

CONTROLLING THE AREA EXPANSION OF A BACKWARDS CENTRIFUGAL FAN
BLADE PASSAGE USING THE PRINCIPLES OF A DIFFUSER
AND COMPUTATIONAL FLUID DYNAMICS

by

Adam Jeffrey Michalson

A dissertation submitted in partial fulfillment
of the requirements for the degree

of

Doctor of Philosophy

in

Mechanical Engineering

MONTANA STATE UNIVERSITY
Bozeman, Montana

July 2021

©COPYRIGHT

by

Adam Jeffrey Michalson

2021

All Rights Reserved

DEDICATION

I would like to dedicate this work to my family. Especially to my wife, Pearl, who has supported me throughout this endeavor. To my children, Nia, Savion, Xavier, and Castiel for being themselves and making me smile.

ACKNOWLEDGEMENTS

I would like to thank my advisors Dr. Erick Johnson and Prof. Kevin Amende. Dr. Johnson for enlightening me on the workings of Computational Fluid Dynamics. Prof. Amende for his industry knowledge and introducing me to ASHRAE where I have been able to observe and learn from the HVAC industry as a whole.

AAON has supported and funded this research for the entirety of my doctoral program, and I am thankful for the opportunity.

Hyalite Cluster on campus provides high-performance computing power to students on campus and without this resource my research would have taken far longer to complete.

The Fluids Computation Lab & HVAC Lab team members have been invaluable in their support and feedback. It is a collection of talented individuals working on engineering problems ranging from fans to atomized sprays. Having multiple perspectives is something every researcher should have.

TABLE OF CONTENTS

1. INTRODUCTION	1
2. BACKGROUND	7
Design Planes	11
Centrifugal Fan Design	12
Stream Theory and the Fluid Path	16
Traditional Blade and Blade Passage Design	17
Mean Fluid Path, Stream Theory	21
Fluid Expansion	23
Computational Fluid dynamics	25
3. APPROACH AND METHODOLOGY	29
Computational Fluid Dynamics	30
Efficiency Parameters	34
Housing Proximity	37
Suction Chamber	38
Passage Assumptions	39
Blade Passage as a diffuser	40
Standard Method of Area Evaluation	41
Alternative, Controlled Area Expansion	44
A New Blade Passage Procedure	49
4. RESULTS	54
System Studies	55
Housing Proximity Study	55
Intrusion Study	58
Controlled Area Expansion	62
A New Blade Passage	64

TABLE OF CONTENTS CONTINUED

Applicability	69
5. CONCLUSIONS.....	77
6. FUTURE WORK.....	80
7. OTHER WORK	82
8. REFERENCES CITED.....	83

LIST OF TABLES

Table	Page
Table 1. Starting s locations for each total expansion angle ϕ	51
Table 2. Intrusion Study Peak Performance Values	62
Table 3. Intrusion Study Minimum Pressure Location at Peak Performance	62

LIST OF FIGURES

Figure	Page
1. Mine ventilation fan from Upper Silesia, German Empire with human for height reference (Matschoss 1907).....	8
2. Generalized fan performance curves with operational range outlined to aid in fan selection	9
3. Basic sections common to HVAC installations with flow directions.....	10
4. Meridional and radial planes and profiles in relation to the fan rotational axis.....	12
5. Blade tip angle with respect to rotation	12
6: Types of centrifugal fans with direction of rotation indicated.....	13
7. Three main components of traditional centrifugal fan design: shroud, blades, and backplate	15
8. Natural path, in orange, and ghost blade, in blue.....	17
9. Comparison for a turbine, ghost, and fan blade in relation to the natural path of a fluid particle	17
10. Fluid Element between two ghost blades with geometry and forces identified	18
11: Velocity triangles for a backward centrifugal fan fluid element	20
12. 7 blade backward curved centrifugal fan	29
13. Radial mesh slice through the system.....	31
14. Efficiency curve	35
15. Pressure curve	35
16. Power curve	36
17. Meridional cross-section of a centrifugal fan with the suction chamber outlined.....	38
18. Steps for finding passage area using graphical method	41
19. Diameter and blade intersections with alternate diameter location that intersects both blades	42

LIST OF FIGURES CONTINUED

Figure	Page
20. Standard method height compared to trapezoid defined by blade height	43
21. Standard method and true area of a blade passage along the mean path	44
22. Cross-sections of a cone diffuser (left) and a blade passage (right) with quasi-1D continuity equation.....	45
23. Comparison of angle limits set by Eck and Bleier.....	46
24. Cone diffuser with geometry parameters	47
25. Area of a cone vs centerline.....	48
26. Area change vs centerline	48
27. Cone expansion angles to 0° to 14° compared to the 9° & 12° angle limit proposed by Eck	49
28. Test fan reference geometry.....	49
29. Blade passage bounds	50
30. Inscribed circles with mean path.....	50
31. Planes perpendicular to the mean path.....	51
32. Ellipsoidal passage.....	52
33. Ellipsoidal passage fan.....	53
34. System study diagram with parameter definitions.....	55
35. Volumetric flow rate vs. efficiency for different housing diameters at 100 Pa.....	56
36. Volumetric flow rate vs. efficiency curves for selected housing diameters	57
37. Meridional cross section of a centrifugal fan with the suction chamber outlined.....	58
38. Suction chamber with motor shroud depths, ranging from (left to right) 0,3,6, and 9 cm with blades removed for visual clarity	59
39. Efficiency curves for introducing a motor shroud at different depths into the suction chamber.....	59
40. Velocity profile within the suction chamber for the 0 cm motor shroud.....	60

LIST OF FIGURES CONTINUED

Figure	Page
41. Pressure profile within the suction chamber for the 0 cm motor shroud	61
42. Pressure distribution along the center line of the suction chamber, coinciding with the axis of rotation where 0 position is at the backplate for the 0 cm, flat backplate case. All cases shows are at peak efficiency	61
43: Meridional velocity profile of the test fan	63
44: Velocity profile of blade passage discharge	63
45. Blade passage volumes of fluid: test, rectangular, ellipsoidal from left to right.....	65
46. Volumetric flow rate vs. efficiency for different passage geometries	66
47. Pressure gradients along blade passages for the area shape cases	67
48. Pressure gradients in the radial plane for the three passages	67
49. Volumetric flow rate vs. efficiency for different expansion angles.....	68
50. Volumetric flow rate vs. efficiency for a 2x scale ellipsoidal blade passage fan	70
51. Volumetric flow rate vs. efficiency at 1900 RPM	71
52. Comparison of ellipsoidal passage to logarithmic passages with the passage inlet area constant	72
53. Comparison of ellipsoidal passage to $\varphi = 0.3$ logarithmic passage and test fan	73
54. Logarithmic passage pressure profile for $\varphi = 0.3$	73
55. Meridional velocity profile for $\varphi = 0.3$	74
56. Comparison of ellipsoidal passage to logarithmic passages with the outlet area constant	74
57. Logarithmic spiral case $\varphi = 0.3$, at three test RPM, 100 Pa.....	75

ABSTRACT

Centrifugal Fans are widespread in today's modern built environment. While a few variations of these fans exist, backward centrifugal fans are an efficient economical option capable of producing the pressure and airflow required for many modern building systems. Even though fans have become necessary piece of building engineering to facilitate occupant health and comfort, fan design almost exclusively relies on approximations to equations that have not changed since the 1950s and can consume, on average, 15% of a building's electrical consumption. Additionally, the approximations made support the ease and low cost of manufacturability. The traditional centrifugal fan design is made from stamped metal parts creating a fan blade sandwich with the blades held between an inlet shroud and a backplate. This rectangular blade passage is where the fluid flows through and picks up tangential acceleration. However, since the 1950s, nearly all advancements in fan design have been through incremental changes that are made by individual companies, and these resulting designs and performance data remain proprietary. This research revisits the foundations of centrifugal fan design with more modern tools and utilizes the concept of the diffuser to strictly control the expansion of the blade passage to improve centrifugal fan efficiency. Computational fluid dynamics was used to evaluate the performance of the new design against a traditionally manufactured fan. Combining the diffuser concept with an elliptical profile for the blade passage better controls the uniformity of the velocity field and pressure gradients through the passageway, while also reducing turbulence. Simulations of the new design against the traditional approach to fan design show an increase of nearly 10% in total efficiency.

INTRODUCTION

I like to breathe. I need to breathe and so do you. We know this to be true, but how often do we think about it? As we breathe it is easy to overlook how fresh the air is or at what temperature it is at. We notice the quality of the air we breathe when it becomes uncomfortable. We notice if there are particulates in the air making us cough. We notice if it is so hot that our lungs burn. We notice if it is cold enough to take our breath away. Since we often find ourselves indoors, we expect the air to be breathable, and even take it for granted. It is easy to forget that breathable air must be delivered to our indoor spaces. In fact, it is required by law and the applicable ventilation rates can be found in ASHRAE standard 62.1(ASHRAE 2019). Heating, ventilation, and air-conditioning (HVAC) systems are responsible for delivering the air that keeps us healthy and comfortable. At the heart of those systems are fans producing the airflow. Without these fans, the modern built environment would not be possible.

A fan is any device that moves air. The hand fan is a classic historical example of a simple surface being employed to move air for human benefit (Cory 2005). With a simple flutter motion, a cooling effect is felt by the user. This simple design has been used for millennia, and while a hand fan is good enough for a single person it is not practical for larger applications such as a building. At a larger scale, hand fans would quickly become unwieldy and the flutter motion inefficient. One solution to moving more air is to mount a surface onto a rotating frame. Now the surface no longer has to pause to repeatedly change directions. The fan surface, known as a blade, now moves continuously as it rotates. This simple change has allowed for more powerful and efficient designs to be created and implemented over the last few centuries.

There are two main classifications of rotating fans, axial and centrifugal, where the orientation of the blades relative to the axis of rotation is used to distinguish between them. Axial fans are the most well-known type of fan. Many people have an axial fan visible in their own homes, such as a ceiling or a box fan. Centrifugal fans on the other hand, are hidden within HVAC systems. Almost every modern building has at least one centrifugal fan, but most people never see them.

For centuries, natural ventilation from windows and loose construction provided plenty of fresh air for human needs. Historical buildings are often described as drafty or leaky, making it difficult to maintain a certain temperature within a building. In colder climates where heating is of particular concern, leaky construction causes heat loss and requires more energy to maintain. In warmer climates, as mechanical cooling came into being, leaky construction let the heat in. By tightening the construction of buildings with more advanced materials and techniques, more heat is kept inside in the winter and more heat is kept out in the summer. People like to be comfortable. The methods of heating and cooling have evolved, but the way humans sense the temperature has not. The ambient air temperature around the human body is a significant factor in how human comfort is gauged. For this reason, most buildings now use a ventilation system to provide both heating and cooling and is one of the fastest growing sources of energy demand (IEA 2018). It is not surprising that the global HVAC Industry market is estimated to be over 100 billion US dollars by the middle of the next decade ("HVAC Equipment Market 2019 Size, Statistics, Growth, Revenue, Analysis & Trends Industry Forecast Report -2025|Market Research Engine" 2020).

Unfortunately, tighter construction also keeps in all of the occupant's exhalations, which include carbon dioxide, moisture, and germs. Poor ventilation gave rise to the term, Sick Building Syndrome. Sick Building Syndrome covers a wide variety of conditions associated with the indoor environment (Pickering 1989). Further, our understanding of indoor air quality has evolved so that we now know that there are other airborne contaminants (Yanagisawa 2017). These contaminants come from outside air pollutions as well as modern materials and chemicals that are brought inside. This has led to the research and study of chemical sensitivity to further explain psychological and physical symptoms. There is clearly a need to provide both clean air and remove contaminated air.

In addition to construction quality, as humans built larger and taller buildings, the volume of air inside became too large for natural ventilation to handle alone. Interior spaces away from the exterior walls would receive less fresh air leading to pockets of stagnate air within larger buildings. This means that there are more parts to the HVAC system as the air flow is being delivered to individual spaces and often times from several different points. Each piece of equipment utilized to make a HVAC system possible adds static pressure to the systems. Static pressure is the resistance from equipment and systems that must be overcome in order for the airflow to move through building. This resistance comes from sources like friction in the ducts used to direct the air to the different occupied spaces, restriction through filters used to remove contaminants and provide clean air, and diffusers that spread the air inside rooms. All of these components resist the airflow through friction and/or a change in the direction of the airflow. Each component or piece of equipment adds up to a total system static pressure that must be overcome by the fan used to drive the airflow and determines the minimum size a fan, or fans

need to be. This is where the centrifugal fan comes into play. Centrifugal fans are the most efficient option that can push air through the building and overcome the static pressure.

Everyday millions of centrifugal fans are in continuous operation across the world constituting a constant draw on energy resources. These fans are an important component to modern heating, ventilation and cooling systems, yet they cost millions of dollars globally to operate every year as ventilation accounts for approximately 15% of the total energy usage within the built environment (EIA 2020). This accounts for 59 million megawatt hours a year, equivalent to 45 million tons of carbon dioxide emissions for cooling operations alone. For a medium sized commercial building with an annual electrical bill of \$100,000, that means that \$15,000 went to pay for the energy required to spin a fan. If the fan averages 60% efficiency, then \$6,000 was spent on losses due to its inefficiencies. Increasing the total fan efficiency by just 2% would save \$500 annually. This seems relatively small until you look at just how many buildings there are in the U.S. alone. The 2018 Commercial Building Energy Consumption Survey (CBECS) includes just under 6 million commercial buildings (EIA 2020). \$500 saved over each of the 6 million buildings results in an annual savings of \$3 billion. In terms of emissions that is roughly the equivalent of 2.8 million tons of carbon dioxide, The CBECS also showed that in 2012 there were roughly 5.5 million buildings with roughly 8 billion square feet of floor space or an average of 1455 ft²/building (EIA 2020). In 2018, CBECS showed an increase within the U.S. commercial sector to almost 6 million buildings with close to 9 billion square feet of floor space or an average of 1500ft²/building. This supports the trend from CBECS data that shows that as we build more buildings, the buildings are getting larger, and larger buildings will require even more ventilation.

We like to breathe clean and conditioned air, and centrifugal fans make it possible in the modern built environment to do so safely and comfortably. They deliver fresh air to keep us healthy and temperate air to keep us happy and productive. They make it possible for us to forget about the quality of indoor air we utilize every day and simply breathe. With centrifugal fans being a significant portion of energy use and imperative to a healthy indoor environment, increasing the efficiency of the centrifugal fan is of great benefit to our society.

The question then is, how do we increase the efficiency of the centrifugal fan? Traditional fan design relies on assumptions from stream theory that hasn't changed since the 1950's (Bleier 1998; Cory 2005; Eck 1973). Stream theory utilized by traditional fan design starts with assuming that the fan has an infinite number of backward curved blades of infinitesimal thickness. These blades have a defined angle of intersection with an inner and outer radius where velocities can be determined. The analysis is then correlated to different blade types where the process of fan design uses blade design as the primary driving factor. The choice of blade type influences the force upon the blades and the predicted flow profile between them. An approximation for the expected fan performance can be calculated before final design tweaks are made by an experienced fan engineer. As such, the designs are heavily influenced by manufacturability. The standard approach is to stamp the fan parts out of sheet metal. The blades are then sandwiched between a backplate and inlet shroud, creating rectangular passageways for the air to move. Centrifugal fan design has only changed incrementally over the last 50 years with most of the data locked away as proprietary knowledge by fan manufacturers and HVAC companies. More recently, some manufactures have begun to use injection molding to reproduce

smaller designs, eliminating weight. These plastic models, however, have the same geometry as their metal counterparts.

The goal of this research is to rethink the design process and take advantage of new manufacturing techniques such as additive manufacturing to open new possibilities into centrifugal fan design where the final product is no longer a blade sandwich. This can be accomplished by shifting the focus away from the design of the blade to the blade passage, or the volume through which the fluid is moving. This perspective introduces the three-dimensional nature of airflow through a fan and, therefore, what is possible in fan design. In this thesis, the blade passage is treated as a diffuser. The area of the passage is shaped and controlled to maximize the pressure gained within the passage without disrupting the velocity profile. In doing so, an increase of nearly 10% efficiency has been simulated over a test fan of the classic design. This would correspond to an annual savings of nearly \$2600 and a reduction of 0.02 tons of carbon dioxide emissions for the previous medium sized building example, or 16 billion dollars and 120,000 tons of carbon dioxide emissions nationwide. While the fan can no longer be stamped, the increased manufacturing cost would be recouped within a few years over the designed 20-25 year life span of centrifugal fans.

BACKGROUND

The earliest use of the centrifugal fan can be traced back to mining ventilation as early as the 15th century (Engeda 1998) (Cory 2005). As people dug deeper into the earth to exploit natural resources, breathable air became difficult to maintain in underground mines. Dust from digging and blasting filled the shafts and pockets of deadly gasses displaced breathable air. Centrifugal fans were employed to deliver fresh air to the deepest parts of the mines and remove the contaminants being produced, making the air in mining environments safer to breathe. These fans were colossal in nature, standing tens of meters tall with large wooden paddle like blades and were often animal powered. In terms of the volume of air delivered they were poor by modern standards, but they paved the way for their modern equivalents.

Centrifugal fans were limited in application and design for centuries. It would take innovations in materials and the industrial revolution to renew interest in centrifugal fans for moving air. The invention of the steam engine allowed for centrifugal fans to transform (Cory 2005). The fans could be spun at higher peripheral speeds and therefore be smaller than their predecessors. Coupled with the industrial revolution and the ability to use metal components, centrifugal fans were faster, stronger, and more reliable. An example of a ventilation fan used around 1900 can be seen in Figure 1 (Matschoss 1907). Due to size and weight of large steam engines, fans still saw limited applicability to predominantly industrial applications. This constraint would change with the introduction of the electric motor, where this innovation decoupled the fans from large scale operations and allowed for smaller designs that could be implemented in a wide range of applications, even to moving air in our homes.

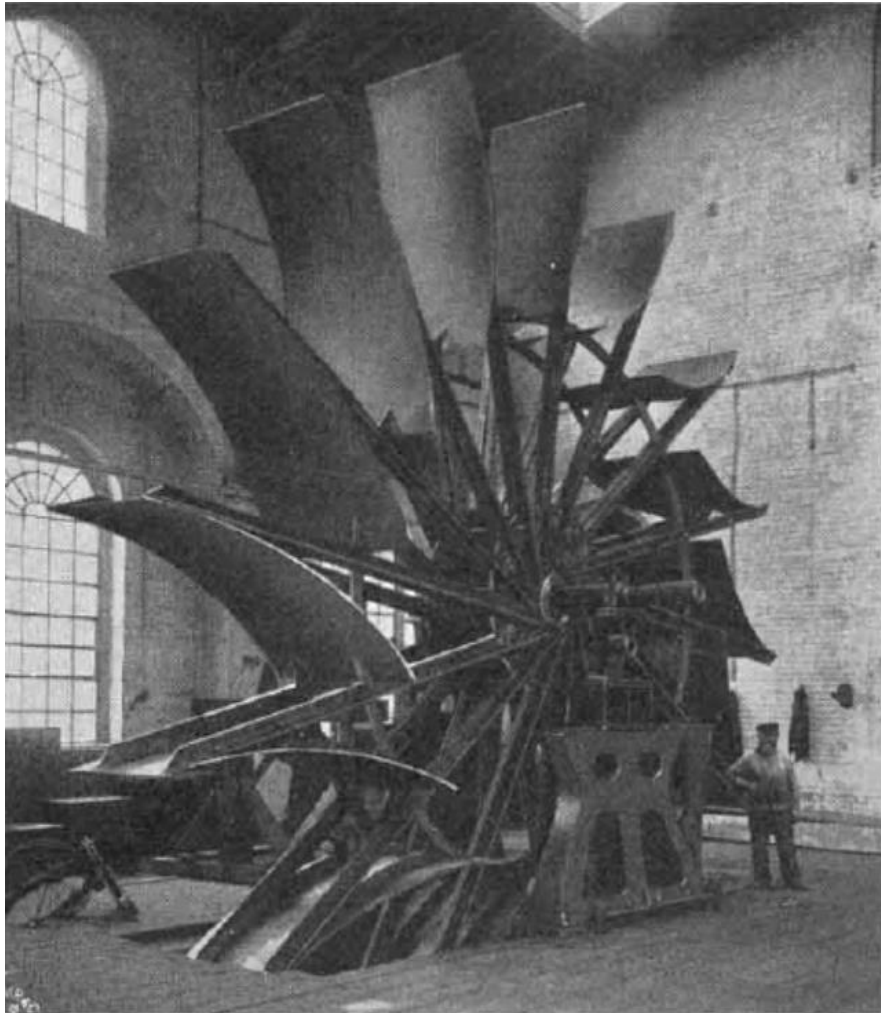


Figure 1. Mine ventilation fan from Upper Silesia, German Empire with human for height reference (Matschoss 1907)

Since fresh air must be delivered to the enclosed environment to keep the occupants comfortable and that comfort requires driving the airflow against the static pressure of the full system, selecting an appropriate fan is important. HVAC designers match the required airflow and pressure to find the fan with the highest efficiency possible out of the selection available from manufactures. Backward centrifugal fans, in particular are able to tick all the right boxes for the industry in a broad range of applications due to their high efficiency, flexibility of

installation and operation, and ease of manufacturing and maintenance. The conditions inside a building fluctuate as people move around and engage in different operations. Fans are selected based on how much air they can move and how much pressure they can overcome. Having a fan that can handle varying conditions, like the backward centrifugal fan, is desirable. Additionally, every fan has a range of operation where it is the most efficient, shown in Figure 2, as the section between the vertical dashed lines.

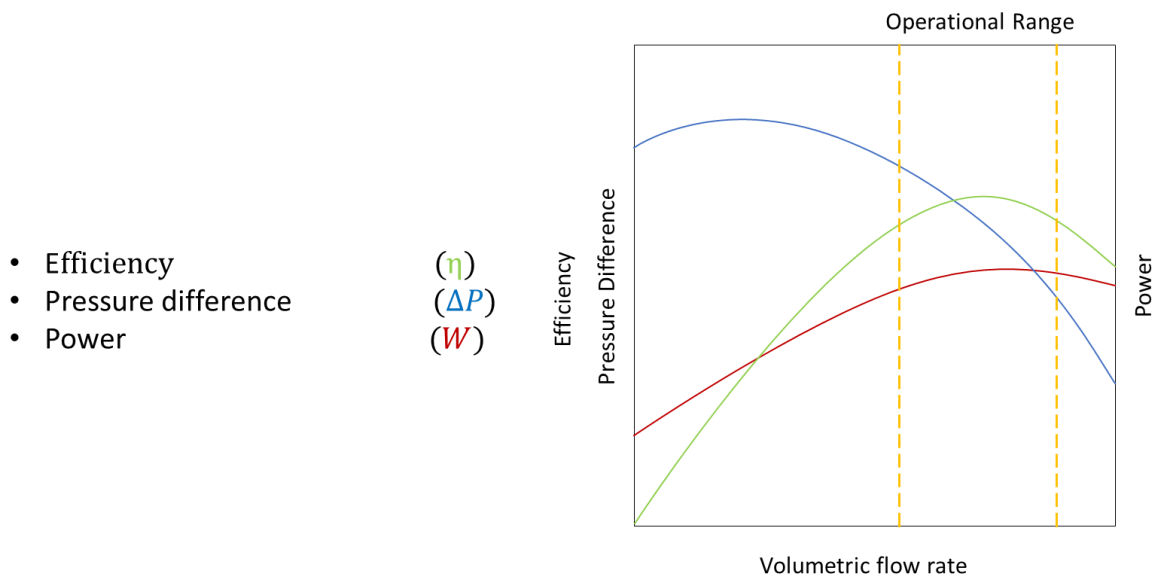


Figure 2. Generalized fan performance curves with operational range outlined to aid in fan selection

When looking at improving how centrifugal fans are implemented it is important to look at the basic system setup. The fan does not operate independent of the system. Different installations produce different performance curves; yet, they do have the same basic pieces. As the flow moves through the system there is an inlet, the fan, housing for the fan and other components in the heating and cooling of air, and an outlet. The inlet and outlet often have ducting attached as seen in Figure 3.

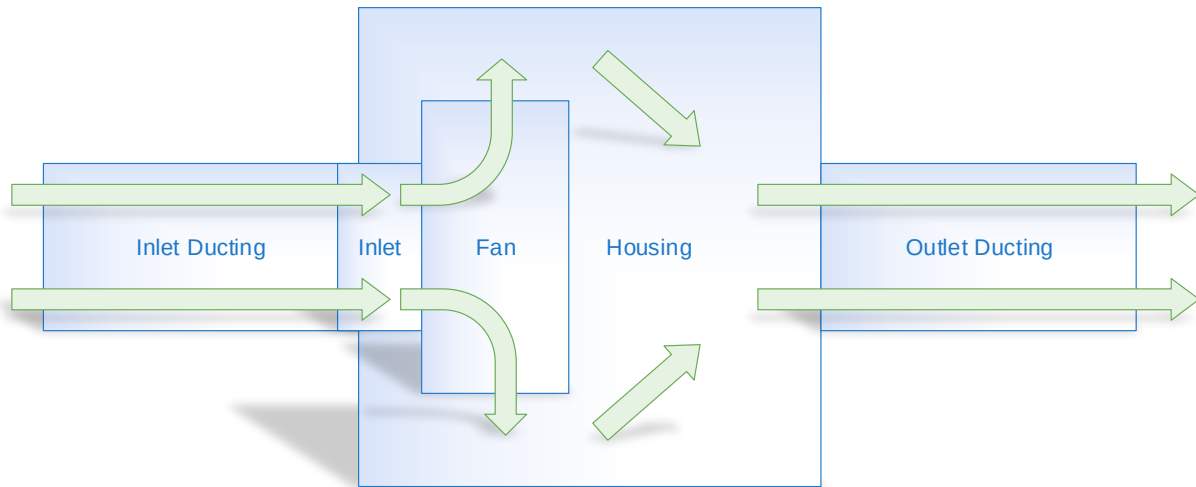


Figure 3. Basic sections common to HVAC installations with flow directions

Fans of all types use a bell mouth inlet also known as a venturi inlet, when transitioning air from the inlet and through the fan. The bell mouth inlet is used because it has high aerodynamic efficiency and smoothly lowers the velocity pressure (Bleier 1998). This results in a more uniform flow entering the fan and reduces the effects of *vena contracta*, where the flow reaches a maximum velocity near the smallest flow area (Bleier 1998). A simple hole orifice with a sharp interface and rapid area change creates excessive turbulence and noise. The curvature of the bell-mouth inlet also influences the fan performance (Son, Kim, and Ahn 2011). Sharp inlets with high curvatures lead to flow separation within the fan while lower curvatures run into size limitations.

The bell-mouth inlet leads to the suction chamber of a centrifugal fan. This region is the cavity inside the fan body where the flow makes a 90° transition into the blade passages of a centrifugal fan. The suction chamber can have mounting hardware or motor shrouds located at

the back of the chamber away from the inlet and is where the drive shaft connects to the fan backplate, which provides the rotation motion to the fan.

The air flow moves through the blade passages, picking up kinetic energy and is discharged into the housing. Centrifugal fan housings can be as simple as a box, called a plenum, or take on a spiral shape called, a volute. The size and shape of the housing directly impacts the efficiency of the fan. Complex housing designs can be leveraged to maximize the potential of the fan, but they are harder to manufacture and implement. Simple designs, like the plenum, offer flexibility in installation and maintenance while sacrificing efficiency and, therefore, increasing long-term operational costs. From the housing, the flow is either transitioned into a duct or allowed to disperse to the environment.

Design Planes

Two reference planes are typically used to describe the flow in and around a centrifugal fan and aid in their design. Bisecting the fan along a radial line through the axis of rotation results in a meridional plane. Slicing through the blades of the fan and perpendicular to a meridional plane gives the radial plane. The meridional plane is used to describe the fan shroud and backplate, while blade designs are shown in the radial plane. These planes can be seen in Figure 4. These profiles are the used to describe the fan and aid in design. The radial plane is the primary focus of traditional design because this is where the blade profile can be clearly seen.

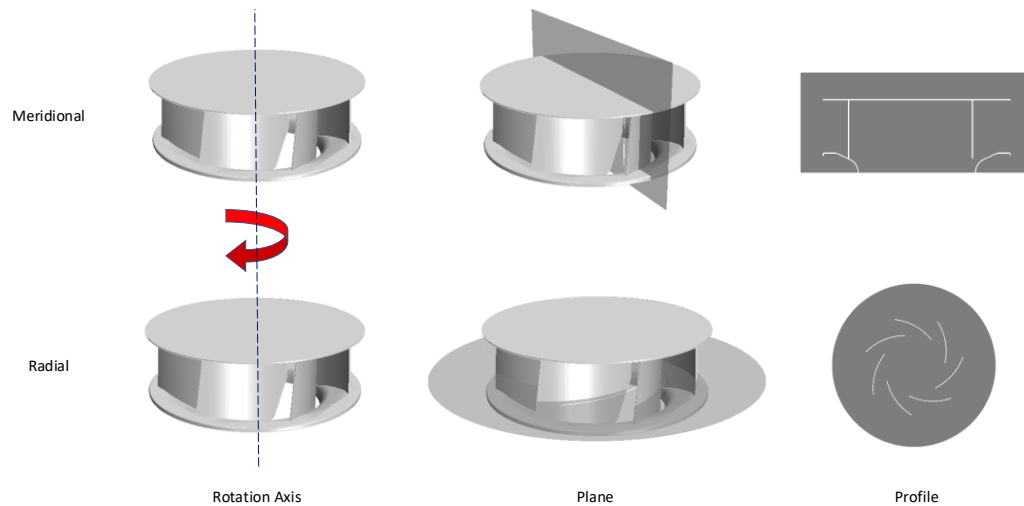
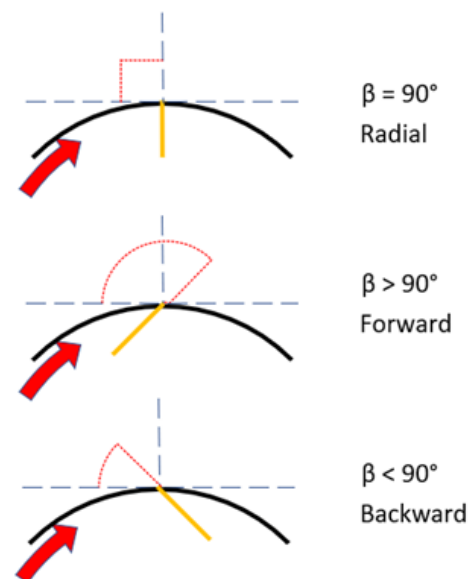


Figure 4. Meridional and radial planes and profiles in relation to the fan rotational axis

Centrifugal Fan Design

All centrifugal fans use centrifugal forces to accelerate the fluid and provide the necessary pressure required to move air through the system. There are three main types of centrifugal fans based on how the blades are oriented in the radial plane with respect to the direction of rotation. They are categorized by the angle the trailing edge of the fan blade makes with the tangent of the fan outlet circumference. The angle, β , is measured in the same direction as the fan rotation (Figure 5). If the blade tip is perpendicular to the tangent, $\beta = 90^\circ$, it is considered a radial centrifugal fan. Radial blade centrifugal fans have flat blades aligned with radial lines coming from the axis of



5. Blade tip angle with respect to rotation

rotation. Radial tip centrifugal fans have curved blades with a $\beta = 90^\circ$. Forward centrifugal blades, $\beta > 90^\circ$, lean away from the radial lines in the direction of rotation with an obtuse angle and can have either a straight blade, also called, an incline blade, or a curved blade. Backward centrifugal blades, $\beta < 90^\circ$, lean into the direction of rotation with an acute angle and can be inclined, curved, or airfoils. The types are then qualified by their blade shape. The most common blade shapes are flat, curved, and airfoil. Examples of the most common combinations of centrifugal fans can be seen in Figure 6.

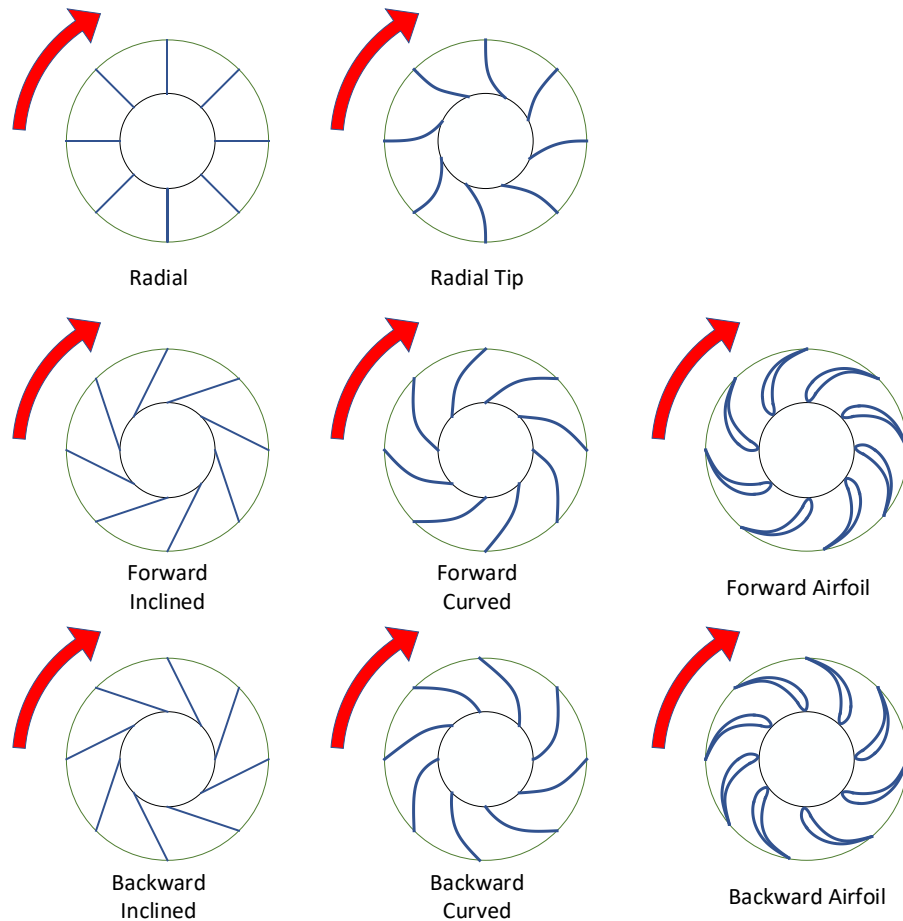


Figure 6: Types of centrifugal fans with direction of rotation indicated

Each combination of centrifugal fan type and blade arrangement results in different approximate efficiencies and acceptable uses (Eck 1973). The radial blade has the lowest efficiency but is well suited for airstreams containing particulates. Radial tip blades have increased efficiency over the radial blade but fall below that of backward centrifugal fans. Forward inclined and curved fans are slightly more efficient than those with a radial blade, but these can only handle clean lower pressure airstreams, or systems below 750 Pa. Forward centrifugal fans are often found in residential homes. Backward centrifugal fans can handle medium to high pressure airstreams; inclined blades being the most robust in terms of airstream particulates and efficiency, while an airfoil shaped blade is the most efficient, though this last variation requires an unsullied airstream and only reaches peak efficiency in a very narrow operational window (Bleier 1998).

A fan's efficiency is dependent on the power supplied, the static pressure it must overcome, and the produced volumetric airflow at an operation point. Because the static pressure and volumetric flowrate are nonlinearly related, a fan's efficiency is not constant over all operational conditions (Bleier 1998). Forward centrifugal fans exhibit an overloading power curve. That is, as power is increased, volumetric flowrate increases. They also have a region of instability that must be avoided. The instability region is caused by the alignment of the slope of the system resistance curve and the fan pressure curve (Improving Fan System Performance: A Sourcebook for Industry 2003). This instability causes a cycling in the fan performance as the fan tries to find a stable operating condition. An increase in pressure causes a reduction in airflow. As the airflow decreases the pressure decreases and the fan then produces more airflow. Backward airfoil fans also have stall regions where the flow separates from the airfoil causing

extra drag and therefore must operate at a stable condition. Conversely, the backward curved centrifugal fan has both a non-overloading power curve and is stable throughout its fan curve. This versatility is the reason that the backward curved centrifugal fan has become the preferred choice in the HVAC industry.

As the preferred fan for building systems for industry the manufacturability of the backward curved centrifugal fan is a priority of the design. The easier the fans are to make, the cheaper they are to produce. The result is that centrifugal fans are mostly made from stamped parts with profiles and curvatures conducive to the punch and die process. In Figure 7, the three main components of the traditional fan design can be seen. This process results in a blade sandwich, with the blades on the inside of two simple circular profiles. While there are injection molded options for smaller diameter fans, the designs are copies of the stamped process.

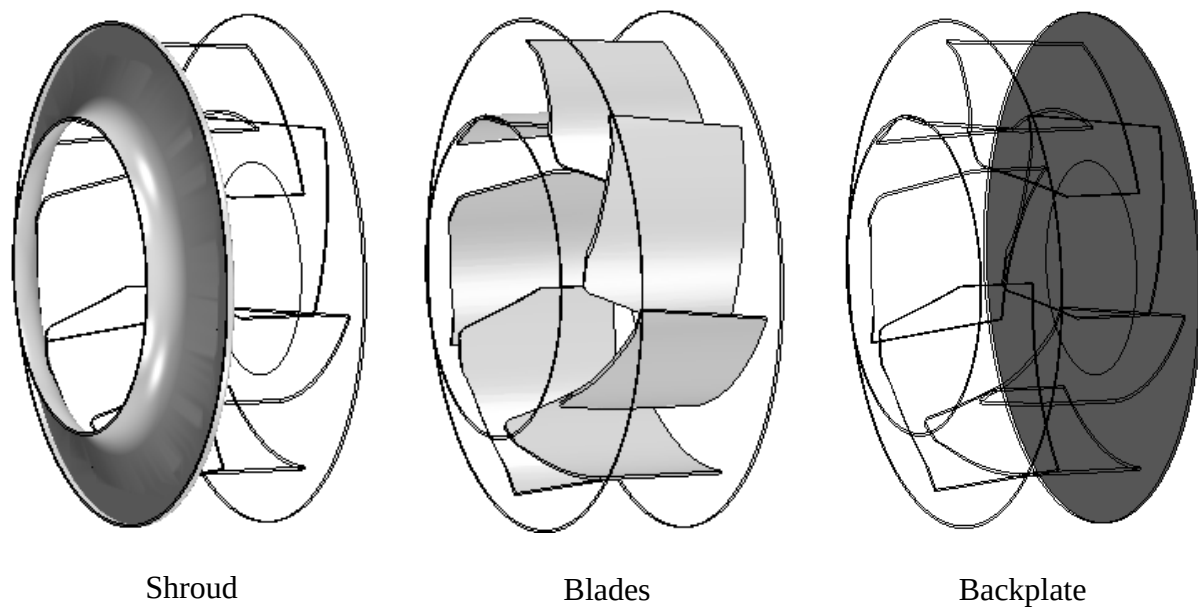


Figure 7. Three main components of traditional centrifugal fan design: shroud, blades, and backplate

Stream Theory and the Fluid Path

Evidence for the effectiveness of the backward curved centrifugal fan can be seen in the liquid application as implemented historically for the centrifugal pump. The initial impetus for the design is rooted in stream theory. Rotodynamic pumps use the backward curved blade because it imparts the most tangential acceleration to the fluid with the least amount of effort (Addison 1948). This results in less work to achieve the exit velocity at the trailing edge of the blade. In two-dimensions, there is a backward curved shape that, for a given rotational speed of the pump, would allow the blade to pass through the fluid without imparting any additional tangential acceleration. This effect occurs because the fluid particle takes a natural path solely caused by the rotation of two infinitely wide discs and the viscosity induced flow of the fluid between the discs. The path that the fluid particle would take is a curved streamline and lies in the direction of rotation. The left side of Figure 8 shows the natural path of a fluid particle as it moves from position a_1 to a_5 , as referenced from a stationary viewpoint above the fan. Any fluid particle starting at a_1 would follow this path for a constant rotation. In a rotating frame with the discs, however, any point along path a from the perspective of the disc rotates away. This path, b , follows the rotating frame. Starting at point 1, a and b are coincident. As the disc rotates and the fluid particle moves out radially, a and b are now coincident at point 2. This continues as the particle moves along path a , tracing path b on the disc. This is the “ghost blade”, or datum blade, of classic rotary pump design, and seen at the far right of Figure 8. A blade in the shape of the ghost blade would not impart any extra force on the fluid element (Addison 1948). In order to impart more force on the fluid to increase fluid acceleration, the blade has to deviate from the datum blade into the direction of rotation Figure 9. Conversely, shifting the blade in the opposite

direction leads to a turbine blade so that the fluid imparts energy to the blade. The more the blade deviates from the ghost blade the greater the imparted tangential force and acceleration, though this deviation requires more power to do so (Addison 1948).

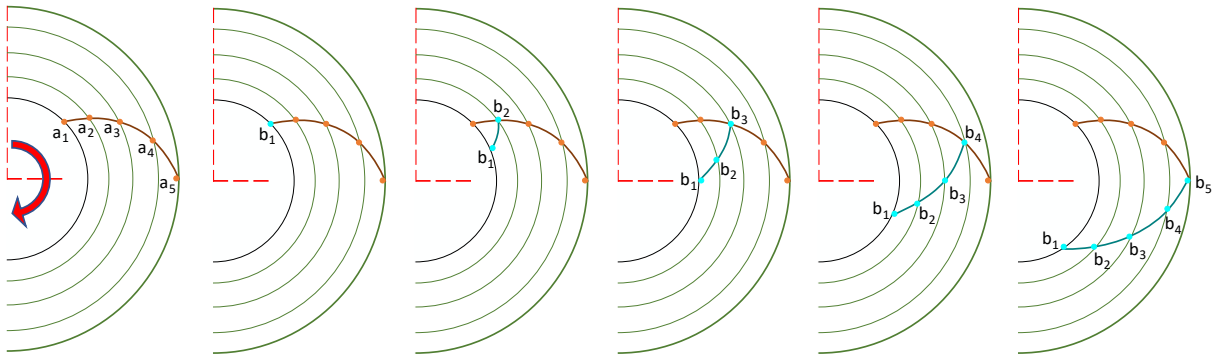


Figure 8. Natural path, in orange, and ghost blade, in blue

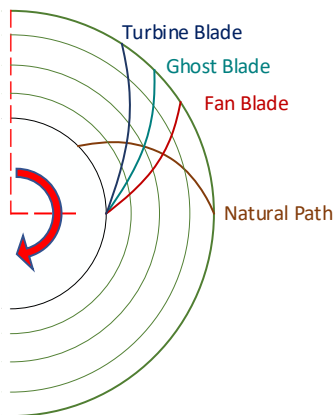


Figure 9. Comparison for a turbine, ghost, and fan blade in relation to the natural path of a fluid particle

Traditional Blade and Blade Passage Design

Traditional blade design makes use of stream theory through the analysis of a fluid element and the resultant velocity vectors as the process moves from the assumption of infinite discs to a system with a finite number of blades and fan radii (Eck 1973). In order for forces to

be transferred to the air there must be a pressure and/or velocity gradients produced within the blade passages. In the radial plane, a coordinate system exists that is always tangential, S , and perpendicular, N , to a streamline (Eck 1973).

A fluid element within the blade passage between a pair of ghost blades experiences both centrifugal and Coriolis forces (Eck 1973). The element has an infinitesimal mass dm . The fluid element, related geometry, and relevant parameters can be seen in Figure 10.

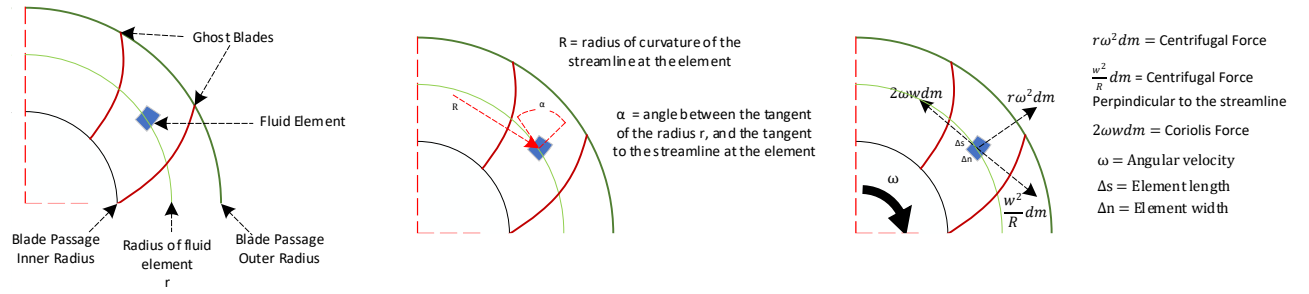


Figure 10. Fluid Element between two ghost blades with geometry and forces identified

Summing the forces in the streamline-perpendicular direction yields an equation for the pressure gradient

$$\frac{\partial p}{\partial n} = \frac{\gamma}{g} \left(\frac{w^2}{R} + r\omega^2 \cos(\alpha) - 2\omega w \right) \quad (1)$$

and in streamline-perpendicular direction an equation for the velocity gradient

$$\frac{\partial w}{\partial n} = 2\omega - \frac{w}{R} \quad (2)$$

where w is the stream wise velocity, γ is the specific weight of the fluid, and g is gravity.

These equations can be solved in limited cases to dictate a specific profile within the blade passage (Eck 1973). The pressure gradient, Equation 1, has been applied in the field of compressors. Equation 2, the velocity gradient, is the starting point for traditional fan blade design. For a straight blade, R , is infinite and the solution is a linear reduction in the distribution of velocity across the passage from the leading blade edge to the trailing blade edge. It is possible to have a uniform distribution of velocity across the passage, $\frac{\partial w}{\partial n} = 0$, where if R being taken as constant, results in circular blades. A uniform distribution results in less turbulence at the discharge. These solutions do not however take into account any three-dimensional considerations from the meridional profile on the flow. It is assumed that forces in the radial plane dominate the system and assumed to be uniform along the height of the blade.

Another approach is to look at the streamline velocity, w , along the path through time, $\frac{dw}{dt}$. Empirical studies have shown that $\frac{dw}{dt}$ decreases along the blade passage and for short passages is nearly linear (Pantell 1949). This is consistent with the Bernoulli equation for an incompressible fluid, as the area of the passage increases towards the outer radius of the fan, the velocity must also decrease. It is then convention to dictate a constant deceleration, Equation (3) (Eck 1973). From here, a blade form can be determined and matched to a meridional profile.

$$\frac{dw}{dt} = \frac{dw}{ds} \frac{ds}{dt} = \frac{dw}{ds} w = \text{constant} \quad (3)$$

The fluid element has a velocity vector, c , which is the result of the passage velocity, w , and the tangential velocity caused by rotation, u (Figure 11). The resultant velocity vector, c , has radial and tangential components, c_r and c_u respectively. The fluid element angle, α , is related to the velocity components by trigonometry, Equation (4).

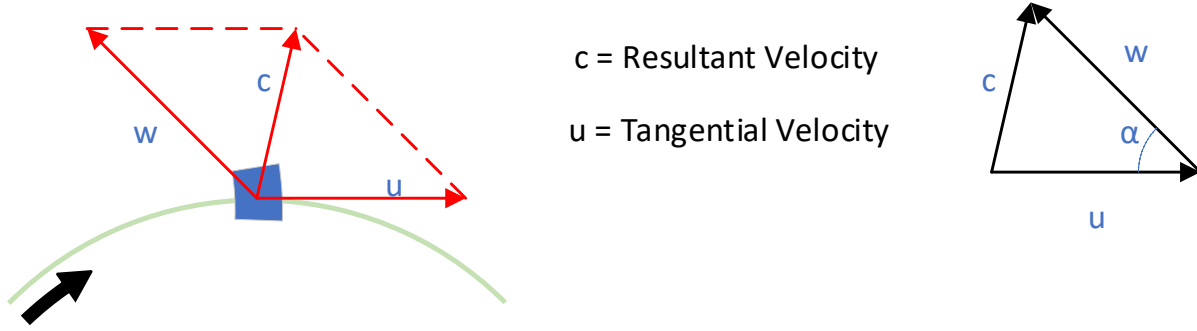


Figure 11: Velocity triangles for a backward centrifugal fan fluid element

$$\tan \alpha = \frac{c_r}{c_u} \quad (4)$$

We therefore know through trigonometry that,

$$\sin \alpha = \frac{c_r}{w} = \frac{dr}{ds} \quad (5)$$

solving for ds , and using Equation (3):

$$\frac{dw}{ds} w = w \sin \alpha \frac{dw}{dr} \quad (6)$$

Equation 6 can be then be integrated to deduce a blade form with certain assumptions. For instance, if α , is held constant, then the blade takes the shape of a logarithmic spiral.

Mean Fluid Path, Stream Theory

A fan can be modeled using a source and a free vortex. The source models the fan inlet, Equation (7), and the free vortex models the rotation of the fluid, Equation (9). V_r is the radial velocity and V_θ is the tangential velocity. λ and Γ are stream potential constants and r is the radius of the flow from the center. Together with the stream function, Equation (11), streamlines for the flow can be determined.

Source Flow	$V_r = \frac{\lambda}{2\pi r}$	(7)
-------------	--------------------------------	-----

	$\lambda = 2\pi r V_r$	(8)
--	------------------------	-----

Free Vortex	$V_\theta = \frac{\Gamma}{2\pi r}$	(9)
-------------	------------------------------------	-----

	$\Gamma = 2\pi r V_\theta$	(10)
--	----------------------------	------

Streamline	$\Psi = \frac{\lambda}{2\pi} \theta - \frac{\Gamma}{2\pi} \ln r$	(11)
------------	--	------

rearranging the stream function to solve for r yields Equation 12

	$r = e^{-\frac{2\pi\Psi}{\Gamma}} e^{\frac{\lambda\theta}{\Gamma}}$	(12)
--	---	------

φ represents the ratio radial to tangential potential which reduces to the ratio of the radial and tangential velocity using Equations (8) and (10) yielding

$$\varphi = \frac{\lambda}{\Gamma} = \frac{V_r}{V_\theta} \quad (13)$$

At $\theta = 0$, the stream function reduces with the substitution of Equation 10 where r_i is the starting radius of the streamline

$$\Psi = \frac{\lambda}{2\pi}\theta_i - \frac{\Gamma}{2\pi}\ln r_i = -rV_\theta \ln r_i \quad (14)$$

substituting Equations (13) and (14) into Equation (12) yields a simple logarithmic function for r

$$r = e^{\frac{2\pi r V_\theta \ln r_i}{2\pi r V_\theta}} e^{\varphi\theta} = r_i e^{\varphi\theta} \quad (15)$$

With Equation (15), a logarithmic path can be constructed. If r_i is specified and the tangential velocity is a function of the rotational speed, this leaves the radial velocity component as the sole design variable.

Fluid Expansion

It is common for centrifugal fans to be installed in a housing. The housing provides a frame for the fan to be mounted as well as help direct the fan discharge if desired. In the built environment, housings are used because the flow is going to be utilized for ventilation or for safety and environmental protection. A plenum housing can take any shape, however, they are often rectangular because of their ease of manufacturing and are utilized to direct the fan discharge to ventilation ducting. Cylindrical plenums provide a more streamlined volume for discharge by eliminating sharp corners. The advantage of a plenum is flexibility; as long as they do not constrict the fan discharge, no standard design criteria exists. Rectangular plenums can have any aspect ratio and the fan can be mounted anywhere inside with adequate clearing between the housing wall and the fan outlet. A special type of housing, a volute housing provides a high efficiency design with specific design parameters. Volute housings are more expensive to manufacture but do provide a volume for controlling the fan discharge for better performance.

Just as the backward curved blade has its root in pump design, so does the volute housing. Centrifugal pumps are universally used in a volute housing. A volute is shaped like a spiral, though while a spiral may continue for multiple revolutions, a volute casing only completes one rotation. Volute housings are used in air movement and are sometimes called spiral or squirrel cage housings in industry. Unlike centrifugal pumps, volute housings are not standard for centrifugal fans and air movement. In a volute housing close to the discharge and near the volute tongue is a source of turbulence with the blade passages (Hariharan and Govardhan 2016). The proximity in the volute is a necessity of the design. One of the drawbacks of the volute housing is the constant noise generation caused by the volute housing (Shepherd

1956). The noise is often over 80 dBA and can be well over 100 dBA in larger diameter fans (American Society of Heating and Air-Conditioning Engineers). OSHA requires hearing conservation programs, including hearing protection, above 90 dBA and limits exposure to 2 hours at 100 dBA (Lehtola 2000). Another drawback to the volute housing is the inability to use variable fan speeds effectively, which is necessary in moving air through buildings to account for different desired flow rates and changes in the static pressure through daily and seasonal variations. The volute design requires operating at a single, specific flow condition for proper performance and efficiency (Bleier 1998).

To avoid the issues introduced by a volute housing the HVAC industry utilizes a plenum housing for the backward curved centrifugal fan used in most systems. This entails mounting the fan within a large volume that captures the discharge and allows the pressure within the plenum to direct the flow toward an outlet and into the rest of the system. The plenum also allows a portion of the sound generated by the fan to dissipate within the larger volume through disruptive noise interference prior to the air moving through the ductwork.

In a plenum, the area of discharge immediately jumps from the circumferential area of the blade passage openings to the much larger flow area provided by the plenum. From the Bernoulli principle it is known that as a fluid velocity decreases, and the pressure increases. Rapid expansion of the flow reduces the amount of kinetic energy that would have been gained within the system due to velocity being converted to pressure through vortex formation and losses to turbulence. In a volute housing the area of discharge from the fan is regulated by the spiral expansion of the volute. While this helps with rapid expansion of the discharge, a fan volute has been shown to cause turbulence and recirculation with the passage between blades

where the housing is in close proximity to the fan outlet (Dilin et al. 1998). Even though there are obvious reasons to not consider the volute housing, converting kinetic energy to pressure, known as static regain in fan design, boosts the performance of the system.

One method to control the expansion of the flow within a plenum is by changing the distance from the fan discharge to the housing. When placed too closely, there is interference from the housing on the airflow (Eck 1973); too far away and the afore mentioned losses to turbulence begin to dominate. Bleier, suggests that the housing should be at least 1.5 times the fans diameter if a cylindrical housing is used (Bleier 1998). The optimal distance from the fan discharge has not been extensively studied, nor has a literature review indicated if there are any effects from fan design rotational speed that would further influence this design choice.

Additionally, the blade passage itself is often a diffuser as the area is usually designed to increase along its length. This feature is a design check for centrifugal fans, the passage should not expand too quickly, otherwise, static regain is minimized in a similar fashion as the issue around plenum sizing. Since centrifugal fans are utilized for their ability to produce pressure, the efficient expansion of the air within the blade passage and into the housing need to be considered in fan design. These effects are not sufficiently captured through two-dimensional analyses. Ensuring that the blade passage acts as an efficient diffuser enables the fluid flow through the passage to retain more of the energy imparted to the fan, improving their efficiency, and reducing the energy required to keep buildings comfortable and healthy.

Computational Fluid dynamics

The discipline of fluid dynamics has been around for as long as people have strived to engineer better ways to use liquids for their benefit. The ancient Greek Archimedes deduced the

Archimedes Principle of the buoyant force exerted on an object immersed in a fluid. In 1738, Daniel Bernoulli published *Hydrodynamica*, where he wrote about the relation between a fluid's velocity and the pressure within the fluid, specifically, the principle that the pressure decreases as a fluid's velocity increases (Bernoulli 1738). This principle was then turned into the Bernoulli Equation by Leonard Euler in 1752 and is still widely used by engineers. Euler also formulated a set of equations in 1757 that describe adiabatic, inviscid flow, the Euler Equations. The Euler Equations are one of the earliest known examples of a partial differential equation (PDE) recorded. These principles were further expanded on by Claude-Louis Navier with the addition of viscosity and simultaneously discovered by Sir George Stokes in the early to mid 1800s. These newly created set of PDE's are now known as the Navier-Stokes equation and remain the backbone of modern fluid dynamics almost 200 years later. This reliance continues despite the fact that the equations are not integrable in their entirety. A 1-million-dollar prize has been offered by the Clay Mathematics Institute to anyone who can prove the three-dimensional, Navier-Stokes equations are unique (Fefferman). Even without being explicitly solvable, the Navier-Stokes equations are a powerful tool in fluid dynamics; yet, in order to do solve them for general problems the equations must be discretized to be used.

There are several methods to discretize a PDE. Within computational models there are several approaches: Finite difference (FDM), finite element (FEM), finite volume (FVM) methods (Anderson 1994), and Lagrangian approaches like, smooth particle hydrodynamics (SPH) (Monaghan 2005), and the Lattice Boltzmann (LB) method (Sudhakar and Das 2020). Since the Navier-Stokes equations are not directly solvable, any method chosen becomes an approximation of the solution. The accuracy of the solution depends upon the applicability of

each model to the system being solved. FDM models are typically used with structured grids making modeling irregular geometry difficult. FEM breaks the domain into simple geometric shapes, or elements, and then shape functions are applied within the elements that are specific to the physics of interest. These elements must adhere to strict rules and are often limited to tetrahedron and hexahedron, in three-dimensions. All of the elements combined together creates a system of equations that must be solved simultaneously. While commonly used in structural mechanics, FEM is non-conservative when dealing with fluid mechanics. FVM also breaks the domain in to elements, though there is no strict restriction on the shape these elements take. Here the flux of several parameters of interest that passes through the faces of one element to the next is calculated. This approach has the advantage of taking into account the conservation laws making FVM the most widely used method for fluid dynamics. Lagrangian methods, such as SPH, that was originally designed for astrophysics or the LB method, track particles as they move through the domain. Since each particle has an assigned mass, the method inherently satisfies conservation of mass. While developed in the 1970s, these methods are seeing renewed interest and are growing in popularity due to their ease of parallelization, but they are incapable of adequately resolving boundary layer flows for most problems due to computational limitations.

The numerical methods in the Eulerian, fixed frame, i.e., FDM, FEM, and FVM, predate modern computational abilities. Solutions were limited to simple problems and the abilities and direction of human computing until the 1960s. In the 70s the potential for computers to aid in solving large scale numerical problems became quickly apparent. The size of computation machines and their cost limited their application to solve engineering problems to a select few

with access to such resources. Since then, a computational machine that could not be moved by a single person now fits in someone's pocket. Parallelization now grants the ability to leverage the power of multiple processors into a single task. Fluid dynamic problems that were impractical to solve 50 years ago are now being solved in days. Whether the domain is divided into elements, or particles, modern high-performance computing allows for millions of calculations to be processed in a fraction of the time it would have taken a room of human computers a few generations ago.

The number of calculations the fastest computers in the world, super computers, can calculate per second has increased a million-fold in the last fifteen years. While this resource is limited to a few, as was high performance computing in its infancy, there are a growing number of resources available. On the campus of Montana State University, the Hyalite High-Performance Computing Cluster enables access to valuable computing power for academic researchers on the campus of a land grant university.

Computational fluid dynamics has been used to investigate aspects of both fan and pump design. It has been used to investigate the use of splitter blades, small blades placed within the blade passage (Yuan and Yuan 2017). Outlet guide vanes to help direct the fan discharge (Senturk and Tas 2015). Inverse design where the designers starts with the desired output and works backward to determine blade shape and size (Kruyt and Westra 2014). As well as looking at the housing performance to improve total efficiency (Dilin et al. 1998; Hariharan and Govardhan 2016). The emphasis in these studies is to improve or augment the classical design approach, i.e., the design based primarily on the radial profile.

APPROACH AND METHODOLOGY

The backward curved plenum fan chosen for comparison of preliminary research has seven stamped metal blades welded between a flat backplate and a curved shroud that transitions the flow from the inlet (Figure 12). This is an example of current designs in production and use today. The fan has seven blades of constant curvature, $R = 20\text{ cm}$. It is 16 cm at the largest meridional height and 10 cm at the fan discharge. The outer diameter of the fan is 40 cm.

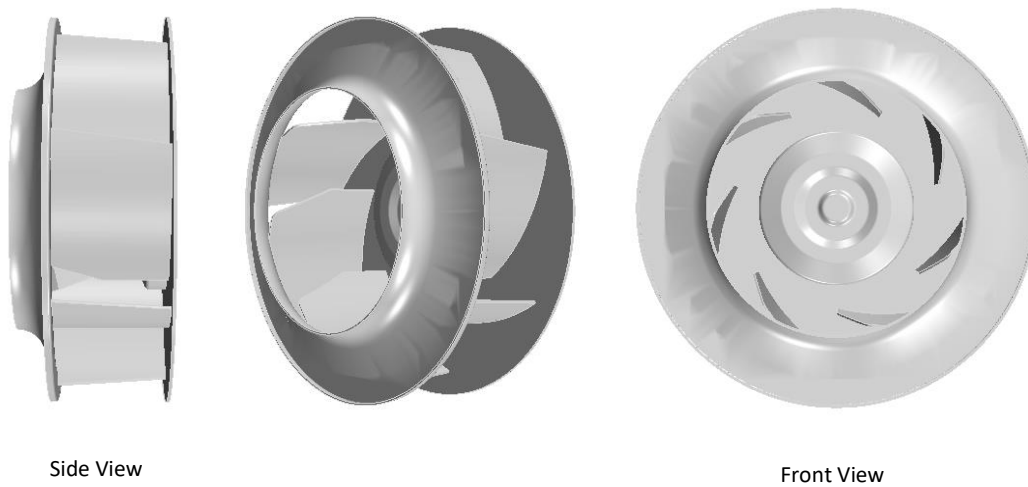


Figure 12. 7 blade backward curved centrifugal fan

For the preliminary studies the fan discharge will fill a cylindrical plenum. If the fan is centered in the middle of the chamber, a cylindrical plenum will give symmetric flow from the fan discharge. The plenum is then transitioned to the outlet through a necking down of the cylindrical plenum to a cylindrical outlet duct. With the surrounding system selected, any changes in the velocity and pressure profiles will be detectable in the fans performance. Analysis can then be performed on the developed flow under different loading conditions and how it is reflected in the performance parameters: pressure, volumetric flow rate, and power.

Computational Fluid Dynamics

Computational fluid dynamics (CFD) allows researchers and designers to simulate fluid environments. By leveraging the computational power and speed of modern computers, these simulations have become an invaluable tool to researchers and developers. While CFD will not replace experiments or theoretical approximations, it is a strong compliment in the process (Anderson 1994).

The computation of the fluid flow for this research is handled within the program Star CCM+ using a Reynolds-Averaged Navier-Stokes (RANS) equations and the $k-\omega$ turbulence model to numerically approximate the fluid motion (Siemens). Ventilation systems are known to have Reynolds numbers well above the laminar regime and are therefore turbulent in nature (Bleier 1998). The simulations were run using an implicit, unsteady solver to capture time-dependent phenomena. In order to properly capture the fluid-flow, the rotation of the fan between computational time steps was limited to no more than 1° of rotation per time step. This means that faster RPM's require smaller time steps, which increases the computational costs.

Preliminary research uses 1000 RPM which translates to a time step of 167 microseconds per 1° of rotation. CFD is dependent on how the geometries are subdivided into elements to make a computational mesh. Larger element sizes can give approximate simplified flows. However, these sizes often fail to capture fluid flow close to surfaces, complex geometries, and turbulence, but, while using small element sizes improves the accuracy of a simulations, they also contribute to large computational costs. Every element within the mesh requires calculation at every iteration. Balancing computational cost with computational accuracy is an important

consideration for time management. To better optimize the simulation mesh, the elements are allowed to change depending on where they are located within the simulation. The smallest

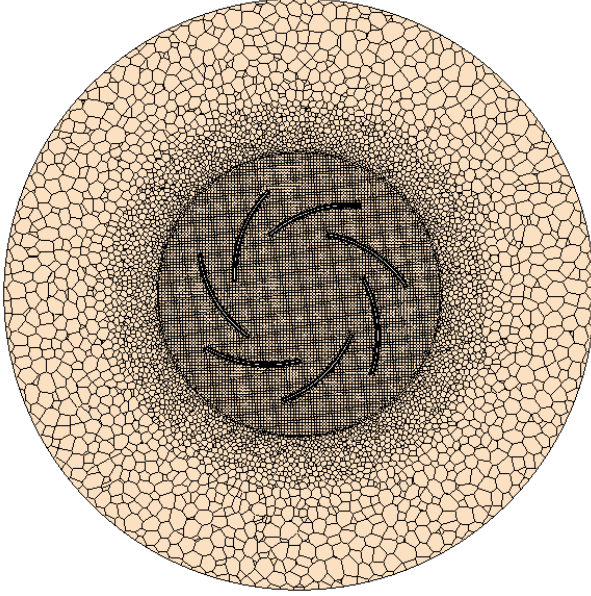


Figure 13. Radial mesh slice through the system

elements exist within a rotational frame enclosing the fan and surrounding surfaces and are allowed to grow as they move away from the fans volume into the housing (Figure 13). This allows the simulation mesh to be tailored so that the minimum number of elements can be used to minimize the computational cost while maintaining a desired level of accuracy for the fluid flow.

Flow within the system falls within the turbulent regime with a Reynolds number (R_e)

at the fan outlet around 15000, where $R_e > 4000$ is considered turbulent for internal pipe flows, where the blade passage is an internal flow bounded by the blades, backplate, and shroud. The Navier-Stokes equations comprise of one equation for conservation of continuity, three equations for the conservation of momentum for three-dimensional flow, and one equation for the conservation of energy, which can be seen in continuum notation in Equations (16), (17), and (18), respectively, where ρ is density, $\mathbf{v}$ is the continuum velocity, $\boldsymbol{\sigma}$ is the stress tensor, $\mathbf{f}_b$ is the body forces, E is total energy per unit mass, $\mathbf{q}$ is the heat flux, and S_E is the energy source per unit volume (Siemens):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (16)$$

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v}) = \nabla \cdot \boldsymbol{\sigma} + \mathbf{f}_b \quad (17)$$

$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{v}) = \mathbf{f}_b \cdot \mathbf{v} + \nabla \cdot (\mathbf{v} \cdot \boldsymbol{\sigma}) - \nabla \cdot \mathbf{q} + S_E \quad (18)$$

Where Equation (17) includes the three equations for conservation of 3D spatial linear momentum. In order for the angular momentum to be conserved the stress tensor must be equal to its transpose.

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}^T \quad (19)$$

RANS and unsteady RANS (URANS) further decompose the Navier-Stokes equations into time-averaged and fluctuating quantities which are then used to predict the solution at the next time step. The fluctuating portion is collapsed into the Reynolds stress tensor and requires at least one more equation to close. RANS models have been used in CFD to model turbulent flow since the 90s and have even been used for the fluid flow within a centrifugal fan system (Dilin et al. 1998). In order to predict turbulence within the solution, there are different methods to close the RANS equations. There are integral, eddy viscosity, and differential methods. The eddy viscosity hypothesis states that local rate of strain is proportional to the turbulent stresses in the fluid (Schmitt 2007; Hanjalić and Launder 2020). Eddy viscosity methods allow for evaluation of the

boundary layer at surfaces. The two-equation eddy viscosity method, k - ω , was chosen for this research as applied to the sheer-stress transport first proposed by Menter and seen in Equations (20) & (21) (Menter 1994), where k is the kinetic turbulent energy, ω is the specific rate of dissipation of k into thermal energy, u is the velocity, x is the spatial location, τ is the shear stress, γ is the specific weight of the fluid, μ is the dynamic viscosity, ν is the kinematic viscosity. The equations contain model coefficients β^* , β , σ_k , σ_ω , and lastly a blending function, F_1 . The k - ω is has the advantage of being better suited to near wall flows, such as those along the blade surface.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (20)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma}{\nu} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (21)$$

An implicit, unsteady model was used for the simulation because it provides more stability and allows for a larger time step which reduces computational costs. The model uses information at the current and future times step and 1st-order differentiation was implemented. Explicit solutions are conditionally stable, as they require sufficiently small time-steps to converge to a solution and remain stable. Various implicit unsteady models generally allow for larger time steps. A combination of large time steps for initializing the simulation that are later reduced for accuracy as the simulation converges to a solution was implemented to reduce total computational time. For this research, the relationship between density, temperature, and

pressure was evaluated using the ideal gas law and gas properties were taken to be that of standard air at sea level with 0% moisture. While a parameter sweep of fluid properties was not conducted for this work, the effects of temperature and pressure on fan performance can be successfully correlated by using the ratio of density at different condition (Osborne 1977). The density ratio can be applied to pressure and power to correct for different operational conditions. Solution of the energy equation was decoupled from that of pressure and velocity. Since the flow within the system falls well below the sonic flow regime, more complicated models taking into account high density ratios were not needed.

Efficiency Parameters

In order to measure the performance of the fan you need to record each parameter within the equation for fan efficiency (Equation (22)); where η is efficiency, Q is the volumetric flow rate, ΔP is the pressure difference developed by the fan, and W is the power required to spin the fan. These parameters can be measured on a working fan with moving airflow. Testing procedures for each parameter can be found in ASHRAE Standard 51 (ASHRAE 2016). The efficiency of the fan has a maximum at a specific flow rate for a given RPM, Figure 14.

$$\eta = \frac{Q \cdot \Delta P}{W} \quad (22)$$

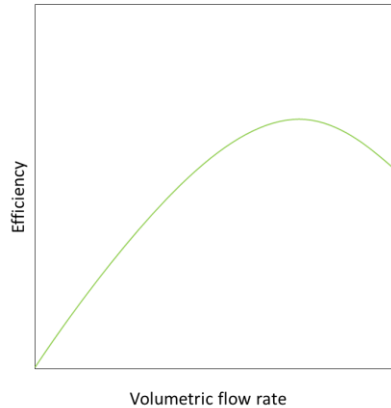


Figure 14. Efficiency curve

Pressure readings are needed upstream and downstream of the fan, as well as the barometric pressure. A differential manometer or a pair of manometers can be used for the fluid pressures and a barometer for barometric pressure. The fan pressures are measured across traverses of a circular duct where measurements at the same radii are averaged. Within the simulations, the average pressure across a selected area is calculated automatically. The pressure curve is the plot of the pressure difference, across the fan from an inlet condition to an outlet condition, vs. the volumetric flow rate for a given RPM, and can be seen in Figure 15. The curves are characterized by having higher flow rates at low pressure differences.

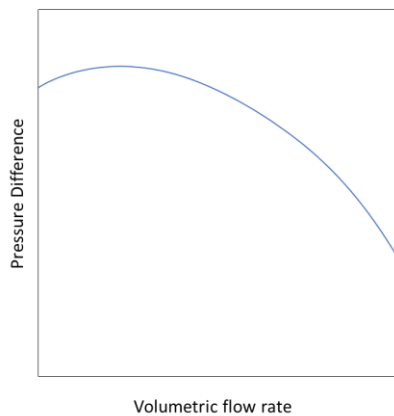


Figure 15. Pressure curve

A power meter can be used to measure the power consumed by the fan, but this must be corrected with a motor calibration curve. By using a motor calibration curve the inefficiency of the motor can be removed from the power readings. A more accurate method is to employ a torque sensor and tachometer. The simulations use the pressure and shear forces from the fluid to calculate the moment about the axis of rotation imparted on the fan, which can then be multiplied by the angular velocity to yield a term with units of power, though without any mechanical or motor losses. The power curve for a backward centrifugal fan is characterized by the non-overloading nature of the curve and can be seen in Figure 16. The fan has a maximum power draw at a specific flow rate. This means that the fan will not continue to draw more power as the flow is increased.

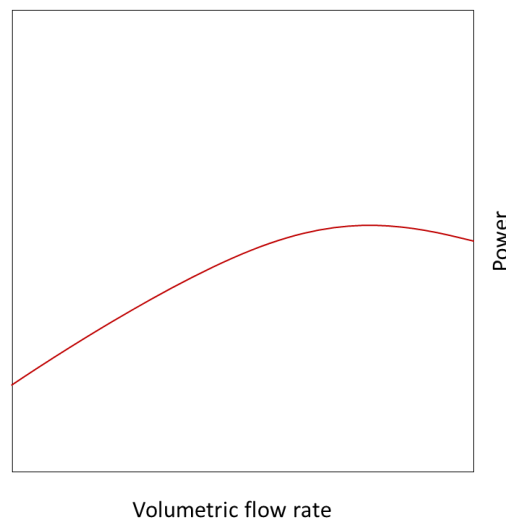


Figure 16. Power curve

A velocimeter can be used to measure the velocity in a circular duct using the same traverse method as for pressure. With a known duct area, a volumetric flow rate can be

calculated. The simulation tracks the mass flow rate leaving the system, which can then be divided by the density of the fluid at the outlet. With these parameters recorded, the aerodynamic efficiency of the fan can then be computed.

The fan is run at a particular operating speed and the airflow is varied from full flow to zero flow using a throttling device. This process produces the *fan curves*. The ASHRAE Standard 51 specifies that 10 test points is sufficient to characterize the curve. Simulations for 1000 RPM and the original fan diameter used 11 test points at static pressures from 0-200 Pa in increments of 20 Pa. A simulation conducted at a faster rotational rate (1900 RPM) using the original fan diameter used 18 test points from 0-680 Pa in increments of 40 Pa. A scaled fan simulation was also performed at 1000 RPM and a geometry with double the original diameter used 20 test points from 0-760 Pa in increments of 40 Pa.

Housing Proximity

The proximity of the housing wall to the fan discharge is not well defined for all housing types within literature. Volute housing is the one design where the design is predicated on a specific shape and relationship to the fan size and operating conditions. This, however, limits the applicability and ease of implementation. HVAC systems use rectangular plenums because of their simplicity. Between these two designs there is a lot of wiggle room. A recommended ratio for the lower limit on how close the housing wall can be to the fan is 1.5 times the fan diameter. This comes with no upper limit beyond the reality of size constraints nor any quantification as to what the effect of the housing wall proximity has on performance. For this research, the housing proximity ratio D_r is defined as the ratio between the housing diameter of a cylindrical plenum, D , to the diameter of the fan d , Equation (23).

$$D_r = \frac{D}{d} \quad (23)$$

To test the effect of the housing proximity different values of D_r were evaluated above and below the recommend ratio. Since values below $D_r = 1.5$ further restricts the volume of the plenum, only two values, 1.25 and 1.375, were tested. Values larger than $D_r = 1.5$, could increase until the fan no longer noticeably interacts with the housing. Values of 1.8, 2.1, 2.5, 3.0, 3.5 were chosen to what observable trend could be seen in the increasing housing diameter.

Suction Chamber

Within a centrifugal fan there is an internal cavity known as the suction chamber where the flow transitions from an axial direction as it enters through the inlet shroud into a radial path dictated by the blade passage and can be seen in the simple diagram illustrated in Figure 17.

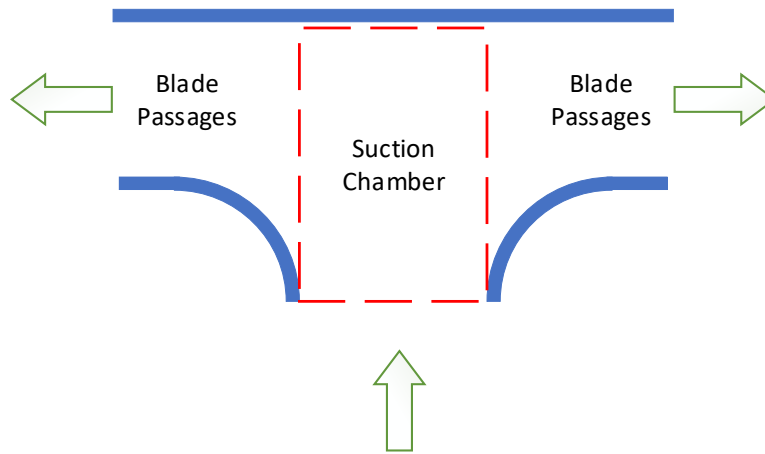


Figure 17. Meridional cross-section of a centrifugal fan with the suction chamber outlined

This chamber is utilized for mounting drive shafts and motors. The trend among industry is to utilize the chamber to shrink the overall fan/motor installation. The test fan for this research has

a 3 cm intrusion into the suction chamber by the motor shroud. In order to test the effect this has on the fan performance and the utility of such intrusions the motor shroud intrusion was tested at four incremental depths of 3 cm starting with a flat back plat with no intrusion and ending with a 9 cm intrusion.

Passage Assumptions

In traditional fan design, streamline theory and the evaluation of the forces on the fluid element within a passage both make the assumption of a 2D solution that is then assumed to be uniform through the height of the blade. The backplate and shroud guide the flow through the 90° transition and are assumed to have minimal impact on the flow if designed correctly (Cory 2005). The flow within the blade passage will be uniform away from these surfaces. Losses from the surfaces are then lumped in with losses from friction, shock losses from the fluid encountering the blade leading edge and so forth.

The reality is that these surfaces disrupt the flow between the passages. They add recirculation zones and reduce the uniformity within the passage. This means that the effective height of the blade is less than the geometric height. The effect of these losses can be minimized by choosing an appropriate design. The starting point of an appropriate design is based on controlling the expansion of the blade passage. This is then checked with experiments and the design is then tweaked by an experienced engineer. These changes are then incorporated in the final design of a fan. Almost exclusively, the changes are not published and are instead kept as proprietary knowledge by individual fan manufacturers. The assumptions made with streamline theory and by analyzing the profiles from the blade element forces provide a solid basis for the initial design.

Blade Passage as a diffuser

Traditional design starts with picking a blade form. The simplest blade form, the straight blade, is still widely used despite having lower efficiency because of durability in particulate contaminated flows i.e., industrial exhaust. This blade form has a simple solution to Equation (6) where the passage wise velocity, w , decreases linearly from the suction side to the pressure side. Constant curvature blades, circular arcs, provide the solution where w is constant across the passage. These two blades illustrate different approaches to blade design. The straight blade has a known linear distribution of w and is a valid solution to Equation (6). The circular arc blade shape assumes that $\frac{\partial w}{\partial n} = 0$ and has a uniform distribution across the passage. A uniform distribution being preferable as it leads to less turbulence. Though in reality the distribution of a circular arc blade is not perfectly uniform. As the blade form becomes more complicated the distribution of w is harder to determine without further assumptions. Regardless of the blade form chosen, the velocity distribution is assumed to be uniform through the height of the blade.

The meridional profile is then chosen to guide the flow through the 90° transition into the blade passage and to help control the expansion of the passage. A straight blade can have a constant breadth meridional profile, i.e. flat shroud with a flat backplate. This makes the fan easy to manufacture. Curved blades will most often have a slanted or curved shroud. A backward logarithmically curved blade can be paired with a hyperbolically curved shroud. In each case, one consideration comes up in literature repeatedly, that the passage should act like a diffuser and should increase in area accordingly. That is, the cross-sectional area should not drastically change from the blade passage inner radius to the outer radius. This last piece in fan design for the height of the passageway is traditionally evaluated after the blade curvature and meridional

profile have been chosen. An area within the passage is compared to the position along the mean path in the radial plane between the blades. The slope of the area along the path should fall between 9 to 12° (Eck 1973). This suggests that there is a sweet spot where the passage is expanding efficiently, neither over nor under expanding. Though the effect is not clearly outlined in literature it would result in a higher maximum efficiency and/or a larger operational range

Standard Method of Area Evaluation

The traditional method of evaluating the area of the passage provides an approximation that is used as a check to evaluate the design. Evaluation of the blade passage area consists of inscribing a circle between the blades in the radial plane. The diameter of the circle is then multiplied by the average height of the passage, which is taken as the height at the center of the circle. This is done graphically and the steps can be seen in Figure 18 (Eck 1973).

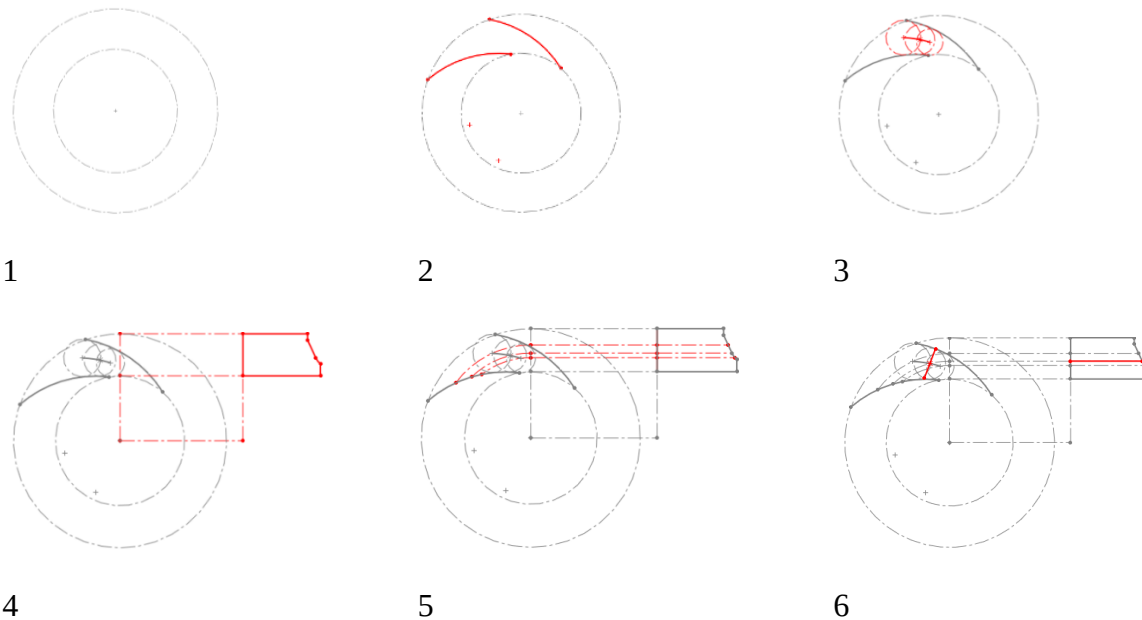


Figure 18. Steps for finding passage area using graphical method

1. Once the diameter of the leading edge and trailing edge of the blade are determined,
2. the blade shape can be constructed outlining the blade passage.
3. Inscribing circles within the passage, use their centers to draw the mean path between the blades.
4. The meridional profile of the fan is projected onto the sketch and
5. the location of the circle centers is transferred to the profile.
6. Finally, the diameter of the inscribed circle is multiplied by meridional height of the passage as determined to give the area.

The area is defined by the inscribed circle diameter; however, the diameter of the circle does not line up with the circles' intersections with the two blades. Figure 19 shows the diameter of the circle as it intersects each blade, $\overline{Aa}$ and $\overline{Bb}$, and the diameter location where it falls within the passage intersecting both blades, $\overline{A'a'}$. Where the $\overline{A'a'}$ intersects the mean path shows that at that diameter, the height within the meridional profile is different than the location of the circle center used in the calculation. The diameter of the circle can in fact be oriented between the blades at multiple angles to the projection line, none of which will have the same meridional height as the circle center.

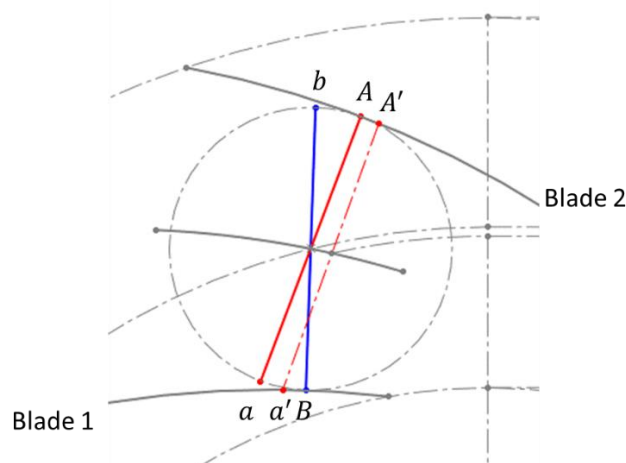


Figure 19. Diameter and blade intersections with alternate diameter location that intersects both blades

Another issue is identifying the average height of the passage as the translation of the center of the circle to the meridional profile. This approach assumes that the passage is roughly trapezoidal, and the area is the base of the trapezoid multiplied by the average height. The circle does not locate the plane the area is supposed to be on nor where it lies on the mean path. For visual reference the plane of the area was located using the intersections of the circle with the blades $\overline{AB}$ in Figure 20. Looking at a slice of the passage defined by $\overline{AB}$, the translated height from the standard method, $\overline{Cc}$ underestimates the area while the trapezoid defined by the height at each blade, $\overline{Cc'}$, overestimates the area.

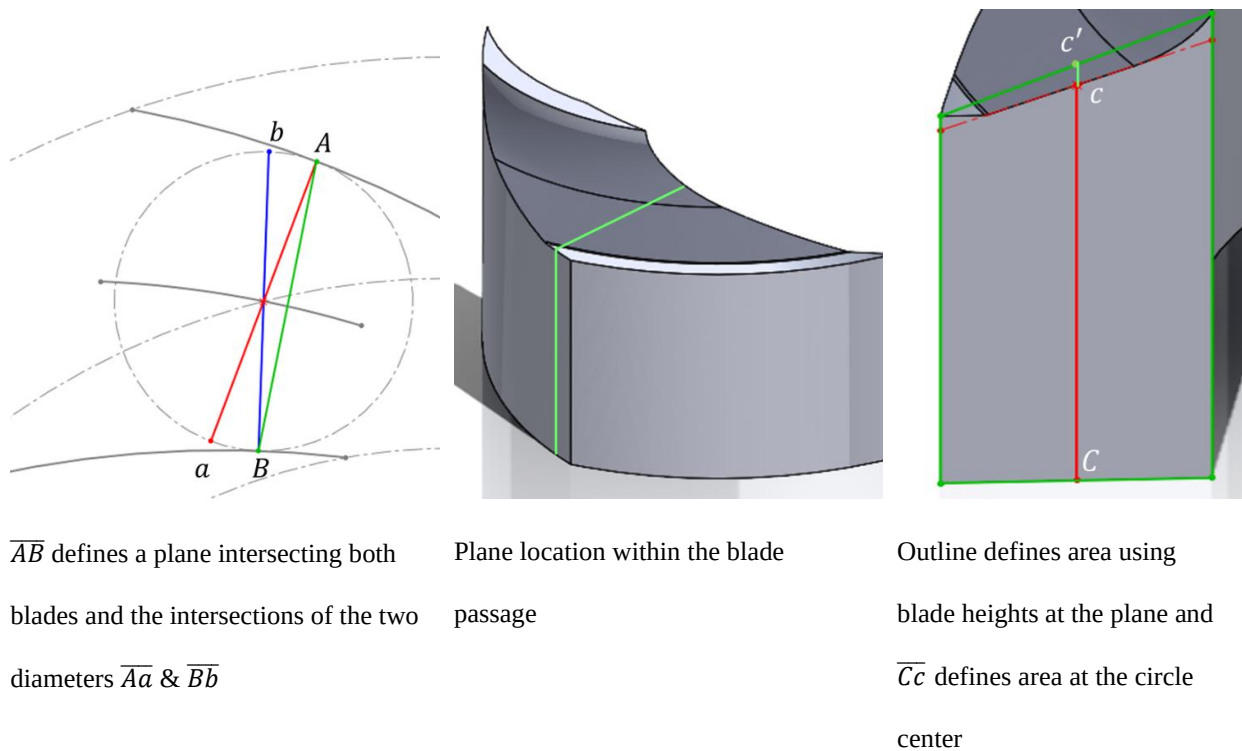


Figure 20. Standard method height compared to trapezoid defined by blade height

Figure 21 shows the plot of the area of the test fan for the standard method and the true area, evaluated using planes perpendicular to the mean path at the circle centers, and the standard

method does capture the area of the passage fairly well. The standard method does underestimate the passage in the middle of the mean path. At the beginning of the passage the true area is much smaller than is accounted for by the standard method. While the slope of the lines for the standard method and the true area within the middle of the passage do adhere to the 9° - 12° angle limit, the beginning and end of the passage deviate. Both approaches demonstrate a sharper increase in area at the beginning of the passage and a decline in area expansion at the outlet.

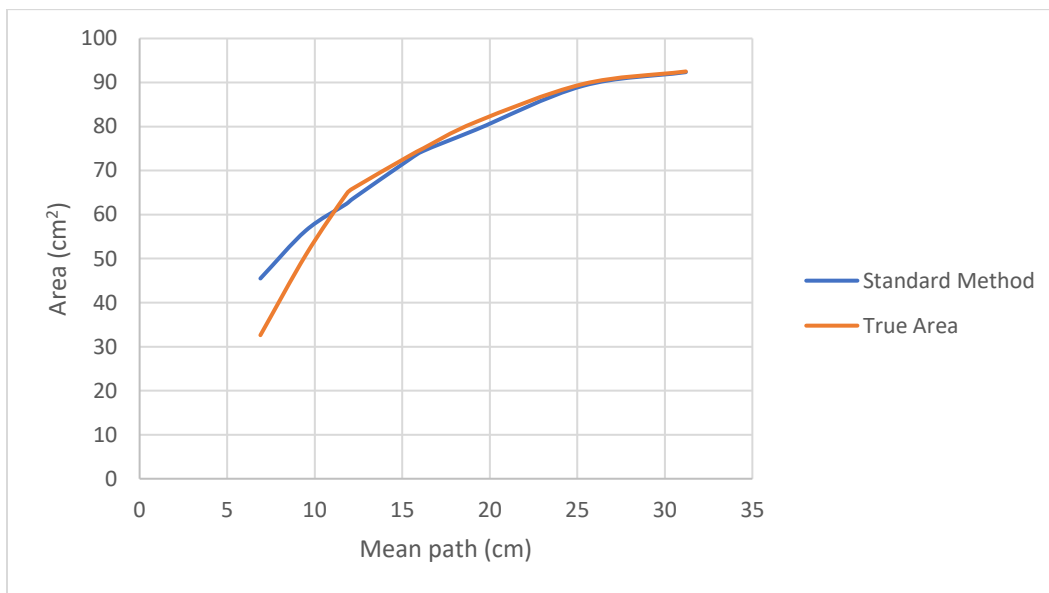


Figure 21. Standard method and true area of a blade passage along the mean path

Alternative, Controlled Area Expansion

While the old method of area evaluation can be used to approximate the slope and check to see if the design adheres to the diffuser principle, it does not use the diffuser principle to drive the design. If the blade passage is supposed to act like a diffuser, why not use the concept of a diffuser earlier in the design process. This entails controlling the area expansion of the blade passage along the mean path. The simplest high efficiency diffuser is that shaped like a cone

(Patterson 1938). The diffuser works off the Bernoulli principle, Equation (24). By comparing the radial profile of a blade passage with the cross-section of a cone the similarities become apparent as

$$P_1 + \frac{1}{2}\rho v_1^2 = P_2 + \frac{1}{2}\rho v_2^2 \quad (24)$$

and seen in Figure 22. Assuming that each passage is quasi-1D in the direction of flow the volumetric flow (Q) is simply the area (A) times the mean velocity (v). For the following analysis the density of the fluid is assumed to be constant for simplicity. Changes in density are taken into account for accurate aerodynamic testing of fans, (ASHRAE 2016) and by the simulations. By continuity, Q is equal at any point in the direction of flow. Since by the geometry of the two profiles the area is continually increasing this leads to a reduction in velocity and an increase in pressure. This increase in pressure is highly desirable in a centrifugal fan.

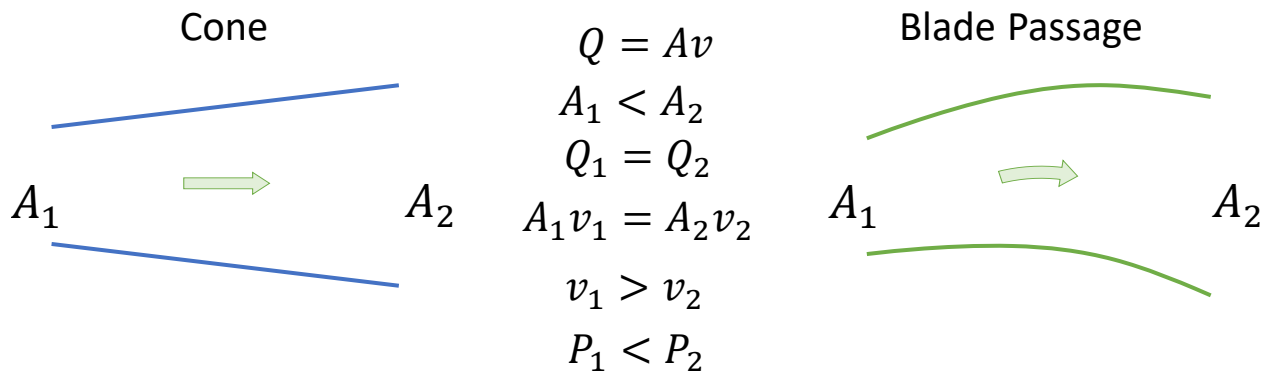


Figure 22. Cross-sections of a cone diffuser (left) and a blade passage (right) with quasi-1D continuity equation

In order to further relate the cone diffuser to the blade passage it is necessary to have a metric for an efficient cone diffuser. The main goal of the diffuser in this case is to increase the pressure at the blade passage outlet as much as possible. How quickly can the cone expand? It turns out not very much. A relatively small angle of 7° or is recommended as measured from the centerline of the cone to one side of the cross section (Bleier 1998), where this limit is for stationary duct diffusers and is well above the recommended range described by Eck as seen in Figure 23. Where the slope proposed by Eck is a straight line the area of cone is a curve.

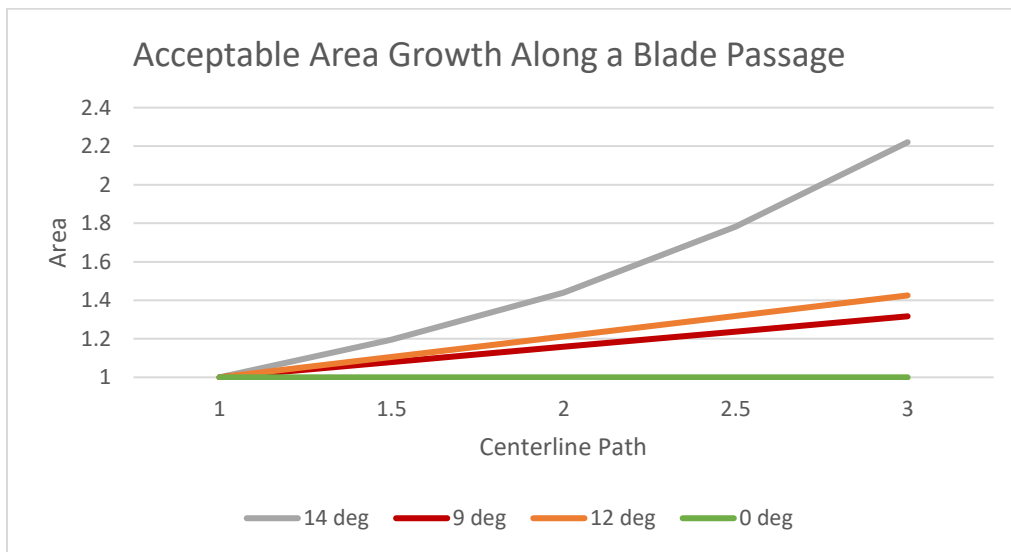


Figure 23. Comparison of angle limits set by Eck and Bleier

These small angle constraints are further supported by aeronautical diffuser testing (Patterson 1938). To use this angle in the blade passage design, the cone and blade passage need be correlated. This is done by transforming the cone parameters along the centerline to the blade passage parameters along the mean path. Figure 24 shows the needed parameters of the cone going forward.

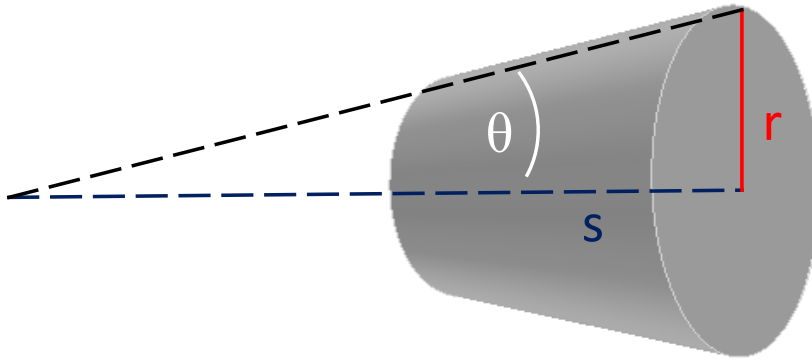


Figure 24. Cone diffuser with geometry parameters

The area of the cone diffuser cross section is that of a circle, Equation (25).

$$A = \pi r^2 \quad (25)$$

This can be related to the half angle of the cone along the centerline, Equation (26).

$$\tan(\theta) = \frac{r}{s} \quad (26)$$

Solving for r and substituting into the Equation (25) gives the area of the cone diffuser as a function of the half angle, Equation (27). Equation (27) can now be used to control the expansion of an area profile along a path s .

$$A = \pi s^2 \tan^2(\theta) \quad (27)$$

If you plot the area of the cone along its centerline as you would in the standard evaluation method, you see that the shape is parabolic as seen in Figure 25 and the rate of change is linear as seen in Figure 26.

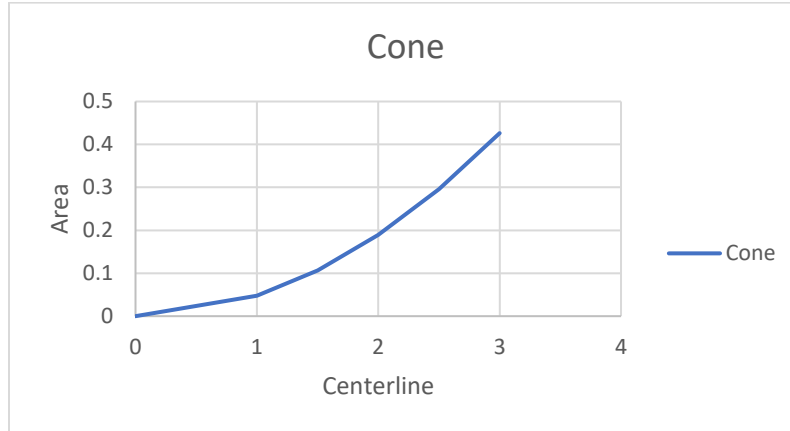


Figure 25. Area of a cone vs centerline

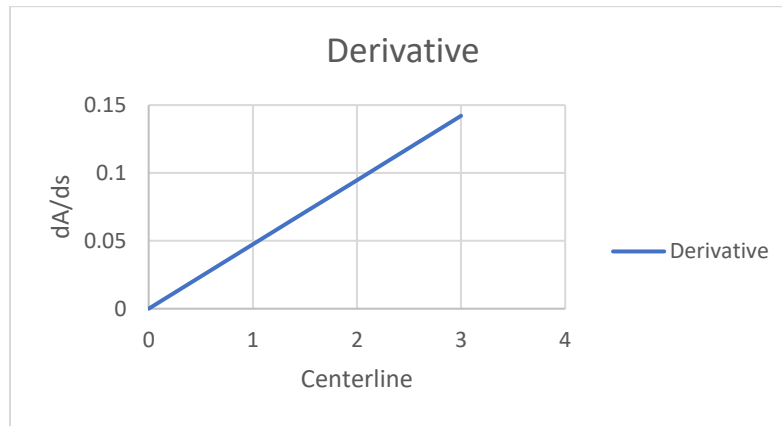


Figure 26. Area change vs centerline

Five angles ranging from 14° of total expansion angle limit for a diffuser described in literature (Bleier 1998), where the total expansion angle $\phi = 2\theta$, to 0° representing no expansion were tested. $\phi = 14^\circ$, 10.5° fall above the recommended range by Eck while $\phi = 7^\circ$, 3.5° fall below, as seen in Figure 27.

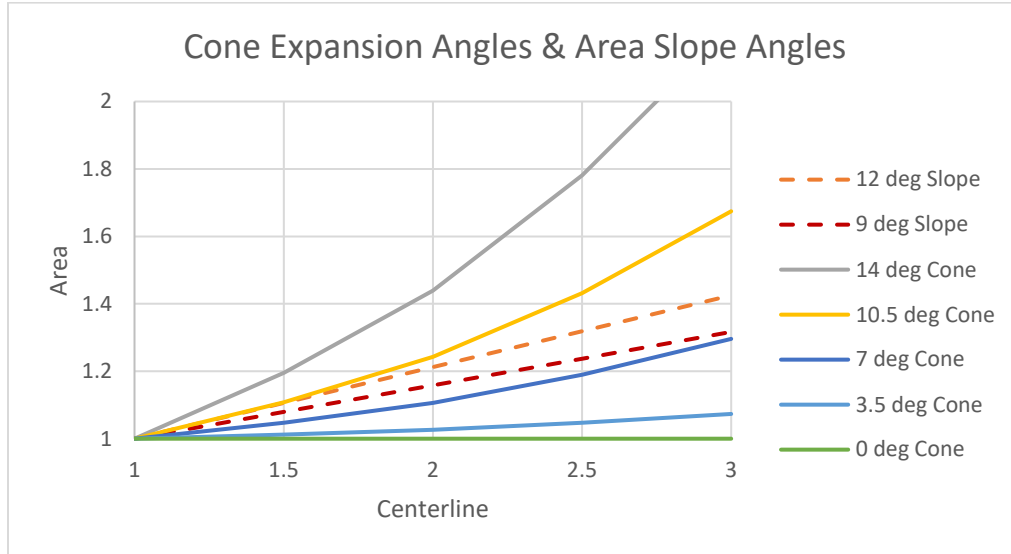


Figure 27. Cone expansion angles to 0° to 14° compared to the 9° & 12° angle limit proposed by Eck

A New Blade Passage Procedure

Building the new blade passage using a controlled area expansion uses similar starting steps to the standard method of area evaluation as seen earlier. For reference, the inner outer radius, and the center points of the blade curvature of the test fan can be seen in Figure 28. Test fan reference geometry. This is the starting point for building the new blade passage. The blade passage for this example will be built up from the back plate.

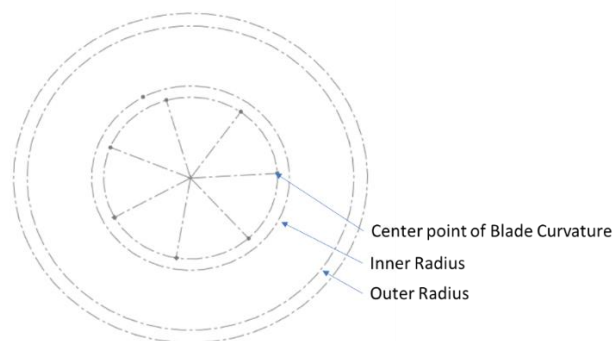


Figure 28. Test fan reference geometry

Two blades are used for the bounds of the blade passage, Figure 29. The blades in this example have constant curvature, $R = 20\text{ cm}$. This matches the curvature of the test fan.

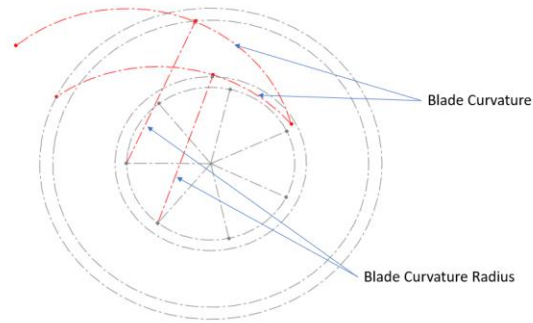


Figure 29. Blade passage bounds

The circle method is then used to find the mean streamline. As before, in the standard method, circles are inscribed between the two blades. The centers of the circles then mark out the blade passage, Figure 30. Planes perpendicular to the mean line are then used for the location of a given area long the mean path as function of Equation (27), and observed in Figure 31.

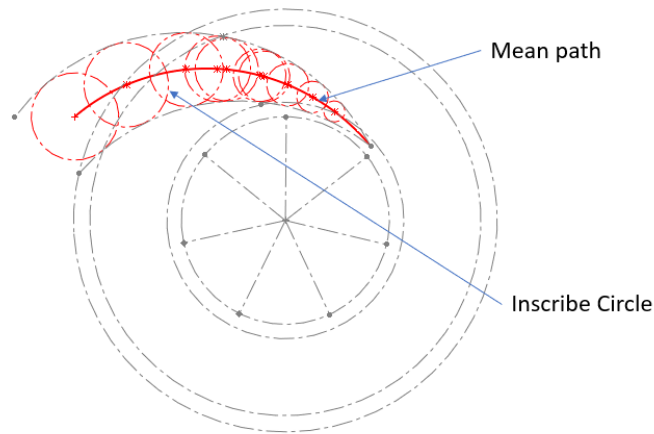


Figure 30. Inscribed circles with mean path

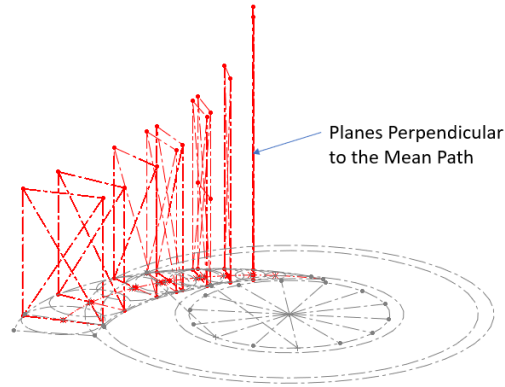


Figure 31. Planes perpendicular to the mean path

For a given area, the location s is different for any θ . The initial area A_i , is constant for all tests. This means that the starting s location s_i , seen in Table 1, is different for each case. Once s_i is known, the length of the mean path is added and a correlated s is then used to calculate the area at each plane.

Table 1. Starting s locations for each total expansion angle ϕ

<i>Case</i> ϕ	s_i <i>cm</i>
<i>0</i>	<i>0</i>
<i>3.5</i>	<i>148.96</i>
<i>7</i>	<i>74.41</i>
<i>10.5</i>	<i>45.53</i>
<i>14</i>	<i>37.07</i>

s_i was calculated by rearranging Equation (27) giving Equation (28),

$$s_i = \sqrt{\frac{A_i}{\pi \tan^2 \theta}} \quad (28)$$

This area can be applied to any geometric shape. With a blade curvature chosen, the width of the passage is known and a fixed value that depends upon the inner and outer radii, the number of blades, and the blade form. The height is therefore dictated by the width and the desired diffuser area. For an ellipse, the area is

$$A_{ellipse} = \pi ab = \frac{\pi AB}{4} \quad (29)$$

where A and B are the length of the major and minor axes, respectively. If B is the width of the passage, then A is the height. This results in an ellipsoidal passage, as can be seen in Figure 32. Here, there is no traditional meridional profile. Each passage is independent of a single meridional profile.

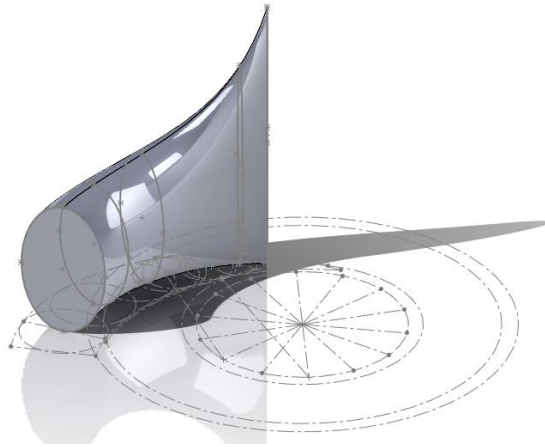


Figure 32. Ellipsoidal passage

The new passage can then be truncated at an inner and outer radius. The resultant shape is then arrayed and subtracted from a cylinder volume resulting in a new fan, with an ellipsoidal example shown in Figure 33. The new ellipsoidal passage fan CAD can then be exported into any CFD or analysis software for fluid and structural evaluations.

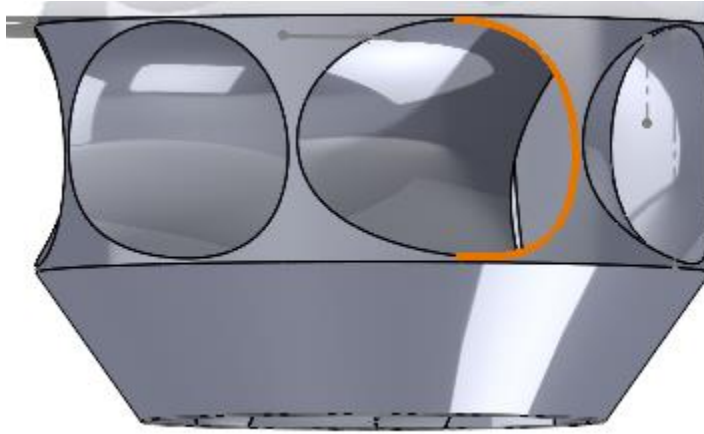


Figure 33. Ellipsoidal passage fan

RESULTS

Multiples studies of the backward centrifugal fan were performed using Star-CCM+ as the CFD software. Performance of a centrifugal fan is impacted not only by the inherent fan design but also the implementation of the fan. How the fan is housed, the rotation driven, even how it is mounted affects the total system performance. Two studies were performed on system geometries that impact the fluid flow. One inside the fan that is not inherent to traditional blade design, and one outside the fan inherent to the discharge. Multiple studies were performed on the blade passage to evaluate the effects of total efficiency. The cross-sectional area shape was tested between the test fan, a rectangular, and an ellipsoidal passage. The expansion angle of the blade passage as it relates to a cone diffuser was tested for the ellipsoidal passage. Fan size and rotational speed were tested to evaluate applicability if the diffuser concept translated to different operating conditions. Last, the blade form was changed from a constant curvature to a logarithmic spiral to test the effects of the diffuser principle on a different mean path.

System Studies

For the system studies, the test fan with seven blades was used. This provided a constant base line for the simulations. The system with study parameters outlined can be seen in Figure 34.

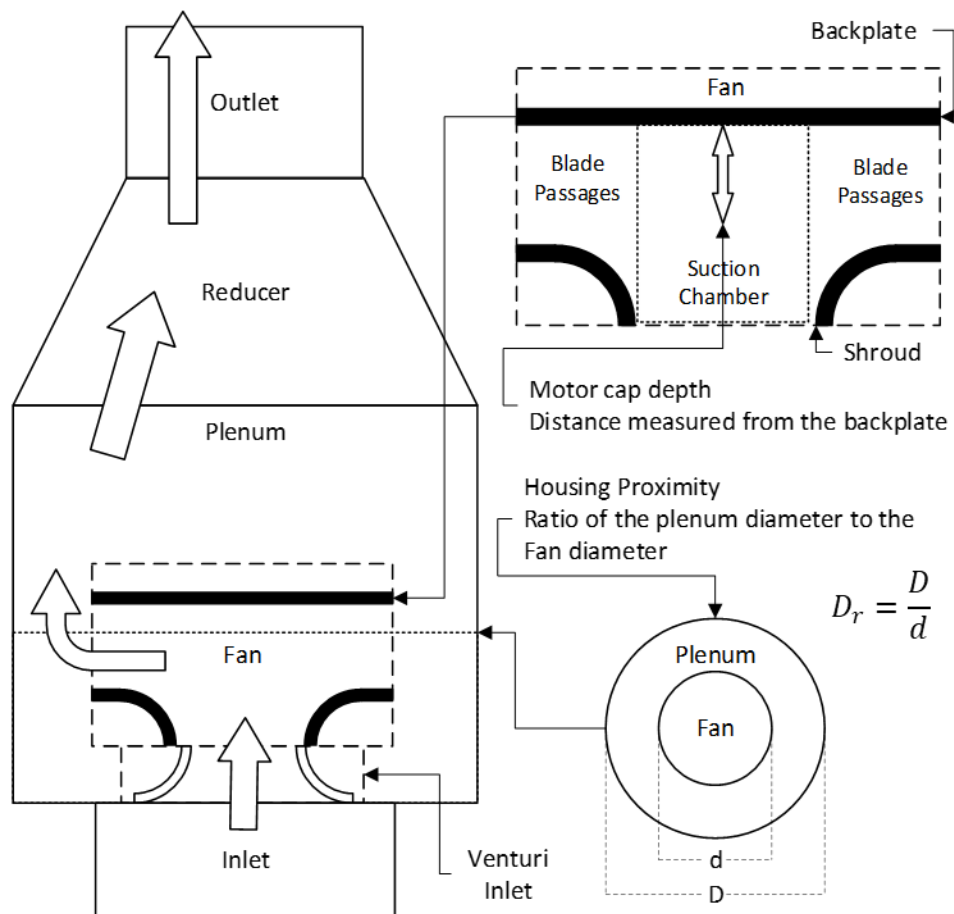


Figure 34. System study diagram with parameter definitions

Housing Proximity Study

The proximity of the housing to the discharge of the centrifugal fan impacts its performance. Intuitively, having the housing surface too close to the discharge unnecessarily constricts the flow by adding an undue amount of resistant that would require a much larger

amount of power to push the same amount of air. Centrifugal fans that have a cylindrical housing call for a minimum housing radius 1.5 times the diameter of the fan with the optimal radius requiring empirical testing (Bleier 1998). A review of current literature has not yielded anything published on the effect of having a larger discharge volume.

Eight different housing cases were simulated at 100 Pa static pressure at the outlet. Figure 35 shows how the change in housing diameter affects the efficiency. A D_r of 1.5 correlates to the highest flow rate. A D_r of 1.375 correlates to the highest efficiency. At the smallest housing diameter tested, the constriction of the flow is evident in the reduction of the both the flow rate and efficiency. Housing diameters larger than the recommended D_r of 1.5 lose both flow rate and efficiency as well, though not as drastically. The effect on the flow rate diminishes as the housing diameter increases.

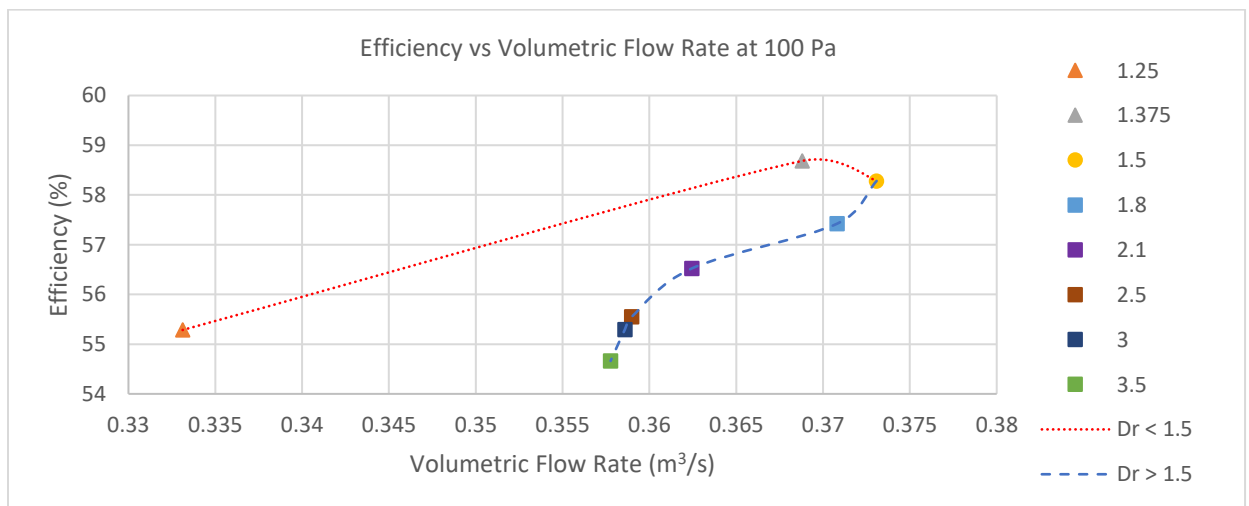


Figure 35. Volumetric flow rate vs. efficiency for different housing diameters at 100 Pa

The efficiency curves for four cases, D_r of 1.25, 1.375, 1.5, and 3.5, were simulated for a deeper comparison across a range of static pressures, see Figure 36. It should be noted that the efficiency boost observed at D_r of 1.375 is now reduced, with peak efficiencies being observed at static pressures other than 100 Pa. The D_r of 1.25 has a clear reduction in performance. This narrowed efficiency curve means that the system has limited applicability. The highest efficiency peak is at a D_r of 1.375, though it has a reduction in performance when compared to the operation ranges of the larger D_r ratios. Well above the recommended D_r of 1.5, the D_r of 3.5 has a similar efficiency curve to the recommended D_r of 1.5. A large plenum with a D_r of 3.5 provides for flexibility in design while retaining most of the fan performance.

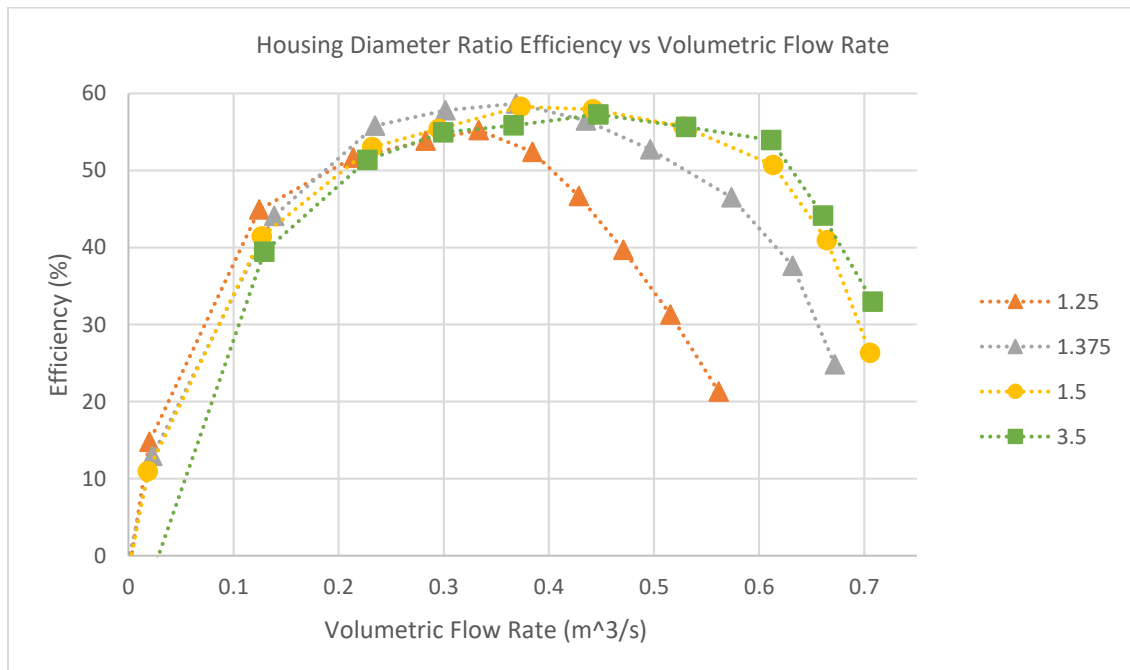


Figure 36. Volumetric flow rate vs. efficiency curves for selected housing diameters

Intrusion Study

The fan suction chamber (Figure 37) is where the flow makes a 90° transition from the inlet into the blade passage. This is also where the fan is mounted to the drive shaft at the back of the chamber. The configuration and geometry of drive shaft mounting varies between manufacturers. It can be as simple as a drive shaft collar or a motor shroud. Many current designs are attempting to push these geometries deeper into the suction chamber. To understand the effect of the current motor shroud on the design a series of CFD simulations were run to see the impact on the internal flow and the performance.

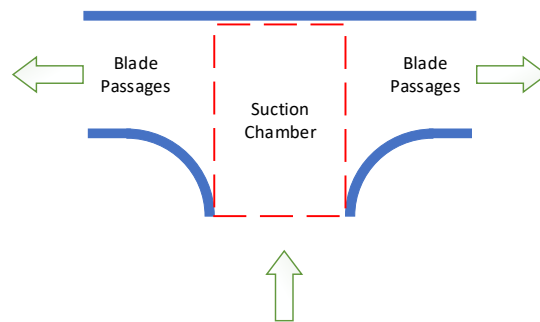


Figure 37. Meridional cross section of a centrifugal fan with the suction chamber outlined

The test fan design has a 3 cm dome shaped shroud that is also used for mounting the fan to the motor assembly and is based off a production model, with a suction chamber that is 15 cm in depth. Four design iterations were simulated with different geometries in the suction chamber. First, the fan design with no motor shroud protruding into the suction chamber and the flat backplate was chosen, then the current 3 cm intrusion into the suction as well as a 6 cm and 9 cm extension of the current motor shroud (Figure 38).

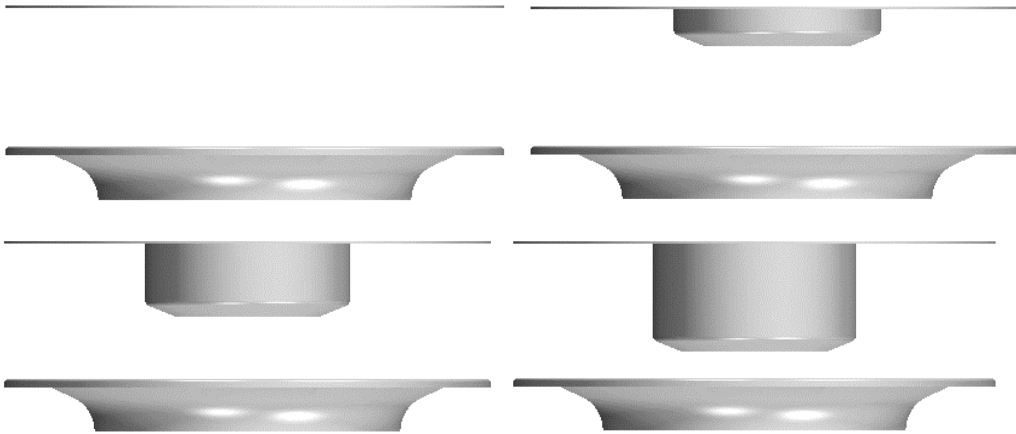


Figure 38. Suction chamber with motor shroud depths, ranging from (left to right) 0,3,6, and 9 cm with blades removed for visual clarity

The resultant efficiencies can be seen in Figure 39. It can be seen that the flat plate and the 3 cm motor shroud efficiency are similar with the 3 cm case having a slightly higher peak efficiency. The 6 cm case has a minimally lower efficiency throughout the performance curve that drops off more quickly at higher flow rates. The 9 cm case is visibly lower at peak efficiency and has a narrower efficiency curve.

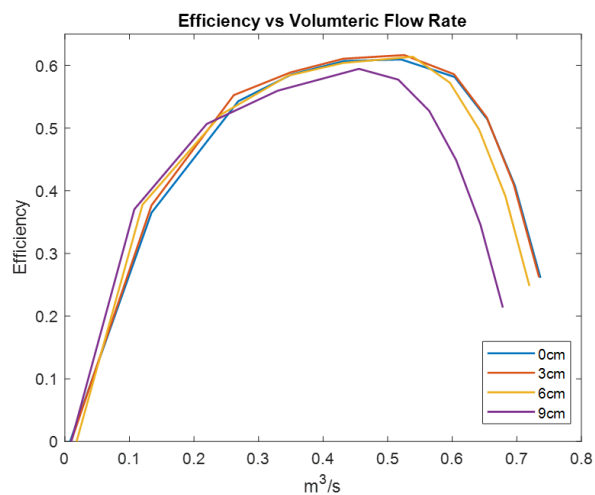


Figure 39. Efficiency curves for introducing a motor shroud at different depths into the suction chamber

Visual inspection of the velocity and pressure profiles within the suction chamber for the flat plate shows a large, low-velocity pocket near the plate with a coinciding increase in pressure around the stagnation point at the center of the plate (Figure 40 & Figure 41). This is a reduction in the energy of the fluid flow. While the stagnation point cannot be eliminated, the effect can be mitigated. The 3 cm case reduces the size of the pocket while still allowing the fan to develop a high negative pressure at the inlet. The 6 cm and 9 cm cases reduce the volume of low-velocity fluid flow, but these cases also reduce developed negative pressure. This can be seen in a plot of the center line pressure from the stagnation point to the inlet (Figure 42).

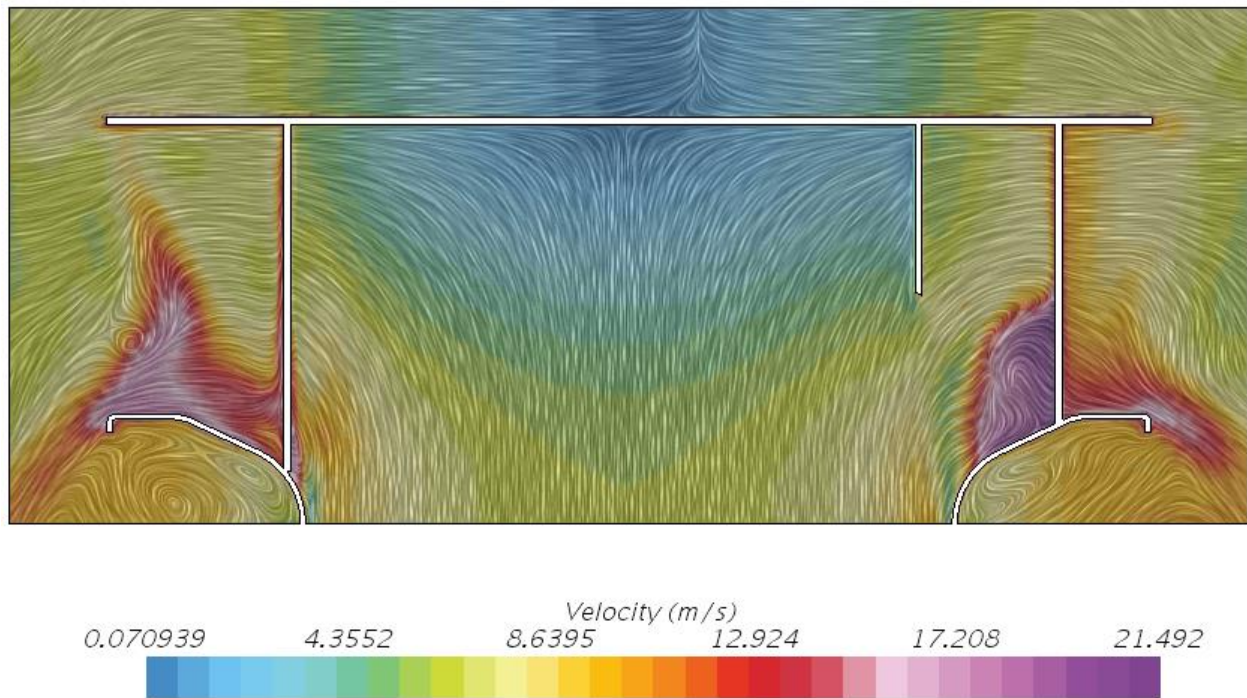


Figure 40. Velocity profile within the suction chamber for the 0 cm motor shroud

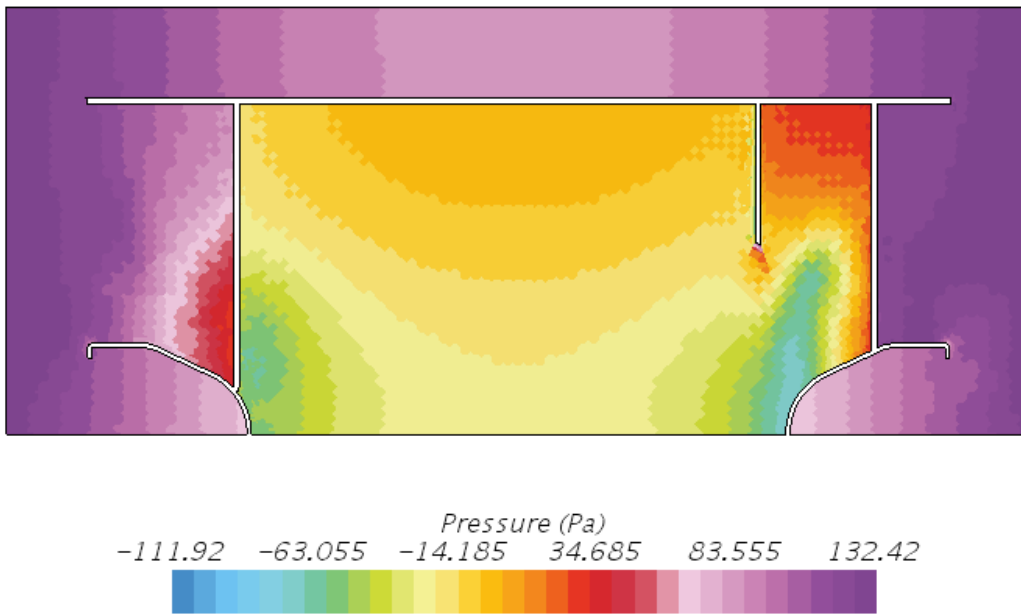


Figure 41. Pressure profile within the suction chamber for the 0 cm motor shroud

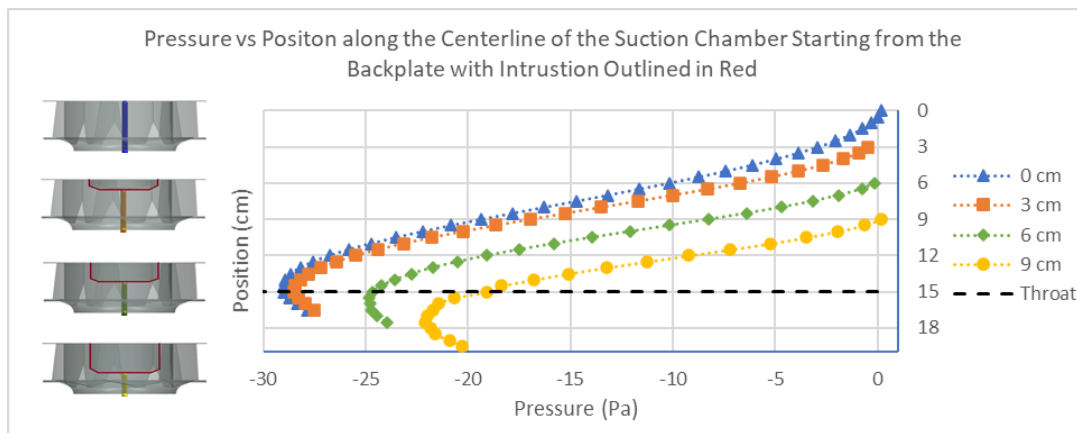


Figure 42. Pressure distribution along the center line of the suction chamber, coinciding with the axis of rotation where 0 position is at the backplate for the 0 cm, flat backplate case. All cases shows are at peak efficiency

The negative pressure developed at the suction chamber affects the breadth and peak efficiency for each case, though the minimum pressure value and location is different. Seen in Figure 42, the minimum pressure and location can be seen in relation to the transition between

the fan and the throat. The closer the minimum location is to the throat the higher the efficiency.

Efficiency values and distance from the throat are tabulated in Tables 2 and 3.

Table 2. Intrusion Study Peak Performance Values

<i>Case</i> <i>cm</i>	ΔP <i>Pa</i>	Q m^3/s	W <i>W</i>	η <i>%</i>
0	100.12	0.521	81.44	60.96
3	100.35	0.526	85.61	61.64
6	101.27	0.539	88.95	61.40
9	100.68	0.516	90.06	59.48

Table 3. Intrusion Study Minimum Pressure Location at Peak Performance

<i>Case</i> <i>cm</i>	<i>Distance from Backplate</i> <i>cm</i>	<i>Distance from Throat</i> <i>cm</i>	<i>Minimum Pressure</i> <i>Pa</i>	η <i>%</i>
0	14.0	1.0	-29.01	60.96
3	15.0	0.0	-28.48	61.64
6	15.5	0.5	-24.83	61.40
9	17.5	2.0	-22.11	59.48

This shows that there is an ability to utilize a portion of the back of the suction chamber without negatively impacting performance. In the case of the 3 cm motor shroud the effects of the stagnation point are reduced while still allowing the development of negative pressure. The 6 cm case developed more pressure and flow rate but at the expense of more power. An optimized shape and size of the intrusion could be achieved through more testing and aerodynamic profiling.

Controlled Area Expansion

Traditional blade design uses the assumption that the 2D profile of the radial plane used in the blade design dominates the system and that the flow is uniform along the length of the blade with minimal impacts from the meridional profile. Figure 43 shows the non-uniformity of

the velocity profile within the passage in relation to the blade height. In fact, there is a considerable turbulent zone caused by the inlet shroud as the flow exits the blade passage into the housing. While the path the flow takes is related to the blade from, backward curved in this case, there is significant influence from the inlet shroud.

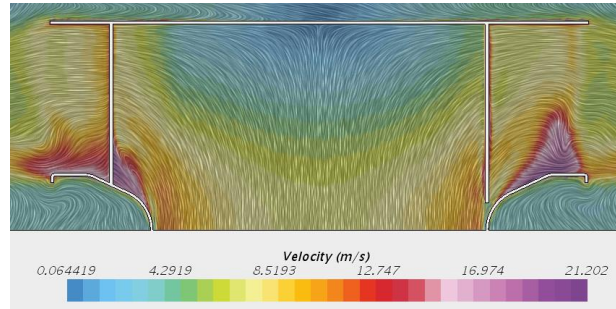


Figure 43: Meridional velocity profile of the test fan

The traditional analysis assumes that the velocity profile between the passages is also constant along the height of the blade. The rectangular shape of the passage outlet can also clearly be seen in Figure 44. Not only is the velocity field non-uniform along the height of the blade, the profile between the blades is significantly different closer to the shroud than it is near the back plate, showing limitations in both of the governing assumptions in traditional fan design.

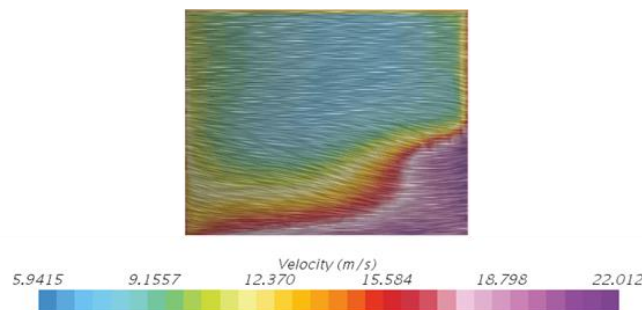


Figure 44: Velocity profile of blade passage discharge

A New Blade Passage

The goal of this research is to design a new blade passage using controlled area expansion along a blade passage mean path. The area expansion is related to the expansion of a cone along the centerline which has a linear rate of change. Traditional blade design does not strictly enforce an area expansion, nor does it enforce a geometric profile. The profile along the mean path is the result of sandwiching the blades between a backplate and inlet shroud. The backplate and shroud are traditionally simple revolved shapes around the axis of rotation. The main components of traditional design being stamped parts, the blades, backplate, and shroud, cater to manufacturability. The new blade passage procedure removes these manufacturing restraints and the resulting blade sandwich which results. Here, the resulting design from the new procedure is not reliant on the traditional manufacturing process. Instead, the profile of the blade passage and the expansion are the driving factors of the design.

Area Shape (Traditional vs Rectangular vs Elliptical) The flow area perpendicular to the mean flow path can be any arbitrary shape. Traditional design leads to a shape closer to a trapezoid. This shape is not constant through the blade passage as it transitions to a rectangular profile at the discharge. This discontinuity in shape means that the flow is not only changing area, it is changing area in an inconsistent manner, moving from an irregular trapezoid to a rectangle. For comparison, the two new blade passages were created that were forced to adhere to a rectangular or ellipsoidal profile. The initial starting area at the beginning of the passage was kept constant between the three models. Each passage width was dictated by the constant curvature of the test fan and the seven blade passages. The rectangular and ellipsoidal passages

used an expansion angle of 7 deg, $\theta = 3.5$ deg. The rotational speed was constant at 1000 RPM.

The fluid volume created by these geometries can be seen in Figure 45.

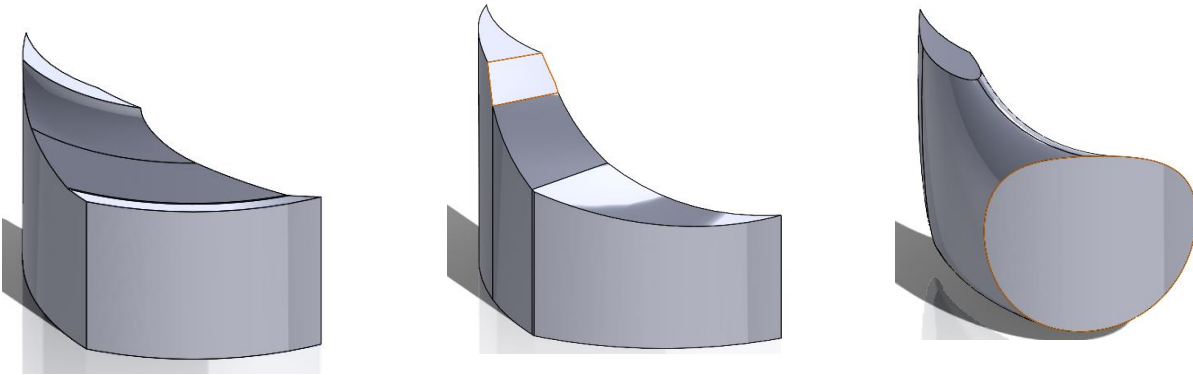


Figure 45. Blade passage volumes of fluid: Test, Rectangular, Ellipsoidal from Left to Right

As can be seen in Figure 46, the ellipsoidal passage achieves a higher efficiency over a larger range of flow rates and can be said to be moving more air efficiently. The rectangular passage has a higher maximum efficiency than the test fan, but it has an operational zone at lower flow rates. The test fan has the lowest maximum efficiency of all of the cases. The ellipsoidal passage has a maximum efficiency of 67% and an operational zone between 50 and 130 Pa, and between 0.25 and 0.65 m³/s. The Rectangular passage has a maximum efficiency of 63% and an operational zone between 50 and 150 Pa, and between 0.18 and 0.53 m³/s. The test fan has a maximum efficiency of 58% and an operational zone between 30 and 150 Pa, and between 0.18 and 0.63 m³/s.

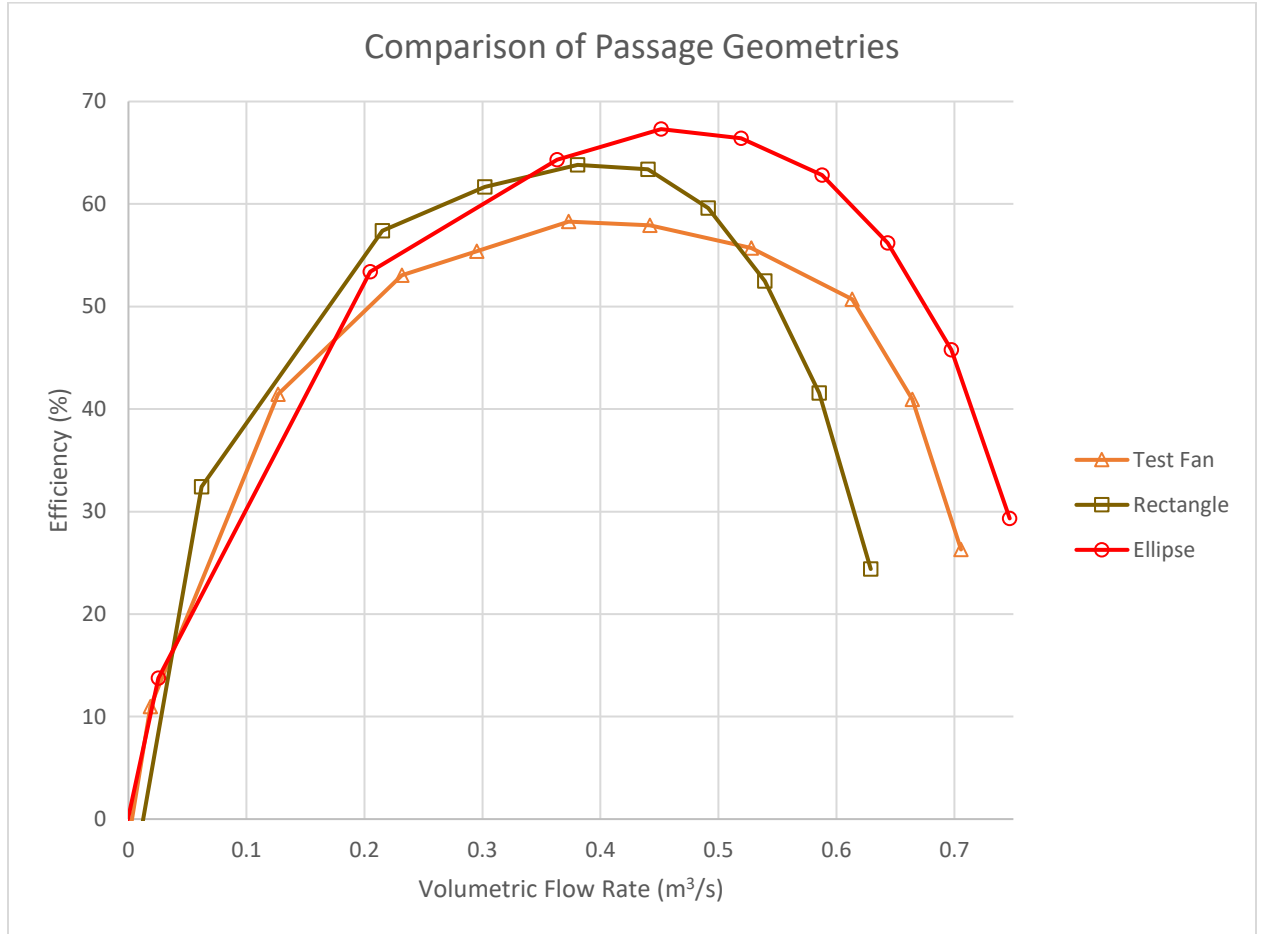


Figure 46. Volumetric flow rate vs. efficiency for different passage geometries

Figure 47 shows the pressure gradient along the different passages tested. The Ellipsoidal passage has the smoothest pressure gradient. This results in less disturbance within the passage. The ellipsoidal cross section removes the hard corners from the area cross-section and has less friction for the same cross-sectional area. This was clearly shown in early experiments for aeronautic diffuser done in the 1930s, where rounded diffusers outperformed rectangular diffusers in empirical testing (Patterson 1938).

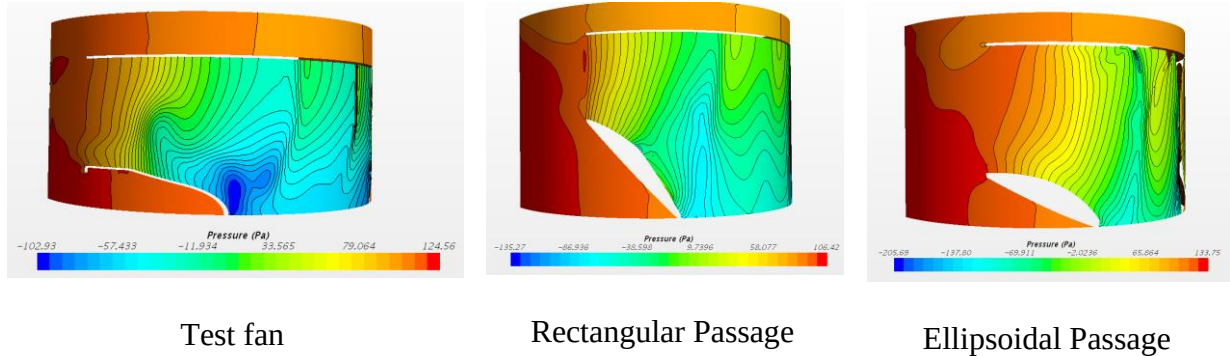


Figure 47. Pressure gradients along blade passages for the area shape cases

Looking at the radial plane of the three passages, disturbances with the test fan passage can clearly be seen (Figure 48). The structures within the test fan passage are indicative of turbulence in the passage. The turbulence is a time-dependent phenomenon, and the progression is caught at different stages in each passage.

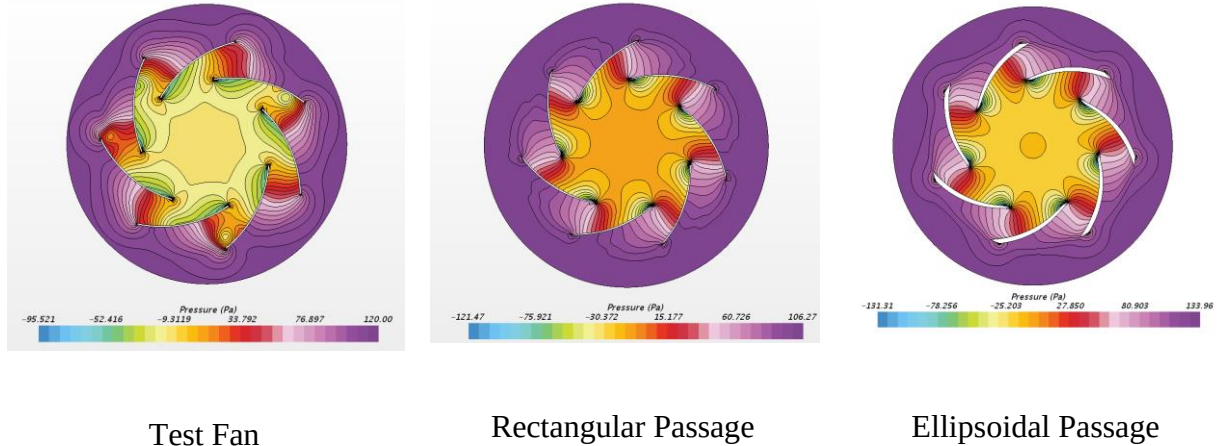


Figure 48. Pressure gradients in the radial plane for the three passages

Expansion Angle The expansion of the blade passage is related to the expansion angle of a cone diffuser. As a cone expands there is a constant angle between the sides of the cone and the cone center line. This Angle can be used as a design parameter for the expansion of an arbitrary

area along a path. The flow area perpendicular to the mean flow path for a cone is a circle, Equation 25. As the ellipsoidal passage was the best performing design from the area shape study, it was chosen to test different angles of expansion. For this study, θ , was varied between 0 and 7° for an overall expansion between 0 and 14°. Where the 14° expansion coincides the maximum angle of efficiency for a converging or diverging cone (Bleier 1998).

The blade passage with 7° of expansion outperforms the rest of the cases, and falls below the lower limit 9° area slope (Eck 1973). It has the highest overall efficiency and has a similar breadth in the operational zone as the other cases, Figure 49. The cases smaller than 7° show an increase in the maximum efficiency as they approach 7° and an increase in the operational zone. Cases larger than 7° show a decrease in the maximum efficiency as they move away from 7° with a decrease in the operational zone. This correlates to data for diffusers shown in the paper by Patterson where there is an optimal angle for a diffuser cross-sectional area with a decrease in performance above or below the optimal angle (Patterson 1938).

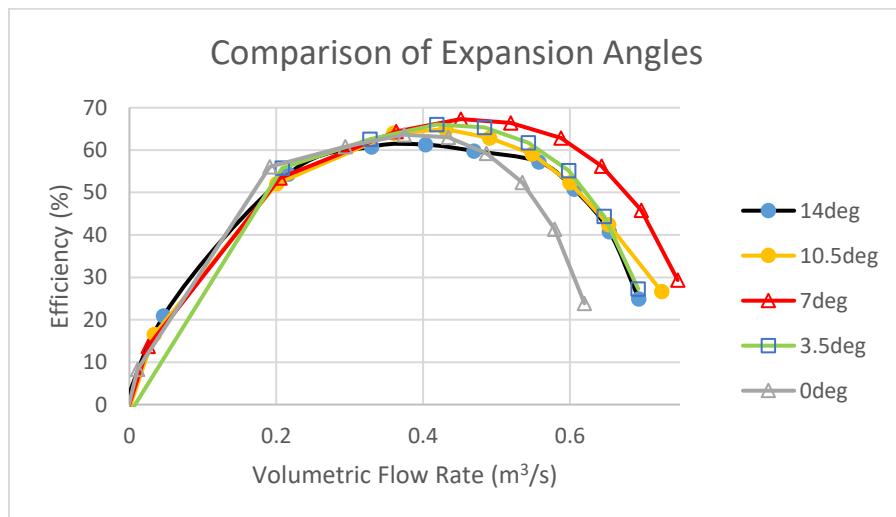


Figure 49. Volumetric flow rate vs. efficiency for different expansion angles

Applicability

Now that a new and improved blade passage has been achieved, how does the design translate to other conditions. Three main concerns are the ability to scale the design, to operate at different rotational speeds, and for the new procedure to be applied to a mean path of different curvature. The test fan has the advantage of being empirically tested. The parameters of the test fan being achieved through the manufactures proprietary process. The main parameters were used in the new process so that the designs could be compared, such as the outside and inside radii of the blade passage and tested RPM. If the new design only improves the efficiency at these conditions, it greatly limits the applicability. Lastly, the 7° ellipsoidal passage utilized a mean path dictated by the constant curvature blade passage of the test fan. If the basis for the mean path is changed from the constant curvature, does the design have an improved efficiency. Therefore, it is important to evaluate if the new design can be applied more broadly.

Fan Size For a constant rotational speed, the fan size will increase both the power and pressure. The ellipsoidal passage fan and the test fan were scaled to two times their original size to see the effects on the efficiency. It is important to check if the area expansion of the blade passage correlates to a different size. If the boost in efficiency is confined to a smaller diameter fan, this greatly limits its applicability. Figure 50 shows that the boost in efficiency holds for the larger diameter fan with the larger ellipsoidal achieving 72% maximum efficiency over the test fans 64%.

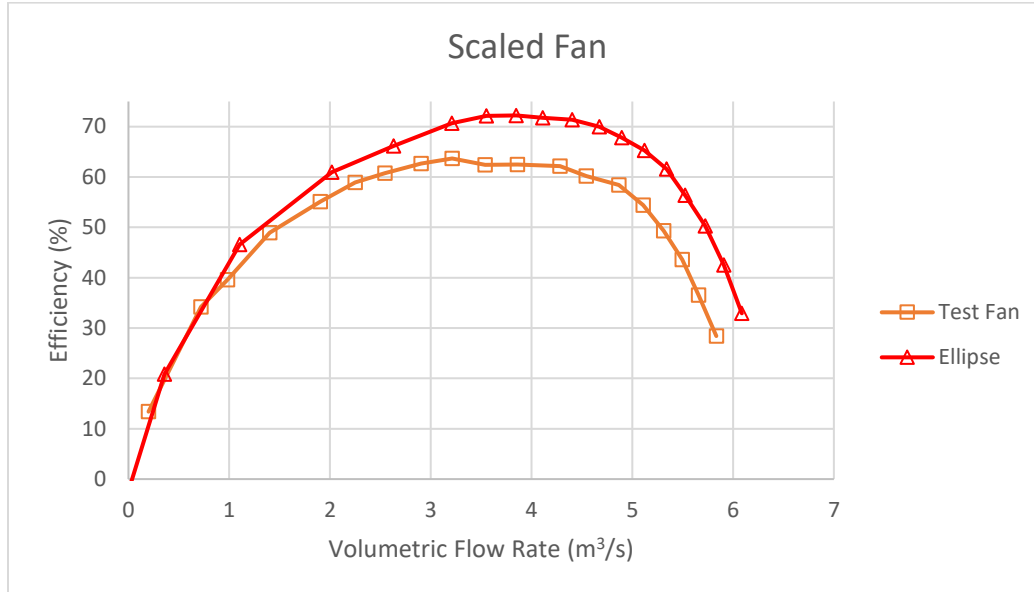


Figure 50. Volumetric Flow Rate vs. Efficiency for a 2x Scale ellipsoidal blade passage fan

It also shows that for this blade shape the larger fan is more efficient. Where the smaller fan tops out at 67% efficiency the larger fan achieves more than 70%. The difference between the test case and the 2x scaled ellipsoidal fan in both instances is similar at roughly 10%. This consistent increase shows that by strictly holding the passage to the diffuser principle better performance can be achieved over the classical sandwich design.

Rotational Speed While every fan has a peak efficiency at a designed rotational speed, in application, fans can be operated under a range of speeds. The upper limit is defined by structural integrity of the material. Spin a fan too fast and they will suffer catastrophic failure and fly apart. Below this upper limit though, the fan can be operated safely. The operational speed is chosen based on the required flow rate and pressure of the system. Since centrifugal fans are often running continuously for long periods of time, ideally the lowest rotational speed is chosen that satisfies these requirements.

This variability in the operational speed necessitates that the efficiency of the fan be evaluated at different rotational speeds. The initial tests used an operational speed of 1000 RPM. This was increased to 1900 RPM to see how the fan performed at a higher rotational speed. 1900 RPM falls within acceptable range of RPM given by the manufacturer. It can be seen from Figure 51, that the new fan outperforms the test fan at the new operational speed as well.

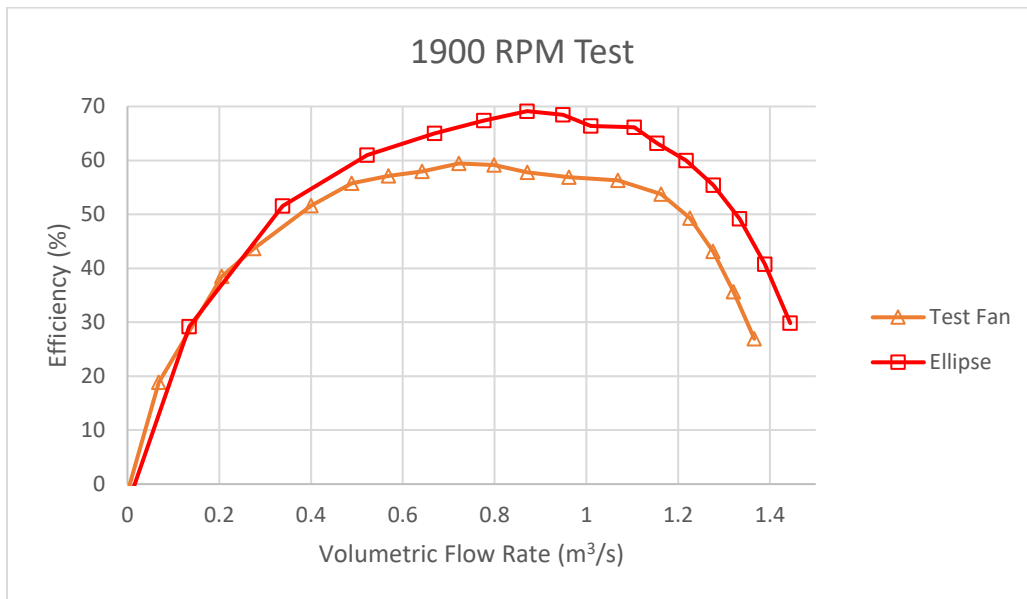


Figure 51. Volumetric Flow Rate vs. Efficiency at 1900 RPM

A New Mean Path: Logarithmic The 7° ellipsoidal blade passage was constructed with a mean path based on constant curvature of the passage bounds in the radial plane. Applying the new procedure to the constant curvature mean path achieved a 10% increase in efficiency. What happens if the procedure is applied to a mean path based on the logarithmic principle from stream theory (in Chapter Mean Fluid Path, Stream Theory)?

Three blade passages were constructed using logarithmic blade forms as a starting point for a new fan design. For ratios of radial to tangential velocities, ϕ , values of 0.3, 0.45, 0.9

corresponding to radial velocities of 6, 9, and 12 m/s and a rotation rate of 1000 RPM, or 20 m/s tangential velocity at the blade passage outer radius. The value of 9 m/s matches the average radial velocity achieved in the simulations of the 7° ellipsoidal constant curvature passage. The passages were constructed with the blade passage inlet area constant as before. The maximum efficiency for all three passages is around 60%. In comparison with the 7° example this is 7% less in maximum efficiency. Furthermore, the effect of changing the radial component only shifts the operational zone from a lower to higher flow rates as the radial component decreases, as seen in Figure 52. In comparison to the test fan, the initial logarithmic fan is close in performance, see Figure 53, where the test fan, and therefore the 7° blade passage fan, have the advantage of optimized parameters such as fan radii, the logarithmic passage starts from theory and remains unoptimized. This initial performance of an untuned design shows promise for producing efficient fans and in a manner that is not predicated on primarily on 2D design derived in the radial plane and proprietary changes.

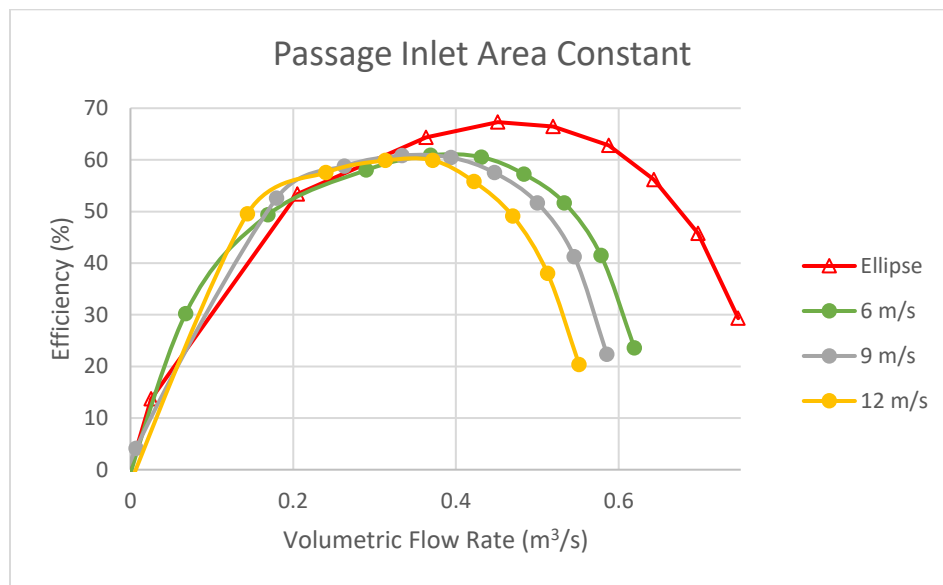


Figure 52. Comparison of ellipsoidal passage to logarithmic passages with the passage inlet area constant

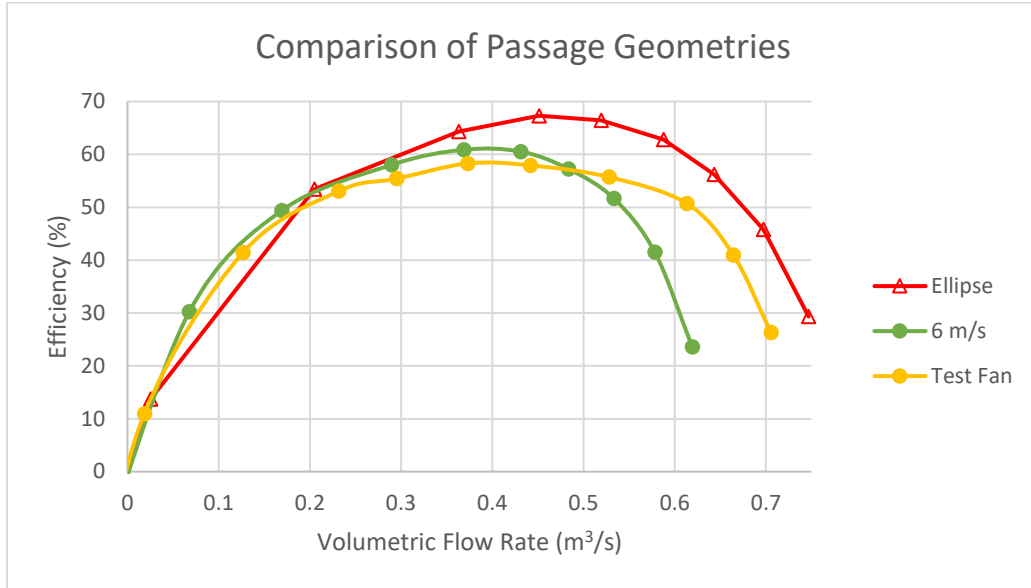


Figure 53. Comparison of ellipsoidal passage to $\varphi = 0.3$ logarithmic passage and test fan

Looking at the pressure gradient with the blade passage there is more disturbance along the backplate within at the blade passage inlet than with the 7° ellipsoidal case, shown in Figure 54. It can be seen in Figure 55 that the velocity profile within the passage shows a clear separation of high and low velocity flow. This is indicative of the fluid not filling the passage effectively. Using the fan inlet area from previous designs provided a good starting point for the design though it fails to achieve higher efficiencies here.

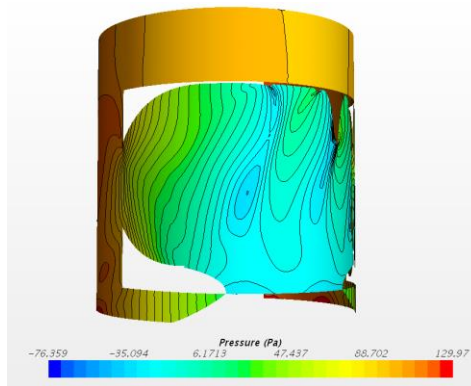


Figure 54. Logarithmic passage pressure profile for $\varphi = 0.3$

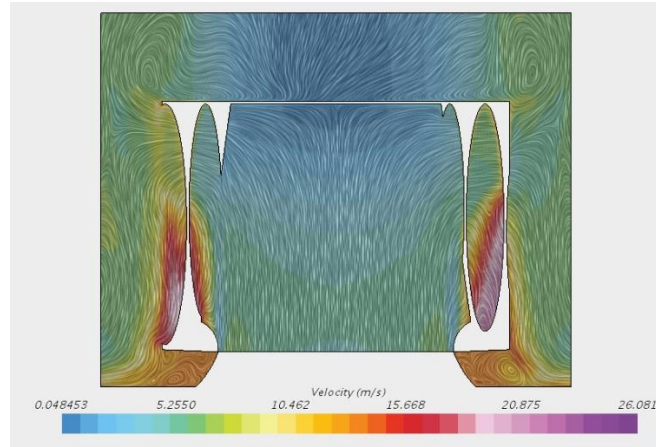


Figure 55. Meridional velocity profile for $\varphi = 0.3$

In an alternative approach, the passage outlet areas were matched to the 7° ellipsoidal passage fan. The area of interest used was from the perpendicular plane that bisected the passage at the fan outer radius. This caused the efficiency to become similar in both maximum efficiency and operational zone, as seen in Figure 56. However, the simulations showed the same deficiencies in the pressure gradient as well as the velocity profile.

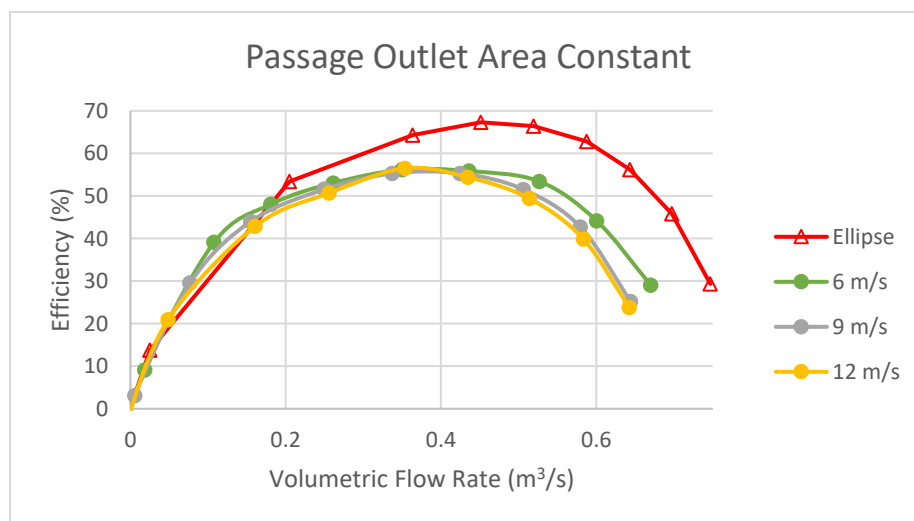


Figure 56. Comparison of ellipsoidal passage to logarithmic passages with the outlet area constant

To investigate the logarithmic passage further, the $\varphi = 0.3$ case was tested at two higher rotational speeds of 1450 and 1900 RPM near the peak efficiency of 100 Pa. Since the passages were constructed for the rotation rate of 1000 RPM, the goal was to see if the with increasing angular velocity, the passages would fill more uniformly. Figure 57 shows as the angular velocity increases, the non-uniformity decreasing within the height of the blade passages. An increase in efficiency can be seen in the 1450 RPM case, $\eta = 57\%$,

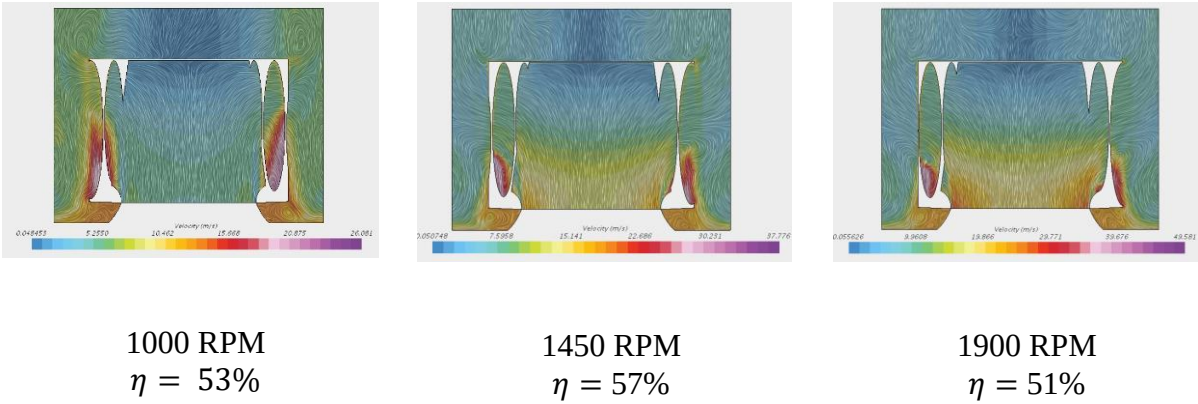


Figure 57. Logarithmic Spiral case $\varphi = 0.3$, at three test RPM, 100 Pa

Since the width of the passages is dictated by the number of blades the difference for the logarithmic passages has been the height at the outer radius that is created when the passage is truncated. A final iteration investigated utilized the height of the 7° ellipsoidal passage. This created a blade opening of similar shape and size to the test and 7° ellipsoidal passage fan. The logarithmic spiral has a longer mean path than the constant curvature of the 7° ellipsoidal and the spiral has a shaper curvature along the passage. This means that the width in the passage is smaller. By restricting the passage to the height of the 7° ellipsoidal passage the blade passage volume for the logarithmic case was greatly reduced. However, the efficiency of the fan

increased to 64% at 100 Pa and 1000 RPM. The best efficiency achieved so far. This came at the cost of overall volumetric flow rate from 0.43 kg/s for the $\varphi = 0.3$ case with matching inlet area to 0.31 kg/s. Again, as an unoptimized blade passage, the fan following the logarithmic achieved a peak efficiency of 64% and compares to the best fan performance of 67% for the blade passage with a constant radius of curvature and is significantly improved over the original test fan, with an efficiency of 58%.

CONCLUSIONS

Using the diffuser concept to dictate the area expansion of the blade passage area provides a clear improvement on the efficiency of the backward-curved centrifugal fan. Traditional designs used the diffuser concept as a design check, where the blade and meridional profiles are sandwiched together. The resultant blade passage changes geometry along the mean path line, nor does it adhere to strict area expansion rules. The design assumes that the performance of the 2D blade profile exists along the full height of the blade. This has been observed to be false close to the inlet shroud.

Fan design does not need to be radically rethought to see significant efficiency gains. By removing the manufacturing restraints and the reliance on a primarily 2D analysis within the radial plane and the resulting blade sandwich allowed for a return to foundational fan design and an emphasis on the blade passage over strictly looking at blade design. The diffuser concept shows the importance of controlling the rate of expansion of the blade passage, with a correct evaluation of the area being necessary. An ellipsoidal cross-section was shown to be the most efficient profile.

The new fan design increases the radial velocity produced within the blade passage. Since the tangential velocity is fixed by the rotation speed, this velocity increase is a clear indication that the passage is acting more efficiently. The goal of shaping the blade passage as a diffuser is to increase the pressure increase from the expansion of the blade passage. The velocity profile uniformity within the passage is also improved. The ellipsoidal passage reduces friction of the passage by eliminating the sharp corners inherent to the traditional design.

The pressure gradient within the blade passage is also improved in the new design. The diffuser concept allows for more of the energy imparted by the centrifugal rotation to be retained by the flow as more of the kinetic energy is turned into pressure. The increase in pressure correlates to an increased performance at the same operating conditions for the test fan.

Using stream theory as basis for a logarithmic passage a fan was created with a maximum efficiency of 64%, above the test fans 58%. This was achieved with very few design iterations. Further refinement could yield an even more efficient design. It is possible to build a new fan from scratch using potential flow theory and a logarithmic mean path with improved maximum efficiency over the test fan. An increase of 5% in total efficiency was achieved.

It is clear that using the diffuser principal to design the blade passage of a backward centrifugal can increase the performance over the traditionally designed fan. While the main passage tested began with a constant curvature blade form this method could be applied to radial and forward centrifugal fans. This would broaden the applicability of the new design to industrial buildings and residential buildings where radial and forward centrifugal fans are commonly found.

The ellipsoidal blade passage cannot be stamped of metal parts, as is the case with traditional fan design. This increases the initial cost of the new fan as new equipment and methods will have to be utilized. As well as material testing and certification for any plastics that may be implemented. Larger designs over a meter across may have to be made out of metal. One advantage of additive manufacturing methods is the reduction in material waste, which would help offset some of these costs in the long term.

A 10% increase in total efficiency will save \$2,600 annually for a medium sized building. With nearly 6 million commercial buildings in the United States that is nearly \$16 billion of savings. This represents a savings of 2 million MWh a year and a reduction of 1.5 million tons of equivalent carbon dioxide emissions. If this fan design approach is applied outside of commercial buildings, the savings will be even more significant.

FUTURE WORK

While the work done in this research was done on a backward centrifugal fan, the applicability to other centrifugal types, such as the radial type, should be evaluated. All fan types utilize the diffuser principle as a check on the traditional design. Therefore, a more strict adherence to the diffuser principal could be applied in any centrifugal case.

This research used the cone diffuser as the basis for the area expansion. This had the advantage of a simple linear rate of change. A parabolic rate of change would lead to a cubic area change vs. the mean path. For a given blade, a different rate of change may be more applicable. The expansion angle study showed that the rate of expansion for the given blade from was optimized at $\theta = 3.5$ deg. A separate study could investigate different rates of expansion to see the effect over a constant mean path length.

The mean path for the studies in this research was constrained to the radial plane. A 3D mean path would take into account the inlet area and 90° flow transition. This would be difficult given that the flow at the bend analogy is not sufficient for this transition. This section also has points of flow separation into the individual flow passages.

The logarithmic blade design was able to achieve an efficiency of 64%, slightly below the 67% achieved by the 7° case, but with a reduction to the volumetric flow rate. The logarithmic passages were fundamentally narrow due to the shaper curvature of the path. By reducing the number of blades the passage would naturally widen. An investigation into the effects of the number of blades on the performance would show the effect on efficiency.

The geometries used in the CFD simulations are difficult to manufacture. The classic blade sandwich manufacturing technique is no longer an option for a design of this type. The

new design would require injection molding, 5-axis milling, or additive manufacturing. Viability and integrity of these manufacturing methods needs to be explored and tested.

OTHER WORK

In preparation

Designing a centrifugal fan blade passage using the principles of a diffuser

Revision submitted for publication for ASHRAE:

Studying the Effects of Suction Chamber Intrusions and Housing Proximity on Backward Centrifugal Fan Performance Utilizing Computational Fluid Dynamics

Presenter, ASHRAE summer conference 2018, Seminar 49:

Using Uncertainty Analysis to Identify the Impact of Different Parameters.

Member, ASHRAE

2017-Present

ASHRAE Standing Standard Project Committee 41: Standard Methods for Measurement voting member 2018-Present

Standards contributed to:

ASHARE Standard 41.1 – Standard Method for Temperature Measurement

ASHRAE Standard 41.2 – Standard Methods of Air Velocity and Airflow Measurement (ANSI Approved)

ASHRAE Standard 41.3 – Standard Method for Pressure Measurement

ASHRAE Standard 41.4 – Standard method for Measurement of Proportion of Lubricant in Liquid Refrigerant (ANSI approved)

ASHRAE Standard 41.9 – Calorimeter Test Methods for Mass Flow Measurements of Volatile Refrigerants (ANSI Approved)

REFERENCES CITED

- Addison, Herbert. 1948. *Centrifugal and other rotodynamic pumps*. London: London, Chapman & Hall.
- American Society of Heating, Refrigerating, and Inc Air-Conditioning Engineers. 2019. 2019 ASHRAE® Handbook Heating, Ventilating, and Air-Conditioning Applications (I-P Edition). American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc. (ASHRAE).
- Anderson, John D. 1994. *Computational fluid dynamics : the basics with applications*. New York: New York : McGraw-Hill.
- ASHRAE. 2016. *ASHRAE 51-16/ANSI/AMCA 210-16 Laboratory Methods of Testing Fans For Certified Aerodynamic Performance Rating*. Atlanta, Georgia: ASHRAE.
- ASHRAE. 2019. *ASHRAE Standard 62.1-2019 Ventilation for Acceptable Indoor Air Quality (ANSI Approved)*. Atlanta, Georgia: ASHRAE.
- Bernoulli, Daniel. 1738. *Danielis Bernoulli Hydrodynamica, Sive De Viribus Et Motibus Fluidorum Commentarii: Opus Academicum*. Argentorati: Dulsecker.
- Bleier, Frank P. 1998. *Fan handbook : selection, application, and design*. New York: New York : McGraw-Hill.
- Cory, W. T. W. 2005. Fans & ventilation a practical guide. In *Fans and ventilation*. Amsterdam; Boston: Amsterdam; Boston : Elsevier in association with Roles & Assoc.
- Dilin, P., T. Sakai, M. Wilson, and A. Whitfield. 1998. "A computational and experimental evaluation of the performance of a centrifugal fan volute." *Proceedings of the Institution of Mechanical Engineers Part a-Journal of Power and Energy* 212 (A4): 235-246. <https://doi.org/Doi> 10.1243/0957650981536763. <Go to ISI>://WOS:000076411100004.
- Eck, Bruno. 1973. *Fans design and operation of centrifugal, axial-flow, and cross-flow fans*. 1st English ed. Translated and edited by Ram S. Azad and David R. Scott.. ed. Oxford, New York: Oxford, New York, Pergamon Press.
- EIA. 2020. "U.S. Energy Information Administration - EIA - Independent Statistics and Analysis." <https://www.eia.gov/>.
- Engeda, Abraham. 1998. "Early Historical Development of the Centrifugal Impeller." ASME 1998 International Gas Turbine and Aeroengine Congress and Exhibition.
- Fefferman, Charles L. "Existence and Smoothness of the Navier-Stokes Equation."
- Hanjalić, K., and B. E. Launder. 2020. *Eddy-Viscosity Transport Modelling: A Historical Review*. Singapore: Singapore: Springer Singapore.

REFERENCES CITED CONTINUED

- Hariharan, C., and M. Govardhan. 2016. "Improving performance of an industrial centrifugal blower with parallel wall volutes." *Applied Thermal Engineering* 109: 53-64. <https://doi.org/10.1016/j.applthermaleng.2016.08.045>. <Go to ISI>://WOS:000384861300006.
- "HVAC Equipment Market 2019 Size, Statistics, Growth, Revenue, Analysis & Trends Industry Forecast Report -2025|Market Research Engine." 2020. MarketWatch. <https://www.marketwatch.com/press-release/hvac-equipment-market-2019-size-statistics-growth-revenue-analysis-trends-industry-forecast-report--2025market-research-engine-2019-11-26>.
- IEA. 2018. "Air Conditioning Use Emerges as One of the Key Drivers of Global Electricity-Demand Growth - News.". <https://www.iea.org/news/air-conditioning-use-emerges-as-one-of-the-key-drivers-of-global-electricity-demand-growth>.
- Improving Fan System Performance: A Sourcebook for Industry. 2003. edited by U.S. Department of Energy. United States.
- Kruyt, N. P., and R. W. Westra. 2014. "On the inverse problem of blade design for centrifugal pumps and fans." *IP* 30 (6): 65003. <https://doi.org/10.1088/0266-5611/30/6/065003>.
- Lehtola, Carol J. 2000. Occupational Noise Exposure: OSHA Standard 1910.95. edited by Occupational Safety and Health Administration: University of Florida Cooperative Extension Service, Institute of Food and Agricultural Sciences, EDIS.
- Matschoss, Conrad. 1907. *50 years of engineering work in Upper Silesia*. Berlin: Knight.
- Menter, F. R. 1994. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA Journal* 32, no. 8: 1598-1605.
- Monaghan, J. J. 2005. "Smoothed particle hydrodynamics." *REP PROG PHYS* 68 (8): 1703-1759. <https://doi.org/10.1088/0034-4885/68/8/R01>.
- Osborne, William C. 1977. *Fans : (in SI/metric units)*. 2d ed.. ed. Oxford, New York: Oxford, New York : Pergamon Press.
- Pantell, K. 1949. "Impeller-Profile of Turbomachinery." *Konstruktion*: p.77.
- Patterson, G. N. 1938. "Modern Diffuser Design: The Efficient Transformation of Kinetic Energy to Pressure." *Aircraft Engineering and Aerospace Technology* 10 (9): 267-273. <https://doi.org/10.1108/eb030365>.
- Pickering, C. A. C. 1989. "Building sickness syndrome." *Respiratory Medicine* 83 (2): 91-92. [https://doi.org/10.1016/S0954-6111\(89\)80219-7](https://doi.org/10.1016/S0954-6111(89)80219-7).

REFERENCES CITED CONTINUED

- Schmitt, François G. 2007. "About Boussinesq's turbulent viscosity hypothesis: historical remarks and a direct evaluation of its validity." *CR MECANIQUE* 335 (9): 617-627. <https://doi.org/10.1016/j.crme.2007.08.004>.
- Senturk, Utku, and Hamza Tas. 2015. "Computational and experimental study on the performance improvement of an in-line centrifugal fan by outlet guide vanes." *SCI TECHNOL BUILT EN* 21 (7): 1002-1009. <https://doi.org/10.1080/23744731.2015.1035602>.
- Shepherd, Dennis G. 1956. *Principles of turbomachinery*. New York: New York, Macmillan.
- Siemens. "STAR-CCM+." Siemens Digital Industries Software. <https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html>.
- Son, P. N., J. Kim, and E. Y. Ahn. 2011. "Effects of bell mouth geometries on the flow rate of centrifugal blowers." *Journal of Mechanical Science and Technology* 25 (9): 2267-2276. <https://doi.org/10.1007/s12206-011-0609-3>. <Go to ISI>://WOS:000294748300013.
- Sudhakar, T., and Arup Kumar Das. 2020. "Evolution of Multiphase Lattice Boltzmann Method: A Review." *Journal of the Institution of Engineers (India) Series C* 101 (4): 711-719. <https://doi.org/10.1007/s40032-020-00600-8>.
- Yanagisawa, Yukio. 2017. *Chemical sensitivity and sick-building syndrome*. edited by Yukio Yanagisawa, Hiroshi Yoshino, Satoshi Ishikawa and Mikio Miyata: Boca Raton, FL : CRC Press.
- Yuan, Ye, and Shouqi Yuan. 2017. "Analyzing the effects of splitter blade on the performance characteristics for a high-speed centrifugal pump." *ADV MECH ENG* 9 (12): 168781401774525. <https://doi.org/10.1177/1687814017745251>.