Design and CFD Analysis of Bladeless Ceiling Fan

By

Haseeb Aslam

Reg # 00000118972

Session 2016-2018

Supervised by

Dr. Majid Ali

A Thesis Submitted to the US Pakistan Centre for Advanced Studies in Energy in partial fulfilment of the requirements of the degree of

MASTERS of SCIENCE

in

THERMAL ENERGY ENGINEERING

US-Pakistan Centre for Advanced Studies in Energy (USPCAS-E)

National University of Sciences and Technology (NUST)

H-12, Islamabad 44000, Pakistan

August, 2020

THESIS ACCEPTANCE CERTIFICATE

Certified that final copy of MS/MPhil thesis written by **Mr. Haseeb Aslam** (Registration No. 118972), of USPCAS-E has been vetted by undersigned, found complete in all respects as per NUST Statues/Regulations, is within the similarity indices limit and is accepted as partial fulfillment for the award of MS/MPhil degree. It is further certified that necessary amendments as pointed out by GEC members of the scholar have also been incorporated in the said thesis.

Signature: ______________________________

Name of Supervisor Dr. Majid Ali

Date: ________________________

Signature: ______________________________

Name of Co-Supervisor Dr. Adeel Javed

Date: ________________________

Signature (HoD TEE): ________________________

Date: ____________________________________

Signature (Dean/Principal): __________________

Date: ____________________________________

Certificate

This is to certify that work in this thesis has been carried out by **Mr. Haseeb Aslam** and completed under my supervision in US-Pakistan Center for Advanced Studies in Energy (USPCAS-E), National University of Sciences and Technology, H-12, Islamabad, Pakistan.

Supervisor: ________________

Dr. Majid Ali
USPCAS-E
NUST, Islamabad

Co-Supervisor: ________________

Dr. Adeel Javed
USPCAS-E
NUST, Islamabad

GEC member 1: ________________

Dr. Adeel Waqas
USPCAS-E
NUST, Islamabad

GEC member 3: ________________

Dr. Muhammad
Bilal Sajid
SCME
NUST, Islamabad

HOD-TEE: ________________

Dr. Adeel Javed
USPCAS-E
NUST, Islamabad

Principal: ________________

Dr. Adeel Waqas
USPCAS-E
NUST, Islamabad

Dedication

I dedicate my thesis to my beloved parents, siblings, and friends for their affection and support during my studies. Especially to my mother for always believing in me, praying for me and giving me courage.

Acknowledgment

Firstly, I would like to thank Allah Almighty for providing me knowledge, determination, opportunity, and strength to complete this venture. Without His blessings, all of it would not have been possible.

I would like to express my gratitude to my supervisor Dr Majid Ali for giving me the opportunity to conduct thesis under his supervision.

I would also like to thank my co-supervisor Dr. Adeel Javed for providing guidance and motivation throughout the research and study.

I also want to thank my GEC committee members Dr. Adeel Waqas and Dr. Muhammad Bilal Sajid. I feel proud and honored that you have accepted to be on my committee

I wish to acknowledge Syed Muneef Ali Shah and Muhammad Zulqarnain Arif for their support throughout the project.

Finally, and most importantly, I express my profound gratitude to my parents for providing me with unfailing support and continuous encouragement throughout my years of study and through the process of researching and writing this thesis.

Abstract

Conventional ceiling fans use convective effect to distribute air to surroundings. The fan blades deflect the air downwards to produce a draft of air resulting in cooling of the room. However, this draft is delivered in form of gusts along with an uncomfortable noise induced by the blade-air interaction and resulting pressure fluctuations due to turbulence and unsteady flow phenomenon. Bladeless fans provide excellent solution in this regard. Bladeless fans are type of fans which do not contain visible blades or vanes. A motor rotor or radial impeller is used to direct flow inside the geometry of bladeless fan which is multiplied at outlet using Coanda effect. Currently, bladeless fans being used are of desk type providing sufficient air for one to two people. For air supply to the whole room, application of bladeless fan as a ceiling fan is studied in this paper. Bladeless fan is designed using Eppler 473 airfoil in SolidWorks. Flow of bladeless fan is simulated inside a room of dimensions 3.5m×3.5m room. Parametric analysis of bladeless fan is performed in ANSYS FLUENT to find optimum conditions at which bladeless fan yields maximum volumetric flow at outlet. Numerical model is validated by comparing numerical results to experimental model. Results showed that volumetric flow rate of bladeless ceiling fan increases with increase in fan to ceiling distance up to 0.3m and start decreasing as ceiling to fan distance increases beyond 0.3m. The volumetric flow rate of bladeless fan increases with increase in fan diameter and decreases with increase in nozzle diameter. Numerical results also showed that bladeless fans have similar outlet flow field as conventional ceiling fans with sufficient volumetric flow rate to be used as ceiling fans.

Table of contents

List of Figures

List of Tables

Nomenclature

Latin

k= turbulent kinetic energy

p = static pressure

t = time

u = fluid velocity

x,y,z = cartesian coordinate

Greek

ε = dissipation rate of turbulent kinetic energy

μ = turbulent viscosity

ρ = density

τ_{ij} = Shear stress tensor

Indices

i,j,k = indices for Cartesian tensor

t = turbulence flow

1,2,3 = indices for direction x,y,z

Chapter 1

Introduction

1.1 Background

During summer season, ambient temperature increases resulting in thermal discomfort for human beings and other living organisms. To avoid thermal discomfort, indoor spaces as well as certain semi outdoor spaces are cooled using various methods such as heat pumps, air conditioning, fans etc. In developing countries, a cost effective and reliable source for cooling is needed that can provide occupants with enhanced thermal comfort at low energy consumption. While air conditioning has certain benefits over fans as it decreases temperature and controls humidity, it is not affordable for majority of population in developing countries due to high operating costs. However, fans, being cheap and having low operating costs are ideal solution for cooling indoor environment. In 2000, fans accounted for 6% of total energy consumption in India which may increase up to 9% in 2020. Previous studies have also showed that using ceiling fans improve perceived air quality, sensation of thermal comfort and productivity [1], [2].

However, fans are used way beyond just cooling our households. Anywhere where we need to move air for any kind of operation, we will need fans. A brief introduction of fans and its types is given in the coming sections

1.2 Fan

A fan is a powered machine used to produce motion in a gas, typically air. Fans have vanes or blades that rotate to induce motion in a gas. This rotating assembly of blades is also sometimes known as impeller, rotor or runner. They are often housed in a hub or scroll for safety purposes. Most fans are powered by electric motors but other sources such as hydraulic motor, internal combustion engines etc. can be used.

Mechanically, any arrangement of revolving vanes or generally any device that produces currents of air can be classified as fans. Fans produce high volume flow rates and low pressure as compared to compressors which can produce large pressure change. Fans blowers and compressors are differentiated on the basis of method used to move air and by system pressure they must operate against. As per American society of Mechanical engineers, specific ratio, i.e., ratio of discharge pressure to suction pressure is used for defining fans and blowers.

Equipments	Specific Ratio	Pressure Rise (mmWg)
Fans	Up to 1.11	1136
Blowers	1.11 to 1.20	1136-2066
Compressors	More than 1.20	

Table 1.1 Classification of Fans according to pressure Rise

While fans are usually used for cooling purposes, they do not provide cooling by reducing temperature of ambient air; instead they move the air to enhance evaporating, cooling of sweat as well as increasing heat transfer from body by increasing convective effect of air. Thus, fans may become ineffective at places where surrounding air is reaching body temperature or in areas of high humidity.

1.3 Types of Fans

There are six types of fans

1. Axial-flow fans
2. Centrifugal fans
3. Axial-Centrifugal fans
4. Roof Ventilators
5. Cross-flow Blowers
6. Vortex or regenerative blowers

1.3.1 Axial Flow Fans

An axial fan is a type of fan that causes gas or air to flow through it in an axial direction, parallel to the shaft about which blades rotate. The flow is axial at entry and exit. The fan is designed to produce a pressure difference, and hence force, to cause a flow through the fan. Factors which determine the performance of the fan include number and shape of the blades.

1.3.2 Centrifugal Fans

A centrifugal fan is a mechanical device for moving air or other gases at an angle to the incoming fluid. Centrifugal fans use the kinetic energy of the impellers to increase the volume of the air stream which in turn moves against the resistance caused by ducts, dampers or other components. Centrifugal fans displace the air radially, changing the direction of the airflow.

1.3.3 Axial Centrifugal Fans

These fans are also called tubular centrifugal fans, in-line centrifugal fans, or mixed flow fans (especially if the fan has a conical back plate). The following two types of fan wheels are used in these fans

1. A fan wheel with a flat back plate. When used in centrifugal axial fans, however the air has to make two 90^o turns, which results in some extra losses.
2. A fan wheel with a conical back plate. This fan wheel is more expensive to build, but the air stream here has to make only two 45^o turns, a more efficient arrangement.

In either case, the fan wheel usually has backward inclined blades or occasionally airfoil or backward curved blades.

1.3.4 Roof Ventilators

Various models of roof ventilators are in common use. Some may have belt drive instead of direct drive, some may have axial fan wheels instead of centrifugal fan wheels, and some may be for up blast instead of radial discharge. While most models are for exhausting air out of the building, some are for supplying air into the building.

1.3.5 Cross Flow Blowers

A cross flow blower is a unique type of centrifugal fan in which the airflow passes twice through a fan wheel with Forward curved blading, first inward and then outward. The main advantage of cross flow blowers is that they can be made axially wider, in fact to any width desired. This makes them practically suitable for air curtains, long and narrow heating or cooling coils, and dry blowers in car wash.

1.3.6 Vortex or Regenerative Blowers

The vortex regenerative blower is another unique type of centrifugal fan. Here the airflow circles around in an annular, torus shaped space, similar to the shape of doughnut. On one side of the torus are rotating fan blades, throwing the air outward. The airflow is then guided back inward by the other side of the torus so that it must reenter the inner portion of the rotating blades.

1.4 Ceiling Fan

A ceiling fan is a mechanical fan that is mounted to the ceiling. It is a type of propeller fan wheel without any shroud or casing. Ceiling fans are a type of propeller fan without any scroll or shroud. Ceiling fans produce cooling effect

Figure 1.1 Ceiling Fan [1]

by discharging air in the downward direction which increases convective heat transfer from the body resulting in a cooling sensation. Furthermore, circulating air causes evaporation of moisture on the skin which can also lead to a cooling sensation.

1.4.1 Working of Ceiling Fan

Ceiling fans have three blades or vanes, typically at a pitch angle of 10^o to 15^o angle from hub to tip. The hub diameter comprises of about 15% of the total fan diameter. An electric motor is used to rotate prime mover of ceiling fan which rotates hub and blades attached to the hub. When blades of the fan rotate in counterclockwise direction, it creates a suction effect above the ceiling fan which leads to discharge of air from top of the ceiling fans to the floor in downward direction. Velocity of air stream moving downward is typically low ranging from 3 m/sec just below the fan to 0.5m/sec just above the floor level. This air glides along the floor to the walls of the room and circulates upward towards the ceiling creating two parallel loops in opposite directions. Ceiling fans are type of axial fans (propeller fans) which means they can handle large volumes of air at relatively low pressure. Their size varies from 36 to 72 inches, capable of handling airflow between 6000 and 72000cfm. Depending on speed, they may consume power between 60 and 200W. They run at relatively low rpm of 270 to 400 as compared to other type of axial fans.

1.5 Bladeless Fan

In 2009, P.D Gammack, Fredric Nicolas and Kevin John Simmonds [3] invented a revolutionary type of fan which has no visible blades and can multiply air flow at the outlet side by up to 15 times utilizing Coanda effect. This fan was named

Figure 1.1 Bladeless Fan [3]

as Bladeless Fan/ Air Multiplier.

1.5.1 Geometry of Bladeless Fan

Bladeless fan assembly consists of two parts

1. Nozzle
2. Means for creating an airflow through the nozzle

The nozzle further comprises of an interior passage, a mouth for receiving air flow from the interior passage and a Coanda surface located adjacent to the mouth and over which the mouth is arranged to direct the air flow. For creating airflow in a bladeless fan, a motor rotor or a radial impeller can be used.

1.5.2 Coanda Effect

A Coanda surface is a type of surface over which the fluid flowing exhibits Coanda effect. Coanda effect is the tendency of a fluid jet to stay attached to a convex surface.

A free jet of air entrains molecules of air from surrounding causing axisymmetric tube of low pressure around the jet. The resultant forces from this low pressure end up balancing any perpendicular flow instability which stabilizes flow in a straight line. If a horizontal surface is placed close to the jet in a parallel direction then the entrainment and subsequent removal of air from one side results in pressure loss causing flow to distort toward the horizontal surface thus exhibiting Coanda effect. A slight change in curvature will allow jet to move alongside the surface because each incremental change in direction brings about the effects described for the initial bending of jet towards the surface. Adding a small lip at the point where jet starts to flow further increases bending towards the horizontal surface due to creation of low-pressure vortex behind the lip.

Applications of Coanda surface include high lift-devices on aircraft where air moving over the wing can be bent down towards ground using flaps and a jet sheet blowing over the curved surface on top of the wing. The bending of the flow results in aerodynamic lift.

1.5.3 Working of Bladeless Fan

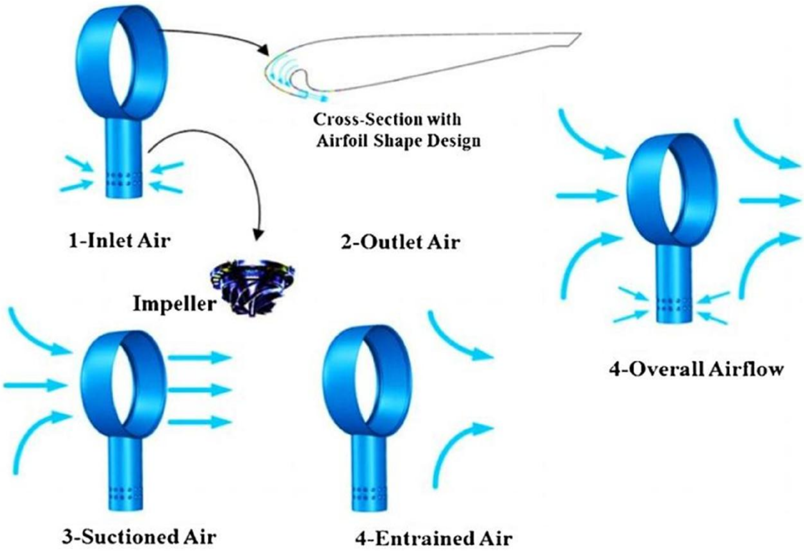


Figure 1.2 Working of Bladeless Fan [4]

A mixed flow impeller connected to a DC brushless motor is used to direct flow into the fan assembly. Mixed flow impeller is a type of impeller that transfers the flow diagonally. Mixed flow impeller rotates to create suction of air from surrounding area. Air is sucked horizontally into the fan assembly and is directed vertically into the interior passage. The interior passage in bladeless fan is circular in shape with hollow inside. The cross section of interior passage is designed by revolving a NACA 0012 airfoil for minimizing flow losses inside the interior passage and maximizing Coanda effect outside bladeless fan assembly. Air entering the interior passage surrounds all the area available and is directed outwards horizontally through the nozzle shaped mouth onto the Coanda surface. Air exiting from the mouth will now be referred as primary flow. Primary air exiting the mouth of bladeless fan exhibits Coanda effect and creates low pressure in the surrounding of the bladeless fan. To fill this low-pressure area air from surrounding (referred from now on as secondary air flow) rushes towards the primary airflow. The surrounding airflow is drawn from the room space, region or external environment surrounding the mouth of the nozzle and by displacement, from other regions around the fan assembly.

The primary airflow combined with the secondary airflow gives the total airflow emitted or projected forward to a user from the opening defined by the nozzle.

The total airflow is sufficient for the fan assembly to create an air current suitable for cooling.

1.6 Bladeless ceiling fan

Bladeless ceiling fan is a type of bladeless fan that can be mounted on ceiling of a room. Desk type bladeless fans with their many advantages are suitable for use by one to three people depending on seating arrangement. By mounting bladeless fan on a ceiling, bladeless fan can be used to supply air flow to the room. This is one of the objectives of this research; to find whether bladeless ceiling fan can supply air to whole room like a ceiling fan and have inherent advantages provided by ceiling fan. Volumetric flow multiplication will also be observed in this research and different parameters will be tested to find suitable parameters with best flow distribution, less noise and maximum volumetric flow rate.

1.7 Problem Statement

Ceiling fans, due to their air-blade interaction produce choppy airflow in the form of gusts, accompanied by an uncomfortable noise. Bladeless fans can provide an excellent solution in this regard. However, bladeless fans currently available are of desk type and supply air to only a specific portion of the room. A bladeless ceiling fan can have inherent advantages of bladeless fans with the capacity to supply air to the whole room.

1.8 Objectives

Objectives of this research are as follows

1. Design a bladeless ceiling fan for a room.
2. Flow field analysis of bladeless ceiling fan using ANSYS CFD
3. Optimization of bladeless ceiling fan using CFD results
4. Experimental analysis of bladeless ceiling fan

1.9 Thesis outline

Thesis outline of this chapter and subsequent chapters is given below.

Chapter 1

A brief introduction on fan, types of fan and ceiling fan is given. Bladeless fans and their working is also explained in this section.

Chapter 2

Literature review on different papers published related to bladeless fans and ceiling fan is provided in this chapter

Chapter 3

This chapter gives a brief introduction on ANSYS Fluent and governing equations that are used to carry out simulations are discussed. Moreover, a methodology flow chart is also given at the end of this chapter.

Chapter 4

An explanation on what modules are present in ANSYS Fluent and how they are used in this research to perform parametric analysis of bladeless ceiling fan is provided in this chapter.

Chapter 5

Result of parametric analysis performed during chapter 4 and explanation behind the behavior observed in parametric analysis is discussed in this chapter. Outlet flow field of bladeless ceiling fan is also compared with conventional ceiling fan.

Summary

A brief introduction on fans and different types of fans and their uses is provided in this chapter. Ceiling fan and its working is discussed. Bladeless fans and their working is explained in detail in this chapter. Coanda effect, a method for air entrainment, and inducement effect leading to increase in volumetric flow at exit of bladeless fan are also discussed in this chapter. In the end a brief summary of next chapters is provided.

References

[1] W. Chen *et al.*, "Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan," *Build. Environ.*, vol. 130, pp. 14–26, 2018, doi: 10.1016/j.buildenv.2017.12.016.

[2] S. H. Ho, L. Rosario, and M. M. Rahman, "Thermal comfort enhancement by using a ceiling fan," *Appl. Therm. Eng.*, vol. 29, no. 8–9, pp. 1648–1656, 2009, doi: 10.1016/j.applthermaleng.2008.07.015.

[3] I. Peter, D. Gammack, F. Nicolas, K. J. Simmonds, and M. Gb, "(12) United States Patent," vol. 2, no. 12, 2008.

[4] M. Jafari, H. Afshin, B. Farhanieh, and A. Sojoudi, "Numerical investigation of geometric parameter effects on the aerodynamic performance of a Bladeless fan," *Alexandria Eng. J.*, vol. 55, no. 1, pp. 223–233, 2016, doi: 10.1016/j.aej.2015.11.001.]

Chapter 2

Literature review

2.1 Numerical Analysis of Bladeless Fan

M. Jafari et al.[5] performed CFD analysis on aerodynamic performance of a bladeless fan by changing five geometric parameters of Bladeless fan. These five geometric parameters are

1. Height of cross section of fan
2. Outlet angle of flow relative to the fan
3. Thickness of airflow outlet slit
4. Hydraulic diameter
5. Aspect ratio of circular and quadratic cross sections

They selected an Eppler 473 airfoil for designing cross section of a fan and placed the bladeless fan in a 4x2x2 m room. Standard k- ε turbulence model was used to perform numerical simulations. These numerical investigations lead to conclusion that thickness of airflow outlet slit (nozzle thickness) is most important and influential parameter for the aerodynamic performance of a bladeless fan. According to simulation results, discharge ratio of bladeless fan increased considerably when nozzle thickness decreased. However, there should be a compromise between generated noise and discharge ratio as decreasing nozzle thickness resulted in increased noise.

Jafari et al.[4] also performed numerical aerodynamic evaluation of bladeless fan via finite volume method as well as noise investigation of bladeless fan by solving continuity and momentum equations and solving noise equations of broadband noise source and Ffowes Williams and Hawkings in both steady and unsteady conditions. They again selected an Eppler 473 airfoil for fan cross section and modeled a 30cm bladeless fan for a 4x2x2 m room. A multiplication ratio of 21 was observed as compared to 15 for NACA 0012 Airfoil used by Dyson. The aero acoustic results showed that produced noise increases with increase in inlet volume flow rate.

2.2 Experimental Analysis of Bladeless Fan

To investigate applications of bladeless fans in industrial use, M. Jafari et al. [6] designed and constructed a 60cm diameter bladeless fan with 6mm outlet slit. At first the flow field of bladeless fan was studied using numerical methods. Thickness of outlet slit had extreme effect in the exit airflow rate. Experimental and numerical results showed an increased volumetric flow rate of 7.2 and 8.3 respectively. The results indicated that outlet flow includes 8.5% inlet flow, 53% sucked air from upstream flow and38.5% from surrounding air. They concluded that large diameter bladeless fans can be applied in various industrial settings such as removing smoke from tunnels.

Hong li et al. [7]also performed numerical and experimental research on the outlet flow field of bladeless fan using RNG K-ε turbulence model and validated numerical results experimentally using constant temperature anemometer and ventilating multi parameter testing instrument. Their research lead to conclusion that the flow parameter distribution in the outlet flow field of the air multiplier are not uniform. The time average velocity of upper zone is greater than the lower zone and pressure in near field is lower than far field.

Hong li et al. [8] also investigated the influence of Reynolds number on the outlet flow field of a bladeless fan. The outlet flow field was investigated at five Reynolds number ranging from 28200 to 40100. Based on their experimental research, Hong li et al. concluded that

1. For each cross section along the axis, the time averaged velocity in the horizontal and vertical direction, all increase with increasing Reynolds number.
2. The time averaged velocity and turbulence intensity on the axis, all increase with increasing Reynolds number as well
3. The complete flow field structure has been obtained as follows

a) The annular jet is ejected outward from the annular slot , entraining the ambient air.

b) Around the axial position of 1.5d, the jet starts to converge, lower part of the flow starts drawing towards the upper part of the flow.

c) This combination finishes at about 3d. However, the flow develops like a classical single jet with velocity peak values decaying and decaying slope increasing with increasing Reynolds number.

4. After finishing self adjustment, the velocity and turbulence intensity reaches self similarity at about 3.5d.

2.3 Flow Field Analysis of Ceiling Fan

Although no research on bladeless ceiling fan has been carried out, extensive research on outlet flow field and thermal comfort provided by ceiling fans is available in literature. Son H. Ho et al. [2] performed 2D numerical analysis on a fan being used in an air conditioned room. With fan being used, strong circulations of air enhance convective heat transfer but restrict removal of hot air from room rather circulating it causing a rise in overall temperature of the room. But as the fan speed increases, the enhanced convection allows for more cooling effect, allowing higher heat load inside while maintaining same level of thermal comfort. This leads to higher energy savings, allowing the user to increase the temperature setting of air conditioning system.

Wenhua chen et al. [1] performed experimental and numerical investigation of indoor air movement with an office ceiling fan. They investigated air movement in an office room by changing different parameters of room. These are

1. Fan rotational speed
2. Fan blade geometry
3. Ceiling to fan depth
4. Ceiling height

Effects of above given parameters on indoor air distribution are as follows

1. Increasing rotational speed of fan increases air speed in the room. Velocity self similarity exists in the main jet zone of ceiling fan for each rotational speed
2. By comparing experimental results, it is implied that standard k-ε turbulence model can suitably predict ceiling fan driven flow pattern when incorporated with Multi reference frame.

3. Blade geometry and ceiling to fan distance may have significant effect on velocity distribution in the area directly below the fan (main jet zone). However, air speed is slightly affected by changing blade design.
4. For the occupied zone (Region in room excluding main jet zone) ceiling height and rotational speed are more significant than blade geometry and ceiling to fan distance.

Shou liu et al.[9] carried out experimental investigations of air speed field induced by ceiling fans. They also found that jet shape remains same at different air speeds however as speed increases, airflow towards the floor increases, resulting in stronger jet and acceleration of flow circulation. For multiple fan case, airflow depends on distance between the fans and speed difference.

Ramadan Bassiouny and Nader S. Korah [10] studied airflow induced by a room ceiling fan using computational methods. Their results showed a downward flow zone having the highest velocities in a room. Moreover, they also concluded that increasing fan's rotational speed will increase downward draft.

Yunfei Gao et al,[11] used experimental setup to increase air speeds around desks and office partitions. They used different table and partition configurations and compared them to airflow inside unoccupied room. They noted that airflow doesn't start spreading until it reaches a distance of 0.3m to 0.5m to the floor. The air speeds above 0.6m height in the area under fan blade diameter are high and outside the region, they are low. In vertical direction, are located within fan diameter and continue to exist 0.6m above the floor.

Yongchao Zhai et al.[12] collected data from 16 human subjects to access human comfort and perceived air quality for warm and humid environment with the use of ceiling fan. The results are as under

1. Using ceiling fans increased thermal comfort of human subjects and increased acceptance of environmental air movement and humidity.
2. Air speed at 1.2 m/sec at head level provided comfort at 30^{o}C and 80% relative humidity.
3. The acceptable air speed increased ASHRAE upper speed limit (0.8 m/sec).

Summary

This chapter contains literature review on two different types of articles. One is relevant research performed on bladeless fan. Bladeless fans were introduced in 2009, so literature for bladeless fan is limited. Parametric as well as experimental analysis is performed on bladeless fan which is discussed in this chapter. The other part is flow field and thermal comfort provided by ceiling fans. Literature review of these researches is also discussed in this chapter.

References

[1] W. Chen *et al.*, “Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan,” *Build. Environ.*, vol. 130, pp. 14–26, 2018, doi: 10.1016/j.buildenv.2017.12.016.

[2] S. H. Ho, L. Rosario, and M. M. Rahman, “Thermal comfort enhancement by using a ceiling fan,” *Appl. Therm. Eng.*, vol. 29, no. 8–9, pp. 1648–1656, 2009, doi: 10.1016/j.applthermaleng.2008.07.015.

[3] I. Peter, D. Gammack, F. Nicolas, K. J. Simmonds, and M. Gb, “(12) United States Patent,” vol. 2, no. 12, 2008.

[4] M. Jafari, H. Afshin, B. Farhanieh, and H. Bozorgasareh, “Numerical aerodynamic evaluation and noise investigation of a Bladeless fan,” *J. Appl. Fluid Mech.*, vol. 8, no. 1, pp. 133–142, 2015.

[5] M. Jafari, H. Afshin, B. Farhanieh, and A. Sojoudi, “Numerical investigation of geometric parameter effects on the aerodynamic performance of a Bladeless fan,” *Alexandria Eng. J.*, vol. 55, no. 1, pp. 223–233, 2016, doi: 10.1016/j.aej.2015.11.001.

[6] M. Jafari, H. Afshin, B. Farhanieh, and H. Bozorgasareh, “Experimental and numerical investigation of a 60cm diameter bladeless fan,” *J. Appl. Fluid Mech.*, vol. 9, no. 2, pp. 935–944, 2016, doi: 10.18869/acadpub.jafm.68.225.24081.

[7] H. Li, H. S. Deng, and Y. Bin Lai, “Numerical and experimental research on the outlet flow field for the air multiplier,” *Appl. Therm. Eng.*, vol. 93, pp. 652–659, 2016, doi: 10.1016/j.applthermaleng.2015.10.029.

[8] H. Li, X. H. Jin, H. S. Deng, and Y. Bin Lai, “Experimental investigation on the outlet flow field structure and the influence of Reynolds number on the outlet flow field for a bladeless fan,” *Appl. Therm. Eng.*, vol. 100, pp. 972–978, 2016, doi: 10.1016/j.applthermaleng.2016.02.128.

[9] S. Liu, A. Lipczynska, S. Schiavon, and E. Arens, “Detailed experimental investigation of air speed field induced by ceiling fans,” *Build. Environ.*, vol. 142, no. June, pp. 342–360, 2018, doi:

10.1016/j.buildenv.2018.06.037.

[10] R. Bassiouny and N. S. Korah, “Studying the features of air flow induced by a room ceiling-fan,” *Energy Build.*, vol. 43, no. 8, pp. 1913–1918, 2011, doi: 10.1016/j.enbuild.2011.03.034.

[11] Y. Gao *et al.*, “Ceiling fan air speeds around desks and office partitions,” *Build. Environ.*, vol. 124, pp. 412–440, 2017, doi: 10.1016/j.buildenv.2017.08.029.

[12] Y. Zhai, Y. Zhang, H. Zhang, W. Pasut, E. Arens, and Q. Meng, “Human comfort and perceived air quality in warm and humid environments with ceiling fans,” *Build. Environ.*, vol. 90, pp. 178–185, 2015, doi: 10.1016/j.buildenv.2015.04.003.

Chapter 3

Numerical Modeling

All simulations were performed using computational fluid dynamics software ANSYS Fluent. ANSYS Fluent is commercial software that solves conservation equations of mass and momentum to simulate compressible and incompressible fluid flows. Air flowing below Mach 0.3 is considered incompressible, so steady state incompressible flow analysis is performed in this research. Due to geometry of bladeless ceiling fan being symmetric (with few minor modifications), 2D analysis of bladeless ceiling fan is carried out inside a 2D room of 3.5m x 3.5m room. While standard k-ε model with standard wall conditions is used to simulate turbulent flow [1].

3.1 Governing Equations

Conservation of mass and momentum are used as governing equations to simulate flow inside the room.

3.1.1 Continuity Equation

Continuity equation states the rate at which mass enters the system is equal to the mass leaving the system plus mass accumulated in the system. Conservation of momentum equation states that momentum of an isolated system is a constant. Continuity equation for a three-dimensional fluid flow is given as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \qquad (1)$$

Where ρ is air density and u_i is velocity in direction i, and i=1,2,3… Momentum equation is given by

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \qquad (2)$$

Where p is the static pressure and τ_{ij} is the stress tensor.

3.1.2 Transient Formulation

Standard k-ε turbulence model with standard wall function is used to simulate turbulent air flow inside the room. Standard turbulence model is a two-equation model with transport equations for turbulent kinetic energy k and dissipation ε [2].

Turbulent kinetic energy is given by

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\rho\varepsilon-Y_M+S_k \tag{3}$$

P_k is the production of turbulent kinetic energy. Y_M represents contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate. S_k is modulus of the mean rate-of-strain tensor for turbulent kinetic energy. For

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\epsilon}\frac{\varepsilon}{k}P_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{4}$$

dissipation ε,

S_ε represents the mean rate-of-strain tensor for dissipation rate Turbulent viscosity is modeled as

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{5}$$

C_μ, $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k and σ_ε are constant parameters where C_μ= 0.09, $C_{1\varepsilon}$= 1.44, $C_{2\varepsilon}$= 1.92, σ_k= 1 and σ_ε= 1.3.

3.2 Methodology

Methodology followed throughout numerical simulations and performing analysis is given in the following figure 3.1.

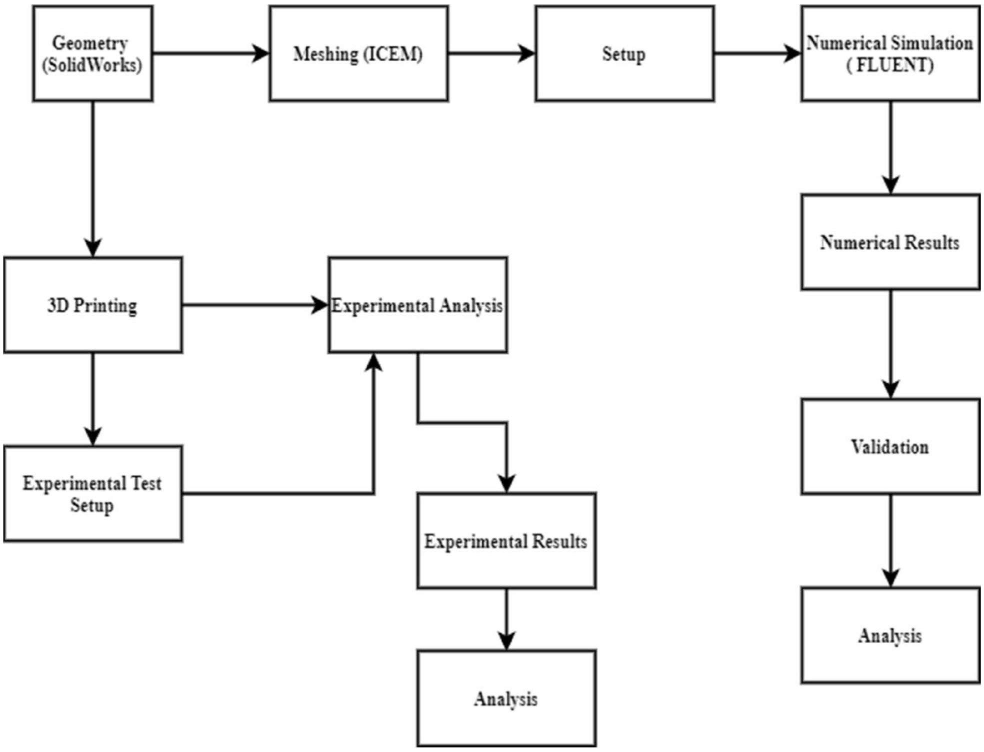


Figure 3.1 Methodology Flow Chart

Geometry of bladeless fan is drawn in SolidWorks, which is then exported to ICEM. Meshing of bladeless fan is done using blocking method. Mesh is exported to setup module of FLUENT where boundary conditions are applied to perform numerical simulation. After the results are obtained, they are validated by comparing them with previously published experimental results.

For experimental analysis, geometry constructed in SolidWorks is 3D printed. An experimental test setup is fabricated. Bladeless ceiling fan is experimentally tested and analyzed. The above-mentioned steps in flow chart are applied and thoroughly explained in subsequent chapters.

Summary

In this chapter, equations governing fluid flow in computational fluid mechanics are discussed. Standard k-ε turbulence model, which is a two-equation model using for simulating flows inside a room is discussed. Finally, a flowchart describing methodology of how simulation is carried out in fluent to calculate volumetric flow rate and velocity contours.

References

[1] N. C. Markatos, "The mathematical modelling of turbulent flows," *Appl. Math. Model.*, vol. 10, no. 3, pp. 190–220, 1986, doi: 10.1016/0307-904X(86)90045-4.

Chapter 4

Numerical Analysis Setup

The main purpose of our project was to design a bladeless ceiling fan, study its flow characteristics using different numerical techniques and perform parametric analysis to find optimum conditions at which bladeless fan will give maximum volumetric flow rate.

4.1 Geometry of Bladeless Ceiling Fan

Bladeless fan geometry was designed in SolidWorks using an Eppler 473 airfoil. Eppler 473 is symmetric airfoil suitable for low Reynolds number. Coordinates of Eppler 473 airfoil were imported from airfoiltools.com to SolidWorks, Fig 4.1.

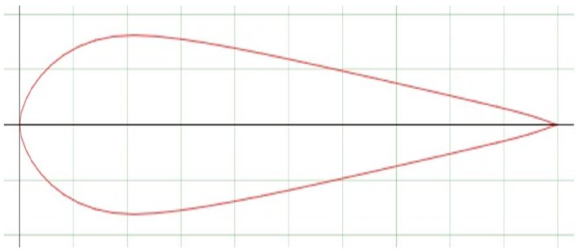


Figure 4.1 Eppler 473 Airfoil [1]

Eppler 473 airfoil was then modeled in the shape of bladeless fan having a length of 10cm and thickness of 1.68cm. Coanda surface was adjusted for maximum airflow, having no sharp edges. A small camber was given at the tail end of airfoil to allow air to diverge after leaving bladeless fan geometry.

After completing bladeless fan geometry on a single airfoil, airfoil was mirrored at a distance of 40cm to create 2D cross section of bladeless fan as shown in Fig 4.2. A room of 3.5m × 3.5m was constructed by drawing a square of same dimensions in SolidWorks around the bladeless fan. 2D plane effect was applied to square and bladeless fan geometry was subtracted from room. Geometry of bladeless fan saved in *.igs format and imported into the design modeler of ANSYS FLUENT.

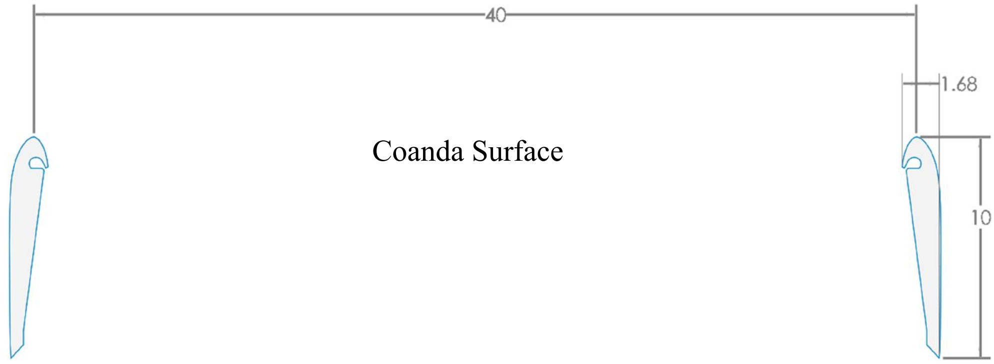


Figure 4.2 Geometry of Bladeless Ceiling Fan

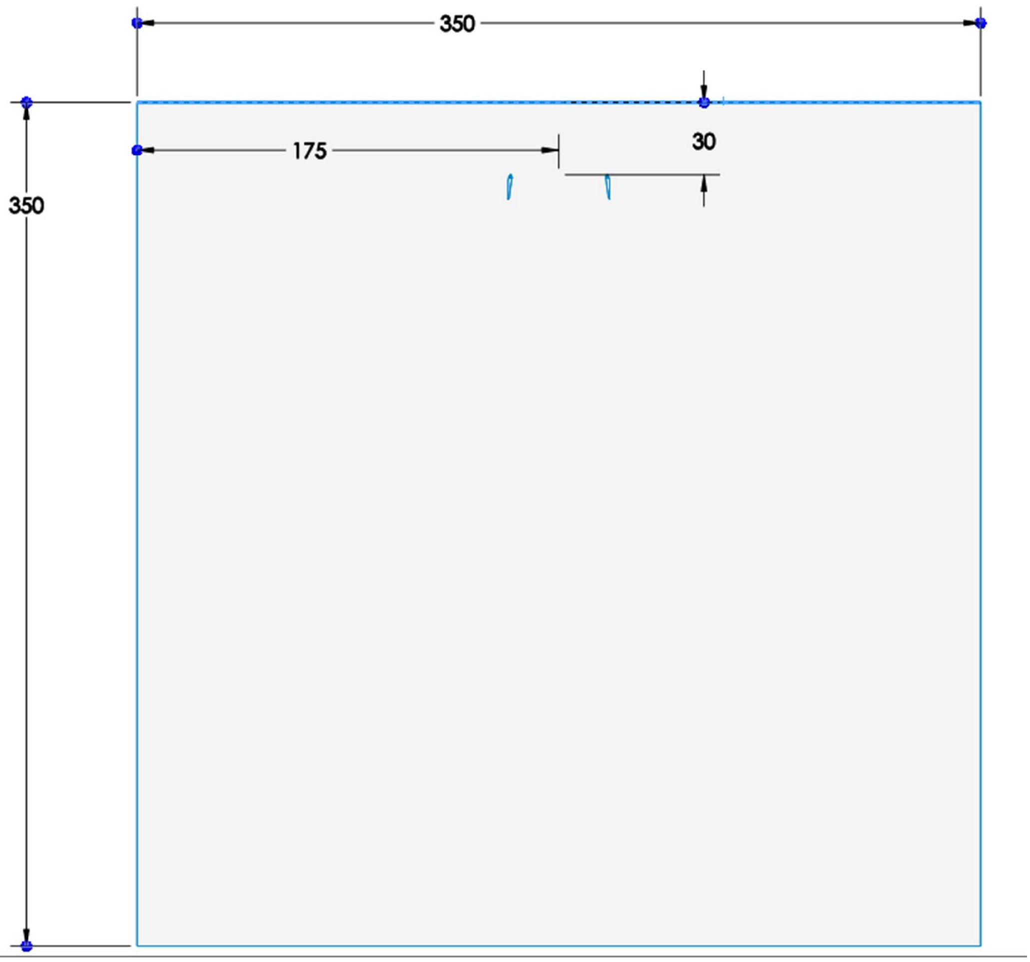


Figure 4.3 Bladeless Ceiling Fan in Room

4.2 Modules of ANSYS FLUENT

ANSYS FLUENT is commercially available computing software for simulating fluid flows using mass and momentum equations.

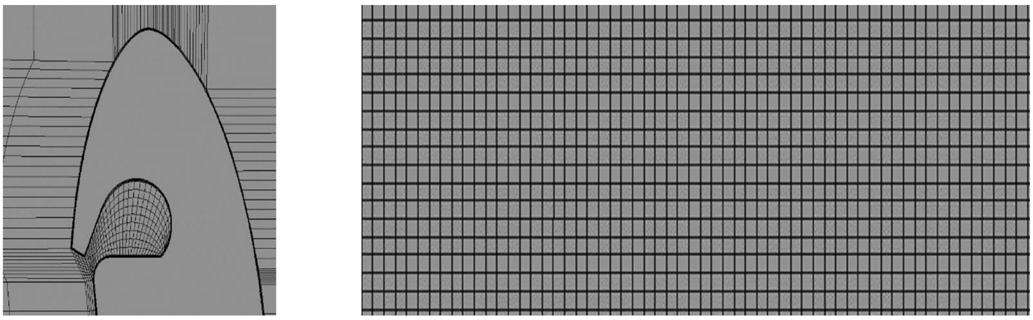
4.2.1 Geometry

Geometry constructed in SolidWorks was imported to FLUENT geometry to prepare bladeless ceiling fan for meshing. Nozzle of both bladeless fans were named 'inlet' for ease of calculations. Geometry was then 'generated' using generate option in design modeler.

4.2.2 Meshing

Due to curved geometry of bladeless fan, automatic meshing was not suitable. Mesh of bladeless ceiling fan was created using ICEM CFD. ICEM CFD in ANSYS is a meshing tool in which mesh can be created manually by creating blocks around desired geometry. Each edge of block can be given individual number of elements to generate quadrilateral mesh elements around curved geometries. Since FLUENT only computes unstructured mesh, mesh generated by blocking method is converted to unstructured format and saved in *.uns file. For each parametric analysis, separate mesh in ICEM CFD is created using same ratio as initial mesh. Initial mesh is shown in Fig 4.4 (a) and 4.4 (b).

=1cm



(a) (b)

H

Figure 4.4 (a) Mesh near Bladeless Fan (b) Mesh Size

4.2.3 Grid Resolution

Grid independency of mesh is carried out before carrying out our simulation. Three mesh sizes with 350592,465558, and 576120 elements for fan with 45cm diameter and 1.15mm nozzle diameter at a distance of 30cm from the ceiling is selected. The velocities of flow exiting from bladeless fan across the whole cross section at a distance of 50cm downstream from the bladeless fan are measured. The velocity profiles for three meshes can be seen in Fig. 4.5. Due to reasonable

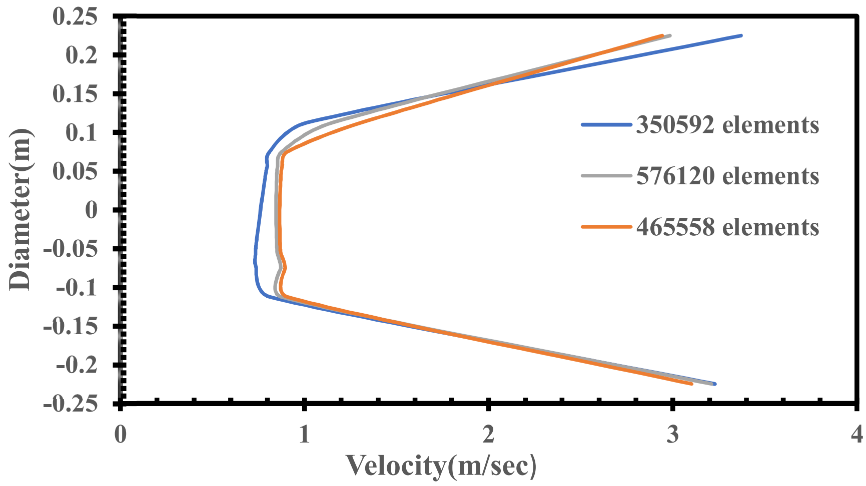


Figure 4.5 Grid Dependence

accuracy and low simulation costs mesh with 350592 elements is selected.

4.2.4 Boundary Conditions

In setup, different inlet conditions can be given to simulate flow inside a room. Unsteady state analysis was performed on bladeless ceiling fan. For transient formulation, standard K-ε model with standard wall conditions was used. No slip conditions were assigned to top, bottom and walls of the room. Bladeless fan nozzle was given different volumetric flow rates and outlet flow rate was measured by drawing a line at exit of bladeless fan cross section. First order implicit scheme was used for transient formulation and SIMPLE algorithm was used for pressure velocity coupling. Solutions are obtained for a time step of 0.01

sec. Inlet volumetric flow rate was obtained from ‘fluxes’ in ‘Results’ toolbar in ‘Setup’ option of fluent.

4.2.5 CFD Post

After numerical simulations are completed, the results obtained were imported to CFD Post. Velocity contours were drawn and reworked for visual presentation. Velocity contours show the velocity of air particles in flow field. While max velocity of flow differed, velocity contours are drawn in range of 0-10 m/sec, as most of flow occurs within this range.

Summary

In this chapter, designing of bladeless ceiling fan and CFD analysis of bladeless ceiling fan using ANSYS Fluent is discussed. Geometry is created in SolidWorks which is exported to ANSYS Design Modeler. After geometry, mesh of bladeless ceiling fan generated using blocking method in ICEM CFD. Different flow conditions are provided to bladeless ceiling fan in ‘Setup’ to perform parametric analysis.

References

[1] M. Jafari, H. Afshin, B. Farhanieh, and H. Bozorgasareh, “Numerical aerodynamic evaluation and noise investigation of a Bladeless fan,” *J. Appl. Fluid Mech.*, vol. 8, no. 1, pp. 133–142, 2015.

Chapter 5

Results and Discussion

Parametric analysis is performed by changing one parameter while keeping other parameters constant. Parameter with best flow field and maximum flow multiplication is selected and next parameter is varied. Optimizing volumetric flow rate by changing only geometric parameters will result in low energy load and less electrical costs. Parametric analysis is performed on following parameters, also shown in Fig 5.1.

a) Distance from ceiling
b) Fan diameter
c) Nozzle diameter

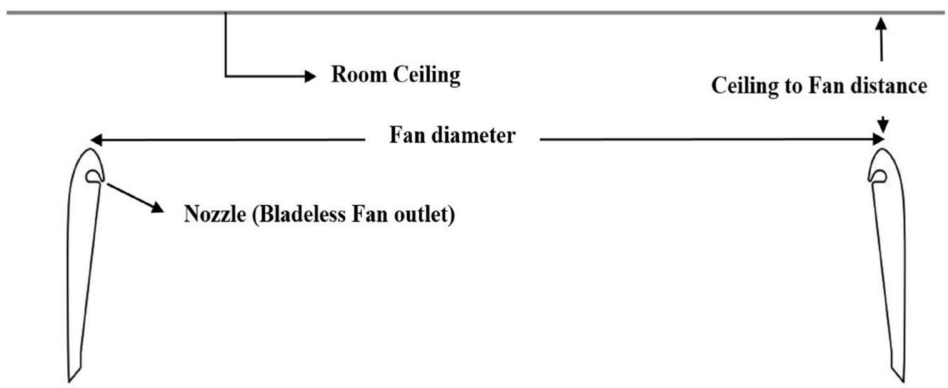


Figure 5.1 Bladeless Fan Parameters

5.1 Distance from Ceiling

A 40cm diameter bladeless fan is modeled at a distance of 0.1m from ceiling. Different inlet volumetric flow rates are given and output volumetric rates are measured. Numerical results showed that volumetric flow rate at outlet increased as distance between fan and ceiling increased up to 0.3m. After 0.3m, increase in ceiling to fan distance resulted in decrease in outlet volumetric flow rate.

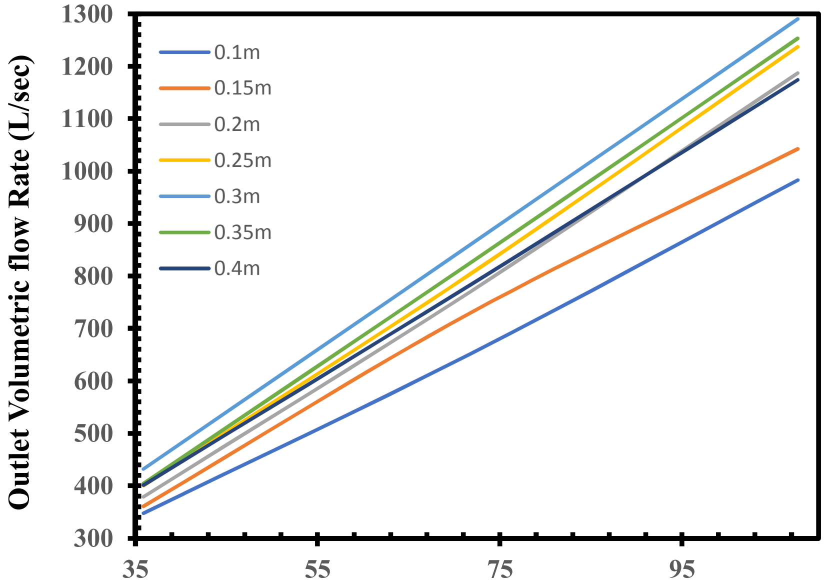


Figure 5.2 Volumetric Flow Rate for ceiling to fan distance

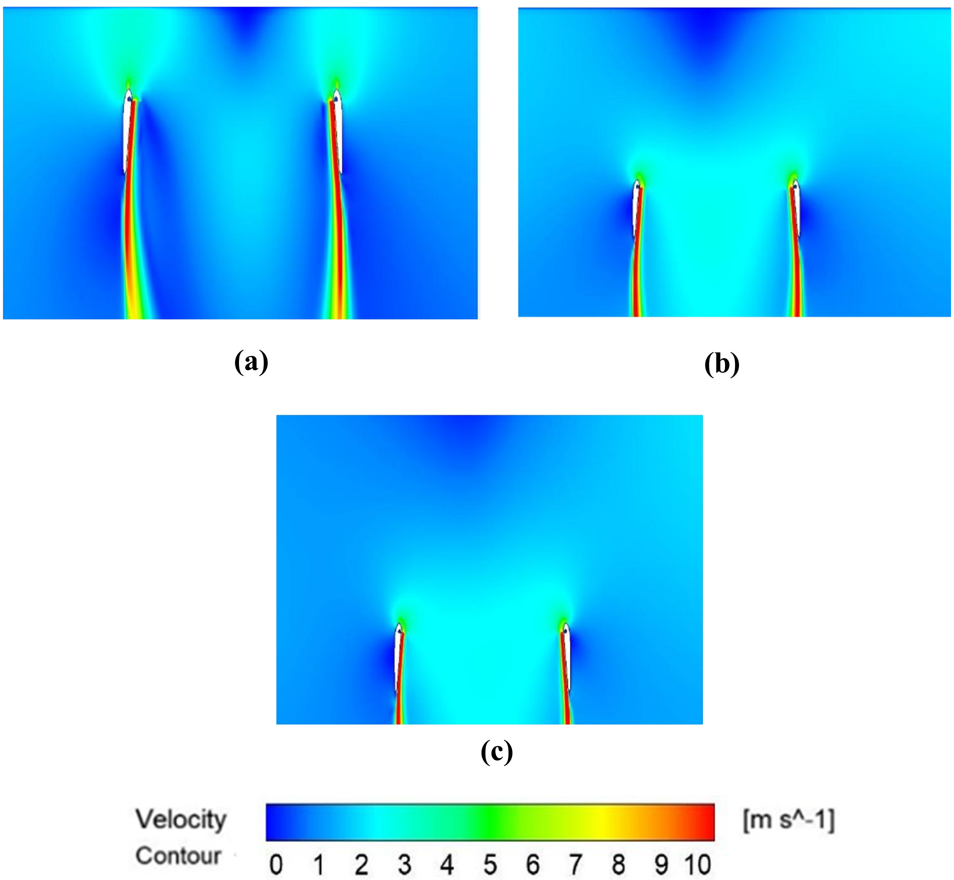


Figure 5.3 Velocity Contours of Ceiling Distance (a) 0.1m (b) 0.30m (c) 0.40m

Initially, at 0.1m due to less distance between ceiling and fan, Fig 5.3 (a), less particles are allowed to enter in the space between fan and ceiling. As distance between fan and ceiling increases, more and more particles are allowed to enter above bladeless ceiling fan until 0.3m, Fig 5.3 (b). As can be seen from Fig 5.3 (c), air particles reaching bladeless fan have low velocity. This may be due to low pressure created above bladeless fan as bladeless fan now has to entrain more particles over larger area.

5.2 Fan Diameter

Bladeless fan is kept at a ceiling distance of 0.3m and fan diameter is varied in these simulations. Different volumetric flow rates at inlet are given and corresponding outlet volumetric flow rates are measured. Volumetric flow rate of

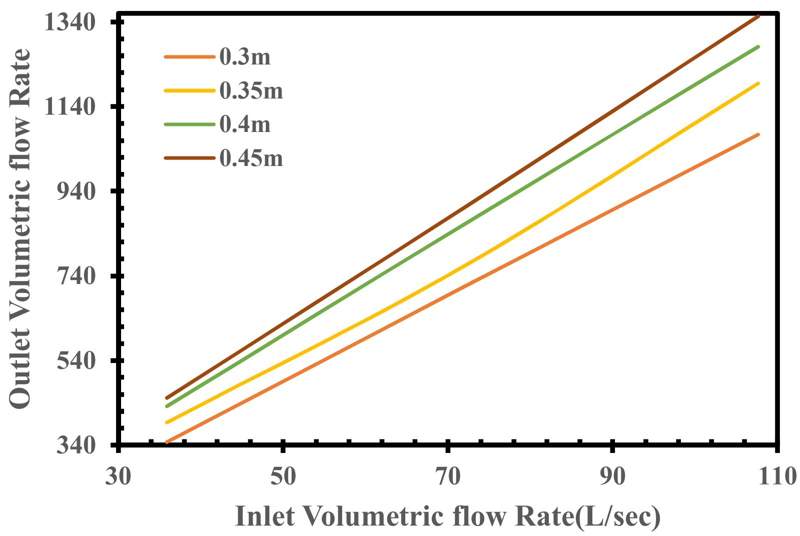


Figure 5.4 Volumetric flow Rate Change with Fan Diameter

bladeless ceiling fan increases with increase in fan diameter as shown in Fig 5.4.

In Fig 5.5 (a), at 30cm diameter, there is less space for air to enter in bladeless fan diameter. Volumetric flow rate at 30cm is lowest as can be seen from Fig 5.4. As diameter increases, there is more space for air particles to enter between bladeless fan diameter. Results from figure 5.4 show that as diameter of bladeless fan increases, volumetric flow rate of bladeless ceiling fan also increases.

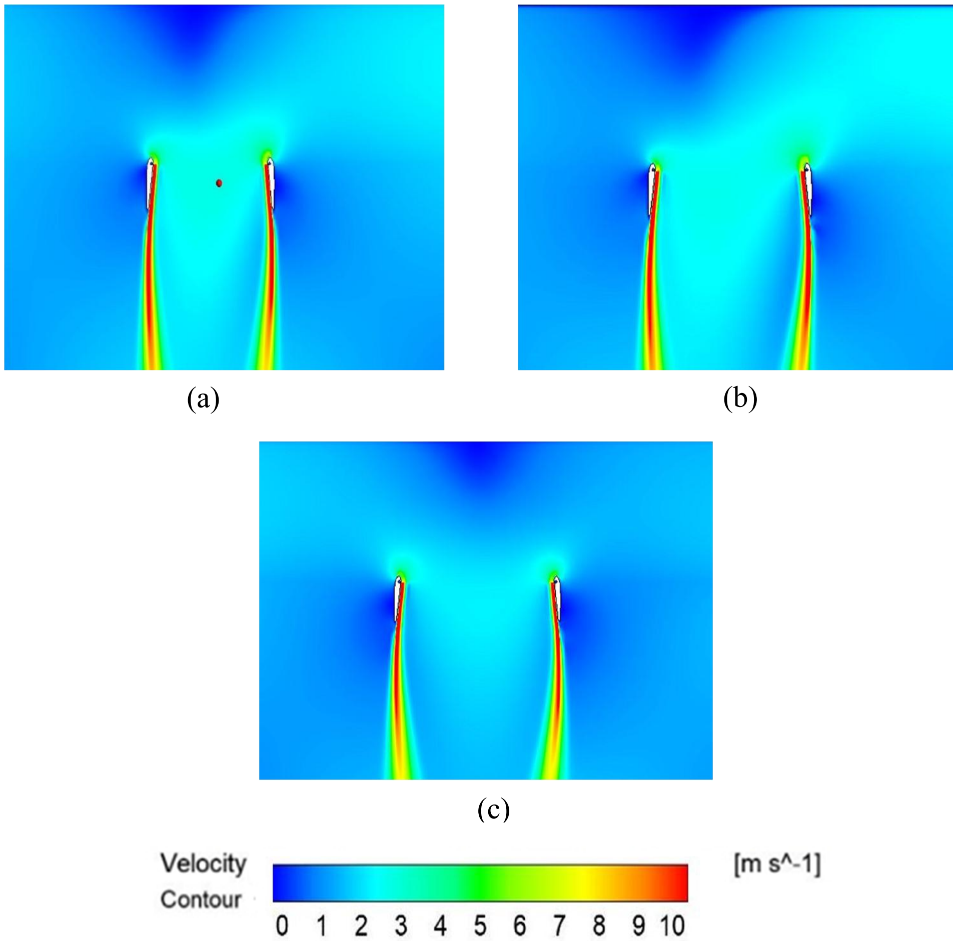


Figure 5.5 Velocity Contours for Fan Diameter (a) 30cm (b) 35cm (c) 40cm

5.3 Nozzle Diameter

A bladeless fan of 40cm diameter with a distance of 0.3m is modeled in these simulations. Different volumetric flow rates are given at inlet and outlet volumetric flow rates are measured. Numerical results showed that increase in nozzle diameter resulted in decrease in outlet volumetric flow rate. At 1mm nozzle diameter, volumetric flow rate multiplication of about 12.5 times is obtained.

As can be seen from velocity contours Fig 5.7 (a), particles induced from behind have larger velocity in case of 1mm nozzle diameter. As nozzle diameter increases, for same volumetric flow, exit velocity decreases. Velocity of jet flow exiting from bladeless fan dictates the amount of induced air particles. If the exit velocity is high, number of induced particles will be high and vice versa.

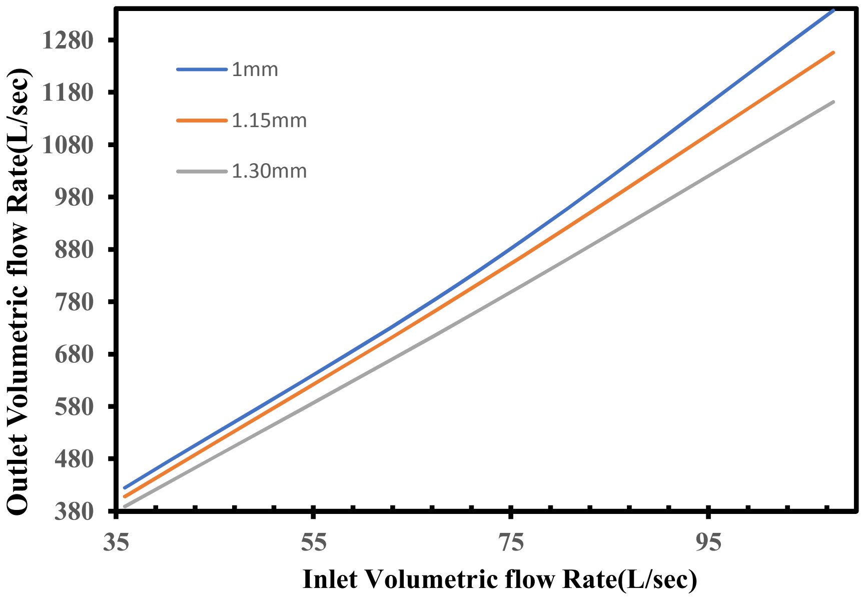


Figure 5.6 Volumetric flow Rate Change with Nozzle Diameter

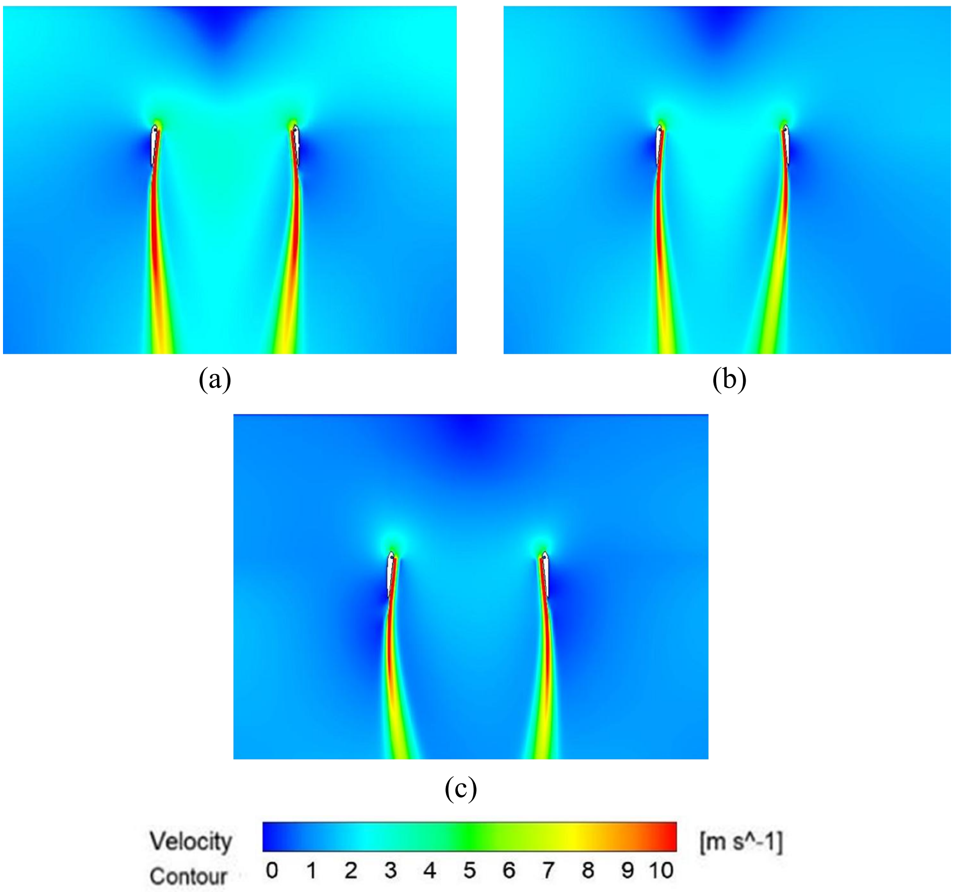


Figure 5.7 Velocity Contours for Nozzle Diameter (a) 1mm (b) 1.15mm (c) 1.30mm

5.4 Validation of Numerical Model

In order to validate numerical model, numerical results are compared with experimental research carried out by Hong li et al. [1] on the outlet flow field of bladeless fan. Hong li et al used a bladeless fan of 30cm diameter with nozzle

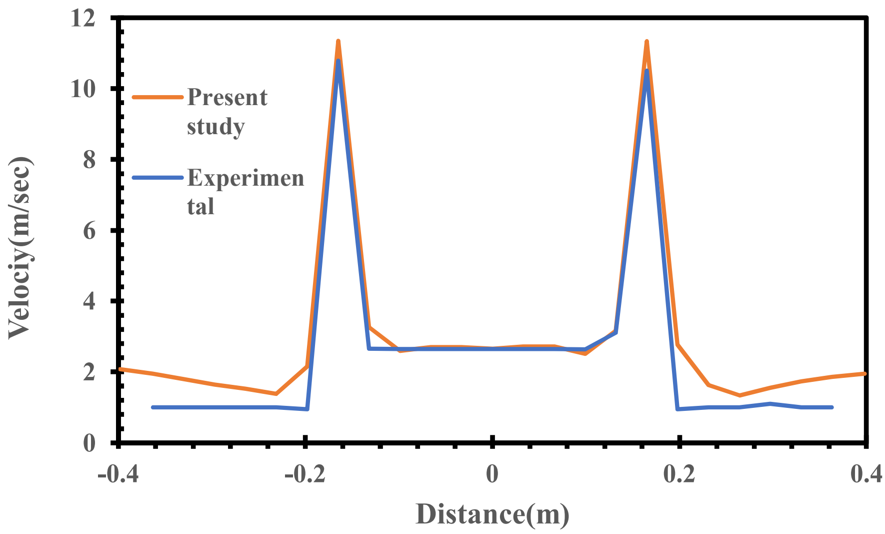
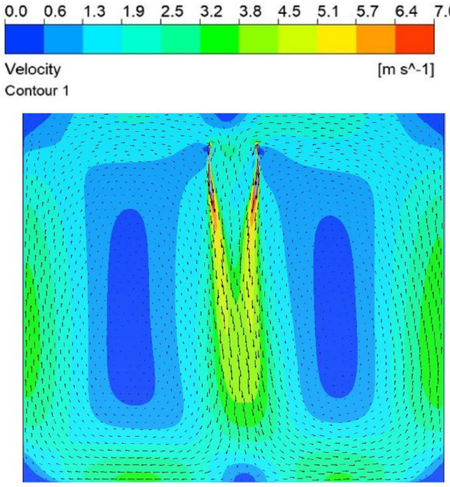


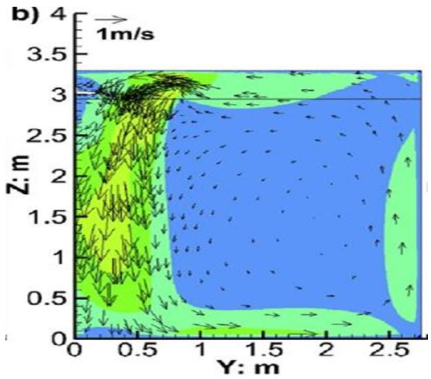
Figure 5.8 Velocity across bladeless fan cross section

diameter of 1.30mm having maximum velocity of 59.4m/sec at outlet. For validation, a bladeless fan of 30cm diameter with nozzle diameter of 1.30mm is simulated reaching maximum velocity of 53.9 m/sec. Velocity at a distance of 15cm from fan inlet measured and compared to velocity measured by experimental results at a distance of 15 cm from fan inlet as seen in Fig 5.8.

5.5 Outlet Flow Field of Bladeless Ceiling Fan

Outlet flow field of bladeless fan is obtained by solving unsteady mass and momentum equations using ANSYS FLUENT. Outlet Flow field of a bladeless fan of 40cm diameter and a ceiling distance 0.30m with outlet volumetric flow rate of 1300 L/sec placed in a 3.5m × 3.5m room is compared with a ceiling fan mounted at a distance of 0.3m from the ceiling revolving at 115 rpm placed in a 5m × 3.25m room [2] . It is clear from following Fig 5.9 that flow originating from bladeless fan follows the same pattern of flow as conventional ceiling fan.





5.6 Experimental Setup

An acrylic glass test room is made with dimensions 1.21m x1.21m x 1.82m. Bladeless fan is first designed in SolidWorks and then 3D printed for experimental use. Bladeless fan is mounted to ceiling of test room at a height of 15cm using nuts and bolts. BENETECH GM 8903 anemometer is used to

for (a) Bladel

g Fan

Figure 5.10 (a) Acrylic Glass Test Room, (b) Anemometer, (c) Bladeless Fan

measure velocity at outlet of bladeless fan. An electric blower is used to supply air to the inlet of bladeless fan. Velocity at outlet of bladeless fan is measured horizontally and vertically at different points. Volumetric flow at outlet is calculated to calculate multiplication factor of bladeless fan at different inlet volumetric flow rates.

5.6.1 Multiplication Factor

To calculate volumetric flow rate of boiler, outlet area of blower was measured using a Vernier caliper. This area was then fed into anemometer to calculate flow rate directly by placing anemometer at exit surface of blower. Blower was run at minimum flow settings and flow was measured at blower exit. volumetric flow of blower at maximum flow settings is provided by manufacturer at 35.5L/sec. Values in between were calculated by using linear interpolation methods. Values at different blower settings are listed in the following table.

Blower Setting	Flow Rate (m3/min)
1	10.83
2	17
3	23.17
4	29.33
5	35.5

Table 5.1 Flow Rate at Different Blower Settings

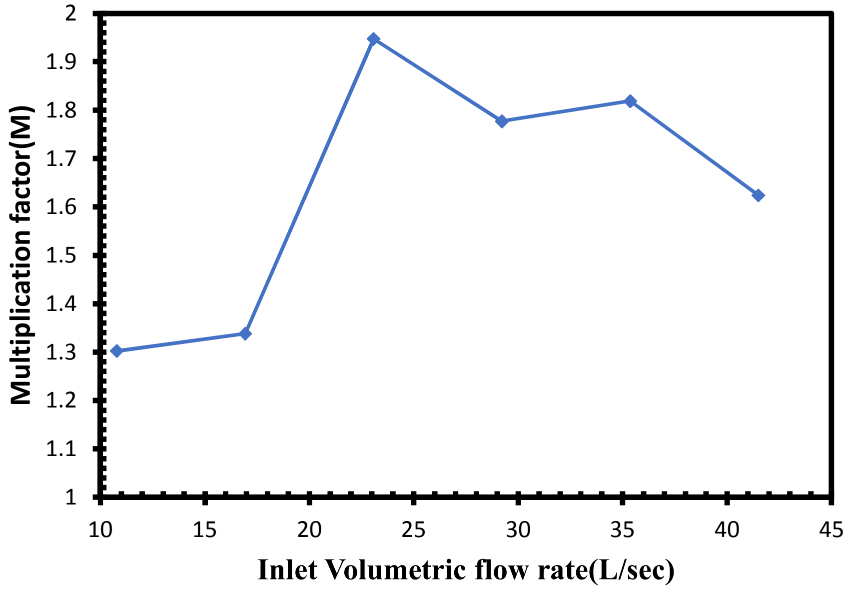


Figure 5.11 Multiplication Factor

After calculation of flow rate at inlet, i.e., blower flow rate, flow rate of air exiting the fan and entrained air is measured to calculate volumetric flow rate at exit surface of fan. Ratio of volumetric flow rate at exit of fan to flow rate is inlet is volumetric flow multiplication factor. Volumetric flow rate multiplication factor or simply multiplication factor at different flow rates is calculated. Figure

5.11 shows that volumetric flow at outlet of bladeless fan increases with increase in inlet volumetric flow rate. For multiplication factor calculations, anemometer is placed directly below surface of bladeless fan at inlet side, center and far side.

Average velocity is calculated at this point and flow rate at exit is calculated by using formula Q=A$\bar{v}$ where $\bar{v}$ is average velocity of flows exiting from bladeless fan. As volumetric flow rate increases at inlet, multiplication factor should increase linearly, but decrease in multiplication factor occurs due to back pressure losses at PVC pipe inlet at higher flow rates.

5.6.2 Velocity Distribution

Velocity distribution in bladeless fan is measured by using hot wire anemometer at different locations within the room. Anemometer is placed at three locations, 0.20m, 0.305m and 0.450m with respect to ceiling distance. Air is supplied to bladeless fan by running blower at different blower settings. Velocity of air flowing out through bladeless fan is recorded at three different locations with respect to bladeless fan geometry. Side of Bladeless fan receiving inlet flow is referred to as inlet side while the other side is referred as far side. From Fig. 5.13

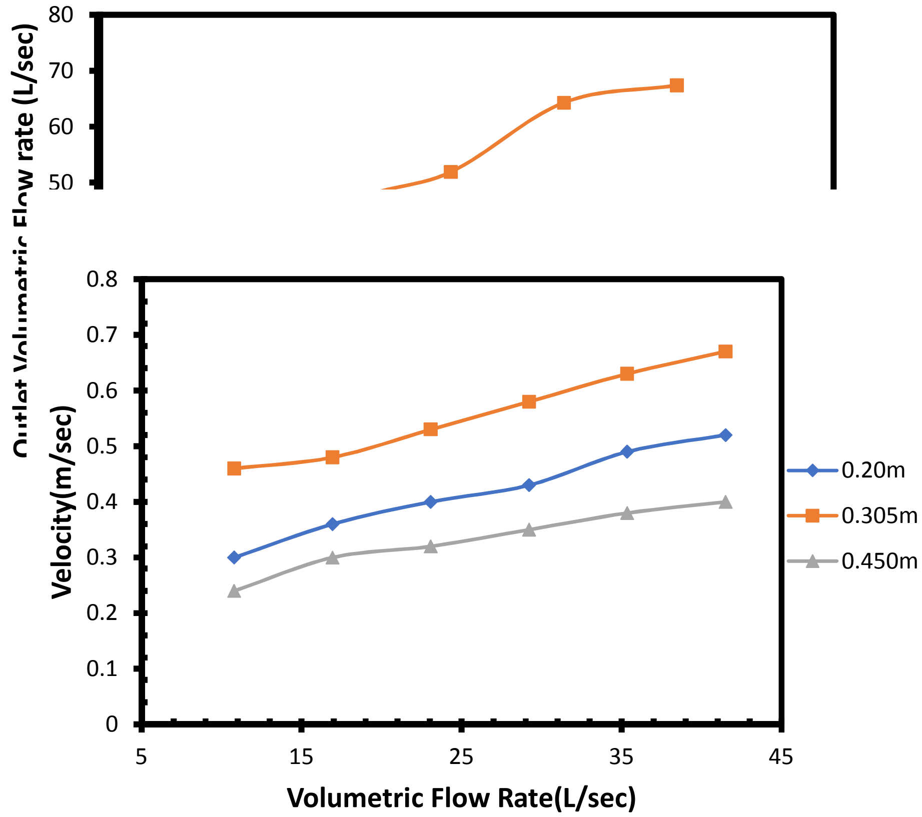


Figure 5.13 Velocity at Bladeless Fan Centre

and Fig. 5.14 it can be seen that velocity at bladeless fan center at 0.20m is greater than at 0.305m. At 0.305m velocity measured is solely due to air entrained by Coanda surface exhibiting Coanda effect. At a distance of 0.305m flow exiting from nozzle combines with entrained air resulting in increase in velocity, while at 0.450m velocity decreases simply due to air spreading at a larger distance.

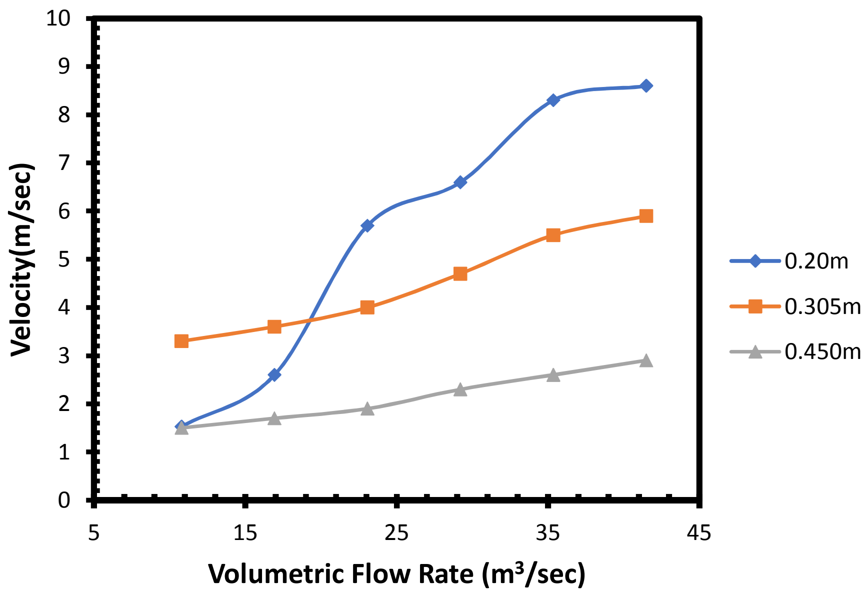


Figure 5.14 Velocity at Far Side of Bladeless Fan

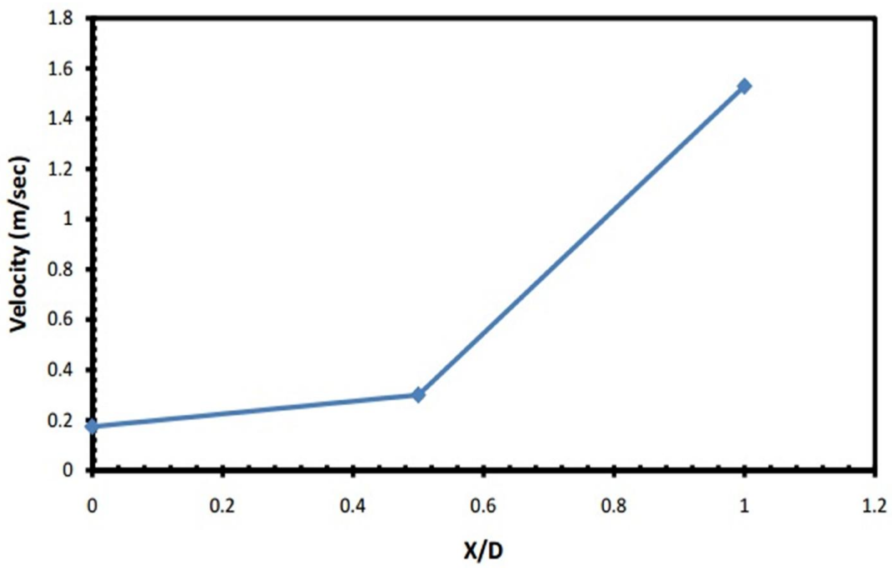


Figure 5.15 Velocity Distribution Across Bladeless Fan

Maximum velocities occur at far side of bladeless fan as air passing through geometry of Bladeless fan converges at far side and then starts flowing in downward direction.

.

Fig 5.15 further confirms that velocity across bladeless fan increases as we move farther away from inlet side. In Fig 5.16, X is displacement from inlet side towards far side while D is diameter of Bladeless fan. Flow entering Bladeless fan from inlet side moves along fan's curvature until it reaches far side; thus,

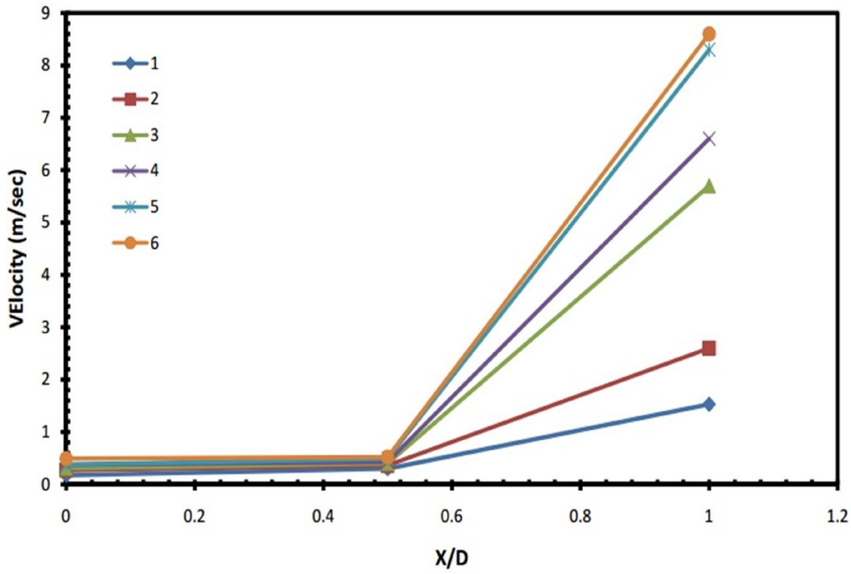


Figure 5.16 Velocities at Different Blower Settings

minimum amount of air exits from inlet side of the bladeless fan.

Fig. 5.16 shows velocities at different points at bladeless fan exits at different blower settings. 1-6 in Fig 5.16 are all blower settings at which velocities are measured.

5.6.3 Limitations

- Losses in Bladeless fan occurred due to back pressure losses and resistance during pipe flow
- An adequate air supply source was not present so power consumption comparison with ceiling fan could not be compared.

Summary

In this chapter, results obtained by performing CFD analysis on bladeless ceiling fan are given. Ceiling to fan distance is first simulated and a ceiling distance of 0.30m is selected. After, ceiling to fan distance, diameter of bladeless fan is simulated and 0.35m diameter is selected due to larger diameter yielding poor flow characteristics. Finally, nozzle diameter is selected, based on compromise between volumetric flow rate multiplication and produced noise. Outlet flow field of bladeless ceiling fan with compared with conventional ceiling fans. Bladeless fans showed similar velocity contours but with larger velocities.

Bladeless fan is then experimentally designed using results from CFD analysis and a test room is constructed to study outlet flow velocities and flow multiplication of bladeless ceiling fan. Results indicate that velocity at far side of bladeless fan is greater as compared to inlet side of bladeless fan. Flow multiplication is obtained but much of it is lost due to pipe resistance and back pressure losses from blower. Improving design, such as having multiple inlets to bladeless ceiling fan and adequate air supply can lead to much better flow characteristics.

References

[1] H. Li, H. S. Deng, and Y. Bin Lai, “Numerical and experimental research on the outlet flow field for the air multiplier,” *Appl. Therm. Eng.*, vol. 93, pp. 652–659, 2016, doi: 10.1016/j.applthermaleng.2015.10.029.

[2] W. Chen *et al.*, “Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan,” *Build. Environ.*, vol. 130, pp. 14–26, 2018, doi: 10.1016/j.buildenv.2017.12.016.

Chapter 6

Conclusion and Recommendation

6.1 Conclusion

In this research, designing and parametric analysis of bladeless fan is performed using CFD software ANSYS Fluent. Unsteady state continuity and momentum equations combined with standard k-ε turbulence model are solved to obtain fluid flow inside a room of 3.5m x 3.5m. Different parameters studied in this research are

1. Ceiling to fan distance
2. Fan diameter
3. Nozzle diameter

We have seen that volumetric flow reaching from backing of the fan is about 7 to 13 times depending on the conditions. Nozzle diameter dictates the velocity of fluid jet escaping from bladeless fan and can be varied to increase volumetric flow rate but decreasing nozzle diameter results in increasing noise. Increasing diameter of bladeless fan results in increasing volumetric flow but for same volumetric flow velocity dissipates quickly before reaching the floor level. Ceiling-to-fan distance is an important parameter which determines how much air particles from backing of the fan can join primary flow resulting in an increase in volumetric flow. However, too much distance between ceiling and bladeless fan can result in fan having to entrain more particles over a larger area which results in decreasing volumetric flow. Comparison of outlet flow field of bladeless ceiling with conventional ceiling fan resulted in similar velocity contours. Bladeless ceiling with increased volumetric flow rate and similar flow field to ceiling fans with inherent advantages such as low noise, smooth streamlined flow and absence of air-blade interaction make bladeless ceiling fans a suitable replacement for conventional ceiling fans.

Experimental analysis leads us to following conclusions

- Volumetric flow multiplication is observed in bladeless Ceiling Fan.
- Maximum velocities occur at far side of Bladeless fan.
- Bladeless ceiling fans can be used for domestic applications.

While there are other parameters of bladeless fans which can be varied such as aspect ratio, thickness of bladeless fan geometry, change in angle of bladeless ceiling fan, height of cross section and use of quadratic cross section, these parameters are not considered as they have very small effect on outlet volumetric flow rate.

6.2 Recommendations

- Further experimental analysis of bladeless ceiling fan on a larger scale should be performed.
- Numerical analysis of bladeless ceiling fan with two symmetric inlets should be performed.
- A proper air supply mechanism for bladeless ceiling fan should be designed with minimal back pressure losses.