic of Type B

Based on Table 6, show that the volume of centrifugal blower type B is 0.02101 $m^3$ and the surface area is 0.57984 $m^2$. Meanwhile, table 6 show the node and elements of centrifugal blower type B. This data is collected after the centrifugal blower done with meshing at ANSYS software. Also, referred from table 5, it is state that the minimum mesh for centrifugal blower type B is 2.5418e-002 while the maximum is 0.99275. Hence the average is 0.78066. Below show the data collected from CFX simulation.

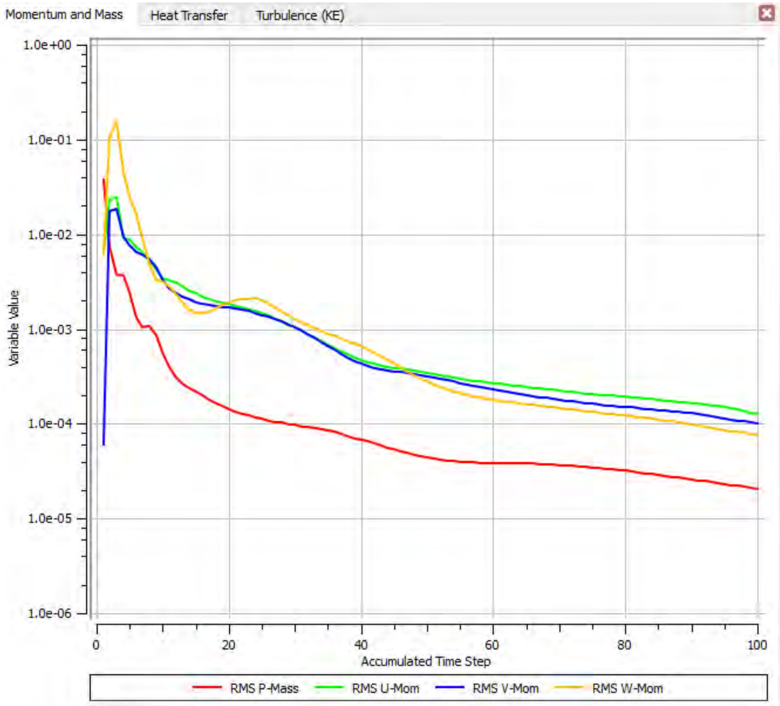



Figure 42: Momentum and Mass (Type B)

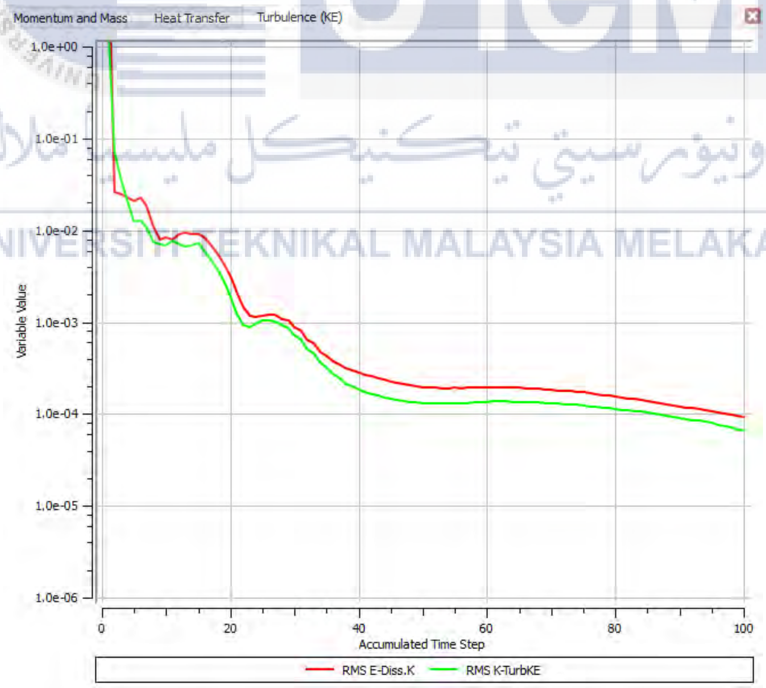



Figure 43: Turbulance (Type B)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Domain Name : Default Domain

| Variable Name | min | max |
|---|---|---|
| Density | 1.19E+00 | 1.19E+00 |
| Specific Heat Capacity at Constant Pressure | 1.00E+03 | 1.00E+03 |
| Dynamic Viscosity | 1.83E-05 | 1.83E-05 |
| Thermal Conductivity | 2.61E-02 | 2.61E-02 |
| Static Entropy | 0.00E+00 | 0.00E+00 |
| Velocity u | -3.51E+03 | 2.02E+03 |
| Velocity v | -1.35E+03 | 2.64E+03 |
| Velocity w | -1.11E+03 | 1.60E+03 |
| Pressure | -4.77E+06 | 6.50E+06 |
| Turbulence Kinetic Energy | 4.75E+00 | 1.66E+05 |
| Turbulence Eddy Dissipation | 3.50E+05 | 5.32E+09 |
| Eddy Viscosity | 1.88E-06 | 5.54E-01 |
| Temperature | 2.98E+02 | 2.98E+02 |

Domain Name : Impeller

| Variable Name | min | max |
|---|---|---|
| Density | 7.85E+03 | 7.85E+03 |
| Specific Heat Capacity at Constant Pressure | 4.34E+02 | 4.34E+02 |
| Thermal Conductivity | 6.05E+01 | 6.05E+01 |
| Static Entropy | -9.09E-13 | 9.09E-13 |
| Temperature | 2.98E+02 | 2.98E+02 |
| Static Enthalpy | -3.20E-10 | 2.47E-10 |

Figure 44: Data recorded (Type B)

Domain Name : Default Domain

| | |
|---|---|
| Global Length | = 2.7535E-01 |
| Minimum Extent | = 1.7000E-01 |
| Maximum Extent | = 6.0000E-01 |
| Density | = 1.1850E+00 |
| Dynamic Viscosity | = 1.8310E-05 |
| Velocity | = 1.3616E+03 |
| Advection Time | = 2.0223E-04 |
| Reynolds Number | = 2.4264E+07 |

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Domain Name : Impeller

| | |
|---|---|
| Global Length | = 1.1435E-01 |
| Minimum Extent | = 9.1000E-02 |
| Maximum Extent | = 4.0000E-01 |
| Density | = 7.8540E+03 |
| Thermal Conductivity | = 6.0500E+01 |
| Specific Heat Capacity at Constant Pressure | = 4.3400E+02 |
| Thermal Diffusivity | = 1.7749E-05 |
| Average Diffusion Timescale | = 7.3668E+02 |
| Minimum Diffusion Timescale | = 4.6656E+02 |
| Maximum Diffusion Timescale | = 9.0146E+03 |
| Temperature Range | = 1.3074E-12 |

Figure 45: Average data (Type B)

### 4.1.2 Centrifugal blower Type C

Below are the data collected from ANSYS software

| Type | Volume, $m^3$ | Surface area, $m^2$ |
|---|---|---|
| C | 0.072181 | 1.3129 |

Table 8: Volume and surface area of Type C

| Statistics | |
|---|---|
| Nodes | 7068 |
| Elements | 29801 |
| Mesh Metric | Orthogonal Quality |
| Min | 2.5418e-002 |
| Max | 0.99275 |
| Average | 0.78066 |
| Standard Deviation | 0.20304 |

Table 9: Statistic of Type C

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Based on Table 8, show that the volume of centrifugal blower type B is 0.072181 $m^3$ and the surface area is 1.3129 $m^2$. Meanwhile, table 6 show the node and elements of centrifugal blower type C. This data is collected after the centrifugal blower done with meshing at ANSYS software. Also, referred from table 5, it is state that the minimum mesh for centrifugal blower type C is 2.5418e-002 while the maximum is 0.99275. Hence the average is 0.78066. Below show the data collected from CFX simulation.

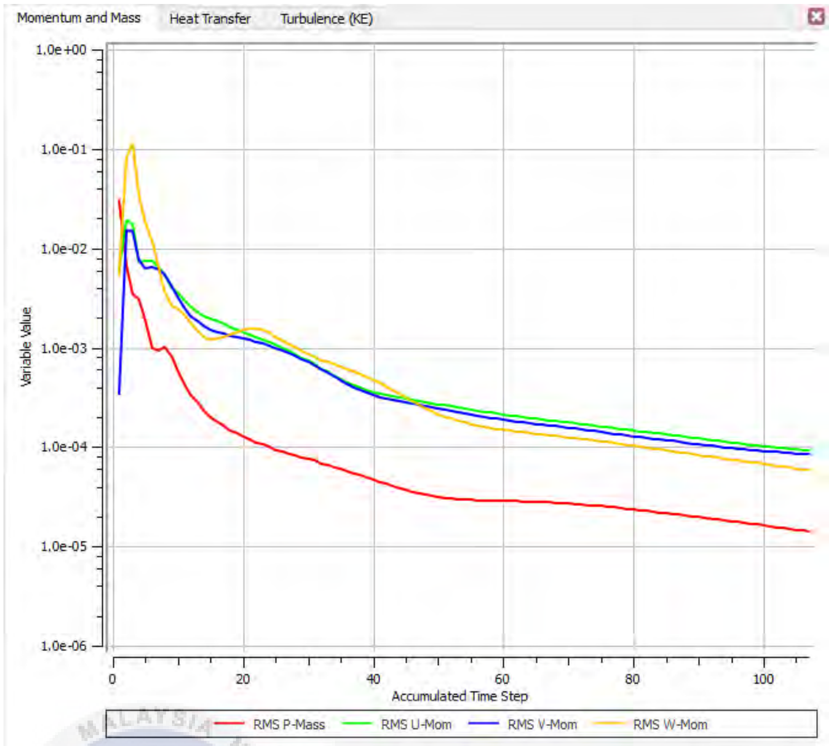



Figure 46: Momentum and Mass (Type C)

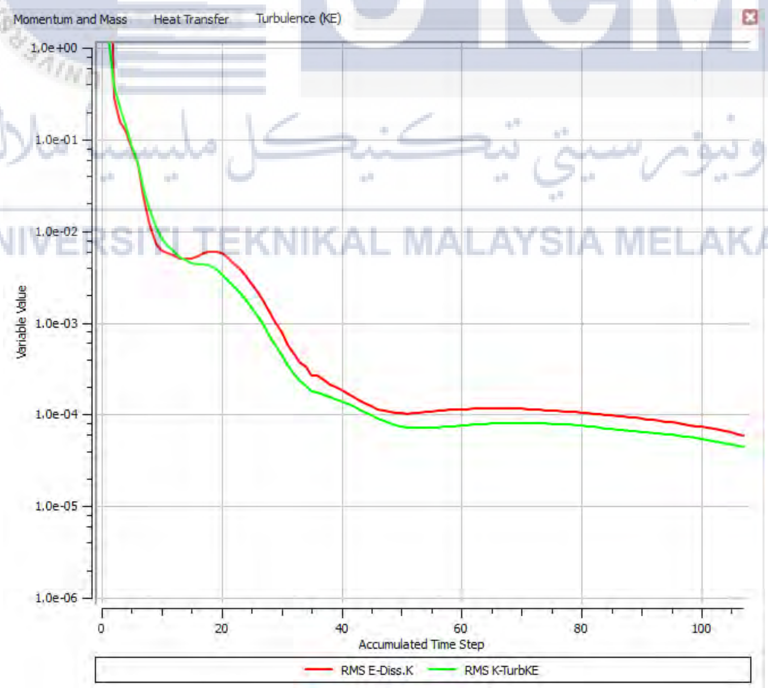



Figure 47: Turbulance (Type C)

Domain Name : Default Domain

| Variable Name | min | max |
|---|---|---|
| Density | 1.19E+00 | 1.19E+00 |
| Specific Heat Capacity at Constant Pressure | 1.00E+03 | 1.00E+03 |
| Dynamic Viscosity | 1.83E-05 | 1.83E-05 |
| Thermal Conductivity | 2.61E-02 | 2.61E-02 |
| Static Entropy | 0.00E+00 | 0.00E+00 |
| Velocity u | -1.57E+03 | 9.09E+02 |
| Velocity v | -5.24E+02 | 1.18E+03 |
| Velocity w | -4.61E+02 | 7.04E+02 |
| Pressure | -7.42E+05 | 1.26E+06 |
| Turbulence Kinetic Energy | 1.40E+00 | 4.27E+04 |
| Turbulence Eddy Dissipation | 1.40E+04 | 4.03E+08 |
| Eddy Viscosity | 3.57E-06 | 4.83E-01 |
| Temperature | 2.98E+02 | 2.98E+02 |

Domain Name : Impeller

| Variable Name | min | max |
|---|---|---|
| Density | 7.85E+03 | 7.85E+03 |
| Specific Heat Capacity at Constant Pressure | 4.34E+02 | 4.34E+02 |
| Thermal Conductivity | 6.05E+01 | 6.05E+01 |
| Static Entropy | -1.01E-03 | 6.10E-04 |
| Temperature | 2.98E+02 | 2.98E+02 |
| Static Enthalpy | -3.01E-01 | 1.82E-01 |

Figure 48: Data recorded (Type C)

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

```
Domain Name : Default Domain
    Global Length                                  = 4.1551E-01
    Minimum Extent                                 = 2.7000E-01
    Maximum Extent                                 = 8.7500E-01
    Density                                        = 1.1850E+00
    Dynamic Viscosity                              = 1.8310E-05
    Velocity                                       = 5.6405E+02
    Advection Time                                 = 7.3666E-04
    Reynolds Number                                = 1.5168E+07

Domain Name : Impeller
    Global Length                                  = 1.5291E-01
    Minimum Extent                                 = 1.4200E-01
    Maximum Extent                                 = 5.9997E-01
    Density                                        = 7.8540E+03
    Thermal Conductivity                           = 6.0500E+01
    Specific Heat Capacity at Constant Pressure    = 4.3400E+02
    Thermal Diffusivity                            = 1.7749E-05
    Average Diffusion Timescale                    = 1.3173E+03
    Minimum Diffusion Timescale                    = 1.1361E+03
    Maximum Diffusion Timescale                    = 2.0281E+04
    Temperature Range                              = 1.1124E-03
```

Figure 49: Average data (Type C)

# CHAPTER 5

# CONCLUSION AND RECOMMENDATION

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## 5.0 Introduction

This chapter will discuss about the conclusion that can be made from the whole this project. Also determine either from result and analysis it is accepted based on the objective or not. On the other side will discuss the recommendation that should be done to make this project more successful.

## 5.1 Conclusion and Recommendation

This project with title "Analysis of The Centrifugal Blower Performance with Different Dimension Configuration by Using ANSYS Software". The objective of this project is to find the difference process of centrifugal blower between difference dimension and to get the skill of using ANSYS software to conduct simulation of Centrifugal Blower. From result and discussion, can conclude that the objective is quite achieve because the result can show the difference performance of centrifugal blower with difference dimension. Also, got skill of using ANSYS software to conduct simulation of centrifugal blower.

Based on the result, can conlude that the centrifugal blower type A is produce more air flow compare to type C. This is because the blade is 15 blade and it can produce more air flow. Also, the size of the impeller can be increase by size to increase the air volume produce from the centrifugal blower.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Recommendation of this project are at the future are do a simulation with difference type of impeller because based on the journal mostly show that each type of impeller have their own characteristics and benefit. Also, do some experiment about the vibration of centrifugal blower during operate. Finally, recommendations for improving this study especially operation of centrifugal blower process. Understanding about the process give more benefit to proceed with any experiment about centrifugal blower.

## REFFERENCE

A.T. Oyelami, S.B. Adejuyigbe, M.A. Waheed, A.K Ogunkoya, D. Illiya. (2012). Analysis of Radial-Flow Impellers of Different Configurations. The Pasific Journal of Science and Technology, 13, 24- 33.

Adgale Tushar Balkrishna, G.R Gogate, R.V. Bajaj. (2014). Static and Dynamic Analysis of Centrifugal Blower Impeller usinf FEA. International Journal of Engineering Research and Technology, 3, 110-13

C. HariHaran, M. Govardhan. (2016). Improving Performance of An Industrial Centrifugal Blower with Parallel Wall Volutes. Applied Thermal Engineering,

Dinesh Tare, Vaibhav Bhagat, Basavaraj Talikotti. (2016). Static and Dynamic Analysis of Impeller of Centrifugal Blower. International Journal of Innovative Science, Engineering and Technology, 3, 547- 553.

Fans and blowers. 1 Oct 2016 retrieved from https://beeindia.gov.in/sites/default/files/3Ch5

اونيۆرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Hudson Daniel Raj.E, Kalaimani. T. Investigation of Key Impeller Parameters of Centrifugal Pump Using CFD. International Journal of Engineering Research and Technology, 2, 2209-2214.

Karthik V, Rajeshkannah T (2014). Experimental Investigation of A Centrifugal Blower by Using CFD. International Journal of Mechanical Engineering and Robotic Research, 3, 39-44.

Li Chunxi, Wang Song Ling, Jia Yakui. (2011). The Performance of Centrifugal Fan with Enlarged Impeller. Energy Conversion and Management, 52, 2902-2910.

Masayuki Takahashi (2002). Characteristics of Centrifugal Blower and Its Effective Use in High Static Pressure Area. Oriental Motor, 1, 1-6.

Siddarth Thakur, Wanlai Lin, Jeffrey Wright. (2002). Prediction of Flow in Centrifugal Blower Using Quasi- Steady Rotor-Stator Models. Journal Engineering Mechanical, 128, 1039-1049.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

V. Krishma, K. Naresh Kumar, M. Prasanth Kumar. (2013). Numerical Analysis of Centrifugal Blower using CFD. International Journal of Engineering Research and Technology, 2, 1461- 1463

Veeranjaneyulu Itha, T.B.S. Rao. (2012). Static and Dynamic Analysis of A Centrifugal Blower Using Fea. International Journal of Engineering and Technology, 1, 1-11.

Young- Tae Lee, Hee-Chang Lim (2016). Performance assessment of various fan ribs inside a centrifugal blower. Energy, 94, 609-622

اونيۈرسيتي تيكنيكل مليسيا ملاك
UNIVERSITI TEKNIKAL MALAYSIA MELAKA