‘I hereby declare that I have read this dissertation and found its content and form to meet acceptable presentation standards of scholarly work for the award of

Bachelor of Mechanical Engineering (Thermal-Fluid)’

Signature :

Name of Supervisor I :

Date :

Air flow distribution of a fan using CFD method

NASRUL HAKIM BIN KHALID

A report submitted in partial fulfillment of the

Requirement for the award of the degree of

Bachelor of Mechanical Engineering (Thermal-Fluid)

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

APRIL 2009

I declare that this thesis entitled *"Air flow distribution of a fan using CFD method"* is the result of my own research except as cited in the references.

Signature	: ....
Name of Author	: ....
Date	: ....

To My beloved mom and dad

For their endless love and support

ACKNOWLEDGEMENT

Alhamdulillah, all praise to Allah, the Most Beneficent and the Most Merciful, who has taught what I knew or not. It is by the grace of the Almighty Allah that this project has been completed successfully.

A deepest appreciation is dedicated to Mr. Shamsul Bahari Bin Azraai as my project supervisor for his extraordinary patience and his enduring optimism. I really admire his knowledge, intelligence and patience. I do appreciate his dedicated guidance, suggestion, critical comments and warm support which have given me the opportunity to develop my project skills. It would have been very difficult to complete this project without the enthusiastic support, insight and advice given by him.

To all my friends or people, who directly or indirectly contribute to this project, I thank all of you for your encouragement, continued support and friendship.

Finally, my sincere gratitude goes to my beloved family especially my parents, Khalid bin Ismail and Aunah binti Mior Yahya for their love, endless encouragement and D'oa. They are truly my inspiration.

ABSTRACT

This paper presents a Computational Fluid Dynamics modeling simulation of air flow distribution from a fan at different angle of blade. A fan consists of a rotating arrangement of vanes or blades which act on the air. Fans produce air flows with high volume and low pressure, as opposed to a gas compressor which produces high pressures at a comparatively low volume. An introduction to the nature of the physical fan air flow problem and its significance was elaborated in order to understand the complications involved in the research and thereafter arrive at the objectives. Improper design of blade will affect the performance of cooling process for household electrical fan. To overcome this problem the optimal angle of blade is determine for enhancing heat transfer rate. Knowing the physical situation is crucial in the application of Computational Fluid Dynamics to numerically model and thereby analyze the simulation by using pre-processor GAMBIT 2.3.16 and solver FLUENT 6.3.26. The tasks undertaken to model the geometries of the fan and its surrounding is the first important step. This is followed by meshing and defining the boundary conditions before numerically solving the variables that represent flow fields of the simulation. The numerical redactions of the variables in the form of velocity vectors and contour plots detailing the flow characteristics are then analyzed, compared and verified according to known physical situation and existing experimental data. In order to increase the flow rate and the static pressure of the axial flow fan, the 26° to 30° of blade angle should be adopted. This study has shown that the Computational Fluid Dynamics simulation can be useful tool in optimizing the design of the fan blade angle.

ABSTRAK

Kajian ini membentangkan pemodelan simulasi pengedaran aliran udara dari bilah kipas yang berbeza sudut dengan mengunakan *Computational Fluid Dynamics* .Pegenalan sifat dari malasah keadaan aliran udara pada kipas signifasinya telah dihuraikan di dalam rangka bagi memahami komplikasi yang terlibat dalam penyelidikan . Kipas menghasilkan aliran udara dengan kelajuan tinggi dan tekanan rendah, berbeza dengan kompresor gas yang menghasilkan tekanan tinggi pada kelajuan yang relatif rendah. Pengenalan sifat dari masalah aliran udara kipas fizikal dan signifikansi yang telah dihuraikan dalam rangka memahami komplikasi yang terlibat dalam penyelidikan dan selanjutnya tiba di tujuan. Rekabentuk yang tidak sesuai dari bilah kiaps akan mempengaruhi prestasi proses pendinginan untuk rumah tangga kipas elektrik. Untuk mengatasi masalah ini sudut optimum bilah kipas ditetukan untuk meningkatkan laju perpindahan panas. Mengetahui keadaan fizikal sangat penting dalam pelaksanaan *Computational Fluid Dynamics* untuk model berangka dan dengan demikian menganalisis simulasi dengan menggunakan pra-prosesor GAMBIT FLUENT 2.3.16 dan *solver* 6.3.26. Tugas yang dilakukan untuk model geometri dari kipas angin dan sekitarnya merupakan langkah penting pertama. Ini diikuti oleh meshing dan mendefinisikan keadaan batas sebelum penyelesaian berangka berbeza yang mewakili bidang simulasi aliran. Suntingan berangka yang berbeza dalam bentuk vektor kelajuan dan plot *contour* menerangkan ciri-ciri aliran ini kemudian dianalisis, dibandingkan dan disahkan sesuai dengan keadaan fizikal dikenali dan data eksperimen yang ada. Dalam rangka meningkatkan laju aliran dan tekanan statik kipas aliran axial, 26 ° hingga 30 ° dari sudut kipas harus dipertimbangkan.kajian ini menunjukkan bahawa simulasi Dinamika bendalir Pengkomputeranadalah alat yang sesuai dalam mengoptimumkan rekabentuk sudut bilah kipas.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF SYMBOLS

Cp	=	specific heat of air, *J/kg.K*
$\dot{m}$	=	mass flow rate of air, *kg/s*
ΔT	=	desired air temperature differential (cabinet to ambient outside air), *K*
°	=	Degree
Q	=	air flow rate, m^3/s
ω	=	angular velocity
r	=	radius
θ	=	angle
$\vec{v}_i$	=	Initial velocity
Pa	=	Pascal
V	=	Velocity, m/s

LIST OF ABBREVIATIONS

CFD	=	Computational Fluid Dynamic
2D	=	Second Dimensions
3D	=	Third Dimensions
AOA	=	Angle of attack

CHAPTER I

INTRODUCTION

1.1 Project Background

A fan is an apparatus that converts electric energy into aerodynamic energy. Some of this energy is useful to other output energy is wasted energy like the air swirl at the fan exit. The fan was basically a blade attached to an electric motor. Mechanically, a fan can be any revolving vane or vanes used for producing currents of air. Fans produce air flows with high volume and low pressure, as opposed to a gas compressor which produces high pressures at a comparatively low volume. A fan blade will often rotate when exposed to an air stream, and devices that take advantage of this, such as anemometers and wind turbines often have designs similar to that of a fan. Using fans to force air having the proper temperature and relative humidity through a crop is a valuable technique for maintaining quality after harvest. The air helps maintain the moisture, temperature, and oxygen content of a crop at levels that prevent growth of harmful bacteria and fungi and excessive shrinkage.

The aerodynamics of fan does not lend itself readily to mathematical analysis and there are no straightforward methods to predicting the air flow around the fan. The aerodynamic of fan research is very important due to promote optimum efficiency. The constant air flows are needed for greater fan performance and reduction in wind noise level.

The design of the fan and its blade type can greatly affect efficiency and power requirements. Laboratory-measured peak fan efficiency may not be the most stable point of operation. If peak efficiency coincides with the peak of the pressure curve then there may be operational problems as volumetric flow rates vary with small changes in system pressure. The designer must consider both curves when selecting the best fan and operating point to optimize reliability and power usage. And fan type may dictate proper selection. Airfoil wheels, while more efficient, may not be a good choice when handling particulate-laden air.

(Gerry Lanham. There's Gold in the Air Power Equation. Retrieved September 10, 2009 from http://www.mfrtech.com/articles/2289.html).

1.2 Problem Statement

Identifying the cause of a fan problem can be difficult due to the wide range of fans and applications as well as the operating points for fans. An air moving device operable to generate a flow of air from a low pressure region to a high pressure region comprising at least one blade operable to generate said flow of air as a result of movement of said at least one blade, wherein said at least one blade includes a rough surface on a side facing said low pressure region and wherein said rough surface is arranged to induce a turbulent boundary layer that enables operation of said air moving device in a manner that would otherwise result in separation of air from said at least one blade. Fan delivers air in an overall direction that is parallel to the fan blade axis.

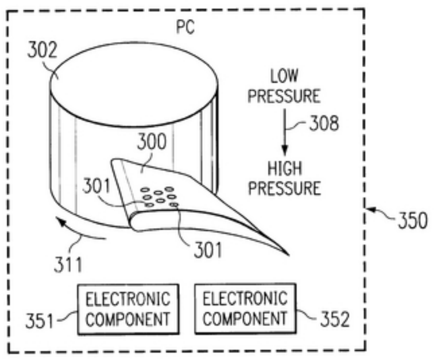

Figure 1.1: The pressure distribution through fan blade
(Source: http://www.freepatentsonline.com/4969799.html)

Fan problems generally fall into two categories which there are mechanical or air performance problems. To solve these problems, an extensive unsteady aerodynamic study is in high demand. Because the fan efficiency and corresponding loading are caused by an aerodynamic behavior which relates to the fan performance, the underlying fluid dynamics must be well understood to accurately determine the geometry and to improve fan efficiency. In order to get the optimum efficiency of a fan air flow rate the design of louvered angle of a blade are important because this part can make difference performance on axial fan. To solve these problems, an extensive unsteady aerodynamic study is in high demand. Because the fan efficiency and corresponding loading are caused by an aerodynamic behavior which relates to the fan performance, the underlying fluid dynamics must be well understood to accurately determine the geometry of blade and to improve fan efficiency

1.3 Objective of the project

The objective of this project was to determine the air flow rate at different angle of blade.

1.4 Scope of the project

The scope for this project includes:

i. To apply the application of CFD fan method- tests the design in computational fluid dynamics (CFD) to measure the drag and lift force. CFD also can determine the aerodynamics flow that the places have turbulence and laminar flow to get the best aerodynamics flow. The software that used in process is Gambit 2.3.16 and Fluent 6.2.26.

ii. To construct the CFD fan geometry-design fan blade consist louvered angle.

iii. To simulate numerically air flow through the louvered angle-collect result and data to observe simulation of air flow

iv. To analysis the results.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Literature review is about to understand the theory of the research. The literature review is very important part to understand first in any research. Since we know the theory of the research, we can proceed it easily. It is will be easy to determine the way of the research will be implementing and also the related theory that has used in previous research. The information about the research can be found through the journal, internet, thesis and reference books. In this research case, all the information and theory that related to the fan performance are needed. In this chapter will discuss about previous research about fan performance.

2.2 Previous Research

In previous research, there are some researches related to determine the air flow rate of a fan distribution. The research title experimental data analysis of five more 10000 class fans used in air cooled heat exchangers that has been executed by Seyedsharif Khoshmanesh (2009), The Research is being conducted at Mechanical Engineering Department, Khormoj Azad University, Khormoj-Boushehr, Iran to produce enough pressure to overcome the friction through the bundle and to prepare proper air volume to cool the sour gas and to determine effect of changing blade angle on static pressure, flow rate, efficiency and energy losses in these fans and comparing these energy losses with cascade test result of Hawell [1] (1942). According to Seyedsharif Khoshmanesh (2009), to check the blade load a complete field run test have been done on the fans. The blade angle has been adjusted to an angle of approximately half that called out on the specification or measured on the unit. The draft gauge is connected to as quiescent a spot in the plenum as possible, preferably in the corner of the plenum ahead of the fan. The fans are start and recorded on the chart provided the blade angle and the static pressure indicated. The blade angle is advanced by one or two degrees and these data are recorded again. The blade angle is increased and followed the procedure until the motor is fully loaded. Figure 1 shows the blade angle versus static pressure and power consumption. It will be noted that the static pressure will be consistently increasing with increased blade angle until the blade loading reaches maximum according to Seyedsharif Khoshmanesh (2009) research.

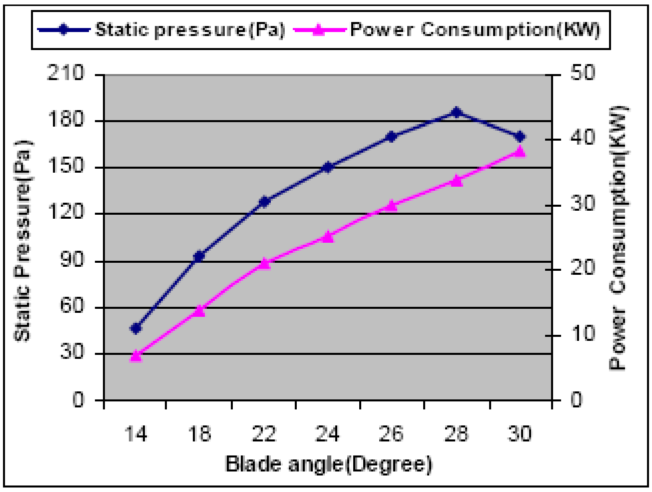

Figure 2.1: Static pressure and power consumption versus blade angle.

(Source: Seyedsharif Khoshmanesh .2009)

The result experimental for the effect of blade angles on flow rate that been done by Seyedsharif Khoshmanesh (2009), shows the blade angle increases the static pressure increase up to maximum value. With more increased in blade angle the static pressure start to drop off sharply. So long as airflow over the blade is smooth and clings to the surface of the blade the little turbulent is present. With increasing the blade angle the air flow breakaway from the convex side of blade. With increasing the blade angle the power consumption and flow rate will increase but in stalling flow while the power consumption increase the increasing rate of the flow decrease and even in some point stop and remain fixed so it causes to reduce the efficiency in stalling flow.

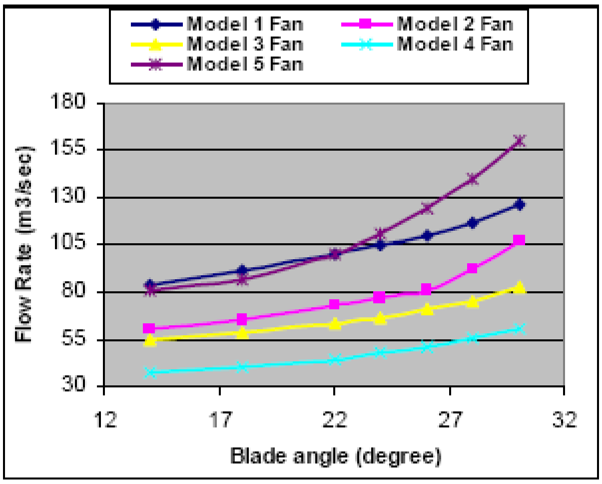

Figure 2.2: Flow rate vs. blade angle for different models of fans.

(Source: Seyedsharif Khoshmanesh.2009)

Form experiment effect of blade angle on efficiency that been done by Seyedsharif Khoshmanesh (2009), show the change in kinematics energy is small so the input power to fan consumes to increase the pressure in reversible case and result from Figure 2.3 show that the maximum efficiency obtains in the range of 22 to 26 degree of blade angle. The efficiency can be defined as the power needed to produce the pressure change to power absorbed in real case that produce the same change in static pressure,