

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

MOHAMAD ASYRAF BIN NAIM

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 : .....

## 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                   : .. .. .

## **DEDICATION**

To my beloved mother and father

## **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.

## 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 memainkan 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.*

## **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



## CONTENT

<b>DECLARATION .....</b>	<b>ii</b>
<b>APPROVAL .....</b>	<b>iii</b>
<b>DEDICATION .....</b>	<b>iv</b>
<b>ABSTRACT .....</b>	<b>v</b>
<b>ABSTRAK.....</b>	<b>vi</b>
<b>ACKNOWLEDGEMENT .....</b>	<b>vii</b>
<b>CONTENT .....</b>	<b>viii</b>
<b>LIST OF FIGURES.....</b>	<b>xi</b>
<b>LIST OF TABLES.....</b>	<b>xiv</b>
<b>LIST OF ABBEREVATIONS.....</b>	<b>xv</b>
<b>LIST OF SYMBOLS.....</b>	<b>xvi</b>
<b>CHAPTER 1.....</b>	<b>1</b>
<b>INTRODUCTION .....</b>	<b>1</b>
1.1 Background .....	1
1.2 Problem Statement.....	2
1.2 Objective .....	3

1.3	Scope of Project .....	3
1.4	General Methodology .....	4
1.4.1	Literature review .....	4
1.4.2	Study .....	4
1.4.3	Experiment .....	4
1.4.4	Simulation .....	5
1.4.5	Report writing .....	5
 <b>CHAPTER 2.....</b>		<b>7</b>
<b>LITERATURE REVIEW .....</b>		<b>7</b>
2.0	Introduction.....	7
2.1	Centrifugal Blower .....	7
2.1.1	Introduction .....	7
2.1.2	Geometrical model of centrifugal blower.....	8
2.1.3	Centrifugal blower analysis .....	11
2.1.4	The performances of centrifugal blower .....	12
2.2	CFD analysis.....	14
 <b>CHAPTER 3.....</b>		<b>17</b>
<b>METHODOLOGY .....</b>		<b>17</b>
3.0	Introduction.....	17
3.1	Design .....	19
3.2	Calculation .....	22
3.3	Simulation .....	24
3.3.1	Import part from SolidWorks .....	24
3.3.2	Meshing .....	26
3.3.3	Setup.....	27

3.3.4	Solution.....	31
3.3.5	Result.....	34
<b>CHAPTER 4.....</b>		<b>37</b>
<b>DATA AND ANALYSIS .....</b>		<b>37</b>
4.0	Introduction.....	37
4.1	Data from calculation and simulation .....	38
4.1.1	Centrifugal blower Type A.....	40
4.1.2	Centrifugal blower Type B.....	44
4.1.2	Centrifugal blower Type C.....	47
<b>CHAPTER 5.....</b>		<b>50</b>
<b>CONCLUSION AND RECOMMENDATION.....</b>		<b>50</b>
5.0	Introduction.....	50
5.1	Conclusion and Recommendation .....	51
<b>REFERENCE .....</b>		<b>52</b>

## LIST OF FIGURES

Figure 1: Flowchart .....	6
Figure 2: Projection view of the centrifugal blower (Tae Lee et al., 2016) .....	9
Figure 3: Two volute shapes with different tongue(Tae Lee et al., 2016) .....	9
Figure 4: Conceptual view of a few types of ribs shapes (Tae Lee et al., 2016).....	10
Figure 5: Conceptual view of the fan (Tae Lee et al., 2016).....	11
Figure 6: Schematic diagram of the fan structure (Chunxi et al., 2011) .....	12
Figure 7: Comparison of blade after extended (Chunxi et al., 2011).....	13
Figure 8: CFD solver result for impeller model (Hudson et al., 2013) .....	14
Figure 9: Pressure distribution for optimized model (Hudson et al., 2013) .....	16
Figure 10: Velocity distribution for optimized model (Hudson.....	16
Figure 11: Flowchart of the methodology .....	18
Figure 12: Radius of impeller.....	19
Figure 13: Isometric view of casing .....	20
Figure 14: Dimension of the casing.....	20
Figure 15: Save the part as IGES format.....	21
Figure 16: Drag out the Fluid Flow (CFX).....	24
Figure 17: Right click on column Geometry and import the geometry .....	25
Figure 18: Rename the part at tree outline .....	25
Figure 19: The part show in the geometry.....	25

Figure 20: At the Mesh column right click and click edit .....	26
Figure 21: Name selections .....	26
Figure 22: Generate Mesh .....	27
Figure 23: Double click at setup.....	28
Figure 24 : Meshing (Impeller) .....	28
Figure 25 : Meshing (Casing).....	29
Figure 26: Type of run and after choose click start run.....	29
Figure 27: Outline of the setup.....	30
Figure 28: View after setting all the setup.....	30
Figure 29: Double click on solution .....	31
Figure 30: Momentum and mass graph .....	32
Figure 31: Heat Transfer graph .....	32
Figure 32: Turbulance graph .....	33
Figure 33: The data of the simulation.....	33
Figure 34: Double click on the result .....	34
Figure 35: Outline.....	35
Figure 36: The velocity streamline.....	36
Figure 37: Pressure view contour .....	36
Figure 38: Momentum and Mass (Type A).....	41
Figure 39: Turbulance (Type A).....	42
Figure 40: Data recorded (Type A) .....	42
Figure 41: Average data (Type A).....	43
Figure 42: Momentum and Mass (Type B) .....	45

Figure 43: Turbulance (Type B).....	45
Figure 44: Data recorded (Type B).....	46
Figure 45: Average data (Type B).....	46
Figure 46: Momentum and Mass (Type C) .....	48
Figure 47: Turbulance (Type C).....	48
Figure 48: Data recorded (Type C).....	49
Figure 49: Average data (Type C).....	49

## LIST OF TABLES

Table 1: L9 orthogonal results (Hudson et al., 2013).....	15
Table 2: Type of dimension.....	19
Table 3: Dimension of the casing.....	21
Table 4: Volume and surface area of Type A.....	40
Table 5: Statistic of Type A.....	41
Table 6: Volume and surface area of Type B.....	44
Table 7: Statistic of Type B.....	44
Table 8: Volume and surface area of Type C.....	47
Table 9: Statistic of Type C.....	47

## LIST OF ABBREVIATIONS

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



## 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

# 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.

## **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.

## **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.

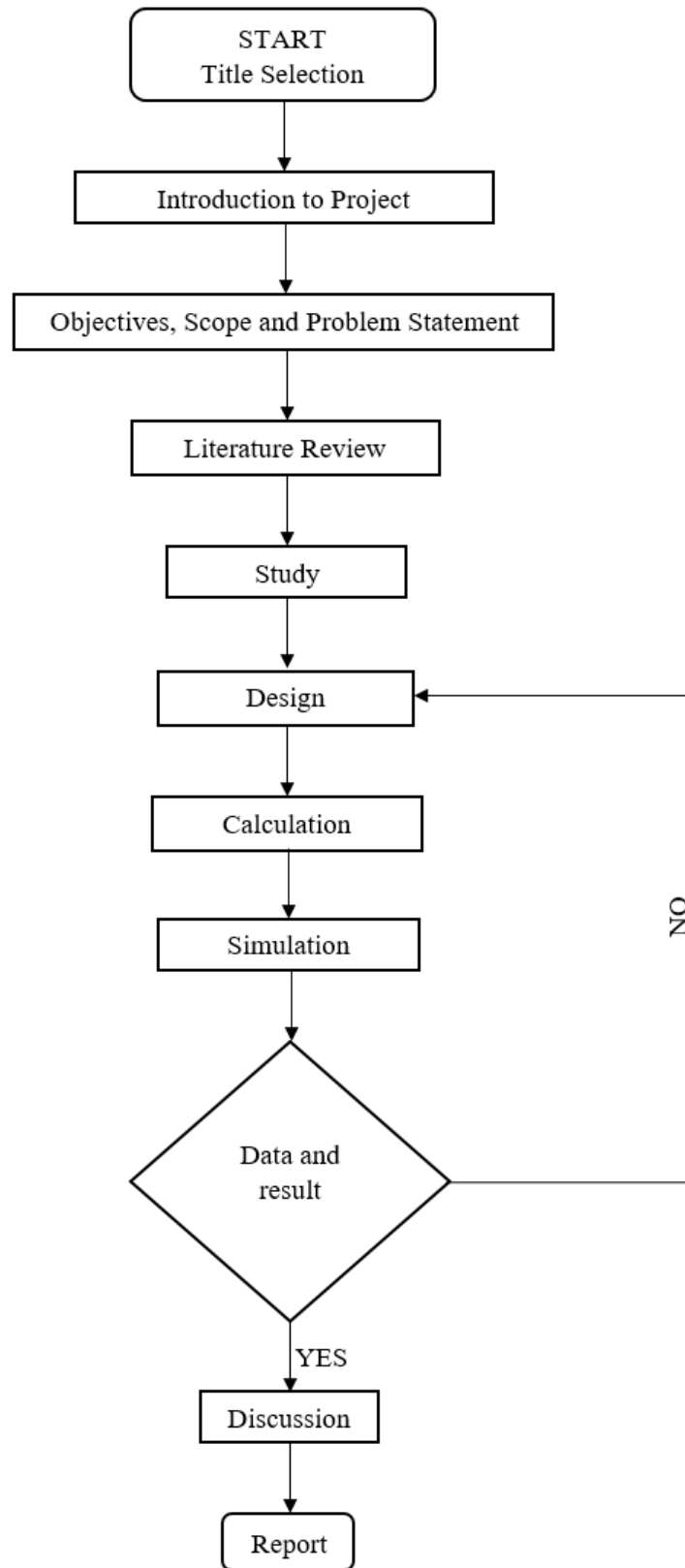


Figure 1: Flowchart

## **CHAPTER 2**

### **LITERATURE REVIEW**

#### **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.