AERODYNAMIC DESIGN AND OPTIMIZATION OF AN ENERGY EFFICIENT CEILING FAN

MUHAMMAD AAQIB AFAQ

RESEARCH CENTRE FOR MODELING & SIMULATION

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY

2013

AERODYNAMIC DESIGN AND OPTIMIZATION OF AN ENERGY EFFICIENT CEILING FAN

MUHAMMAD AAQIB AFAQ

Research Centre for Modeling & Simulation

A thesis submitted to the National University of Sciences & Technology

in partial fulfillment of the requirement for the degree of

Masters of Science

2013

STATEMENT OF ORIGINALITY

I hereby certify that the work embodied in this thesis is the result of original research and has not been submitted for a higher degree to any other University or Institution.

_______________	_________________________
Date	Muhammad Aaqib Afaq

Dedication

This thesis is dedicated to my grandmother (Late) and parents who have always been a source of inspiration for me ☺.

Acknowledgements

Put your heart, mind, intellect and soul even to your smallest acts. This is the secret of success. ~ Swami Sivananda

I would never have been able to finish my dissertation without the guidance, love and blessings of Allah Almighty. Who blessed me with good life.

I would like to express my deepest gratitude to my advisor Dr. Adnan Maqsood, for his excellent guidance, caring, patience and providing me with an excellent atmosphere for doing research. He was always like a brother and his friendly behavior shows me the best path not only in academics but also in my social and professional life.

I am highly grateful to my parents and my Family, without their love and prayers it was impossible for me to complete my studies. They make me smile every time whenever I am frustrated in my life☺.

After the whole day hectic work, a cup of tea with my best friends and research fellows Arooj Fatima and Ammar Sohail provides me the solution to my problems. I really enjoyed there company☺. I would like to thank Faqeed Haider for making me addicted to desi food available in Rawalpindi and Islamabad. Thanks to my roommate Fahad Azad for providing me the homely atmosphere in hostel.

I would like to thank my lab mates and seniors Nafees Mumtaz, Ammar Mushtaq, Junaid Ahmad Khan, Nabeel Qazi and Aqib Chishty for their guidance.

I would like to thank Dr. Khalid Parvez, Engr. Sikandar Hayat Mirza, Dr. Imran Akhtar and Dr. Ahmed Ejaz Nadeem for providing me the facilities and guidance.

I would like to thank Malik Izhar Awan owner of Star Co Fan, Gujrat for their hospitality and helping me to collect experimental data.

Summary

Ceiling fans are the most used resource for providing indoor comfort in the hot climates because of factors like low cost, easy availability/maintenance and less electric consumption compared to air conditioning units. The fan industry of Pakistan is well-renowned on the national scale. This sector is also earning significant foreign exchange through its exports. However, the global share of Pakistan in international market is merely 1.3%. Therefore, a huge potential exists in enhancing Pakistani exports through improving global competitiveness and improving Research & Development infrastructure. Currently there is no design setup related to fan-blades in Pakistani fan-industries. The existing blades are developed through trial and error method and are mostly sub-optimal designs. Therefore, the potential of improving design of ceiling fans warrants detailed investigations.

The objective of the research is to understand the flow field of ceiling fans inside the room and carry out parametric study of several design parameters. A hybrid methodology is used that entails design and optimization through rigorous computational techniques duly validated by experimental data for the baseline geometry. The facility at StarCo fans, a subsidiary of Pakistan Electric Fan Manufacturers Association (PEFMA) is used for experimental testing. A commercial Computational Fluid Dynamics package is used for design investigations. The parameters studied in the fan blade geometry for the purpose of design improvement are rake angle, bent angle and bent position.

The results indicate that the fan efficiency can be increased by increasing the rake angle by one degree from the baseline geometry. Moreover, the bent position, if varied from baseline case to proposed modification will result in the 18% reduction in revolutions per minute without penalizing any important parameter. The results will help PEFMA to meet Pakistan Standard – 1 (PS-1) certifications.

Table of Contents

List of Figures

List of Tables

CHAPTER 1

Introduction

1.1 Background

Thermal comfort during summers is generally done using either air conditioning or air circulation through fans. Humans have a long history of using fans as trivial source of thermal comfort in hot climates. A punkah [1] was a type of fan frequently used in subcontinent and its usage is traced back to 500 B.C [2]. It was primarily a hand-held fan as shown in Figure 1-1.

.

Figure 1-1 Punkah

In 1882, Philip H. Diehl (January 29, 1847 – April 7, 1913) [3], a German-American mechanical engineer gained fame for inventing the first electric ceiling fan. He engineered the electric motor of Singer sewer machine and used the same technology in the ceiling fan. He attached the blades to the Singer electric motor and hanged to the ceiling completely powered by electricity. This laid the foundation of first electric powered ceiling fan [4].

In the late 1890s to the early 1920s [5] domestic electric fans made of brass blades were first sold in USA. In the 1920s, to make the fans cost effective and allowing home-owners to easily afford them steel was used to make fan blades. In 1930s, first art deco fan named as swan fan was designed [5]. In the 1950’s fans were manufactured with bright eye catching colors. In 1960s, with the arrival of air conditioning systems, ceiling fans fell out of favor in USA but they were still getting popularity in the hotter regions of the world[3]. During the energy crises situation in America in 1970s[6], ceiling fans business again got a boost because of cost effectiveness. The ceiling fan manufacturing companies marketed the product throughout the world. The people realized that it was much cheaper to use ceiling fan than to use the new not so cost effective and not so energy efficient ceiling fan. In 1970s, Victorian style ceiling fans started to became popular.

In 1986, Ronald John Rezek (born October 31, 1946, Oakland, California, USA) designed the first contemporary styled ceiling fan. In 2008, he launched a new company named as “The Period Art Fan Co.” after designing several numbers of ceiling fans whose theme is now globally showcased [7] [8] [9].

In twenty first century, the Coanda effect bladeless fans (Figure 1-2) are introduced but still they have not gained popularity [10] because of poor flow circulation in the room.

Figure 1-2 Bladeless Ceiling Fan [10]

A brief timeline snapshot of the historical perspective discussed above is shown in Figure 1-3.

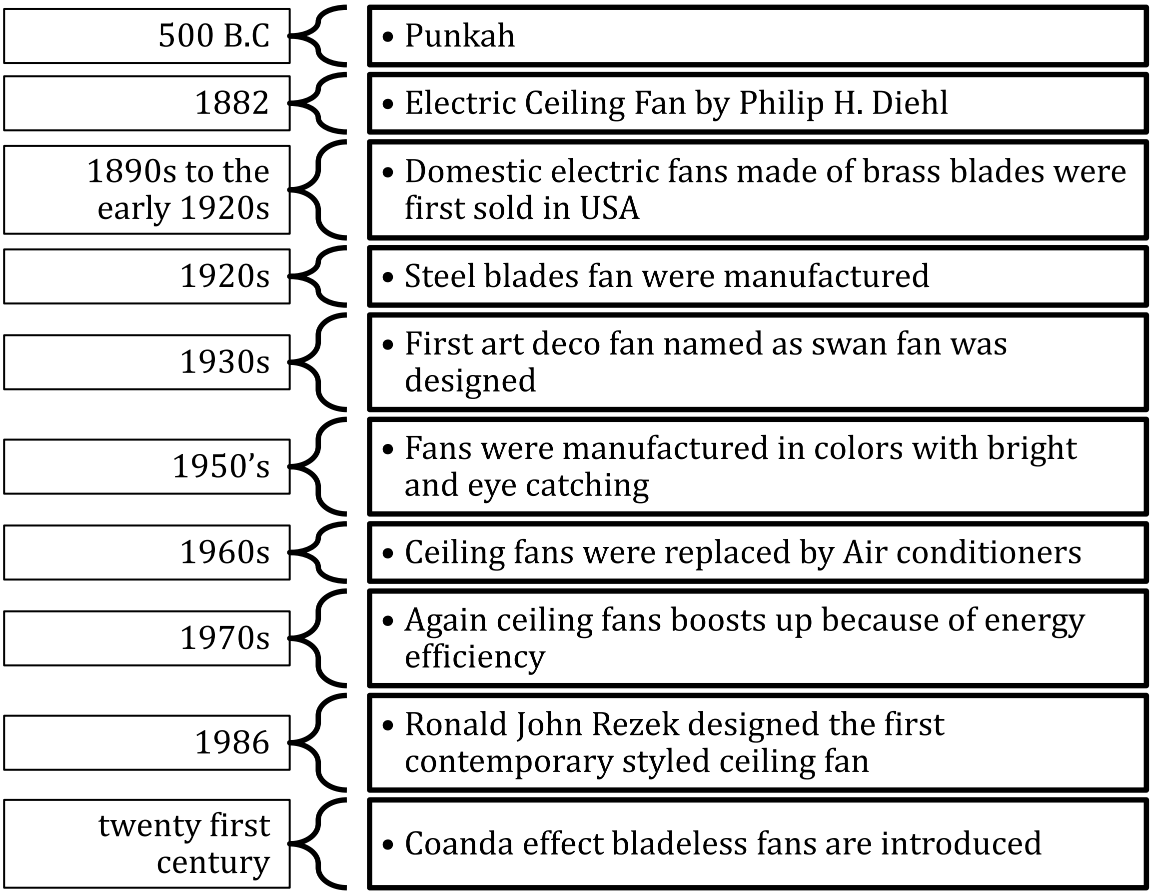


Figure 1-3 Timeline for Design Evolution of Ceiling Fans

In this chapter, the background and the area of research are classified. The objectives, scopes and methodology used in the research are also briefly described. A systematic outline of the report is given at the end of the chapter.

1.2 Area of Research

1.2.1 Fan Industry of Pakistan

The fan industry of Pakistan is well-renowned on the national scale and earning significant foreign exchange through its exports. Pakistan exported fans worth US$ 34.311 million in 2009-2010. The export growth in 2010-2011 was increased by 10.9% with total worth of US$ 38.046 million [11]. Gujrat, Pakistan, a hub of fan industry hosts about 300 Small and Medium Enterprises (SME). However, the downside is that Pakistan export share in global exports is only 1.3%. Pakistan is unable to capture the markets of first world because of multiple reasons that will be discussed in this thesis.

1.2.2 Energy Optimization through Design Refinement

The basic design of ceiling-fan blades has changed little over the years. Currently Pakistan fan industries have no design setup of fan blades. The existing blades are developed through trial and error method and are mostly sub-optimal or old designs. An efficient fan-blade can help to improve the air quality in the room. It has been shown experimentally [12] that significant amount of energy can be saved while maintaining the same air quality standards by changing the profile of the fan-blade. Therefore in this research endeavor, efforts are made to develop a profile of the fan-blade that can provide same air quality at reduced revolutions per minute. This will help the fan manufacturers of Pakistan to develop energy efficient fans and capture bigger share in international export market.

1.2.3 Experimental Studies

The purpose of experimental studies is to collect benchmark data for validation and verification of computational setup. For this purpose, a setup at Star Co Fans Gujrat, a subsidiary of Pakistan Electric Fans Manufacturing Association (PEFMA), is used. Four Different designs of ceiling fans are tested. The fan having best performance parameters is selected for further design improvement and optimization. The instruments used for the experimental data collection includes vane anemometer, digital inclinometer, bevel protractor, tachometer, thermometer and stop watch.

1.2.4 Computational Fluid Dynamics Studies

A finite-volume based commercial Computational Fluid Dynamics (CFD) code is used for computational modeling and detailed investigations. Specifically, ANSYS FLUENT ® software package is used in these studies. The flow field around ceiling fans is modeled as a three-dimensional steady state incompressible case. Reynolds-Averaged-Navier-Stokes (RANS) system of equations with an appropriate turbulence model (Spalart Allmaras) is solved using the coupled-implicit formulation. The hardware used for simulations is HP core i-7 with 16 GB of RAM.

For validation studies, the experimental data was compared with the computational data. The design variables of the ceiling fan geometry are characterized and studied for the refinement purposes. Specifically, the rake angles, bent angles and bent positions are varied.

1.3 Research Objectives

The objectives expected to be achieved in this research endeavor are:

- Understand the flow field of fan blade through rigorous modeling and simulation techniques.
- Characterization of fundamental design parameters in fan blade geometry.
- Carry out parametric study of design parameters.
- Optimization and development of fan-blade geometry to improve air flow/circulation in room with better energy efficiency (reduced revolutions per minute).

1.4 Methodology

In this thesis, a hybrid methodology is adopted that entails design and optimization through rigorous computational techniques duly validated by experimental studies. A conceptual methodology flow chart is shown in Figure 1-4. After comprehensive literature review and computational setup modeling, experimental data is collected for the baseline geometry. Then detailed parametric studies are conducted and flow analysis of various design configurations is carried out. Finally, in the end the performance of proposed modification is compared with the baseline geometry.

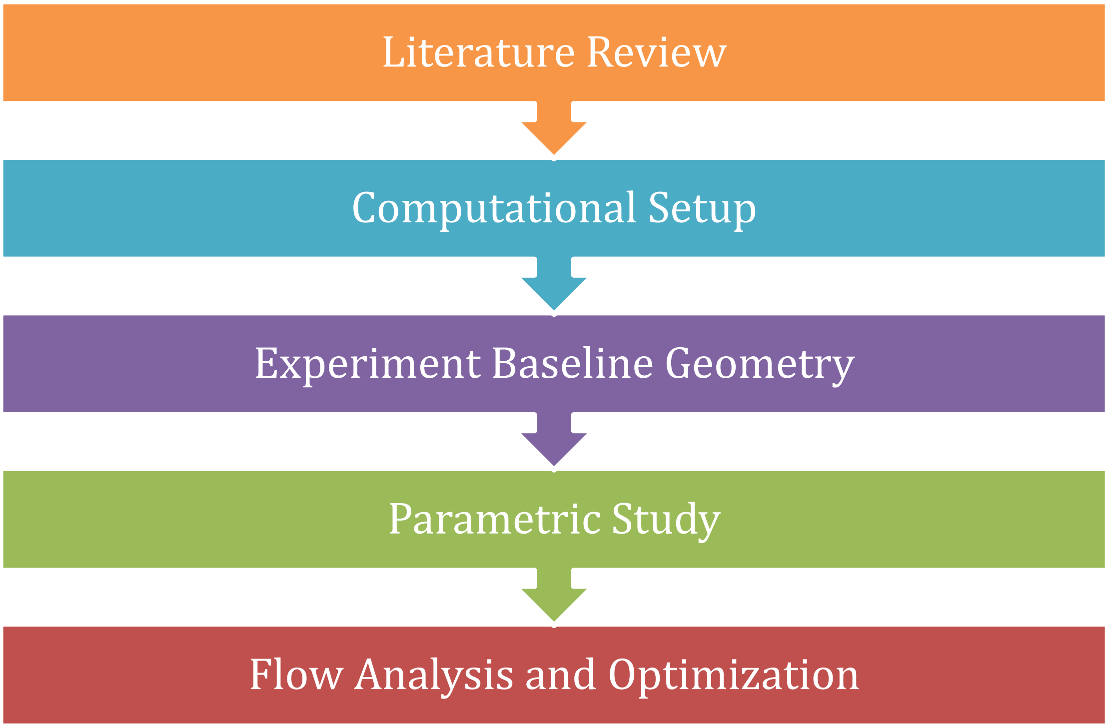


Figure 1-4 Methodology Flow Chart

1.5 Contributions

The investigations carried out during this research are conducted in close collaboration with Pakistan Electric Fan Manufacturing Association (PEFMA), Gujrat. It is envisioned that the outcome of this research will directly help the fan manufacturers of Pakistan to develop energy efficient fans and capture bigger export share in the international market. Following contributions are made to the existing body of knowledge in academia and design refinement through improving energy efficiency to the industry setup of Pakistan.

- Computational modeling using RANS: According to the literature review most of the research on fan optimization is either based on experimental studies through trial and error approach or based on an empirical technique known as linear momentum theory. This thesis holds the honor of seminal contribution to the computational modeling of ceiling fan using a high fidelity CFD algorithm. Specifically Reynolds Average Navier Stokes (RANS) simulations are carried for design refinement and optimization studies using commercial CFD package.
- Design Refinement: The design refinement of the selected ceiling fan frequently manufactured by various Small and Medium Enterprises (SMEs) of PEFMA is carried out with the perspective of energy improvement. Overall 17.81% reduction in RPM is achieved that will directly contribute to less power consumption. The efficiency of the system is enhanced by increasing rake angle and varying bent position from root to tip. It should be noted that the proposed improvement is for the class of 56" diameter ceiling fans.
- The research has contributed in one conference contribution in International Bhurban Conference for Applied Sciences & Technology (IBCAST).
- The outcome of the research from this thesis has generated one journal manuscript.

1.6 Organization of the Thesis

This thesis comprises of seven chapters. The brief outline of each chapter is discussed as follows:

Chapter 1 --- Introduction

In this chapter, the background and the area of research are classified. The objectives, scope and methodology used in the research are documented. A systematic outline of the report is given at the end of the chapter.

Chapter 2 --- Literature Review

A comprehensive summary of the literature study is given in this chapter. Identification of challenges from the literature survey is also presented as the driver for the research.

Chapter 3 --- Experimental Setup and Data Collection

In this chapter, the experimental setup is discussed. The procedure used to obtain the air flow characteristics is described.

Chapter 4 --- Computational Modeling

In this chapter, computational model was developed for flow visualization and further design refinement and optimization studies.

Chapter 5 --- Rake and Bent Angle Studies

In this chapter design and analysis studies for various rake and bent angle configurations is carried out. Initially a three dimensional model was created for validation then modifications were made in the fan blades. The discussion encompasses parametric comparisons.

Chapter 6 --- Bent Position Studies

Based on the rake and bent angle studies, detailed investigations on the variation of bent positions at the root and tip of the blade is carried out. A response surface is generated by computing a total of 16 geometries. The values are compared with the PS-1 standards. After the analysis, an optimal blade configuration is recommended.

Chapter 7 --- Conclusions and Future Work

Conclusions from the current research presented in dissertation are derived and recommendations for future line of action are laid down in this chapter.

CHAPTER 2

Literature Review

2.1 Background

A comprehensive summary of the efforts deployed round the globe for the design improvement of ceiling fans is discussed. The fundamental design parameters are elaborated along with the existing state of Pakistan fan industry. Identification of the missing links from the literature survey are also presented that gives the direction for the future avenues of the research.

2.2 Design Parameters

In this section, the design parameters associated with the ceiling fans are discussed. The experiment to assess the performance of the ceiling fan is carried out in the square room as shown in Figure 2-1.

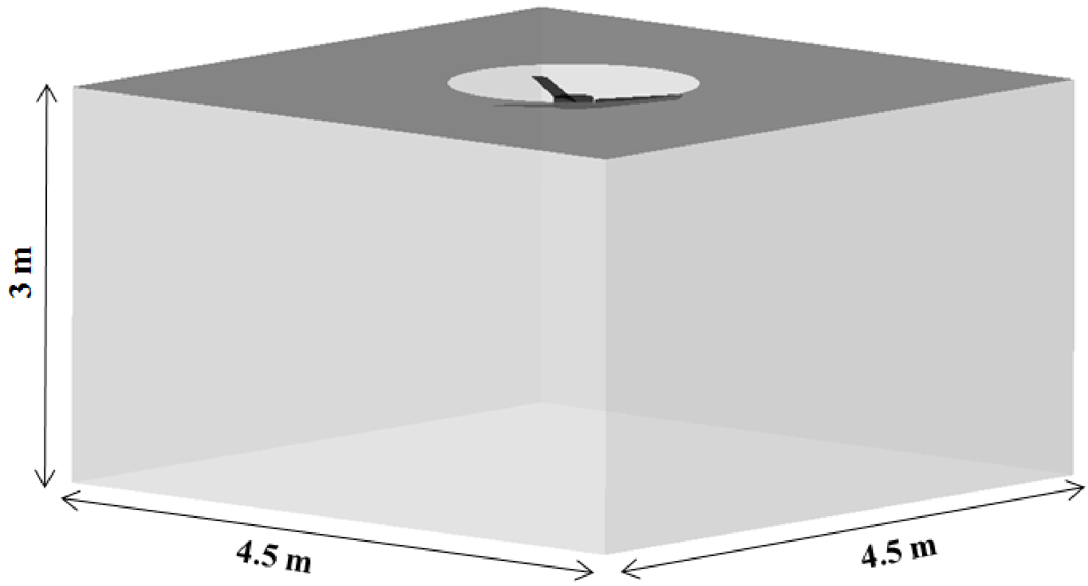


Figure 2-1 Test Room Dimensions

The diagonal view of test room can be seen in Figure 2-2. The dimension of the room is 6.5 m and the fan is installed at the height of 1 m below the ceiling. The distance between the ceiling fan and the floor is 3 m.

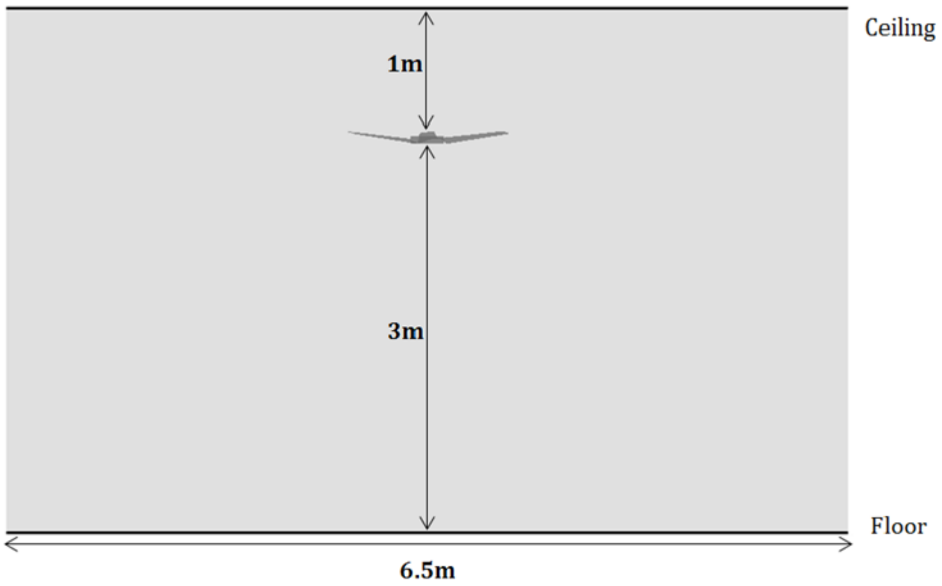


Figure 2-2 Test Room Dimensions

The hub and blade of the fan are shown in Figure 2-3. The central rotating part from which the blades are attached is called fan hub. The purpose of fan hub is to house electric motor. Moreover, the blade section that is attached with the hub is called blade root whereas the other end is called blade tip.

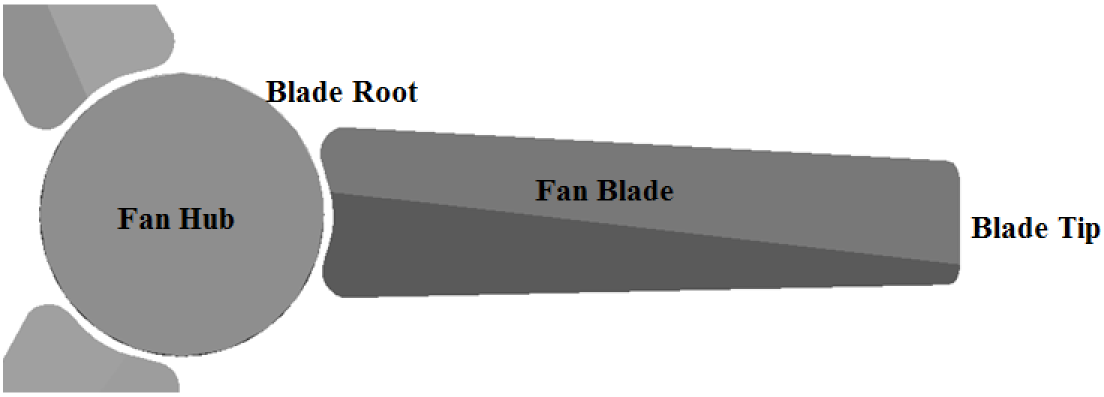


Figure 2-3 Ceiling Fan Components

The ceiling fan used in the research for benchmark purposes belongs to the standard class of 56 inch diameter fan as shown in Figure 2-4. The diameter is based on the calculated circular area swept by the ceiling fan during operation.

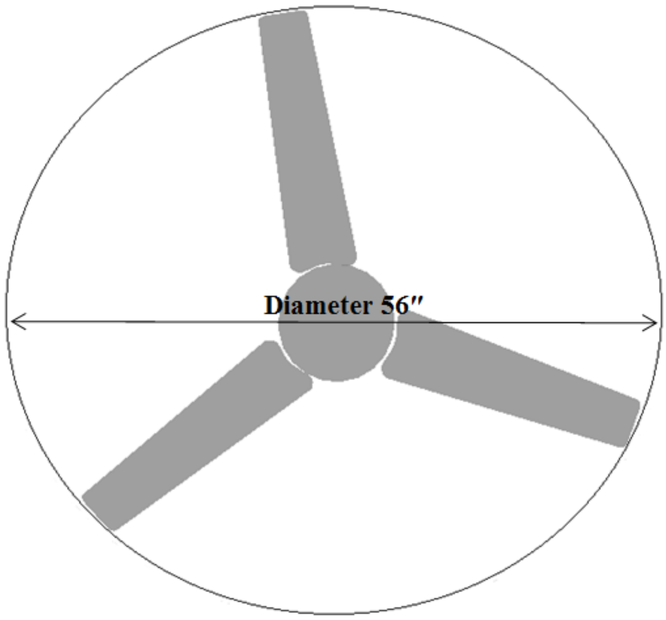


Figure 2-4 Ceiling Fan Diameter

Flow schematic pattern is displayed in Figure 2-5. During the operation, the air is pushed downwards typically perpendicular to the plane of rotation. Moreover near the blade tip, the tip vortices are formed because of pressure leak, As a result, the air from the lower surface swirls on to the upper surface.

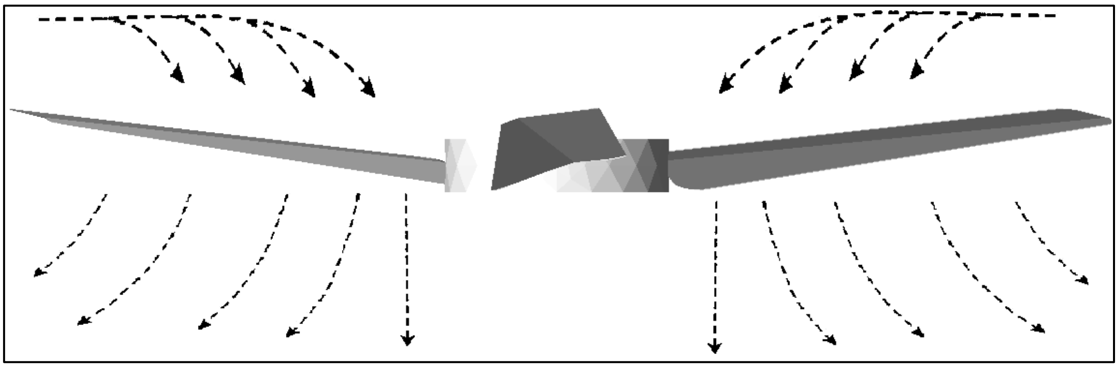


Figure 2-5 Air Flow around Ceiling Fan

The dihedral angle (negative rake) is the angle between a horizontal plane containing the root chord and a plane midway between the upper and lower surfaces of the blade as shown in Figure 2-6 and Figure 2-7. The negative rake spreads the air downwards. If the blade lies below the horizontal plane, it is termed an anhedral angle (positive rake) as shown in Figure 2-7. The positive rake throws the air inwards. If the blade lies in the horizontal plane, it is termed as zero rake angles. Rake angles play an important role in the room air flow. The zero rake angles throw the air vertically downwards as shown in Figure 2-7.

Figure 2-6 Ceiling Fan Rake Angle

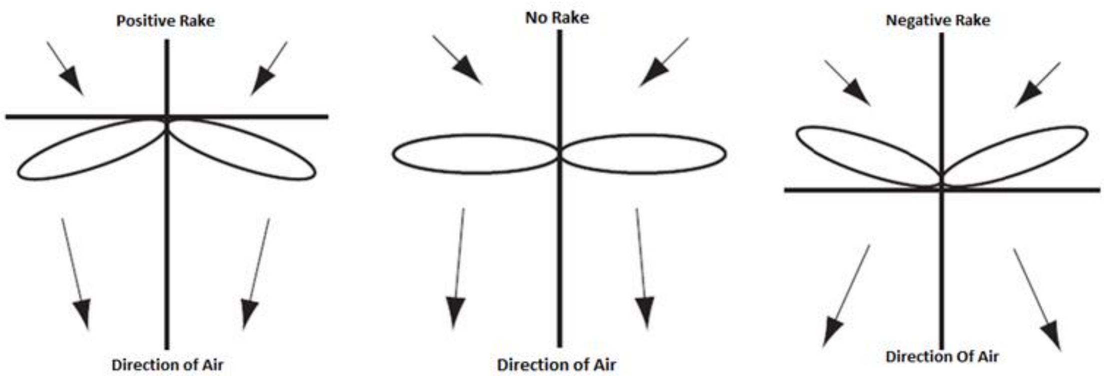


Figure 2-7 Effect of Rake Angle on Air Flow

Bent angle is defined as the angle between two surfaces of blade. The function of this angle is to push more air downwards in the room as shown in Figure 2-8.

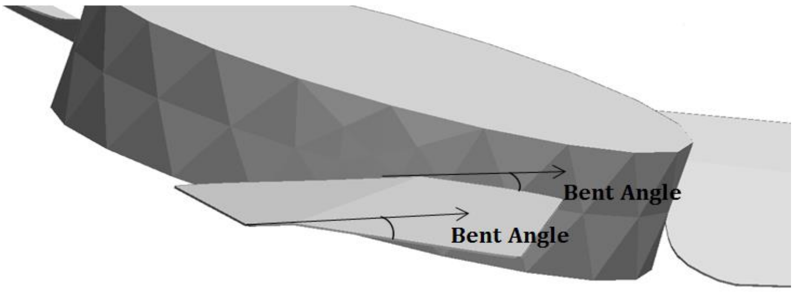


Figure 2-8 Ceiling Fan Bent Angle

Connector angle is termed as the angle between the root chord and hub. The function of connector is to attach hub and blade at specific angle of attack. The cross section of the fan blade is similar to an airfoil as shown in Figure 2-9. Airfoils include leading edge, trailing edge, chord line and mean camber line.

The leading edge is the part of the blade that first contacts the air; alternatively it is the foremost edge of an airfoil section. The trailing edge of an aerodynamic surface such as a blade is its rear edge, where the airflow separated by the leading edge rejoins as shown in Figure 2-10. The chord line is defined as the straight line connecting the leading and trailing edge as shown in Figure 2-9.

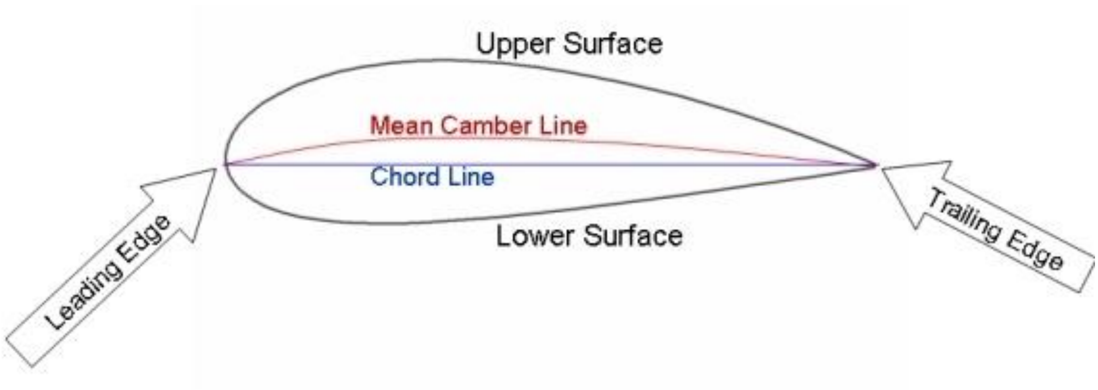


Figure 2-9 Air Foil Crossection

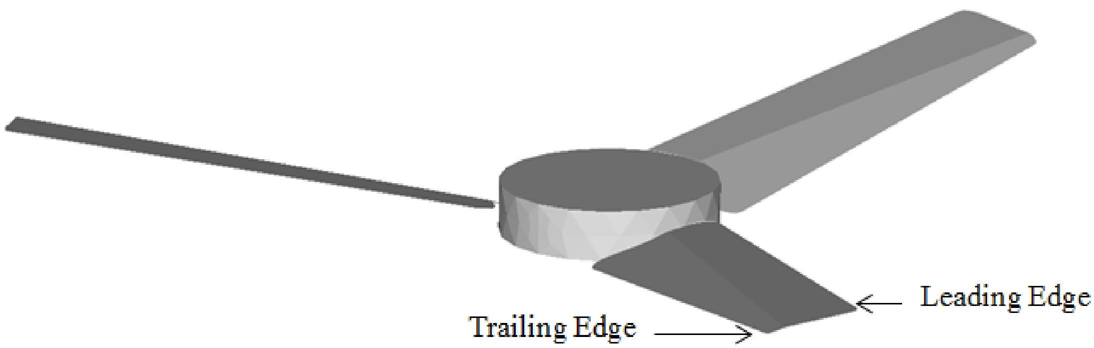


Figure 2-10 Ceiling Fan

2.3 Related Work

In developed countries, the focus of the research is on making air conditioning systems more energy efficient but due to the present energy crises situation in developing countries, most of the people are unable to achieve thermal comfort by air conditioner as they are very costly and consume more energy. As an alternative, ceiling fans are preferable as they have low investment cost and are cheaper in terms of energy consumption. However, a few efforts have been poured to make them energy efficient.

2.3.1 Fan industry setup in Pakistan:

There are 450 small and medium enterprisers (SMEs) fan industries in Pakistan. Out of these, 300 SMEs are housed in Gujrat while the remaining are in Gujranwala, Lahore and Karachi. The annual production of fan units from these factories is about 8 million fans with a total estimated investment of Rs 20 Billion [11]. Fan industry is not only contributing in earning valuable foreign exchange but also in the national economy. This industry provides approximately 30,000 employments to medium and low income families. The average export price of fans made in Gujrat and Gujranwala is around Rs 2300 to 2500. 90% of the produced units are consumed domestically while the rest are exported. The capacity of an average fan manufacturing unit is up to 400 fans per day which cannot be compared with China where average production per day is reported to be about 40,000 fans. The industries in Pakistan are exhibiting this lackluster performance because of shortage of skilled staff, poor infrastructure, inconsistent raw material, low levels of productivity, inadequate technology and poor energy efficiency.

The Pakistan fan industry produces a variety of fans [11] which include, ceiling fan, bracket fan, exhaust fan, pedestal fan, tilting box louvre fan, circomatic fan, louvre pedestal / table fan, louvre bracket fan and ventilation fan. The ceiling fans hold the major share (63%) among the type of produced units. Ceiling fans are further classified based on their utilization as consumer fans and industrial fans. Consumer fans consume less than 125 watts of energy and are categorized under home appliances. On the other hand industrial fan consume over 125 watts of energy.

Pakistan's fan industry mainly deals in consumer fans. Fan export is a significant proportion of Pakistan's economy. Pakistan exported ceiling fans worth US$ 34.311 million in 2009-2011. The export was increased by 10.9% in 2011-2012 worth of US$ 38.046 million [11] but it is only 1.3% of the global export. Major importers of Pakistani fans were Sudan, Yemen, UAE, Bangladesh and Nigeria.

In order to export ceiling fans, Pakistani fan manufacturers have to meet certain quality standards. Generally all countries have their own standards such as

- UL for the USA
- CE for Europe
- SASSO for Saudi Arabia
- SABS for South Africa
- SIRIM in Malaysia and Sri- Lanka
- SONCAP for Nigeria

Pakistan has also got its standards for home appliances (PS-1). The standards for ceiling fans proposed by the quality assurance regulatory body entails following:

1. Rated Power Input (Watts) = 80 (+10% Tolerance)
2. Rated Air Delivery (m^3/min) = 220 (-10% Tolerance)
3. Minimum Service Value (m^3/min/watt) = 3.12 (No Tolerance)
4. Maximum Speed (RPM) = 330 RPM with Load

Rated power input is the electric power provided to fan motor to rotate and rated air delivery of the fan is the volume of the air on a specific surface inside the room. Service value provided by the fan is the ratio of rated air delivery to power input. The maximum speed for the fan to revolve in one minute is 330 cycles.

2.3.2 Literature Review

The computational work for the understanding of flow features of air by a room ceiling fan can be attributed to Ramadan and Nader [13]. The focus of the study is on numerical simulations of fluid flow. A finite element based commercial code was used to simulate the flow. The results showed that flow pattern has different features as it detached the fan blade and moves toward the floor. The configuration considered consists of a simple ducted fan inside a room. The air flow variation and swirl behavior is investigated. It has a divergence angle of almost 150^0. The computational data is compared with experimental work of Ankur [14]. The authors report in the loss of flow momentum as air moves outsides the fan diameter.

Momoi et. al. [15] experimentally as well as computationally investigated the air velocity measurements around a ceiling fan in a large room. The RPM of the ceiling fan was fixed at 160. He used the boundary condition on the basis of air velocity measurements. These boundary conditions were validated by comparing CFD data and experimentally measured data. The study concluded that distance between ceiling fan and floor has a significant effect on the air flow inside the room and air velocity near the center of the ceiling fan is greater.

Watanabe et. al.[16] used RANS CFD code for the simulation of flow around two conventional propeller of different type. These simulations were executed at non-cavitating and cavitating operating condition. In this study a multiphase flow model was used. This model is based on a “full cavitation model” suggested by Singhal et. al. [17]. This study predicted thrust and torque coefficient values. These values were in good agreement with the calculated values in above mentioned operating condition.

Ankur et al. [14] studied the flow field of a ceiling fan inside an empty room through a series of experiments. Different flow regions in the space through smoke visualization were identified. A comparative study was conducted for the fan blades with and without winglets to improve the air circulation and quality inside the room. The idea was to reduce any induced flow generated at the tips of the fan blades by suppressing the blade tip vortices. As a result of spikes and winglets the downward air velocity was increased by 13% with same power consumption.

Schmidt and Patterson [18] experimentally compared the performance of steel and wooden blade fans. Steel blade was preferred due to their high air quality. New 4 blade prototype was developed with high angle of attack which achieves the rated airflow at 140 RPM as compared to the conventional ceiling fan that shows the same airflow at 270 RPM. An aerodynamic efficient fan blade was developed by incorporating various design features such as blade profile, geometric twist and taper ratio. The study concluded that the new ceiling-fan design can decrease the power consumption by a factor up to three, compared to the conventional ceiling fans. However a parametric study detailing the effect of each design variable on the overall performance was lacking.

Danny [12] investigated the different ceiling fan flat plate models and showed the improvement for the low speed fan through computational studies. New design includes twice the airflow, less vibration, less noise and energy efficiency.

Lee and Kim [19] numerically optimized the low speed axial fan through response surface method. Gradient based search algorithm was used to search the optimal design from the constructed response surface method. They concluded that modification of blade lean is more effective to improve the efficiency than the blade profile.

Falahat [20] made an effort to find the best angle of attack and rotational speed of a flat blade axial fan. Blade angles were varied from 30^0 to 70^0 and rotational speed was varied from 50 to 200 rad/sec. Number of blades was varied from 2 to 6 and come up with the results that 4 blade fans are efficient when angle of attack is varied from 45^0 to 55^0.

Rohles et al. [21] experimentally explained that fan manufactures claims that air of 28^0 C with a running fan will provide same comfort as 24^0 C without a fan. Major comfort producing feature are turbulence and variable characteristics of the air plume. Rohles stated that light objects, loose papers and hairs started to blow at an air speed of 0.8 m/s.

Rohles et al. [22] noticed the effect of using a ceiling fan running below a false high ceiling in an open zone at different heights above the floor. They concluded that ceiling has no effect on air velocities and that air velocity reduced rapidly at almost 40% of the height from the hub.

Son et al. [23] numerically investigated the thermal comfort on a human being standing below the ceiling fan along with an air conditioner inside the room. He showed that without ceiling fan or at low speed of ceiling fan thermal comfort is dependent on the position of the air conditioner's inlet unit but as fan speed is increased thermal comfort is increased throughout the room as it circulates the air flow in the room. They also informed that by the increase of fan speed energy consumption is less than the energy consume by air conditioner. So they concluded that fan air speed is the key factor for thermal comfort and less energy consumption.

2.4 Missing Links in Literature

Many decades have been passed and the basic design of ceiling-fan blades has changed little over the years. Currently there is no design setup related to fan-blades in Pakistani fan-industries. The existing blades are developed through trial and error method and are mostly sub-optimal designs. An efficient fan-blade can help to improve the air quality in the room. It has been shown experimentally that by changing the profile of fan-blade, significant amount of energy can be saved while maintaining the same air quality standards.

To understand the blade design physics and its optimization a computational approach Reynolds Averaged Navier Stokes (RANS) can be used. This approach predicts better flow physics and this systematic way will save the excessive experimental time as well as its cost.

CHAPTER 3

Experimental Setup & Data Collection

3.1 Background

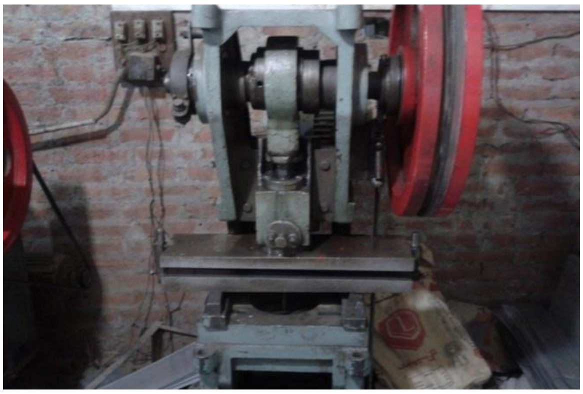
This chapter comprises of dimensions of fan geometry, room geometry, experimental apparatus and its calibration. To collect the experimental data experiments were performed in Star Co Fan industry test room. Profiles of the results extracted from experimental data are included in this chapter. A brief survey of industrial manufacturing process is also described to understand the manufacturing process of fan blades.

3.2 Fan Manufacturing in industry

In Star Co Fan Gujrat, aluminum is used to manufacture fan blades. Pre-cut blade is placed in the drilling machine and three holes are made at root by the drilling machine. Blade is then placed in bent angle machine where bent is given to the blade. Connectors are placed in the drilling machine and different holes are made to screw rivets on hub and blade. Connectors are attached to the blade with the help of rivets. Rake angle is given manually by labor. Blades are segregated through their weight measured by weighing scale sampled in appropriate rakes. Three blades of same weight are attached to a rotating hub 120^0 apart from each other. Weight balancing of fan is done before packing. After successfully passing the weight balancing test, fan is packed for customers as shown in Figure 3-1.

(a) Blade drilling machine

(b) blade drilling



(c) Bent angle pressing machine

(d) Bent angle

(e) Connector drilling machine

(f) Connectors

(g) Attaching blade and connector

(h) Labor giving rake angle

(i) Blade weight measurement

(j) blades in paint section

(k) Blade before and after paint

(l) Blades ready for the weight balancing test

(m) Fan balancing test (n) Blade packing

Figure 3-1 Ceiling Fan Manufacturing Process

3.3 Description of the Fan Geometry

The Star Co Fan Super Delux model shown in Figure 3-2 is selected to get the coordinates of blade profile shown in Figure 3-3. The length of the fan blade is 0.568 m, chord length at root is 0.16 m and 0.115 m at the tip. Aluminum material was used to manufacture blade with the thickness of 1 mm. The diameter of the fan hub is 0.2564 m and total diameter of the fan including the hub was 1.4224 m that categorizes it into 56″ class ceiling fan.

Figure 3-2 Super Delux model

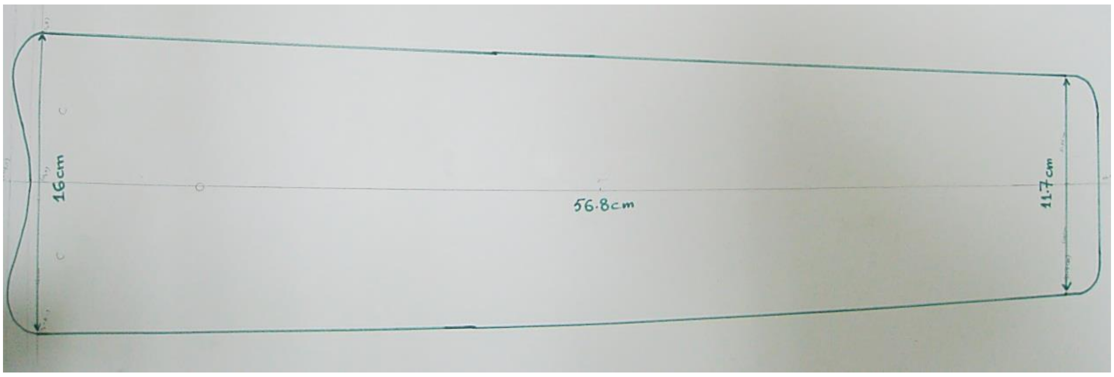


Figure 3-3 Blade measurements

The selected fan consists of 3 aluminum blades with three different angles i.e bent angle of 11.8°, rake angle of 5° and connector angle as 8° measured by digital inclinometer and bevel protector. Bent position is 63% at root chord and 17% at tip chord. All the three blades are at 120° apart from one other. The dimensions of the fan are shown in Table 3-1.

Geometric Attributes	Dimensions
Fan hub diameter	0.2564 m
Fan blade length	0.568 m
Fan diameter	1.4224 m
Fan blade thickness	0.001 m
Fan blade material	Aluminum
Fan blade bent angle	11.8^0
Fan blade rake angle	5^0
Fan blade connector angle	8^0
Fan blade root chord	0.16 m
Fan blade tip chord	0.115 m

Table 3-1 Geometric dimensions of the ceiling fan

3.4 Experimental Setup & Instrumentation

3.4.1 Test room dimensions

Experiment was conducted in the Star Co Fan industry Gujrat test room. It was a square wooden room with length, width and height as 4.5 m, 4.5 m and 3 m respectively. Fan is installed 1m below the ceiling in a wooden duct of diameter 61.6″. Fan distance from the floor was 3m. The geometric dimensions of the room are furnished in Table 3-2.

Geometric Attributes	Dimensions
Room Length	4.5 m
Room Width	4.5 m
Room Height	3 m
Fan Duct	1.6494 m
Fan Distance from Ceiling	1 m
Fan Distance from Floor	3 m

Table 3-2 Geometric dimensions of the room

3.4.2 Instruments for Experiment

The details of the instruments for experimental data collection are described below.

Digital Inclinometer

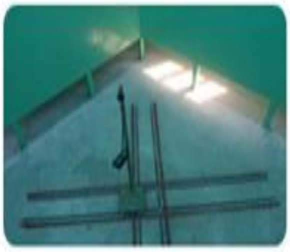
Digital inclinometer is used to measure the blade rake angle, bent angle and connector angle.

Bevel Protector

Bevel Protector is used to measure the blade rake angle, bent angle and connector angle.

Vane Anemometer

Vane anemometer "PROVA AVM-07" is used to capture air velocity magnitude and temperature inside room.



Vane Anemometer Stand

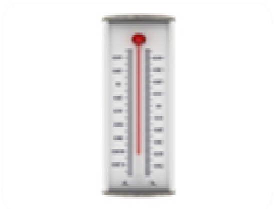
A metal stand is used to carry the vane anemometer at a specific height.

Input Voltage

Input Voltage is varied by the help of regulator to set certain RPM.

Tachometer

Tachometer is used to capture the RPM of ceiling fan.



Thermometer

Thermometer is used to measure the room temperature.

Stopwatch

Stopwatch is used to set time interval for each measurement.

Figure 3-4 Experimental Instruments

3.5 Calibrations

3.5.1 Sensor Calibration

3.5.1.1 Tachometer Calibration

One tachometer was placed above the ceiling fan and the second one was a hand-held infrared tachometer placed just below the fan to compare the fan speed in both instruments. The difference between the two datasets was found insignificant.

3.5.1.2 Vane Anemometer Calibration

Vane anemometer "PROVA AVM-07" and "VA 8020" was used to measure the air speed at 1.5 m above the floor. The difference was found insignificant between two sources. The length of the stand to carry Vane anemometer is set to 1.5 m above the floor.

3.5.1.3 Radial Distance and Centre Point Calibration

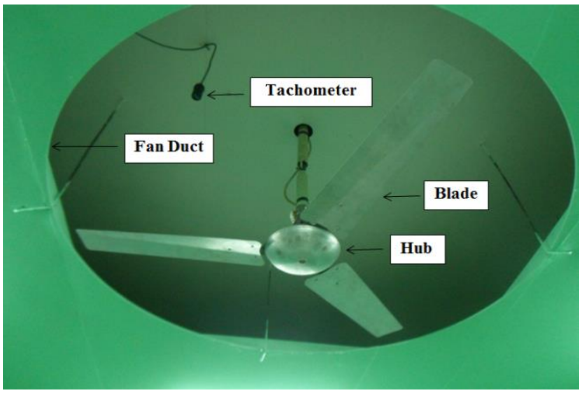
A thread is hanged at the middle of fan hub to mark the center point of the room at floor. Measurement scale lines were marked diagonally on the floor from the center point named as A, AA, B and BB. Fourteen marks are placed on each diagonal line of room. Diameter of Vane anemometer sensor is measured 60 mm so the first point should be marked half the diameter of sensor and afterwards all marks should be marked equal to the diameter of the fan blade. The first point was marked at 30 mm from the center point, the second point was marked 60 mm from the first one and similarly all other marks were 60 mm apart from each other. This point marking operation was done for all four diagonal lines named as A, AA, B and BB. So finally 56 points were marked to collect the air velocity for various fan speed (RPMs).

3.5.2 Room Clearance

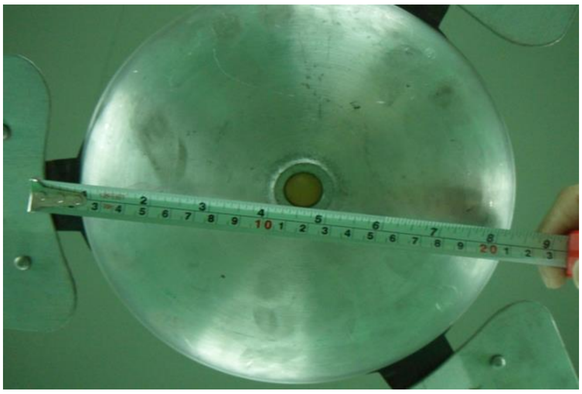
The baseline model is installed in the middle of test room 1 m below the ceiling. A wall clock and thermometer is fixed on side wall. Tachometer was installed above the fan. For the undisturbed flow the test room was emptied from furniture (ladder, chair and table) and windows and doors are perfectly sealed.

3.6 Conducting the Tests

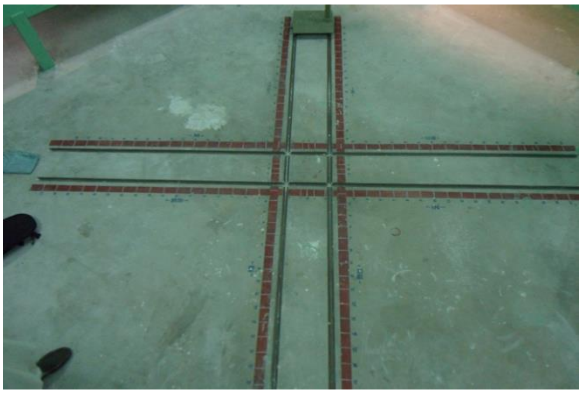
Early in the morning the ceiling fan is switched-on 2 hours before taking the readings for the fully developed flow. The metal stand is placed at the first point on the line A. Seven different RPM’s settings 175, 200, 250, 300, 325 and 365 are set for the measurements one by one. For each reading the anemometer is set so that it can automatically capture the magnitude of maximum velocity in 2 minutes placed on the marked position. After every two minutes the sensor was placed on the next marked point. The input/output voltage and input/output current are shown by electronic display panel. At the end, the fan blades are put on a table and their rake, bent and connector angles are measured for the development of subsequent computational model. A pictorial of the procedure adopted to conduct the experiment is shown in Figure 3-5.



(a) Fan installed in the test room

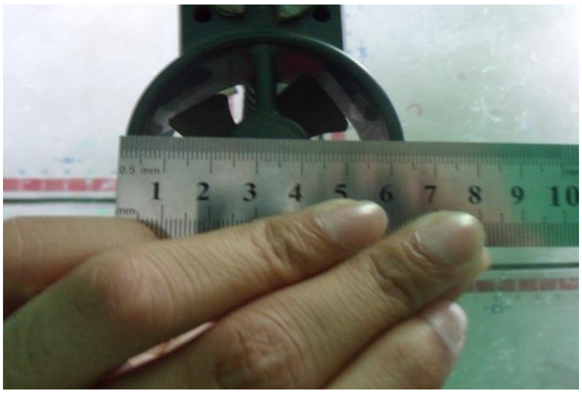


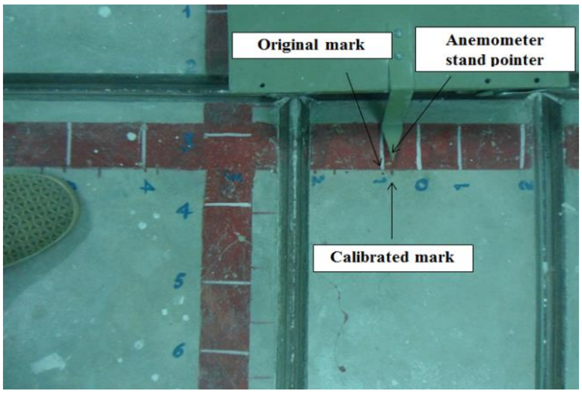
(b) Fan hub diameter



(c) Bent angle measured by Bevel Protector

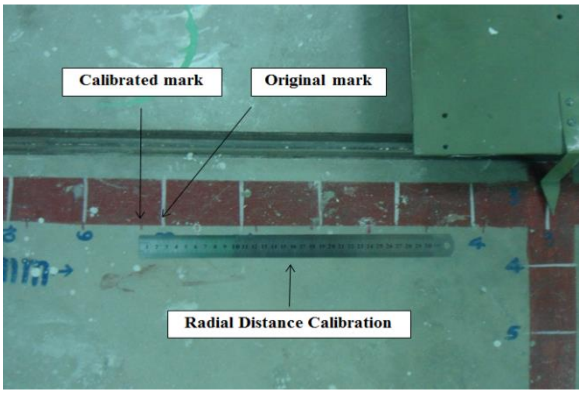
(d) diagonal Floor marks





(e) Diameter of Vane anemometer sensor

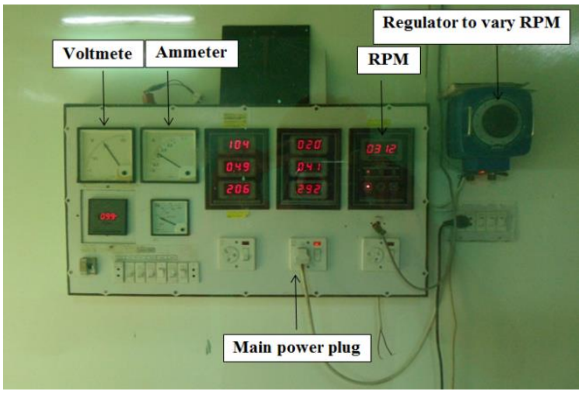
(f) Calibration of radial distance



(g) Calibration of radial distance

(h) Sensor placed below fan

(i) Distance between side walls and floor



(j) Electronic panel

Figure 3-5 Test Room

3.7 Data Collection

Spreadsheets are prepared to collect the experimental data as shown in Table 3-3. For specific RPM power, voltage, current, station number and velocity are recorded. This data is subsequently used in understanding the flow pattern in room and validating the baseline case.

Sr. No	**RPM**	**Power (Watt)**	**Voltage (Volts)**	**Current (Amp)**	**Station (mm)**	**Velocity (m/s)**	**Comment**
1	**175**						
2	**200**						
3	**250**						
4	**300**						
5	**325**						
6	**350**						
7	**365**						

Table 3-3 Experimental Data Collection Chart

3.8 Velocity Profiles

The velocity profile for seven different RPM's is shown in Figure 3-6. The flow patterns clearly show that by increasing the RPM, velocity is increased from fan hub to blade tip. However, for a certain RPM, the maximum axial velocity is recorded from 0.2 m to 0.5 m. After 0.5 m, a sharp decrease in axial velocity is observed.

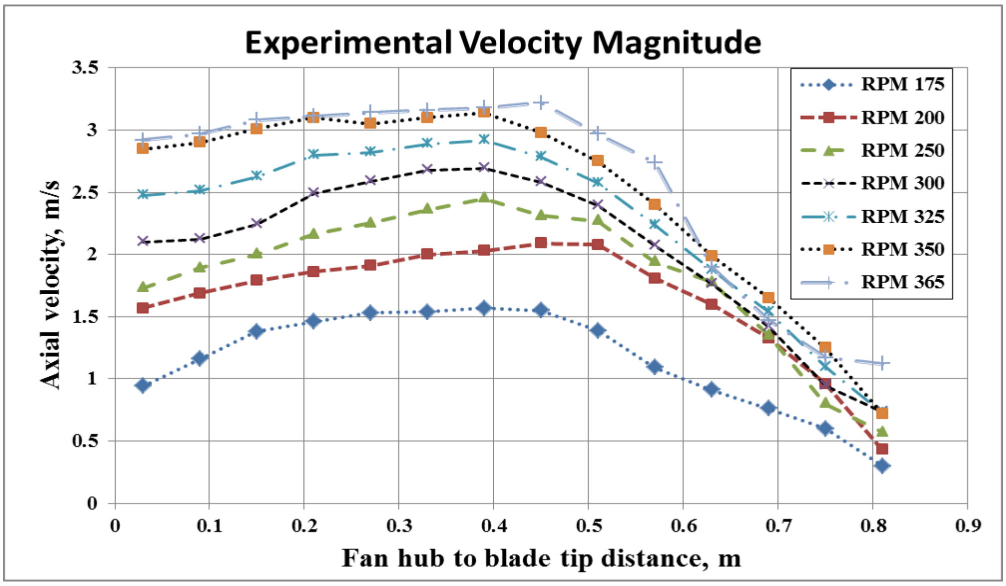


Figure 3-6 Experimental Data for various RPM's

Figure 3-7 shows the linear relationship between RPM and power. As RPM is decreased, less power is consumed by the fan motor to rotate. Therefore, for energy efficient fan power consumption should be decreased for more rated air delivery.

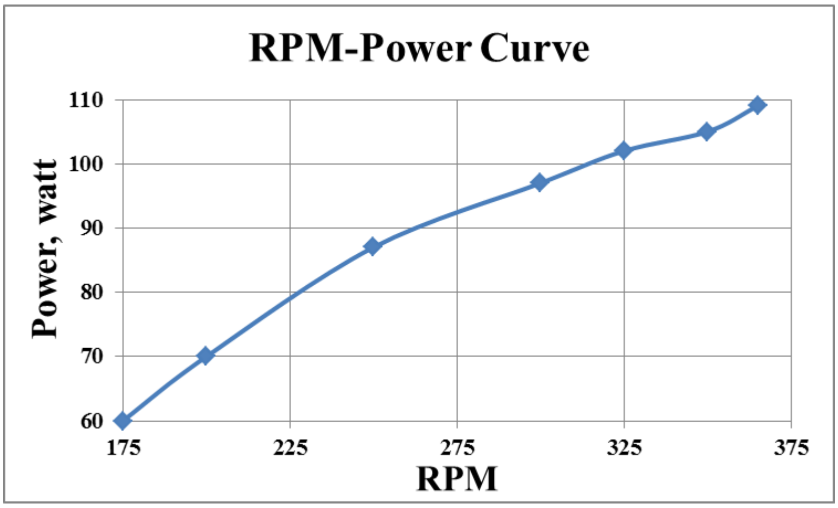


Figure 3-7 RPM Power Curve

Table 3-4 represents the Pakistan regulatory authority standards for ceiling fans. Motor should consume 80 watts for 330 RPM and should deliver 220 m^3/min rated air delivery. Minimum service value should be 3.12 m^2/N.min and is the ratio of rated air delivery to torque produced by motor.

PS-1 Standards	
Rated Power Input (Watts)	80 (+10% Tolerance)
Rated Air Delivery (m^3/min)	220 (-10% Tolerance)
Minimum Service Value (m^3/min/watt)	3.12 (No Tolerance)
Maximum Speed (RPM)	330 RPM with Load

Table 3-4 Pakistan Standards 1

Table 3-5 represents the comparative analysis of Pakistan standards with collected experimental data. For various RPM operations, the power consumptions is recorded from 80 watts to 108 watts. The red colored quantities show that the standard is breached whereas the green figures represent that the standard requirement is satisfied. It can be observed that the ceiling fan fails to satisfy PS-1 standards at all RPM values.

PS-1		Experimental Values						
Maximum Speed (RPM)	330	175	200	250	300	325	350	365
Rated Power Input (Watt)	80	61	72	87	97	102	105	108
Rated Air Delivery (m^3/min)	220	145.33	182.54	217.65	259.45	280.33	292.82	308.18
Minimum Service Value (m^3/min/watt)	3.12	2.38	2.54	2.50	2.67	2.75	2.79	2.85

Table 3-5 Analysis of experimental data

3.9 Concluding Remarks

In this chapter, the experimental procedure used to obtain the air flow characteristics is discussed. Moreover, the details of instruments and their calibration are made for further analysis. The experimental data is collected at different RPM starting from 175 to 365. It is observed that the ceiling fan fails to qualify PS-1 standards at all RPMs. The experimental data collected in this chapter will be used ahead to validate the computational setup for further design refinement/parametric studies.

CHAPTER 4

Computational Modeling

4.1 Background

In this chapter, computational model is developed along with turbulence model study. Solution methodology, preprocessing, grid independence, boundary conditions, validation, mesh is described. A brief description of flow process of modeling setup is also discussed in this chapter.

4.2 Solution Methodology

Aerodynamic problems in engineering applications are usually solved by three approaches, namely:

- Analytical approach
- Experimental approach
- Numerical approach

Multi-physics problems, in general, are often too complex to solve analytically, hence they need to be analyzed by means of experiments or numerical simulation. Since, the purpose of the present study is to focus on the numerical simulations of the ceiling fan, therefore Computational Fluid Dynamics (CFD) has been employed to predict the performance and conduct parametric studies.

4.2.1 Computational Fluid Dynamics (CFD)

CFD is a numerical approach in which computers are engaged to analyze problems in fluid dynamics. There are three stages to the CFD simulation process:

- Pre-processing
- Solution process
- Post processing

In the first stage, the geometry is discretized into number of finite volume by applying the computational mesh over it. In the solution process stage, suitable and appropriate algorithms are applied for the approximate solution of the Navier-Strokes equations. In the final stage analysis are performed to interpret flow behavior through contour plots, animations, etc.

Using CFD is cost effective as compared to conduct experiments which help in optimizing design of the ceiling fans. These tools allow flow visualization around the fan to understand complex flow physics. In other words, careful CFD analysis can predict the performance of the fan with sufficient accuracy so as to avoid experiments to compute performance parameters.

4.2.2 Governing Equations

CFD uses numerical methods and algorithms to solve governing equations of fluid flow and analyze fluid dynamics problems. The governing equations for the flow field are the Navier-Strokes and the continuity equation. Flow is assumed to be incompressible; therefore the equations take the following form:

Continuity Equation

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Momentum Equations

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \rho}{x_i} + \mu \frac{\partial^2 u_i}{\partial u_j u_j} \tag{2}$$

where *i*, *j*=1, 2, 3. u_i represent the Cartesian velocity components (u, v, w); p is the pressure, ρ is the density and μ is the dynamic viscosity of the fluid.

4.3 Preprocessing

Fluent® [24] is a computational fluid dynamics (CFD) software package to simulate fluid problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models, such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is performed using GAMBIT® [25] which is the preprocessor bundled with Fluent®.

The methodology which is adopted in order to simulate the flow behavior is divided into two parts; (a) Meshing methodology and (b) Solver methodology. Following steps are followed in order to apply the computational mesh over the domain in the pre-processing stage

1. Create inner domain mesh in GAMBIT
    a. Create geometry of fan.
    b. Join vertex points with NURB line and then delete all the points.
    c. Create faces.
    d. Create unstructured mesh around fan.
    e. Create remaining mesh.
    f. Define boundary types.
2. Create outer domain mesh in GAMBIT
    a. Create outer domain geometry.
    b. Create mesh.
    c. Define boundary types.

3. Export the mesh in the form of *.msh* format

The exported mesh file is then imported into the ANSYS FLUENT 13 where following procedure is followed for the solution.

a. Read mesh file created from Gambit into ANSYS FLUENT 13.
b. Define solver setting.
c. Define flow conditions.
d. Define interfaces.
e. Define boundary conditions.
f. Define solution controls.
g. Initialize the flow field.
h. Solve steady state using the steady solver.
i. Once steady state solution converges.
j. After desired number of cycles, plot graph, contours and animations.

4.4 The Computational Model

In the present study, GAMBIT® is used for the discretization of the geometry into number of finite volumes. We employed Fluent® for accurate solution of problems involving internal flows.

This study includes two domains inner and outer domains. Inner domain consist of ceiling fan and outer domain consists of room geometry

4.4.1 Ceiling Fan and Room Geometry

Ceiling Fan includes three blades and a hub in a rotatory disk. The coordinates are taken from Super Delux model geometry and was reproduced in GAMBIT in cm units. Subsequently, the geometry was scaled to m when exported to commercial CFD package Fluent. The geometry of the blade is shown in Table 4-1. For Simplicity the rivets and connectors were removed in the computational domain because they have negligible effect on the flow, the main study revolves around the bent, rake and connector angle.

Geometric Attributes	Dimensions
Fan Hub Diameter	0.2564 m
Fan Blade length	0.568 m
Fan Diameter	1.4224 m
Fan Blade Thickness	1 mm
Fan Blade Material	Aluminum
Fan Blade Bent Angle	11.8^0
Fan Blade Rake Angle	5^0
Fan Blade Connector Angle	8^0
Fan Blade Root Cord	0.16 m
Fan Blade Tip Cord	0.115 m

Table 4-1 Fan Blade Geometry

A square room was modeled with dimensions given in the Table 4-2. After importing the file in Fluent the fan geometry was appended in the room disk so that it can rotate.

Geometric Attributes	Dimensions
Room Length	4.5 m
Room Width	4.5 m
Room Height	3 m
Fan Duct	61″
Fan Distance from Ceiling	1 m
Fan Distance from Floor	3 m

Table 4-2 Room Dimensions

Figure 4-1 illustrate the sequence of the work for the development of computational model. For the calculation of parameters geometeries are meshed and boundary conditions are specified. These three steps work as an input for the fliud solver. Refined mesh is considered more appropriate for accurate values of required parameters in computations. Therefore, grid independence study is is carried out for improvement in the accuracy of the model. After grid independence studies, an appropriate turbulence model is selected to incorporate viscous and near wall effects.

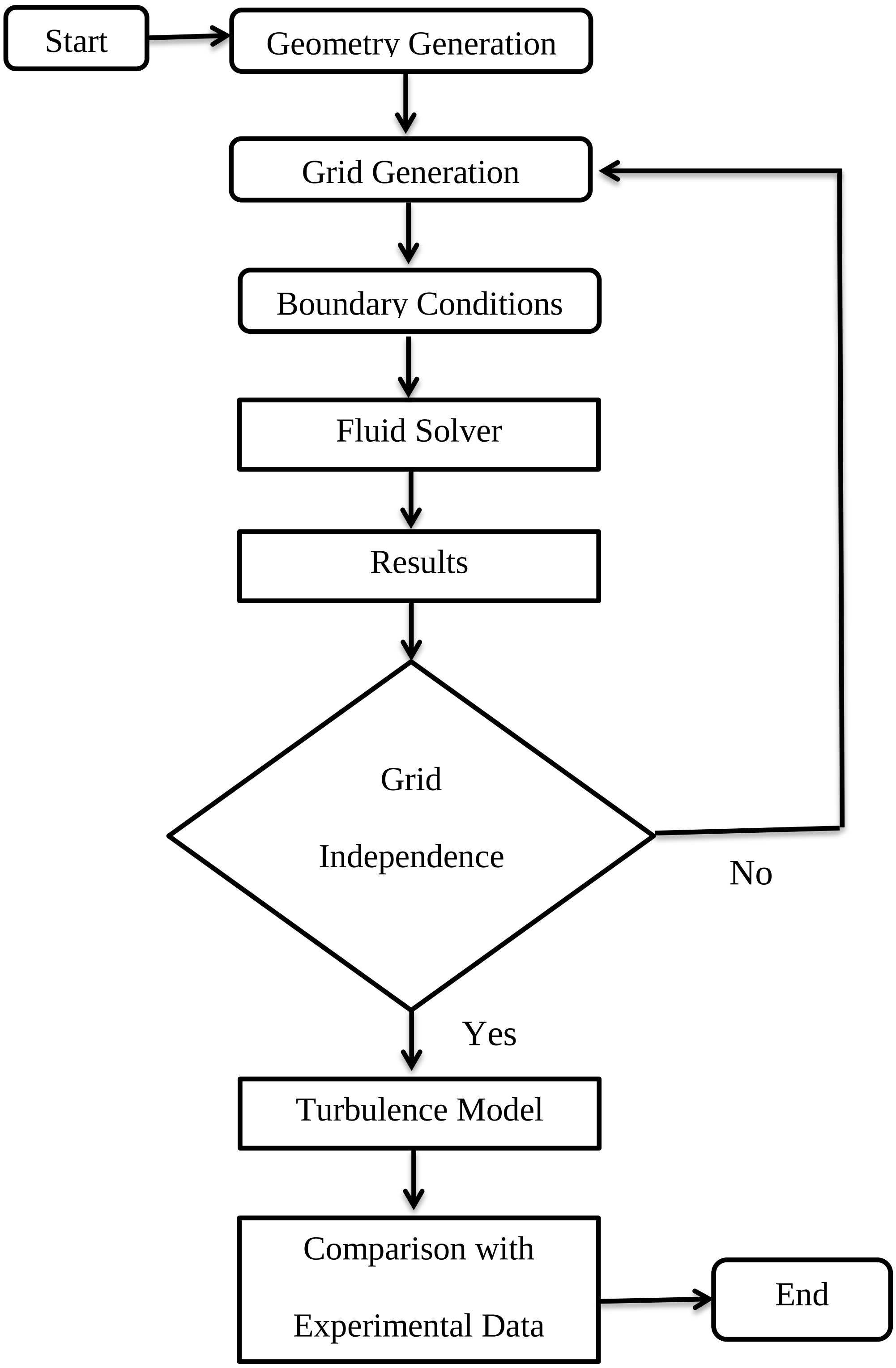


Figure 4-1 Flow Chart

4.4.2 Grid Generation

Structured quad map elements are used to mesh the flow domain of stationary domain (test room) shown in Figure 4-3. Unstructured tetrahedral mesh elements are used to mesh the flow domain of rotating domain (fan disk). The generated grid is illustrated in Figure 4-2. Grid independence is carried out by developing three different grids. The grid with best results in terms of flow profile and computational efficiency has been selected for further analysis. Grids are often non-matching in the boundaries between the two blocks. Non-matching grids may result in the discontinuity of flow parameters. To accurately simulate the phenomenon matching grid has been made at the interface of two grid blocks.

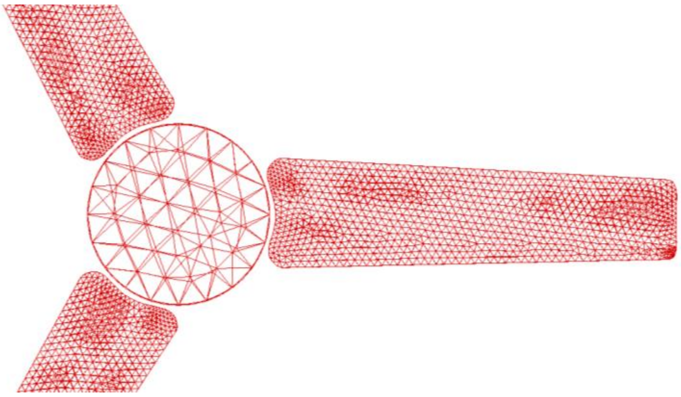


Figure 4-2 Fan Unstructured Mesh

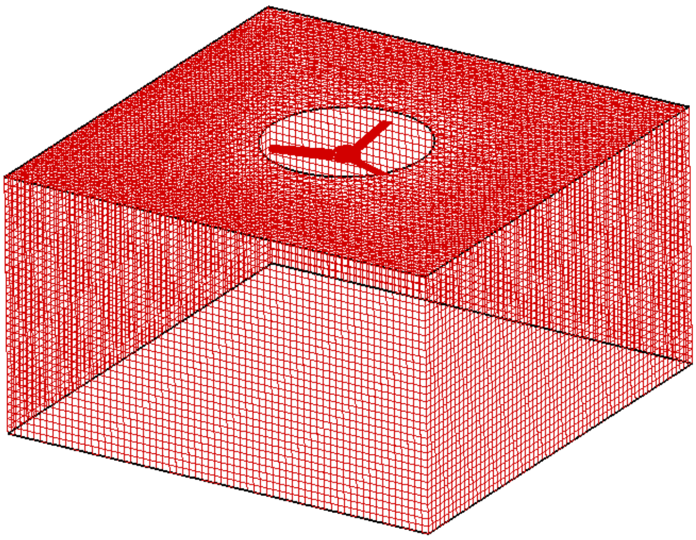


Figure 4-3 Room Structured Mesh

4.4.3 Fluid Solver

Fluent®, finite-volume based CFD code along with its preprocessor Gambit® is used and three-dimensional steady state incompressible Reynolds-Averaged-Navier-Stokes (RANS) system of equations, with turbulence model is solved using the coupled-implicit formulation. To capture viscous effects in the flow the one equation Spalart-Allmaras model is used as turbulence model. The selection of this turbulence model is due to its results that are near to the experimental results. Experimental data was gathered for comparison. Finally, computations with 0.7 million grid sizes was iterated for different dihedral rake angles from 0^0 to 10^0, bent angles from 9.8^0 to 12.8^0 with the increase of 1^0 respectively and several bent positions were varied with the change in root and tip chord on HP core i-7 with 16 GB of RAM.

4.4.4 Boundary Conditions

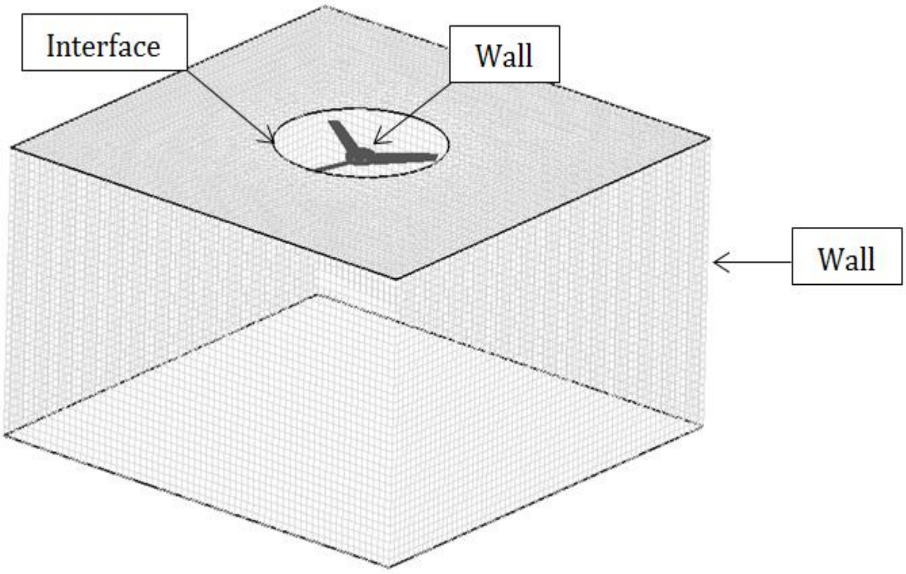


Figure 4-4 Boundary Conditions

No-slip velocity and adiabatic wall boundary condition is enforced at the surface of blades, and room walls (Figure 4-4). Total pressure (101325Pa), total temperature (288K), and 300 RPM are specified for the rotating disk in order to simulate the system.

Fan Components	Boundary Conditions
Fan Hub and Blade	Wall
Fan Disk	Interface
Roof	Wall
Floor	Wall
Room	Wall

Table 4-3 Boundary Conditions

After ensuring that the computational model is predicting the correct flow physics the next step is to ensure that the computational results do not vary with the mesh size. For this purpose three meshes were made, with number of cells 0.5, 0.7 and 0.9 million in coarse, medium and fine mesh respectively.

4.5 Spalart–Allmaras Model

S-A model [26] is one equation model that solves modeled transport equation for eddy viscosity. It is very effective for aerospace applications involving wall bounded flows. The model gives good results at low *Re*, however, it requires very fine meshing near the surface. The transport equation in S-A model is as follows;

$$\frac{\partial(\rho \tilde{v})}{\partial t} + \frac{\partial(\rho \tilde{v} u)}{\partial x_i} = G_v + \frac{1}{\sigma_{\tilde{v}}}[\frac{\partial}{\partial x_i}\{(\mu + \rho \tilde{v})\frac{\partial \tilde{v}}{\partial x_i}\} + C_{b2p}(\frac{\partial \tilde{v}}{\partial x_i})^2] - Y_v + S_{\tilde{v}} \quad (3.2)$$

where, G_v is production of turbulent viscosity, Y_v is destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. And C_{b2} are constants, ν is molecular kinematic viscosity and S is user defined source term.

4.6 Grid Independence study

The purpose of grid independence study is to achieve a set of grid points for which the solution and the flow physics of the computational case does not further change if the grid points are increased in the computational domain.

For this very reason, three types of grid sets were created in the commercial package grid generator and modeler GAMBIT and were analyzed computationally in the FLUENT solver. The three grid sets are 0.5, 0.7 and 0.9 million.

Axial velocity versus fan to wall distance for the grid independence study are shown in Figure 4-5. Plots indicate that all three grid sets are in 98% agreement with each other. All three sets were computationally solved using S-A turbulence model. 0.7 million grid was chosen as the baseline grid size for all computational cases in this research thesis.

Level	Cells	Faces	Nodes
0	775557	1599852	151754

Table 4-4 Grid Information

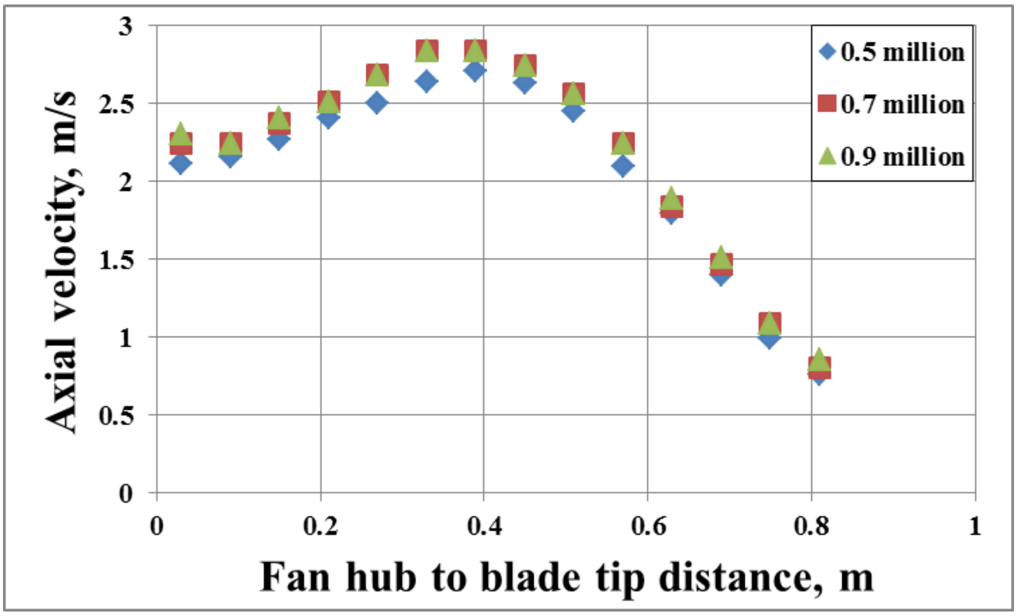


Figure 4-5 Grid Independence Study

4.7 Computational Validation

For successful validation, the experimental data gathered from Star Co Fan Gujrat was compared with the computational data shown in Figure 4-6. The 10% error bars are applied to the experimental data. The figure shows a reasonable agreement in the velocity variation. This shows that velocity is decreasing as the distance is increasing from the fan axis towards the wall.

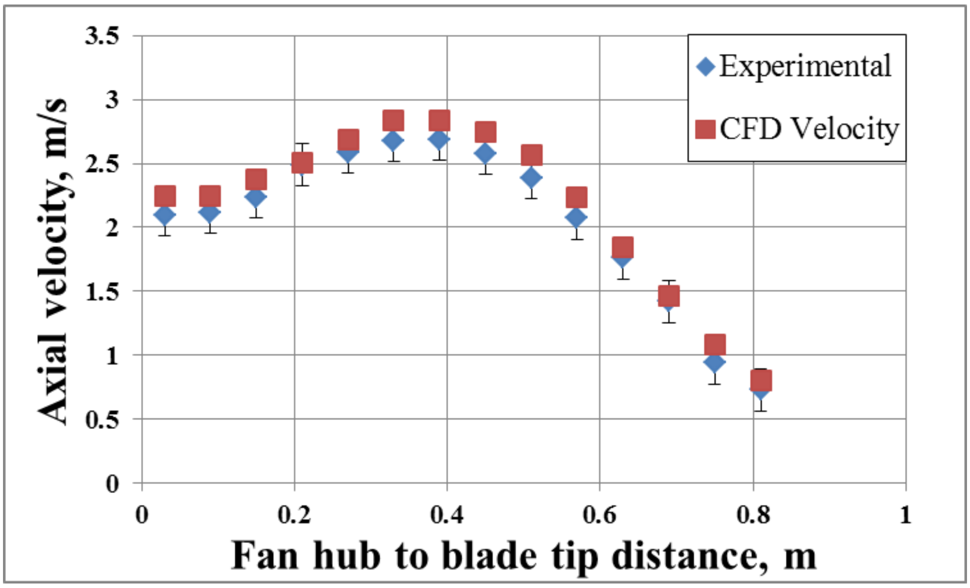
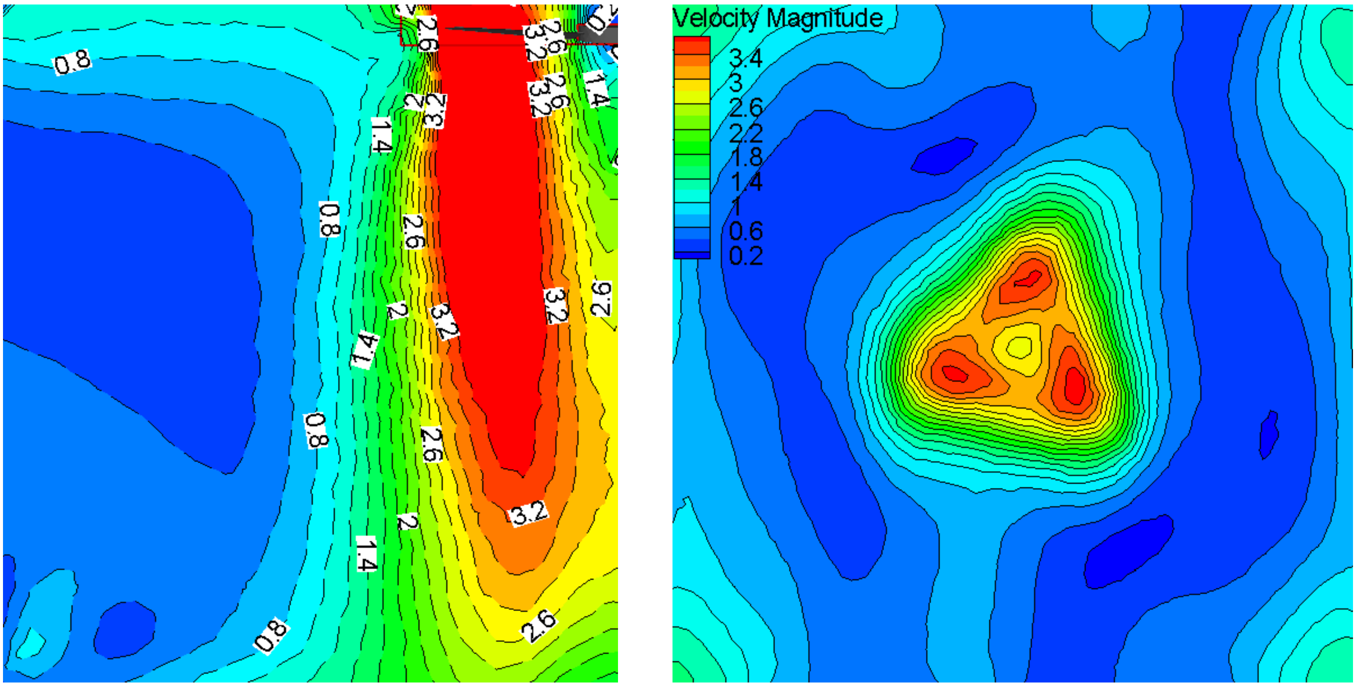


Figure 4-6 Comparison of Experimental and Computational Model (S-A) at 300 RPM

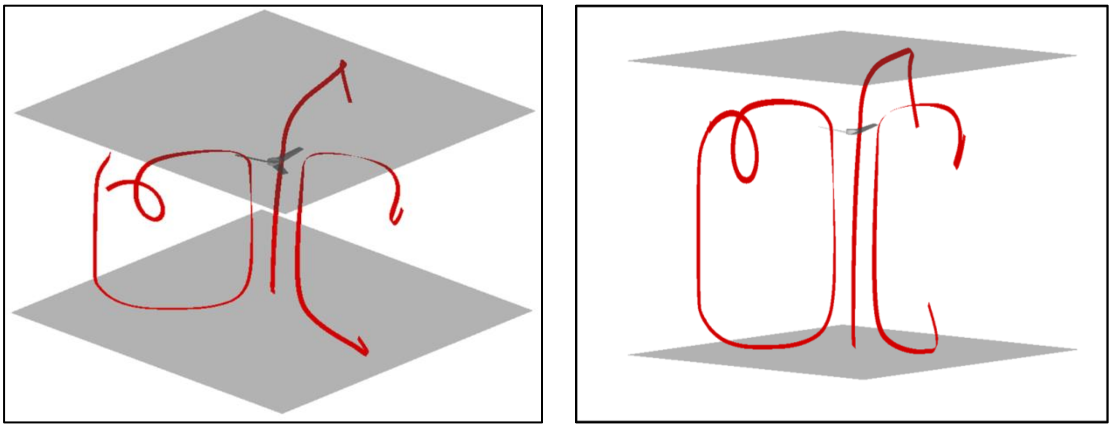
Velocity contours in the room mid plane are shown in Figure 4-7 (a) at 300 RPM and similarly at 1.5m above the floor are shown in Figure 4-7 (b). The contours clearly show the circulation of air in the room.



(a) Velocity Contours in Mid Plane (b) Velocity contours 1.5m above floor

Figure 4-7 Velocity Contours

Air flow ribbons from 3 blades are shown in Figure 4-8 that show the circulation of air inside a room.



(a) Air Flow Ribbons Top View (b) Air Flow Ribbons Side View

Figure 4-8 Flow Ribbons

4.8 Turbulence model sensitivity study

Five models were applied during this analysis on the computational grid of 0.7 million. They were

- Laminar Model
- k-ω Model
- k-ε Model
- Large Eddy Simulation (LES) Model
- Spalart Allmaras (S-A) one equation Model

The comparative plots for five turbulence models are given in Figure 4-9. Plots indicate that all the models depict the same behavior and aerodynamics velocity magnitudes inside room except LES model as it requires large number of mesh size. S-A model is chosen as it provides the best values comparative to the experimental values.

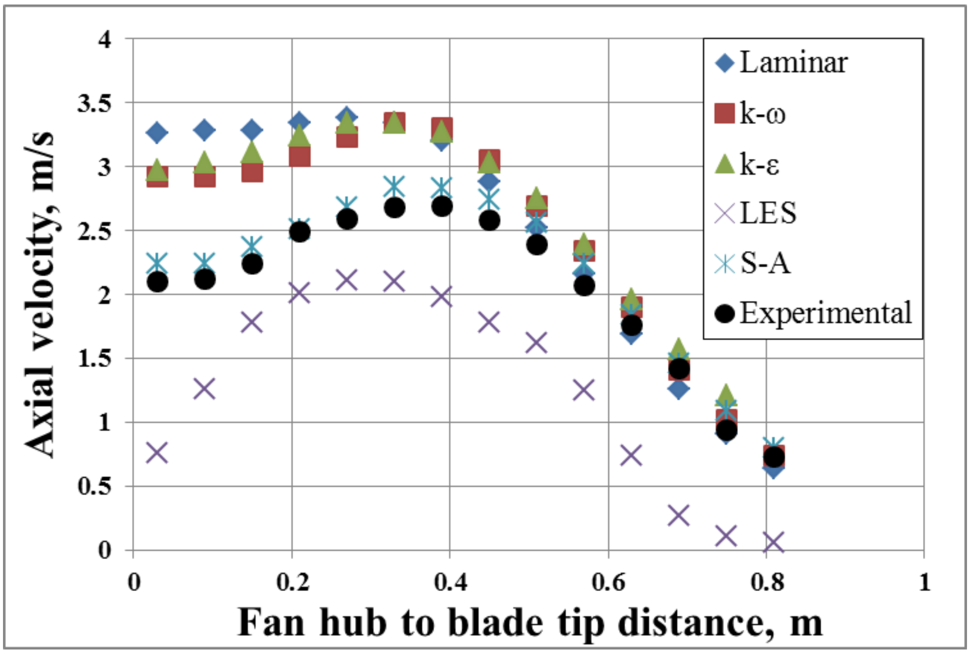


Figure 4-9 Comparison of Different Turbulence Models

Parametric study involves the bent angle, rake angle and bent position variation. Comparison of velocity magnitudes, mass flow rate, rated air delivery, torque and service value is done in the next chapters. An overview of this parametric study is shown in Figure 4-10.

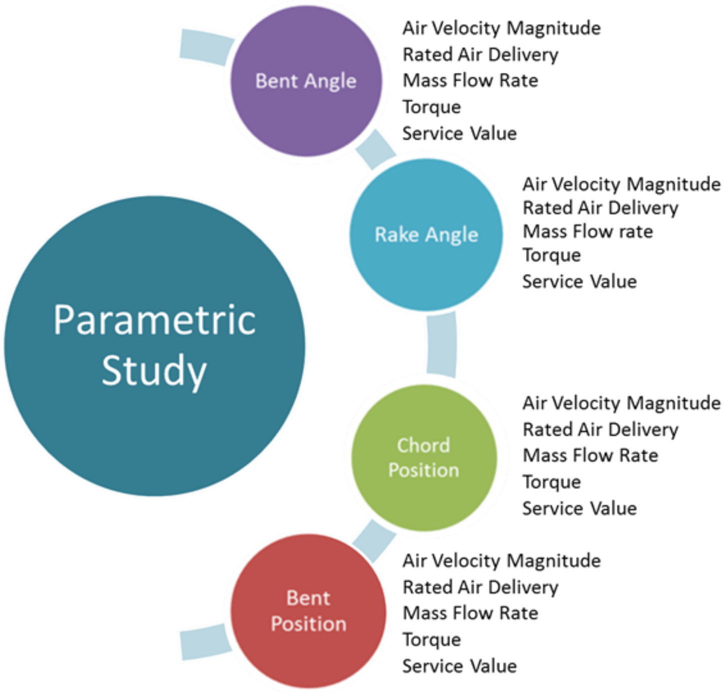


Figure 4-10 Parametric Study Flow Chart

4.9 Concluding Remarks

After carrying out the detailed study, the following conclusion can be drawn;

- Grid independence study for 0.7 million and 0.9 million grid sets shows the same aerodynamic variations. Therefore, the grid size of 0.7 million is used for further studies.
- Turbulence model sensitivity study revealed that one equation S-A model shows better results than other models. Hence, the S-A turbulence model was chosen for future numerical calculations.

CHAPTER 5

RAKE AND BENT ANGLE STUDIES

5.1 Background

This chapter comprises of brief description of bent and rake angles parametric study. The mass flow rate, rated air delivery, torque and service value are discussed along with their geometries. A recommendation is made on the basis of the parametric study.

5.2 Parameters for Performance Evaluation

There are several parameters to gauge the performance of certain design modification on the overall performance. These parameters are discussed below:

5.2.1 Air Velocity Profile

The velocity magnitude approximately 1.5 m below the rotary disc of ceiling fan gives the air velocity profile. The velocity profile is used to assess the spread of velocity from fan hub to blade tip. Different profiles can be compared to see the effectiveness of certain design features at various points in the profile.

5.2.2 Rated Air Delivery

Rated air delivery is the volumetric rate of change of air per unit time and is measured in $\frac{m^3}{min}$.

Rated Air Delivery = Area * Average velocity * 60

5.2.3 Mass Flow Rate

Mass flow rate is the mass of the air that passes from the specified room surface per unit time and measured in $\frac{kg}{m^3}$.

$$m = \rho v.A \tag{2}$$

where m is the mass flow rate, ρ is the mass density, v is the velocity field of the mass elements flowing and A is the cross sectional area of fan disk. Indoor comfort will increase as the mass flow rate increases.

5.2.4 Torque

Torque is the tendency of a force to rotate an object about an axis and measured in N.m.

$$\tau = r \times F \tag{3}$$

where τ is the torque vector, r is the displacement vector and F is the force vector. For energy efficiency torque should be reduced so that ceiling fans are more affordable.

5.2.5 Service Value

Service value is the ratio of the rated air delivery and torque and measured in $\frac{m^3/_{min}}{watt}$.

$$\text{Service Value} = \frac{Rated\ Air\ delivery}{Torque} \tag{4}$$

For energy efficiency, an increase in service value is desired.

5.3 Effect of Rake Angle

The rake angle is briefly illustrated in chapter 2. All the Rake angles studies are conducted on 300 RPM. The rake angles studies were conducted for 0^0, 2.5^0, 5^0, 5.5^0, 6^0, 6.5^0, 7^0, 7.5^0 and 10^0. Originally the Star Co fan rake angle was 5^0. Result in Figure 5-1 shows that the best rake angle for ceiling fan blade is 6^0. So as a recommendation 6^0 rake angle was fixed for further studies. From the Figure 5-1, the velocity is measured 1.5 m above the floor which shows that the velocity decreases at some distance away from the fan blade.

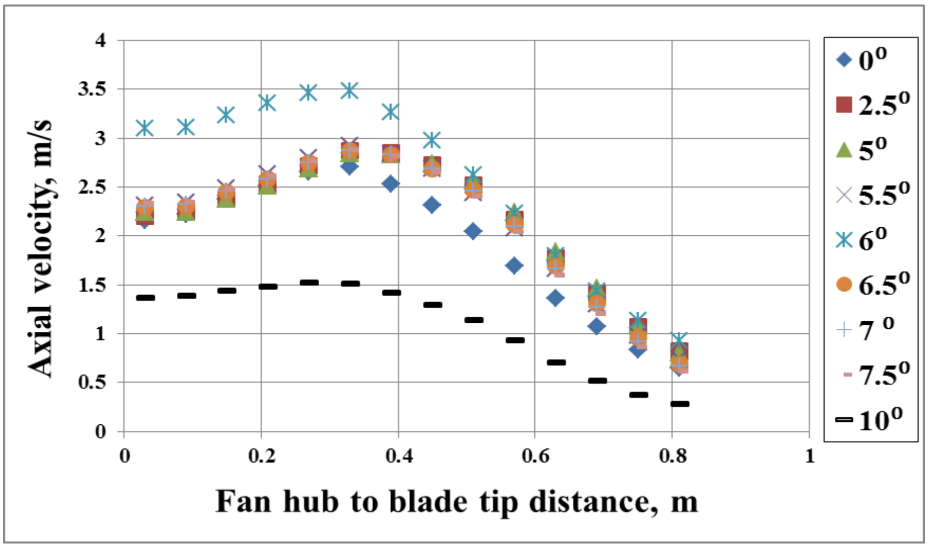


Figure 5-1 Comparison of Different Rake Angles

Figure 5-2 shows the patterns of mass flow rate 1.5 m above the floor which predicts that 6^0 blade rake angle show the maximum mass flow rate inside the room.

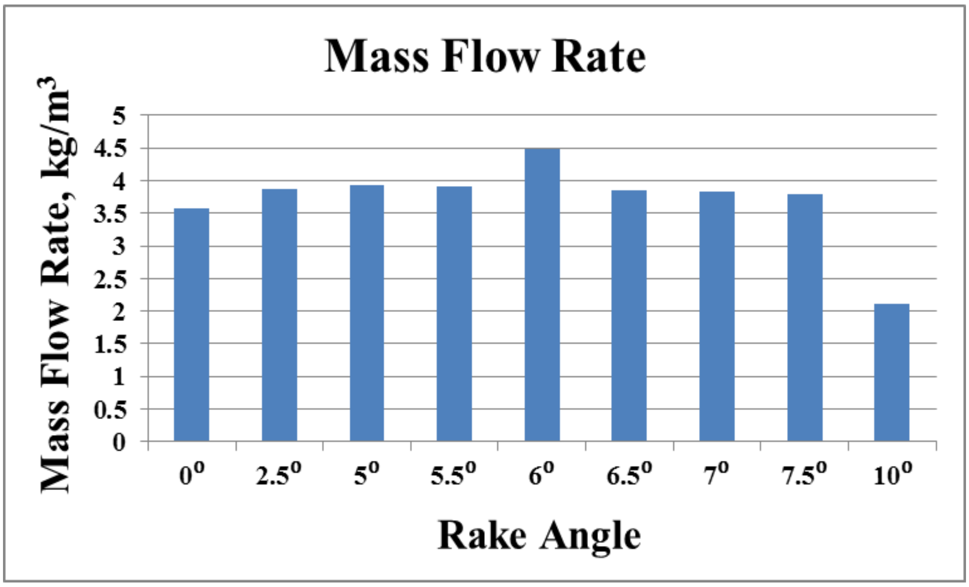


Figure 5-2 Comparison of Different Rake Angles Mass Flow Rate

Figure 5-3 shows rated air delivery at 1.5m above the floor which clearly predicts that 6^0 blade rake angle show the best rated air delivery compared to other.

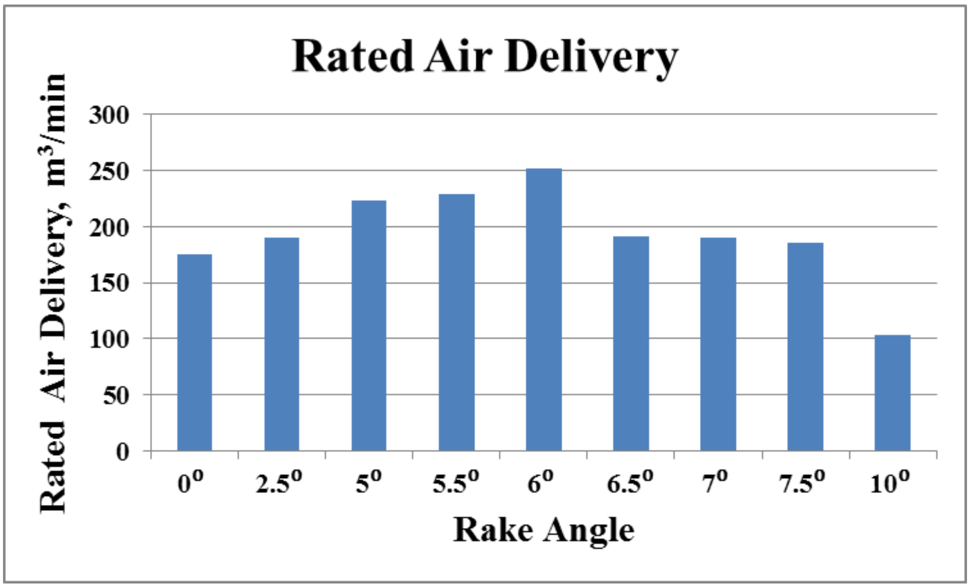


Figure 5-3 Comparison of Different Rake Angles Rated Air Delivery

The higher the RPM higher will be the torque so to make the fan energy efficient torque should be reduced with increased air delivery. The rated air delivery and mass flow rate shows that the 6^0 rake angle is the best angle. So we are more interested in 6^0 rake angle effects.

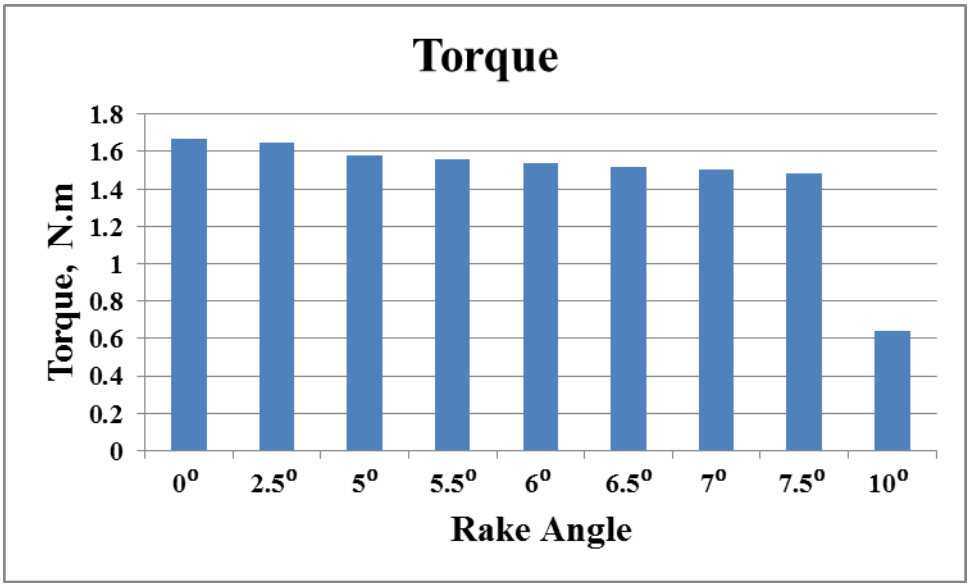


Figure 5-4 Comparison of Different Rake Angles Torque

In Figure 5-5, 10^0 blade rake angle show the best service value but as it has poor rated air delivery and mass flow rate so 10^0 rake angle effects are exempted. 6^0 rake angle is still giving better service value.

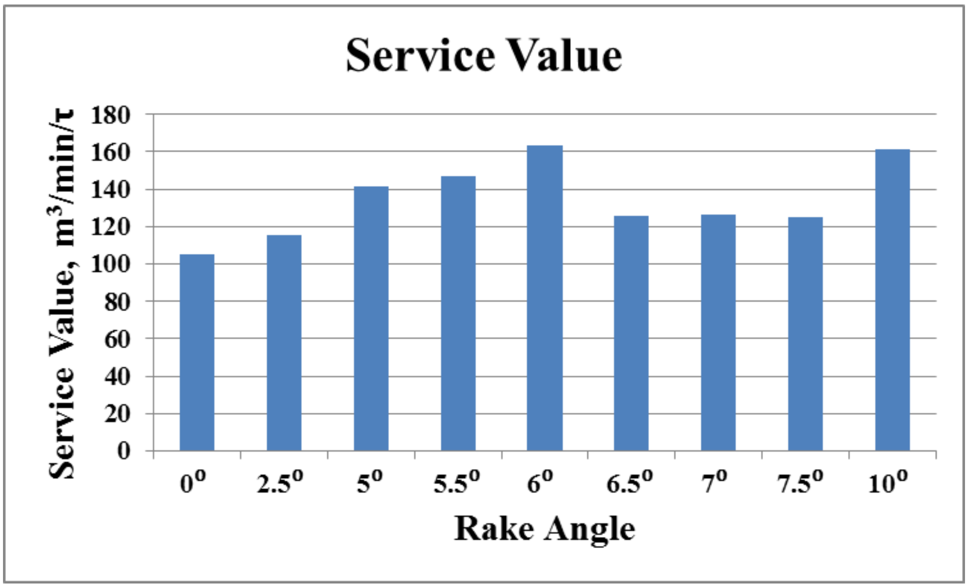


Figure 5-5 Comparison of Different Rake Angles Service Value

The simulated geometries for rake angle are shown in Figure 5-6. Rake angle is varied from 0^0 to 10^0 respectievely.

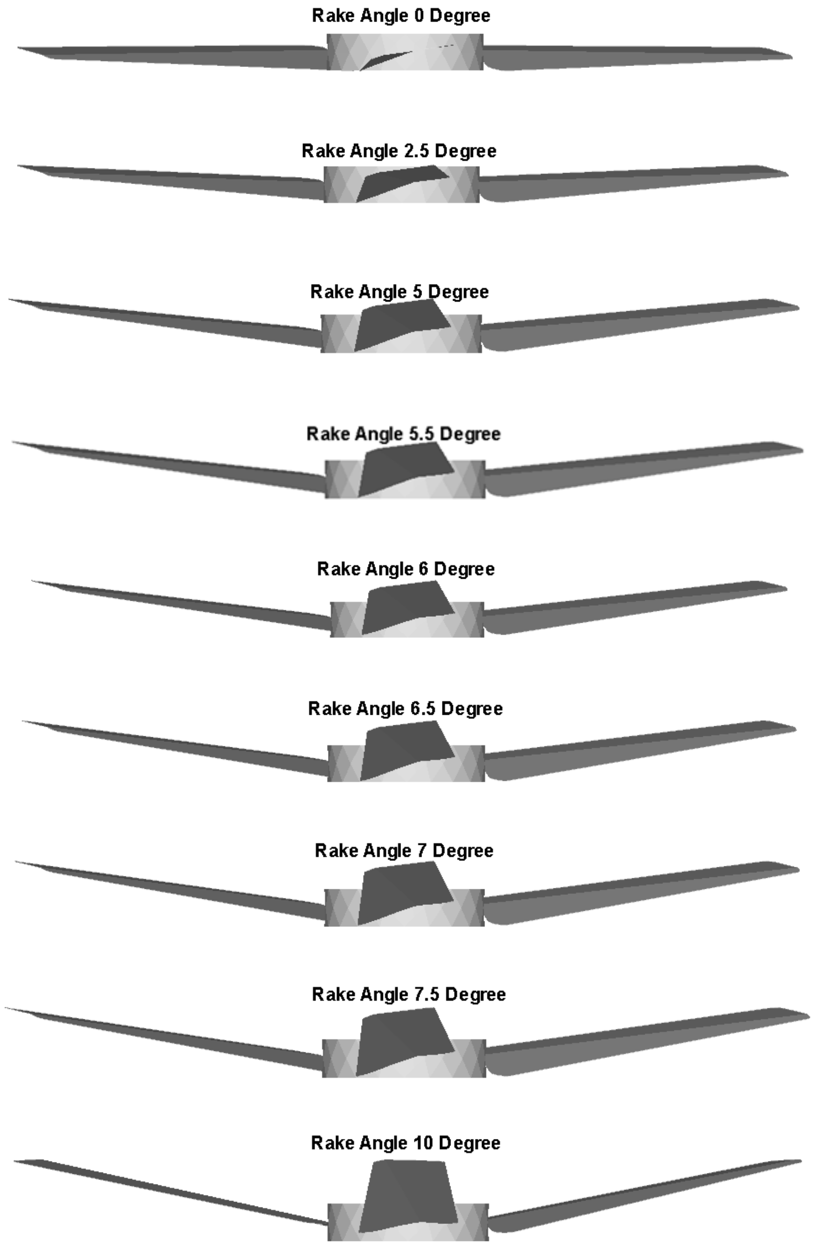


Figure 5-6 Ceiling Fan Rake Angles Comparison

5.4 Effect of Bent Angle

The blade bent angles is the key for the efficiency of fan. The blade bent angle was varied from 9.8^0 to 12.8^0 with interval of 1^0 respectively as shown in Figure 5-7. Rake angle is fixed at 6^0 for the bent angle studies. The mass flow rate, rated air delivery, torque, and service values are calculated. The simulation was performed at

300 RPM. 11.8^0 blade bent angle show the maximum air velocity as compared to the other bent angles.

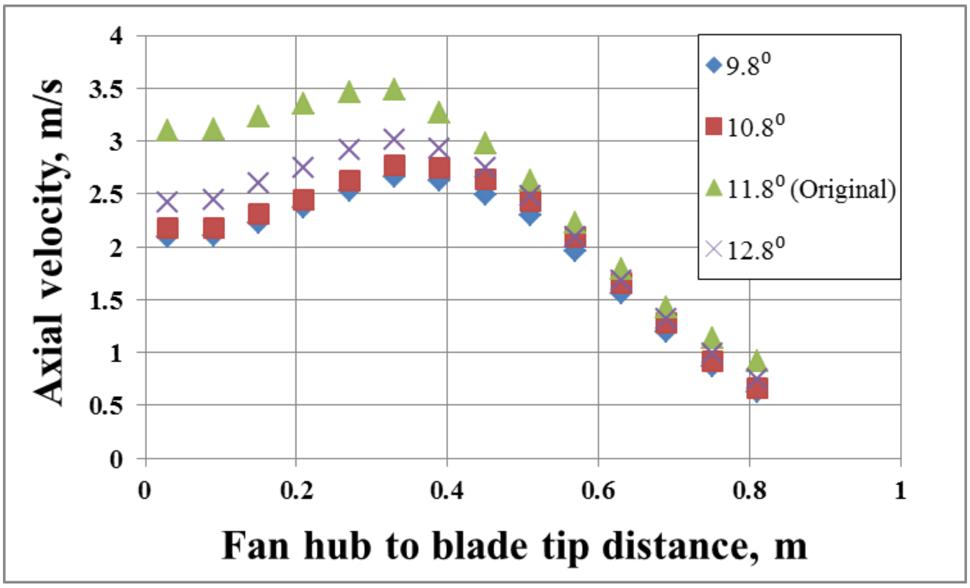


Figure 5-7 Comparison of Different Bent Angles at 300 RPM

Mass flow rate is the mass of the air that passes from the specified room surface per unit time. 11.8^0 blade bent angle show the maximum mass flow rate inside the room surface as shown in Figure 5-8.

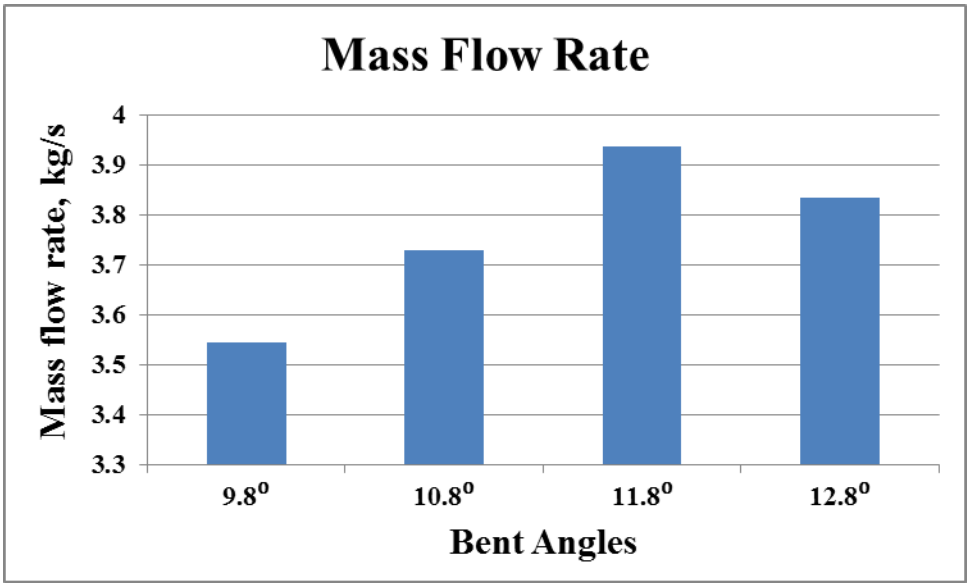


Figure 5-8 Comparison of Different Bent Angles Mass Flow Rate

Rated air delivery is the volumetric rate of change of air per unit time. The Figure 5-9 clearly predicts that 11.8^0 blade bent angle show the best rated air delivery compared to other.

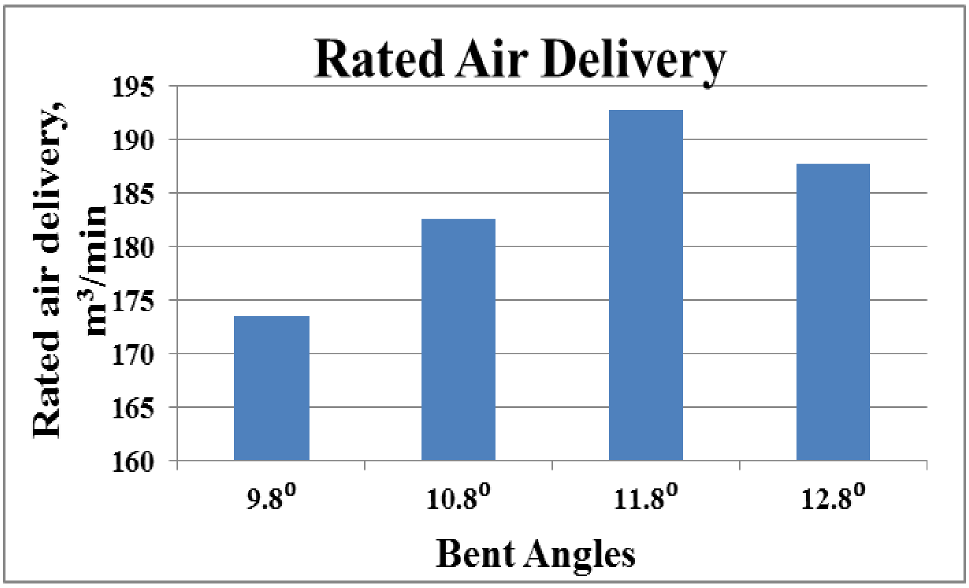


Figure 5-9 Comparison of Different Bent Angles Rated Air Delivery

Torque is the turning effect of blades about an axis and measured in N.m. The Figure 5-10 clearly predicts that 10.8^0 blade bent angle show the minimum torque as compared to other bent angles.

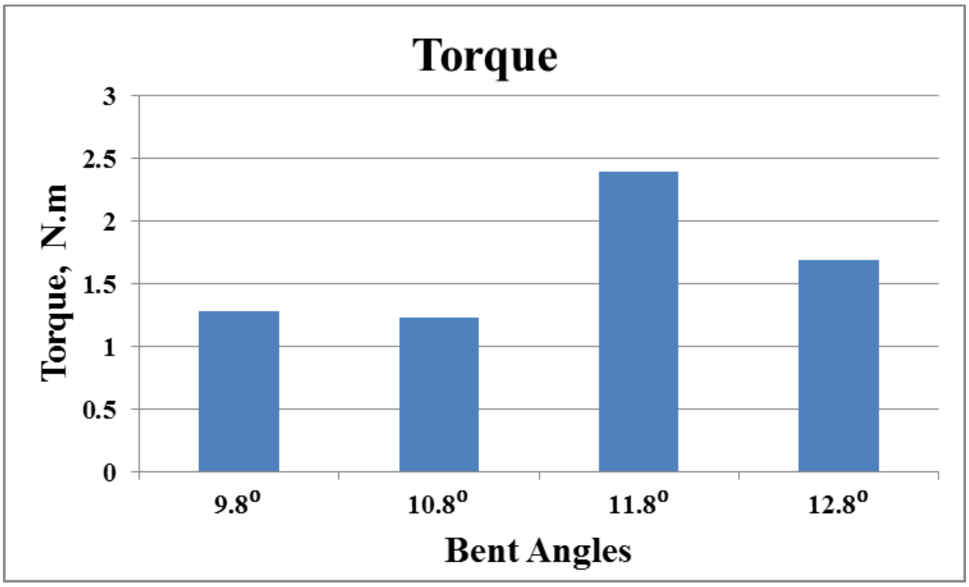


Figure 5-10 Comparison of Different Bent Angles Torque

Service value is the ratio of the rated air delivery and torque. Here 10.8^0 blade bent angle show the best service value in Figure 5-11.

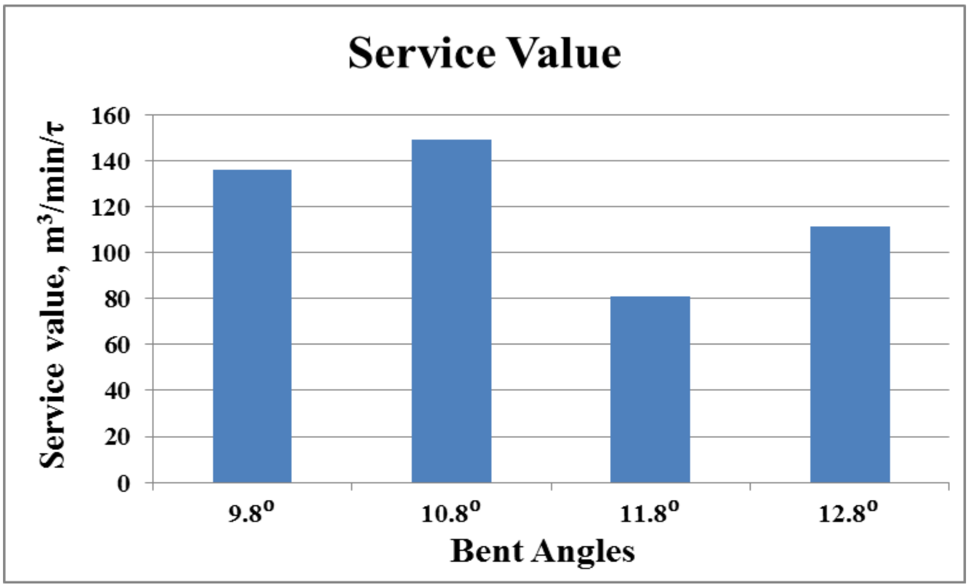


Figure 5-11 Comparison of Different Bent Angles Service Value

5.5 Concluding Remarks

After carrying out the detailed bent and rake angles study, the following conclusion can be drawn;

- Aerodynamic efficiency can be increased by increasing the rake angle by one degree from the baseline geometry.
- The bent angle measured in the Star Co Fan industry is the optimized one along with 6^0 rake angle. So the existing Bent angle is perfect for future production.

CHAPTER 6

BENT POSITION STUDIES

6.1 Background

This chapter includes a brief understanding of bent position variation on the blade along the chord as well as on the existing diagonal line provided by the Star Co Fan. Initially the bent position is varied linearly along the span. Subsequently, oblique variation in the bent position along chord is performed as well. For each blade shape the air velocity, mass flow rate, rated air delivery, torque, service value and energy efficiency is briefly explained.

6.2 Linear Bent Position along Chord

During the initial phase, the parametric study of bent position was carried out by linearly varying the bent position from 0.25c to 0.75c along the chord shown in Figure 6-1. A total of three cases are considered for comparison with the original oblique bent position.

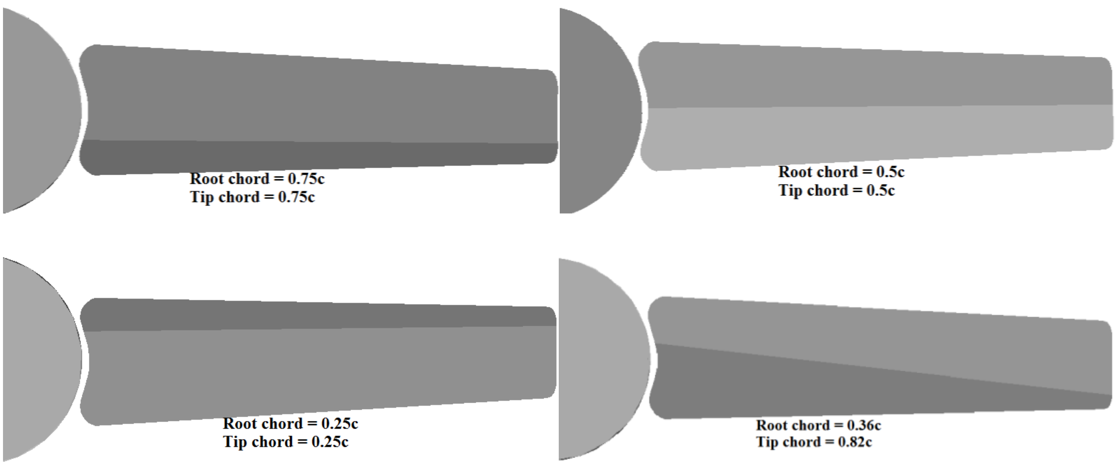


Figure 6-1 Bent positions (lower bent, middle bent, upper bent and original bent)

The comparison of velocity profiles in Figure 6-2 of these three cases with original case shows that the oblique (original) bent position have better flow speed than the three linear bent positions considered. Since, the linear bent position variation resulted in no relative increase of velocity magnitude; the research marched into investigating oblique bent position variation.

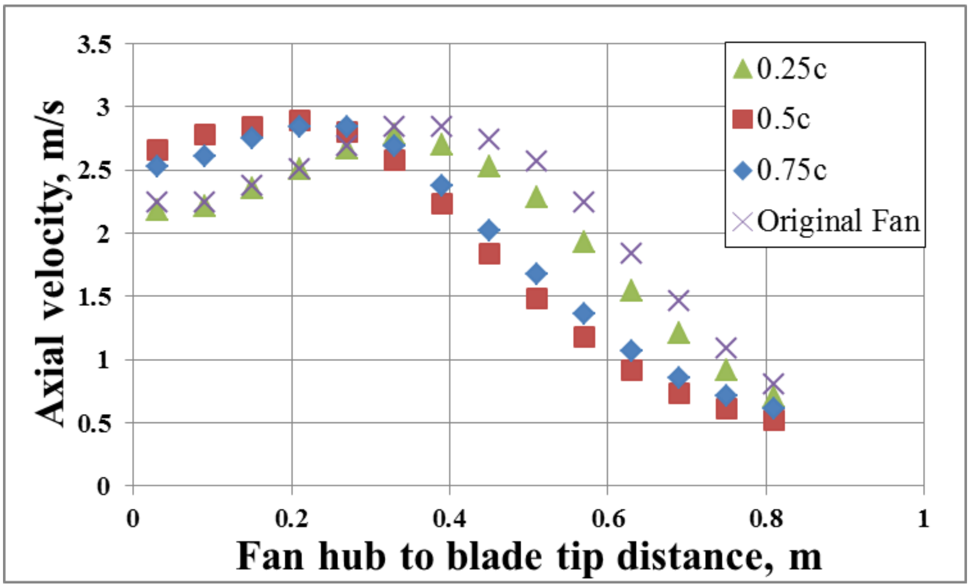


Figure 6-2 Comparison of Velocity Magnitude of Bent Positions at 300RPM

6.3 Oblique Bent Position along root to tip

For oblique bent position variation, different root and tip chord positions are uniformly varied. A 4x4 matrix was created and total of sixteen cases are simulated including the original one at 365 RPM (Figure 6-3). These sixteen geometries are divided into four groups. In each group there are four tip chord positions for one root chord position. To find the optimized geometry after extracting the results, spline interpolation technique is used. A 31*31 matrix is interpolated using MATLAB® spline fitting algorithm. An optimized geometry is selected based on the simulated cases. Result show that the optimized fan blade has the efficient air speed and mass flow rate as compared to the original one. It has minimum torque and maximum service value.

		Tip Chord			
		0.56c	**0.64c**	**0.72c**	**0.82c**
Root Chord	**0.18c**	Geometry 1	Geometry 2	Geometry 3	Geometry 4
	0.24c	Geometry 5	Geometry 6	Geometry 7	Geometry 8
	0.30c	Geometry 9	Geometry 10	Geometry 11	Geometry 12
	0.36c	Geometry 13	Geometry 14	Geometry 15	Geometry 16

Figure 6-3 Bent Position Variations

6.3.1 Simulated Geometries

Figure 6-4 shows the first group, in this group the root chord position is fixed to 0.18c and tip chord is varied from 0.56c to 0.82c. Maximum air velocity is captured for "Geometry 1" in the first group as shown in Figure 6-5

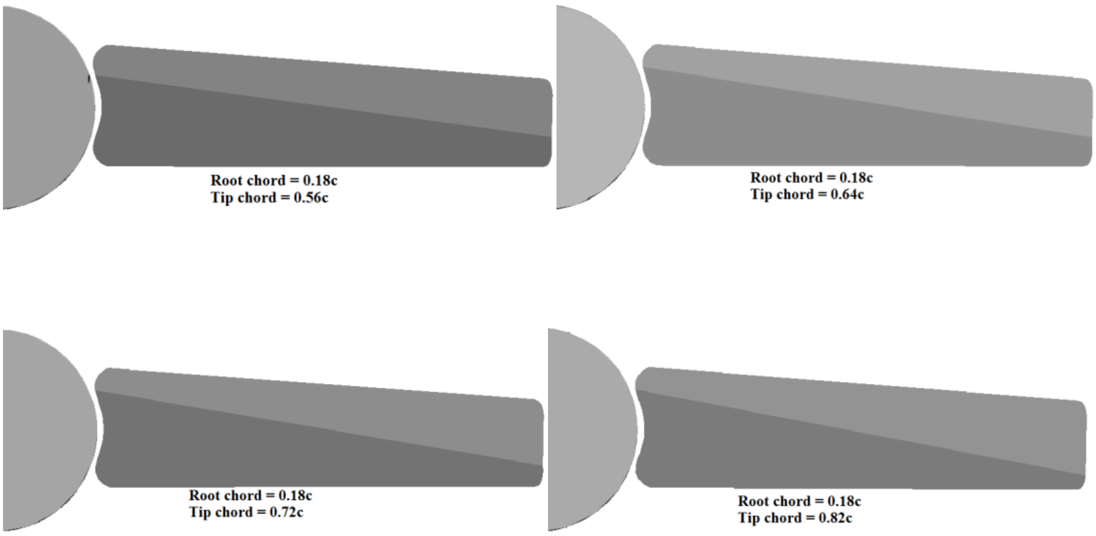


Figure 6-4 Geometry 1, Geometry 2, Geometry 3, Geometry 4

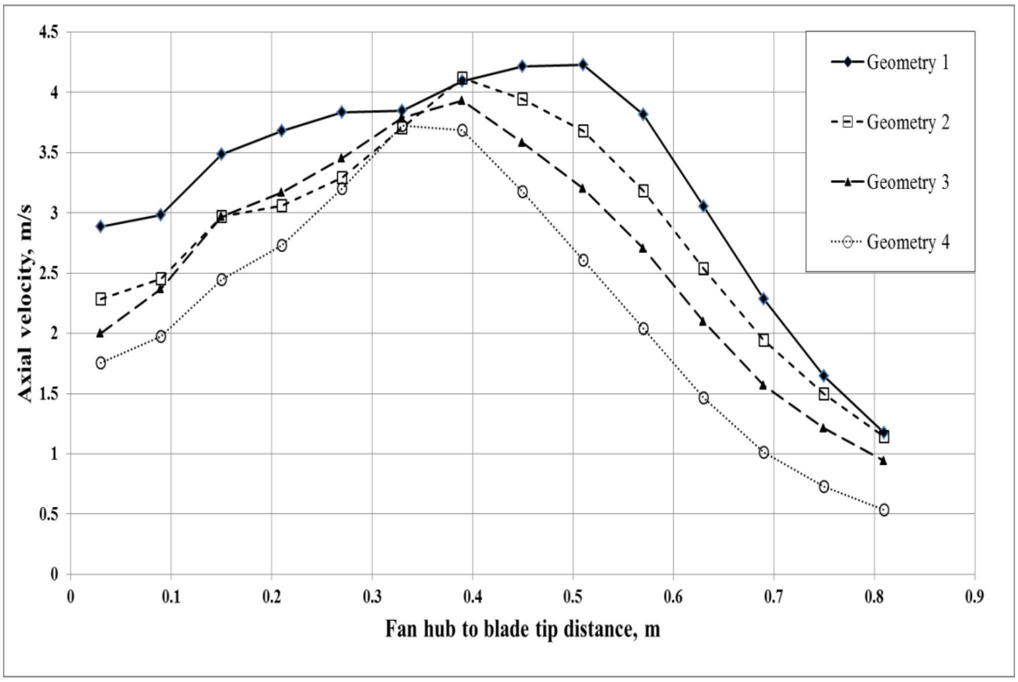


Figure 6-5 Comparison of Velocity Magnitude of Bent Positions at 365 RPM

Figure 6-6 shows the second group of simulated geometries. In this group the root chord position is fixed to 0.24c and tip chord is varied from 0.56c to 0.82c. Maximum air velocity is captured for "Geometry 5" in the second group shown in Figure 6-7.

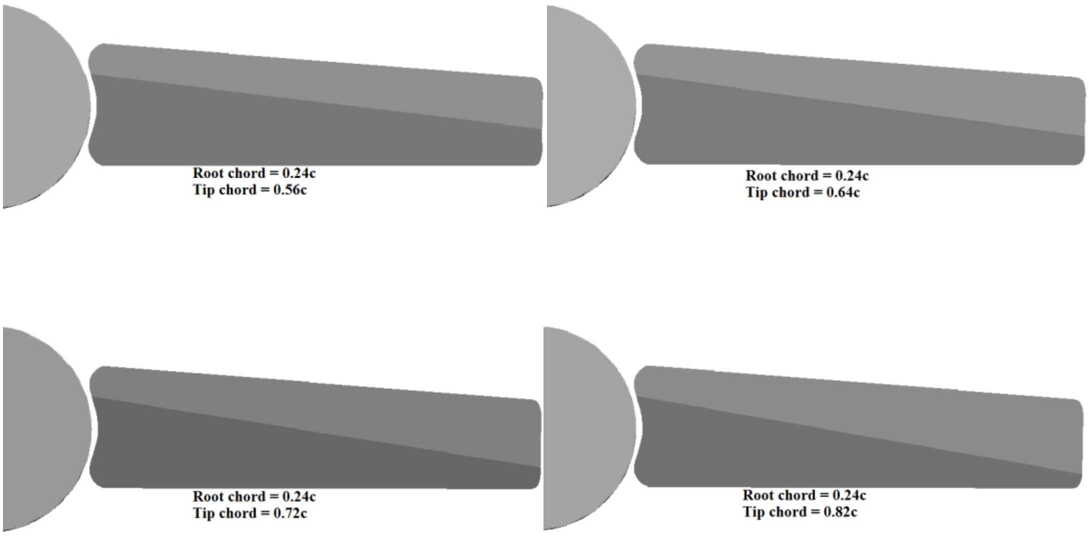


Figure 6-6 Geometry 5, Geometry 6, Geometry 7, Geometry 8

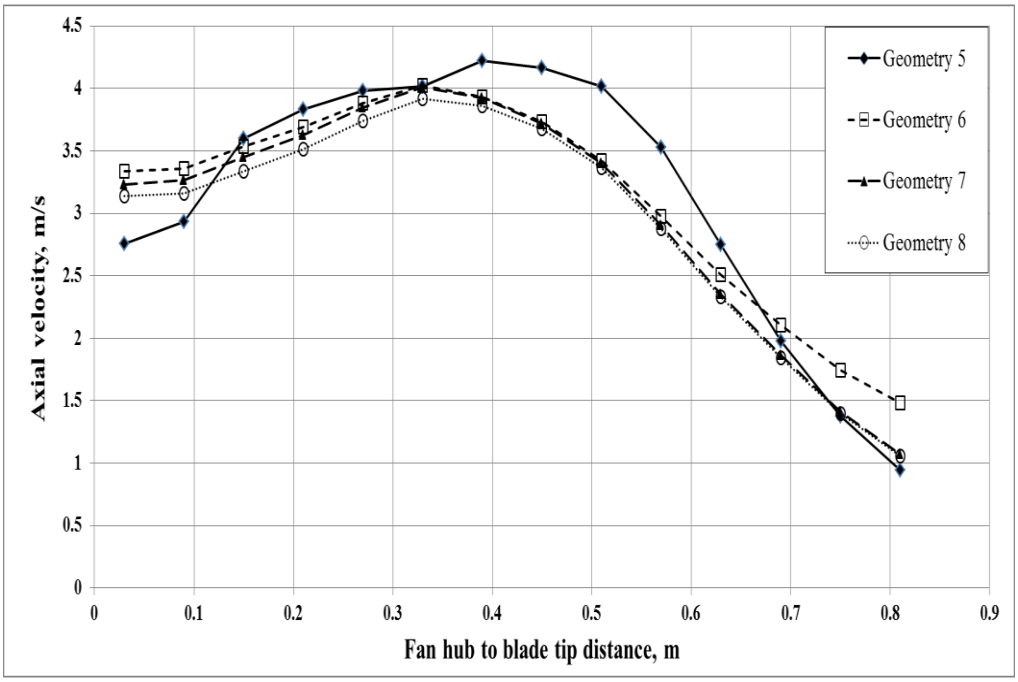


Figure 6-7 Comparison of Velocity Magnitude of Bent Positions at 365 RPM

Figure 6-8 shows the third group in which the root chord position is fixed to 0.30c and tip chord is varied from 0.56c to 0.82c. Maximum air velocity is captured for “Geometry 9” in the first group as shown in Figure 6-9.

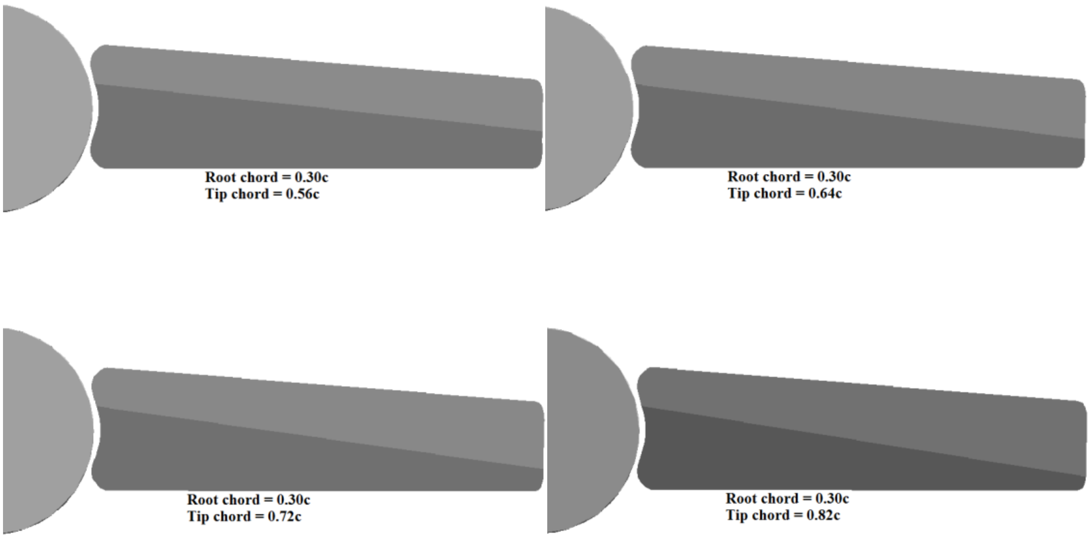


Figure 6-8 Geometry 9, Geometry 10, Geometry 11, Geometry 12

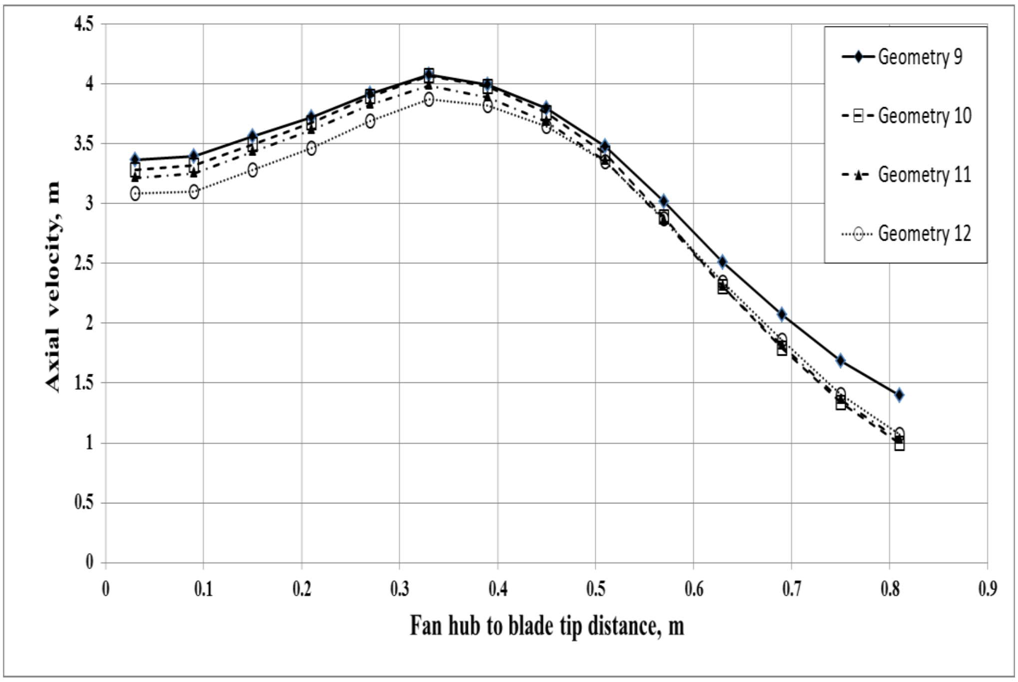


Figure 6-9 Comparison of Velocity Magnitude of Bent Positions at 365 RPM

Figure 6-10 shows the fourth group in which the root chord position is fixed to 0.36c and tip chord is varied from 0.56c to 0.82c. Maximum air velocity is captured for “Geometry 13” in the first group shown in Figure 6-11.

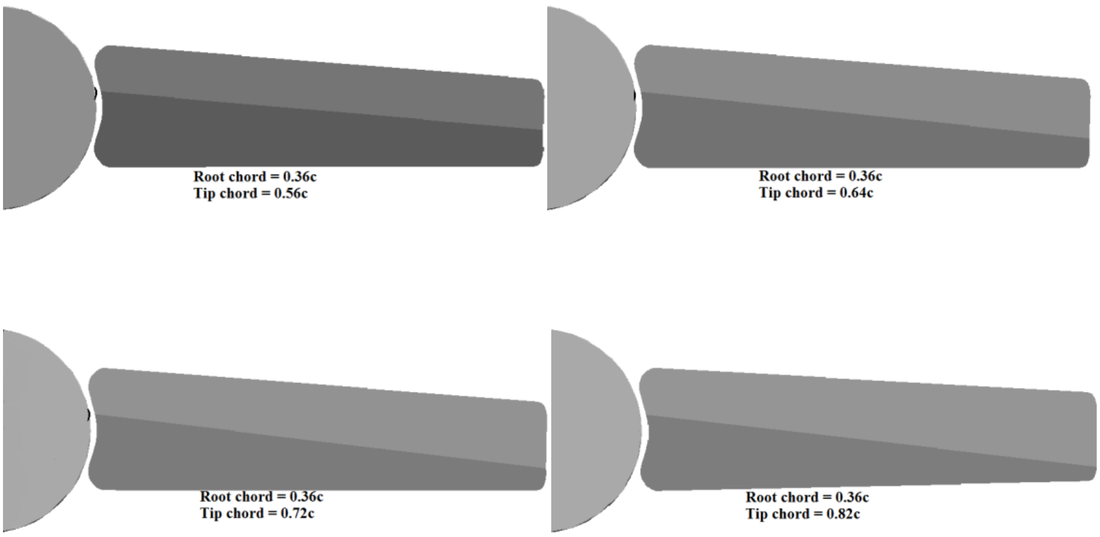


Figure 6-10 Geometry 13, Geometry 14, Geometry 15, Geometry 16

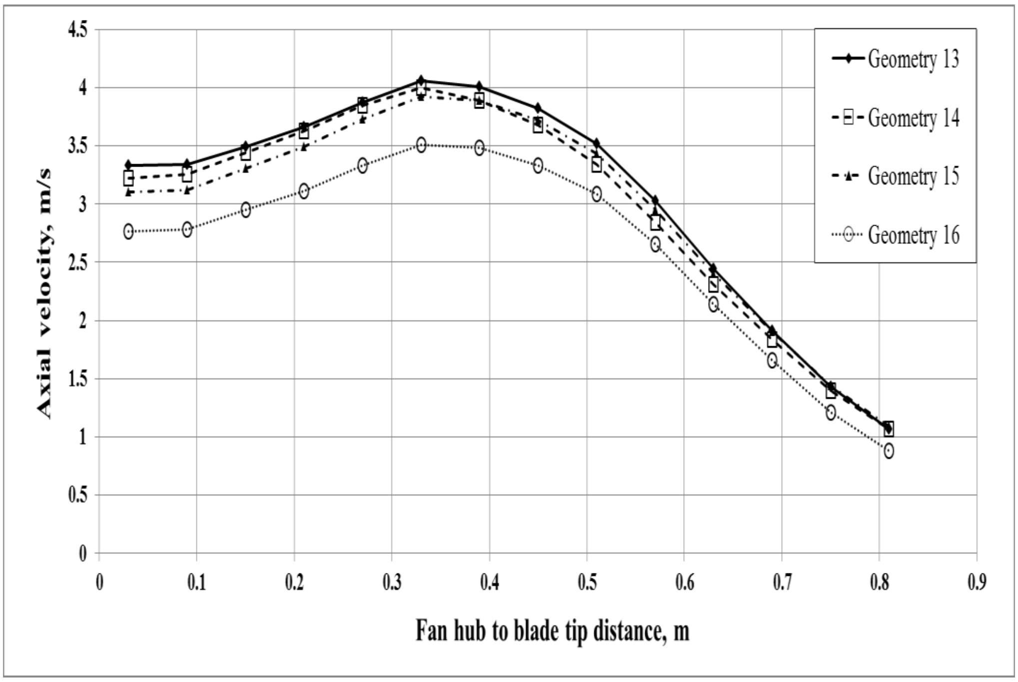


Figure 6-11 Comparison of Velocity Magnitude of Bent Positions at 365 RPM

6.3.2 Parametric Analysis

For the parametric analysis, a 31*31 matrix is interpolated using MATLAB spline command code. A maxima is highlighted in the surface plot shown in Figure 6-12. The rated air delivery, also known as volume flow rate, is plotted as a function of root and tip chord.

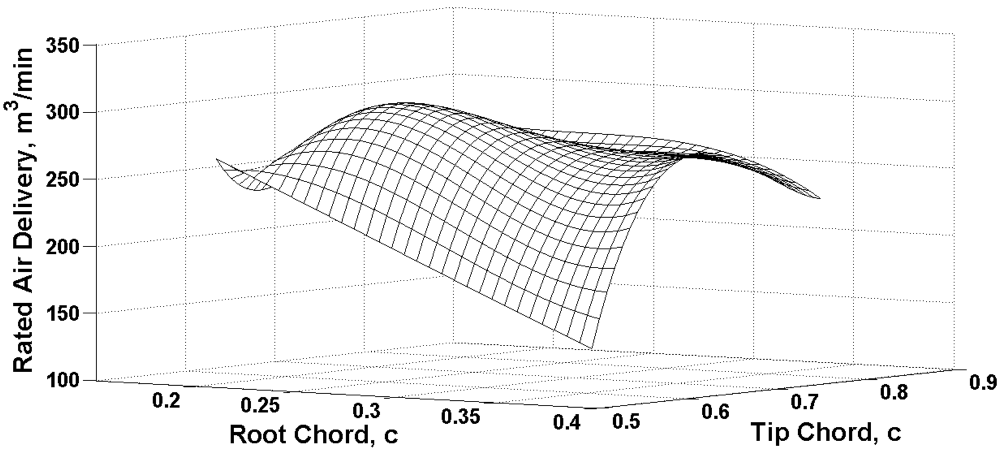


Figure 6-12 Rated Air Delivery

The response surface plotted in Figure 6-12 is mapped on a two dimensional plane in Figure 6-13. The contour plot shows that maximum rated air delivery is in region containing root chord between 0.23c to 0.25c and tip chord between 0.63c to 0.67c. The contour colors differentiate the intensity of the rated air delivery with different tip and root chord positions.

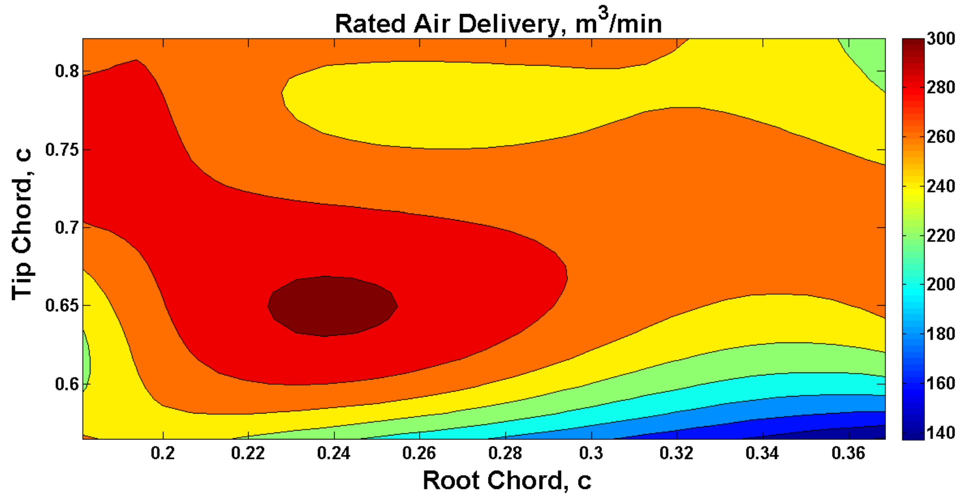


Figure 6-13 Rated Air delivery

The associated mechanical torque (Figure 6-14) results in a slope surface. The slope increases as the bent position is moved towards the leading edge of root and tip chord.

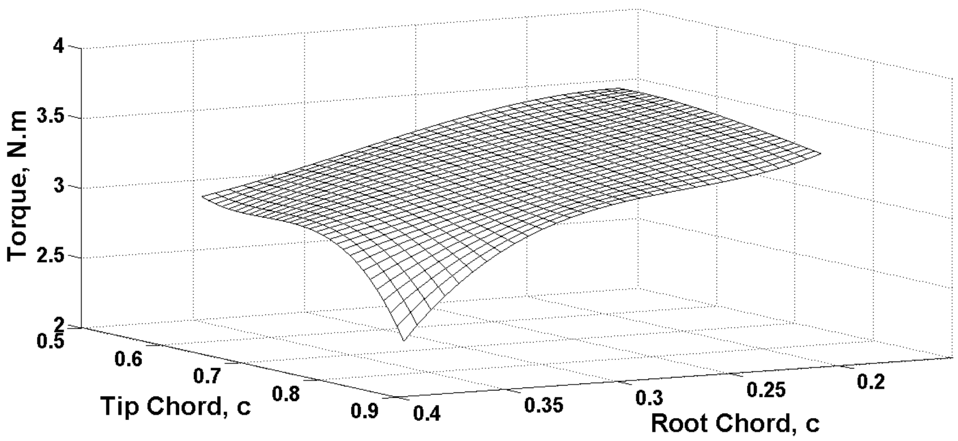


Figure 6-14 Torque

The response for mechanical torque requirement in Figure 6-14 is mapped on a two dimensional contour plot in Figure 6-15. The results indicate that the magnitude of the torque linearly increases as the root and tip chord increases. Contour colors are shown to differentiate the intensity of torque.

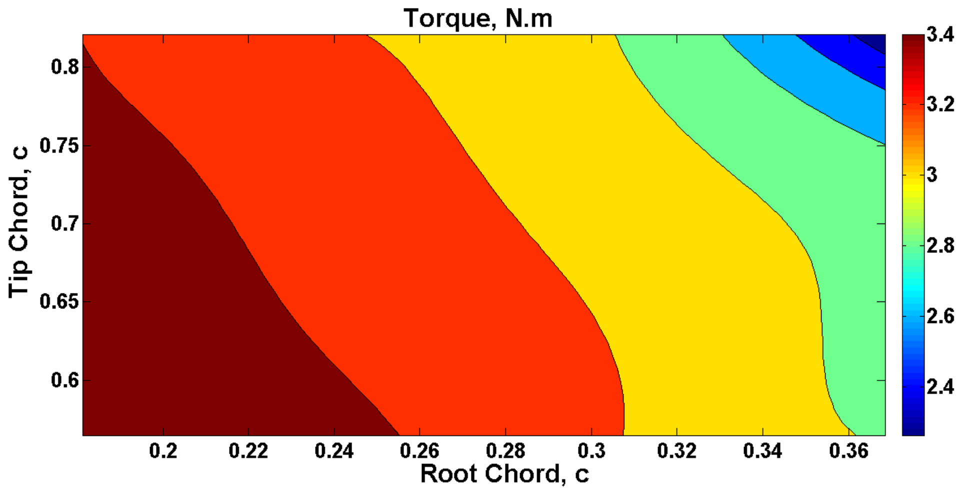


Figure 6-15 Torque

The Figure 6-16 indicates maxima in the surface plot at root chord 0.24c and tip chord 0.64c. One peak is at root chord 0.37c and tip chord 0.9c. The peak associated with the later geometry cannot be considered because of poor rated air delivery as shown in Figure 6-12 and Figure 6-13.

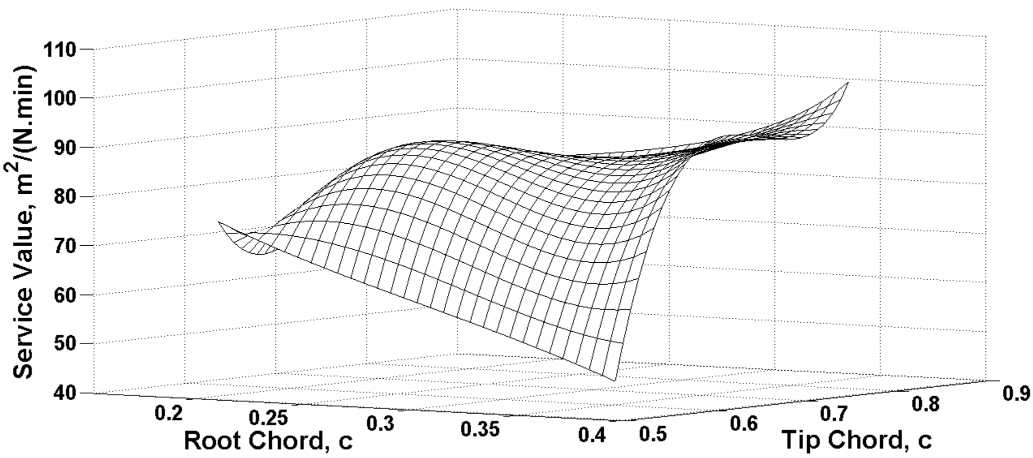


Figure 6-16 Service Value

The Figure 6-17 indicates that the magnitude of the service value is maximum at root chord 0.24c and tip chord 0.64c. Colored magnitudes are shown to explain the intensity of the service value.

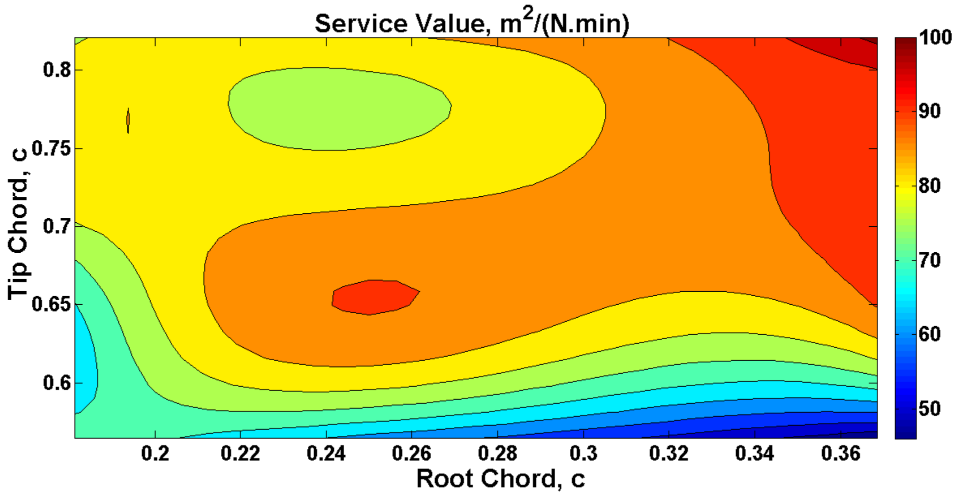


Figure 6-17 Service Value

6.3.3 Final Recommendation

From parametric analysis, an efficient improvement in performance of ceiling fan is obtained. This modified ceiling fan-blade geometry contains root chord 0.24c and tip chord 0.64c as shown in Figure 6-18. Advantages of this modified geometry are

- Pakistan Standards (PS-1) will be satisfied as elaborated ahead
- Rated air delivery is increased
- Energy Efficiency is achieved by reducing RPM

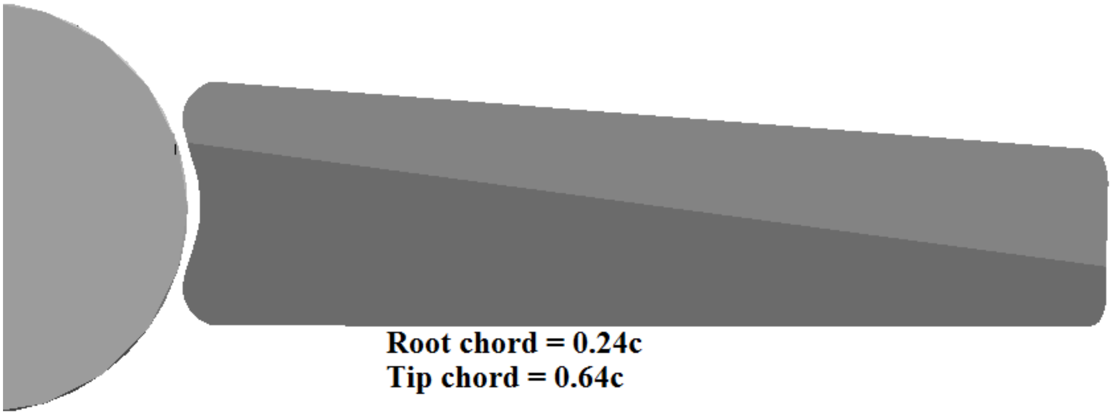


Figure 6-18 Modified Ceiling Fan

6.3.4 Comparison between baseline and modified geometries

The simulated results of both blades original and the modified one at 300 RPM and 365 RPM are shown in Figure 6-19 and Figure 6-20 respectively. It shows that a significant amount of air speed is increased inside the room especially from fan hub to blade distance of 0.3 to 0.8. However, the increase in axial velocity from hub to 0.2 m is negligible.

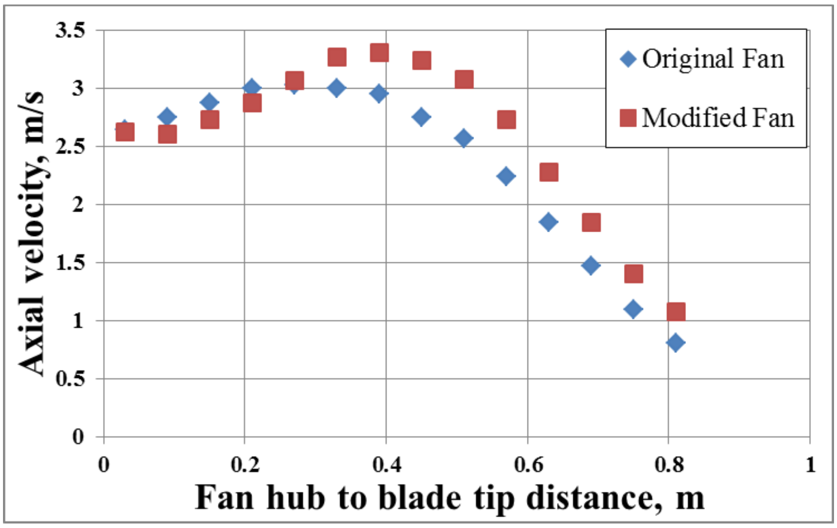


Figure 6-19 Numerical Comparison of Original and Modified Fan at 300 RPM

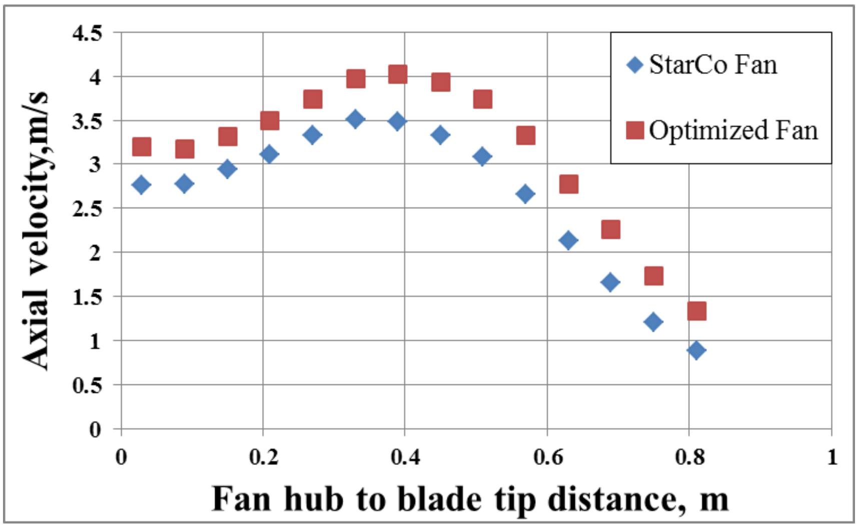


Figure 6-20 Numerical Comparison of Original and Modified Fan at 365 RPM

6.3.5 Performance Analysis

Different scenarios of operation of modified ceiling fan are considered and compared with original ceiling fan as shown in Table 6-1. In the second column, the performance parameters (rated air delivery, torque and RPM) are furnished as a ready reference.

- The Scenario 1 is considered first. The fan is operated at the same RPM to original fan. As a result 30% improvement in the rated air delivery is observed. However, the increase in RAD comes at the cost of increment of mechanical torque of 45%.
- For the scenario 2 case, the RPM of the ceiling fan with modified blades is reduced by 10% to the new value of 330. The reduction in RPM reduces the RAD increase to only 12%. However, the penalization on the mechanical torque is significantly reduced to only 19%. This shows that the 10% reduction in RPM has helped to reduce the mechanical torque by 26%.
- For the scenario 3 case, the RPM reduction is made 18%, i.e. the fan is operating at 300 RPM. The desired criterion for the RAD of 220 is satisfied

without any penalization of torque. Moreover the maximum RPM criteria of 330 is also achieved. The overall reduction of 18% RPM will directly help in significant power savings in spinning the motor.

Table 6-1 Performance Analysis under different scenarios

	PS-1 Standards	Original Fan	Proposed Modification		
			Scenario 1	Scenario 2	Scenario 3
RAD	220 (± 10%)	231.8686228	302.86	259.034	235.0924136
Torque	N/A	2.3238482	3.3736	2.7826207	2.3033063
RPM	330	365	365	330	300
Results			**RAD: +30%** **Torque: +45%** **RPM: 0%**	**RAD: +12%** **Torque: +19%** **RPM: 10%**	**RAD: +1.4%** **Torque: -1.1%** **RPM: 18%**

6.3.6 Cost Analysis for Energy Efficiency

The cost of electricity is calculated by the given formula.

$$\text{Cost(Rs)} = \frac{watts*hours*unit\ price*days}{1000}$$

It is assumed that one fan is utilized on average 13 hours around a year so it saves Rs 339.2675. Similarly Star Co fan is producing 0.1 million ceiling fans in a season so energy saved by these fans will be Rs 33926750 /- means that Pakistan can save energy worth 34 million by modifying the ceiling fan blade of a single SME.

Table 6-2 Cost Analysis Comparison

Sr. No	RPM	Power (Watts)	Unit Price	Cost (Rs)	Comparison
1	365	108	6.5	3330.99	Original Fan
2	300	97	6.5	2991.7225	Modified
Total saving per annum for 1 fan (Rs) 339.2675					
Total saving per annum for 0.1 million fan (Rs) 3,39,26750					

6.4 Concluding Remarks

A modification in the blade geometry for the 56″ ceiling fan class is proposed. The improvement is made without any penalization on the torque. Rather, a negligible improvement of 1.4% is achieved. The modified geometry will satisfy the PS-1 standards for rated air delivery (RAD) as well as maximum RPM. RPM reduction is achieved upto 18%. Substantial power savings without penalizing RAD and Torque are achieved. The mechanical torque with the proposed modification will be higher for the same RPM. However, it is conjectured that the main power consumption is for spinning the hub rather than overcoming the aerodynamic drag.

CHAPTER 7

CONCLUSIONS & FUTURE WORK

7.1 Background

The conclusions drawn from the work presented in the previous chapters can be summarized in three different sections.

7.1.1 Conclusion from experimental studies

Air velocity for seven different RPM was captured in Star Co Fan industry. Dimensions of Room, Fan Model were measured and different instruments functions were understood and few were calibrated. Experimental data is very helpful in developing the computational model with accurate measurements and is validated successfully. The air flow patterns were same for the experimental as well as computational model.

7.1.2 Conclusion from rake & bent angle studies

Aerodynamic efficiency can be increased by increasing the rake angle by one degree in the original one. The bent angle measured in the Star Co Fan industry is the optimized one along with 6^0 rake angle. So the existing bent angle is acceptable for future production.

7.1.3 Conclusion from bent position studies

Bent position plays an important role in improving the present blade design. A modification in the blade geometry for the 56″ ceiling fan class is proposed. The improvement is made without any penalization on the torque. The modified geometry satisfies the PS-1 standards for Rated Air Delivery as well as maximum RPM.

Overall, the RPM is reduced upto 18%. Therefore, substantial power savings without penalizing RAD and Torque are proposed.

7.2 Future Work

Some recommendations for future research based on the work in this dissertation are as follows:

- Nonlinear profile can be generated through Design of Experiments (DOE) and Response Surface Method (RSM).
- Novel Passive Flow Control Strategies can be investigated by including surface roughness and vortex generators on the fan blade to reduce the torque.

BIBLIOGRAPHY

1. ; Available from: http://en.wikipedia.org/wiki/Punkah.
2. *Ceiling Fan Reviews by Ceiling Fantasia*. 06-Dec-2013]; Available from: http://www.ceilingfantasia.com/blog/2008/05/01/ceiling-fantasia-blog/ceiling-fans-throught-the-ages/.
3. Fans, M.a.C.C. 06-Dec-2013]; Available from: http://www.cool-fans.com/2012/01/30/a-brief-history-of-ceiling-fans/.
4. *Story of Philip Diehl*. 06-Dec-2013]; Available from: http://en.wikipedia.org/wiki/Philip_Diehl_(inventor).
5. *Fans by Wikipedia*. 06-Dec-2013]; Available from: http://en.wikipedia.org/wiki/Electric_fan.
6. *The History of Ceiling Fans by Mary Lamphere*. 06-Dec-2013]; Available from: http://voices.yahoo.com/the-history-ceiling-fans-1778605.html?cat=6.
7. SCELFO, J., *Fans That Look Cool*, in *The New York Times*. 2009.
8. Jamieson, T., *CALIFORNIA DESIGN ELEVEN*. 1971: Anderson, Ritchie & Simon.
9. Moore, E., *California Design '76, a Bicentennial Celebration*. 1976: Anderson, Ritchie & Simon.
10. Parrack, D., *Bladeless ceiling fan uses vortex airflow to regulate room temperature by Dave Parrack*, in *gizmag*.

11. (TDAP), T.D.A.o.P., *Engineering Division's Report on Fan Industry of Pakistan*. 2011: Karachi.

12. Danny, S.P., H.S. Guan, and D.H. Bart, *United States Patent*. 2000.

13. Ramadan, B. and S.K. Nadar, *Studying the Features of Air Flow Induced by a Room Ceiling-Fan.* Energy and Buildings, 2011. **43**.

14. Ankur, J., et al., *Experimental Investigation of the Flow Field of a Ceiling Fan.* ASME Heat Transfer/Fluid Engineering, 2004.

15. Momoi, Y., et al., *Modeling of Ceiling Fan Based on Velocity Measurement for CFD Simulation of Airflow in Large Room*.

16. Watanabe, T., et al., *Simulation of Steady and Unsteady Cavitation on a Marine Propeller using a RANS CFD Code*, in *Fifth International Symposium on Cavitation*. 2003: Osaka, Japan.

17. Singhal, A.K., et al., *Mathematical Basis and Validation of the Full Cavitation Model.* ASME, 2002. **124**.

18. K.Schmidth and D.J. Patterson, *Performance Results for a High Efficiency Tropical Ceiling Fan and Comparisons with Conventional Fans: Demand Side Management via Small Appliance Efficiency.* Renewable Energy, 2001. **22**(1-3).

19. Ki-Sang, L., Y.K.K. wang, and A. samad, *Design Optimization of Low-Speed Axial Flow Fan Blade with Three-Dimensional RANS Analysis.* Mechanical Science and Technology, 2008. **22**.

20. Falahat, A., *Numerical and Experimental Optimization of Flow Coefficient in Tubeaxial Fan.* International Journal of Multidisciplinary Sciences and Engineering, 2011. **2**.

21. F.H. Rohles, S.A.K., B.w. Jones. *Ceiling Fans as Extenders of the Summer Comfort Envelope*. 1983: ASHRAE.

22. F.H. Rohles, J.E.L., T.E. Shrimlin. *Assessing Air Velocities from the Industrial Ceiling Fans*. 1983: ASHRAE.

23. Son, H.H., R. Luis, and M.R. Muhammad, *Thermal Comfort Enhancement by Using a Ceiling Fan.* Applied Thermal Engineering, 2009. **29**.

24. *Computational Fluid Dynamics Software Package 6.2.16*. 2006: New Hampshire.

25. *Geometry and Mesh Generation Software Package, Ver. 2.3.16*. 2006.

26. Spalart, P.R.a.A., S. R., *A One-Equation Turbulence Model for Aerodynamic Flows*, in *AIAA*. 1992, AIAA.