INVESTIGATION ON EFFECTS OF AIRFOIL VARIATION IN A SLOTTED PROPELLER

NIRRESH PRABU A/L RAVINDRAN

SCHOOL OF AEROSPACE ENGINEERING

UNIVERSITI SAINS MALAYSIA

2019

INVESTIGATION ON EFFECTS OF AIRFOIL VARIATION IN A SLOTTED PROPELLER

by

NIRRESH PRABU A/L RAVINDRAN

Thesis submitted in partial fulfilment of the requirement for the degree of

Bachelor of Aerospace Engineering

June 20

ENDORSEMENT

I, Nirresh Prabu A/L Ravindran hereby declare that all corrections and comments made by the supervisor and examiner have been taken consideration and rectified accordingly.

____
(Signature of Student)

Date:

____
(Signature of Supervisor)

Name:

Date:

____
(Signature of Supervisor)
Name:

Date:

DECLARATION

This thesis is the result of my own investigation, except where otherwise stated and has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any other degree.

____

(Signature of Student)

Date:

ACKNOWLEDGEMENT

First of all I would like to show my sincere gratitude and appreciation to everyone for aiding, assisting and helping me to wrap up my final year project. Their never ending kindness to make an effort whenever I needed in order to for me to make better progress of my project towards completion.

Foremost, I would like take this opportunity to express my profound gratitude and deep regards to my supervisor Ir. Dr. Parvathy Rajendran, for the continuous support on my final year project from the initiation process till the thesis completion, for her patience, guidance, monitoring, enthusiasm, and immense knowledge. Her guidance and blessing have helped throughout this project.

Morever I would also like to take this opportunity to express my sincere thanks to Mr. Arvind Seeni and Miss Merryisha, postgraduate students as my mentor for they have been helpful, supportive and were able to give me valuable information, in spite of their busy schedule. Besides, I would also like to express my gratitude to my friends Linghan, Bohendiran, and Saravanan for their constant support on finishing both the project and thesis.

Last but not the least, my research would have been impossible without the guide and support of my parents for the unceasing encouragement, support and attention whose blessings are with me in whatever I pursue.

SLOTTED PROPELLER DESIGN USING CFD

ABSTRACT

The usage of slots has gained renewed interest in aerospace particularly on propeller design. Most of the works have focused on improving the aerodynamic performance and efficiency. Modern research on propeller design aims to design propellers with high thrust performance under low torque conditions without any weight penalty. This paper discussed computational fluid dynamics method to predict propeller performance for a small scale propeller. In addition, the performance of the slotted blade designs is presented, in terms of thrust coefficient, power coefficient and efficiency. The performance of slotted propeller blade is influence by the different types of airfoils being used on the blade. Thus, the shape of the slot is fixed which is a square shaped slot and the position of the slot is also fixed at 62.5% of the Chord length. Although research on slotted design has been done before, none has been done for different Airfoils on the propeller blade. Thus, this study aims to provide an extensive research on slotted propeller design with different Airfoils of different properties such as high Reynolds number, low Reynolds number, symmetrical, asymmetrical high lift and low drag. The basis for the propeller will be the APC Slow Flyer. The flow simulations are performed through three-dimensional computational fluid dynamic software (ANSYS Fluent) to determine the thrust coefficient, power coefficient and overall efficiency measured in advancing flow conditions.

MEREKA BENTUK BEBALING BERSELIT MENGGUNAKAN PERISIAN DINAMIK BENDALIR

ABSTRAK

Penggunaan lubang selit telah mendapat minat baru dalam ruang angkasa terutamanya pada rekabentuk bebaling pesawat. Kebanyakan penyelidikan yang dijalankan telah memberi tumpuan kepada peningkatan prestasi dan kecekapan aerodinamik. Penyelidikan moden mengenai reka bentuk bebaling bertujuan untuk mereka bebaling dengan menjanakan prestasi teras yang tinggi di bawah tork yang rendah. Maka penyelidikan ini membincangkan kaedah perisian dinamik bendalir untuk meramal prestasi bebaling bagi bebaling bersaiz kecil. Di samping itu, prestasi reka bentuk lubang selit akan , dari segi pekali teras, pekali kuasa dan kecekapan. bebaling apabila dipengaruhi oleh lapan jenis airfoil yang berlainan yang digunakan dalam bebaling. Oleh itu, bentuk bebaling yang tlah ditetapkan adalah empat segi dan kedudukan selit juga ditetapkan pada 75% daripada panjang kord. Walaupun penyelidikan mengenai reka bentuk selit telah dilakukan sebelum ini, tidak ada yang dilakukan untuk aerofoil yang berbeza pada satu jenis bebaling. Oleh itu, kajian ini bertujuan untuk menyediakan penyelidikan yang luas mengenai reka bentuk bebaling berselit dengan aerofoils yang berlainan sifat berbeza seperti nombor Reynold yang tinggi, nombor Reynold yang rendah, simetri, asimetri, daya angkat tinggi dan daya seret rendah. Dasar untuk reka bentuk bebaling ialah APC Slow Flyer. Simulasi aliran dilakukan melalui perisian dinamik bendalir tiga dimensi (ANSYS Fluent) untuk menentukan pekali teras, pekali kuasa dan kecekapan keseluruhan yang diukur dalam menambahbaikan keadaan aliran udara keliling aerofoil.

Table of Contents

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

K_T	Propeller Coefficient of Thrust
K_Q	Propeller Coefficient of Torque
K_P	Propeller Coefficient of Power
η	Propeller Efficiency
T	Propeller Thrust
P	Propeller Power
Q	Propeller Torque
n	Revolution per second of propeller
D	Propeller diameter
ρ	Density of fluid (1.225 kgm^{-3})

CHAPTER 1: INTRODUCTION

1.1 Overview

Propeller aircraft have come a long way since the early days of fixed pitch wooden propellers that were used by the Wright brothers on the ‘Wright Flyer’. Da Vinci's "helical spiral" helicopter is considered an ancestor of air propellers and helicopter rotors. However, the first idea of the propeller was French mathematician JP Paucton. Paucton envisioned an airplane with two propellers for propulsion and flight maintenance.

A propeller may be considered as a rotating wing that assembles airfoils collectively, comparable to the cross-section of the wing of an aircraft. A basic propeller configuration includes a minimum of two blades, attached together to the central hub. Thrust is generated to push the aircraft forward through the air, by converting the rotation power from the engine shaft into translational motion. The chamber shape of the airfoil causes the airflow in front of the blade to travel at higher speed which is based on Bernoulli’s principle. As the air flows around the airfoil that creates a pressure difference between the upper surface and the lower surface of the airfoil due to difference in airflow speed. The pressure difference between the front and back section of the propeller creates thrust force in forward directions, allowing it to overcome the drag experienced by the aircraft.

Since the usage of aircraft propellers was pioneered, the usage of propeller never diminished due to its main advantage of low fuel consumption at low Reynolds number flight. Propeller manufacturers are constantly coming up with new innovative ways to keep the propellers in the aerospace market despite the evolving advancement of aerospace technology. The propellers characteristics such as the diameter, the pitch, number of blades, shape of blade and airfoil selection used have been altered in a quest to produce a highly efficient propeller.

The study of the implementation of slots in airfoils has been first done by Braslow according to (Seeni et al., 2018). The slots are placed on airfoils in an intend to alter the flow around the airfoil for better aerodynamic performance. The slotted propeller design concept works on a principle where the groove created along the length of the propeller blade will slow down the airflow above the airfoil by creating a flow separation. The reduced velocity above the airfoil will cause the higher air pressure below the airfoil to create an upward force producing lift this creating more thrust at lower torque.

Slots can be grouped into active slots and passive slots. Active slots are slots that can be modified in terms of size, shape, number and position on the airfoil during flight. Example of active slots are conventional wing flaps and slats used today as illustrated in figure 1.1.1. These high lift devices are altered during flight during different flying phases such as take-off, climbing, cruise flight and landing in order to give an advantage for the airplane in terms of its performance. The Passive slots are fixed and cannot be changed in terms of shape, size, number and position. This study mainly focuses on passive slots and its effects on airfoils with different aerodynamic properties using CFD simulation.

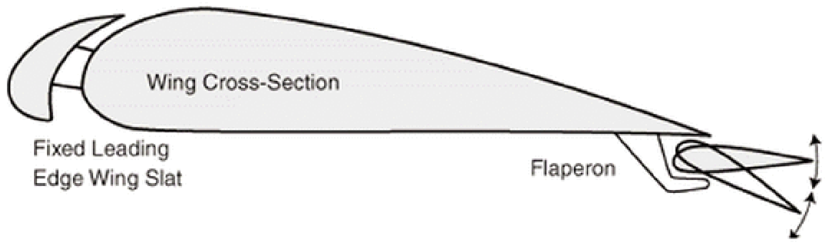


Figure 1.1.1: Flaps on conventional wing

1.2 Problem Statement

Previous patented researches mainly focus on slot characteristics and how changes in in terms of shape, size, number, and position of the slots on the airfoil will affect the performance of the blade. Reducing fuel consumption has become a driving factor for conducting research on improvement of propeller design with high thrust at low torque. Instead of focusing on the slot characteristics, this study solely focuses on a slotted design of the blade using different airfoils that varies with aerodynamic and physical properties such as symmetrical, asymmetrical, low Reynolds number, high Reynolds number, high lift and low lift. Through this study, the overall best performing slotted airfoil type can be identified.

1.3 Research Objective

In this research, there three main objectives where each one is achieved step by step in order to keep the on track:

I. To study aerodynamic parameters that influence the slotted propeller design performance
II. To investigate the effect of airfoil variation on a slotted propeller design

III. To recommend suitable airfoil selection for a slotted propeller design.

1.4 Research Scope

This study mainly focuses on how the variation of Airfoil affects the slotted propeller unlike previous researches where they focus on the modification of the propeller slot characteristics in terms of its size, position on the airfoil, quantity, and shape. The propeller will be numerically analyzed using Computational Fluid Dynamics by running a simulation on the propellers. The input value would be the Free stream velocity and the revolution per minute of the propeller and in return the output value is the thrust, power and efficiency.

CHAPTER 2: LITERATURE REVIEW

2.1 introduction

The main goal of propeller design is to create or improve an existing model in order to enhance the performance of the propeller. The mechanism of the propeller can be divided into two motions, rotation and translation []. Figure 2.1 shows the forces acting on the propeller when it rotates. The blade can be seen to act much similar to an airplane wing in terms of the blade tilted at a certain angle called the pitch and it is composed of an airfoil. The pitch similar to the working principles of angle of attack on an airplane wing will produce a forward force due to the deflection of air. The airfoil further enhances this force using Bernoulli's principle where the low air pressure above the airfoil cause a pressure difference between the two blade surfaces producing a forward reaction force. Thus, when the propeller rotates, the dynamic pressure behind the propeller is higher than the atmospheric pressure causing thrust.

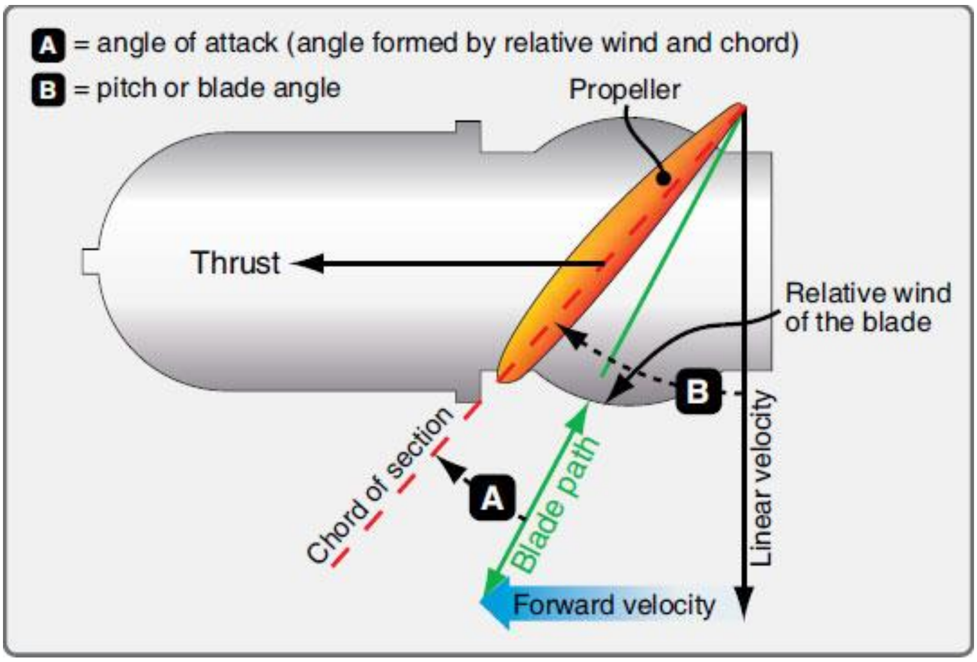


Figure 2.1: Propeller Forces

The design of the propeller can be done by altering any existing feature that the performance of the propeller or adding a new feature on the propeller with an intention of improving its performance. For instance, increasing the number of blades has a positive impact on the performance of the blade since the distribution of thrust and power are even at the wake of the propeller. Therefore, the efficiency is slightly improved but not very significant. However, increasing the number of blades will demand more power from the engine in order to produce thrust. For a given power and thrust, the propeller blades will be narrow as the number of blades increase.

Besides, having a large diameter propeller can have a significant influence on the performance especially the efficiency of the propeller. This is due to is ability to bring in a greater fluid volume and has a better distribution of thrust and power than smaller diameter propeller. However, more power will be needed to rotate the propeller can may cause high consumption of fuel or if it’s an electric aircraft, the motor will potentially burn out.

Furthermore, having the right distribution of lift and drag coefficient along the propeller blade often points out to the composition of airfoils in the blade. When the blade rotates, the propeller blade tip rotates faster than the section of the blade closer to the hub. The selection of airfoil along the blade is very important due to this very reason. The efficiency of the propeller will reduce if the distribution of lift coefficient and drag coefficient is not convenient which may cause the propeller to stall. For maximum performance, the airfoil’s L/D should be maximum during operation.

2.2External Factors Effecting Propeller Performance

2.2.1 Effect of Reynolds Number

The Reynolds number is an important parameter that is used in many applications of fluid flow. The Reynolds number determines the pattern of fluid flow in different situations. Since airfoil works by the means of fluid flow around it, its aerodynamic performances will be affected at different of Reynolds number. This is true based on this (Bartl et al., 2019) where the performance of the wing section affected by the eight Reynolds number in a wind turbine is studied. The wing section is composed by NREL S826 airfoil and the 8 Reynolds number ranged from 0.5×10^5 to 6.0×10^5. Numerical simulation is also carried out in order to compare the results with the experimental method. The experimental method is carried out using a wing section made of polyurethane board which is CNC-milled. The main aim is to measure the lift and drag coefficient of the wing section. The results of the study showed that the lift and drag of the wing section is being influenced by Reynolds number lower than 0.7×10^5.

Another paper (McTavish et al., 2013) has done the similar study but on a wind turbine rotor. A series of experiments were carried out to investigate the wake expansion and thrust coefficient of the wind turbine rotor affected by range of Reynolds number. The experiment is carried out on two scaled down wind turbine rotors, one two bladed and another three bladed at Reynolds number ranging from 3620 to 31400. The results showed that the wake expansion increase for the three bladed propeller while for the two bladed rotor the wake expansion is 30% to 50% narrower. The thrust coefficient for the three bladed rotor increases with increasing Reynold number. For the two bladed rotor, the thrust coefficient for is 25% lower compared to its medium scaled version tested at 30000 Reynolds number and it was 60% lower when it was tested 12800 Reynolds number. The thrust coefficient is reduce for any geometrical scaled propeller when the Reynolds number reduced.

Another similar to both previous studies had been conducted in order to investigate the effects of Reynolds number and the tip losses on the optimal aspect ratio of straight bladed Vertical Axis Wind Turbine (Zanforlin and Deluca, 2018). Airfoil profile of the turbine blades are made of NACA 0015 airfoils. A computational fluid dynamic simulation is done using ANSYS Fluent and required 6 months to run. The Reynolds number tested were ranged between 2.20×10^5 to 1.63×10^7. The result shows that Reynold number strongly effects smaller sized wind turbines AR< 0.8. As the size of wind turbine decreases the effects of tip loss is somehow cut off by the effects of Reynolds number, thus does not effect the variation of power coefficient of power.

2.3 Different Design of Propellers

2.3.1 Number of Blades

Blades is a part of the propeller that is responsible in generating thrust. This is due to the twist and the airfoil composition of the blade. Researches such as (Ahmadi Asl et al., 2017), (Singh and Nestmann, 2011) and (J. A. Lieser, 1997) have demonstrated that number of blades have an impact on the performance of the propeller.

Based on this (Ahmadi Asl et al., 2017), an Experimental investigation of blade number and design effects has been done for a ducted wind turbine. In this experiment, 9 impellers composed of 3 different blade types and 3 different number of blades from 2 to 4 blades have been tested in wind tunnel with angle attack for every 15° of angle of attack from 0 ° to 90°. The velocity magnitude was set at $3.18ms^{-1}$. The results show that as the number of blades increases, the rotational speed decreases.

A similar study has been done by (Singh and Nestmann, 2011) where the author investigated the influence of blade height and blade number on the performance of low head axial flow hydro turbines instead of wind turbine. 3 turbines A, B and C are designed where A and B are made of 5 blades, but C has 6 blades. The results were contrary to the previous study since the propeller efficiency reduced as the number of blades increases.

(J. A. Lieser, 1997) has done a study to exploit mainly the acoustic performance of the propeller instead of aerodynamic performance affected by the number of blades. Three different propellers with 2-blade, 4 blades and 6 blades analyzed based on aero acoustic calculations using on blade element method, panel method and Euler method. The result shows that the propeller with higher number of blade has better acoustic performance than less number of blades.

2.3.2 Shape of Slots

Researchers (Ni et al., 2019) and (Belamadi et al., 2016) have proved that the implementation of slots on turbine rotors have significantly effected their performance. However, (Rong et al., 2015) worked on slotted centrifugal fans which also showed the same results.

(Ni et al., 2019) tested 9 different slotted blade using numerical method and experimental method in order to analyze the effects of slots on the performance of the blade. for the numerical method, the NACA 63_4- 021 airfoil is used to construct the CAD model of the blade and slotted shaped using 2 circles has been implemented on the blade. 9 Different design slotted blade are constructed by altering the position of the circles and are simulated in ANSYS Fluent. In the experimental method, the slotted blades are 3D Printed and a tested in a towing tank filled with water. A transducer is used to measure the lift and drag forces. The results showed that the slotted rotor design has increased the lift coefficient and the lift to drag ratio of the blade.

However, (Belamadi et al., 2016) findings are contrary to the previous study. In this article the slotted airfoil design implemented in wind turbine blades using numerical analysis. The blade is composed of NREL S809 airfoil and the slot is straight and diagonal creating the passage between the upper and lower surface of the blade. The blade is simulated in ANSYS Fluent with different angle of attacks, different slot positions and different slot size. The result shows that the lift coefficient is reduced when the slot is near to the leading edge and increases when the slot is close to the trailing edge. Moreover, the efficiency of the blade is inversely proportional to size of the slot.

(Rong et al., 2015) article presented positive results on the slotted centrifugal fan. Numerical analysis has done onto the slotted centrifugal fan. The shape of the slot is wavy and connects the pressure side of the and the suction side of the airfoil. The blade is simulated in ANSYS Fluent. The author has concluded that the implementation of the slotted design has significantly improved the aerodynamic performance of the fan. It is further explained that the slot plays and important role in manipulating the boundary layer.

2.3.3 Propeller Blade Profile

Studies done by (Maizi et al., 2018) and (Liu et al., 2017) proved that the change profile of blade have improved the performance of the horizontal wind turbine Similarly, the study done by (Jinsoo Cho, 1998) had the same purpose but on a helicopter propeller.

The article written by (Maizi et al., 2018) showed the influence of different blade shapes of a horizontal wind turbine for noise reduction. The turbine blade sued in this study is the NREL Phase VI blade. The blade profile is altered in terms of the blade tip. Three different blade tip shape where used for the numerical analysis which is the shark tip, reference tip and the original tip. The numerical simulation is done using ANSYS CFX. The author has concluded that the shark tip blade had the best acoustic performance by reducing the overall noise by 7%.

In Contrast to the previous study, this article (Jinsoo Cho, 1998) focuses on optimizing the whole profile of the blade rather than only the tip. The blade profile is optimized using 2 methods, the vortex lattice method and the 3D panel method. The blade used for the optimization process is Purdue model which is composed of NACA0010 airfoil along the blade. The obtained results are then compared with already existing theoretical results. This paper has concluded that the optimized propeller shape showed improvement in efficiency and aerodynamic performance.

Investigation done by (Liu et al., 2017) was similar in subject but difference in approach compared to both previous studies. This article showed the effects of twist on the blade and nacelle shape on the performance of the horizontal axis tidal turbine. The turbine rotor consists of 3 blades that are composed of NACA 9618 airfoil along the blade. The Blade Element Theory Method is applied to optimize the performance of the propeller. Furthermore, CFD simulation has been carried out using ANSYS ICEM in order to verify the Blade Element Theory Method results. It has been concluded that the twisted blade produces 1.9 % higher power and 7.8% higher thrust than the untwisted blade.

2.3.4 Airfoil Profile

Generally, researchers test the effectiveness of airfoils used in propellers by interchanging different existing airfoils on to fixed blade profiles and analyzing the propeller’s performance as demonstrated by (Panigrahi and Mishra, 2014) where a study has been done for the selection of blade profile in order to improve energy efficiency in axial flow mine ventilation fans using CFD Simulation. 6 different airfoil profiles are used to compose the fan blade. ANSYS Fluent is used for the simulation of the fan blade at different angles of attack from 0° to 21° at an interval of 3°. The result shows that the presence of airfoil in the fan blade had better performance than original blade. out of the 6 airfoil NACA 747A315 had the highest lift to drag ratio which is 13.329 and provides better efficiency, thus minimizing the energy consumption.

However, (Wang and Zhao, 2019) have done slightly similar study but with a different approach. The original rotor is composed of NACA8H12 airfoil and had an aspect ratio of 17.42. Instead of interchanging the airfoil with another existing airfoil, the original airfoil profile is altered and optimized to create a new airfoil shape. Both CFD simulation and experimental method have been carried out for the result verification. It has ben deduced that the new design of blade had higher lift to drag ratio and produced more thrust compared the original design.

2.4 Mesh of Propeller

2.4.1 Mesh Independency Study

The mesh independency study is process of finding the right mesh resolution in order to get accurate results by reducing the errors produced by the simulation. The accuracy of the result can be determined by comparing the results with existing experimental results as done by (Almohammadi et al., 2013) and (Wang et al., 2019). Both researches have proven that having higher mesh resolution gives the best simulation results. (Almohammadi et al., 2013) research is about the techniques of mesh independency for a straight blade vertical axis wind turbine for Computational Fluid Dynamics. The simulation is done using ANSYS Fluent with 2 different turbulence models, the k-ω SST and k-ε RNG with a goal to find the power coefficient of the wind turbine. 7 different mesh resolutions were produced and based on the results produced show that a finer mesh resolution is needed in order for the power coefficient to have less errors and more accurate with the actual results.

(Wang et al., 2019) did a similar study that presents the numerical simulation of propeller exciting force induced by milling-shape ice. The propeller used for this research is the 4 bladed R-class icebreaker propeller. Before running the simulation, a mesh independency study has been done on the propeller by producing three mesh resolutions, 5.5 million cells, 13.8 million cells and 20.1 million cells. The simulation is ran for three of the propellers and compared only the thrust coefficient to the experimental results. The results shows the errors between the computational and experimental results decreases as the number of cells increases.

Another similar study done by (Scuro et al., 2018) is slightly different in terms of choosing the best mesh resolution. The article is about A CFD analysis of the flow dynamics of a directly-operated safety relief valve. The software used for this simulation is ANSYS CFX. A mesh independency test has been carried out to find the best mesh resolution for the simulation. the mesh independency test results shows that the physical quantities being measured increased as the number of cells increased until it reached 3.4 million cells where the value physical quantities remain constant indicating that simulation results have reached mesh independency instead of comparing with experimental results.

2.4.2 Mesh Grid Shape

Mesh can be generated using two shapes tetrahedral and hexahedral. These shapes have advantages for different purposes based on the geometrical contruction of the parts being simulated. In terms of accuracy, researchers (Li et al., 2017), (Rupak Biswas 1998) and (Bahramian, 2019) demonstrated that the hexahedral mesh produced more accurate results compared to the tetrahedral.

(Li et al., 2017)'s study has been carried out on the reliability of turbulence models and mesh types for Computational Fluid Dynamics simulation of a mechanically ventilated pig house containing animals. The software involved in this simulation is ANSYS Fluent. The simulation for the ventilated pig house is done based on different turbulence models, different mesh resolution and different mesh type which is tetrahedral and hexahedral mesh. The experiment concluded that the hexahedral mesh had better accuracy but there is no significant effects shown in terms of performance. However, the tetrahedral mesh can be used for meshing complex shapes since more cells are generated compared to hexahedral.

Similar results have been obtained on the study done by (Rupak Biswas 1998) which is based on tetrahedral and hexahedral mesh adaptation for Computational Fluid Dynamics. Two problems are tested based on mesh adaptation. The first problem is the simulation of flow around a wing composed of NACA 0012 airfoil. The second problem is the simulation of a helicopter rotor model UH-1H to determine its acoustic performance. The results show that the tetrahedral mesh had higher mesh elements compared to hexahedral mesh in the same condition. It is also concluded that the hexahedral mesh contributed to the improvement to the solution accuracy compared to tetrahedral mesh.

Similarly to both previous study, (Bahramian, 2019) has experimented on the effects of mesh refinement, grid configuration and the wall boundary condition on the reliability and accuracy of numerical results of microparticle fluidization in a conical bed. To types mesh, hexahedral and tetrahedral are generated on the conical bed using Gambit and ANSYS and Computational Fluid Dynamic simulation has been conducted.it is concluded that results from the hexahedral mesh were close to the experimental results compared to tetrahedral.

2.5 Computational Fluid Dynamics Simulation

2.5.1 Turbulence Model

The turbulence model is an important mathematical construction that needs t be selected in order to determine the turbulence in a fluid flow simulation. Widely used turbulence models such as the variants of k-ε and k-ω have been tested by (Ayadi et al., 2018), (Abdolrahim Rezaeiha, 2019) and (Fu et al., 2018)

(Ayadi et al., 2018) concluded that the k-ε model is the best turbulence model due to its good agreement to the experimental results. This article is about the effects of turbulence model on simulation of airflow of a solar chimney. Computational Fluid Dynamics simulation is done using ANSYS Fluent. Five turbulence models such as Standard k-ε , Transition-k-kl-ω, RNG k-ε, Realizable k-ε and Transition-SST models were tested in order to find the best one in terms airflow characteristics such as velocity, temperature, pressure and turbulence characteristics.

Similar study has been done by (Abdolrahim Rezaeiha, 2019), he investigate the accuracy of the turbulence models for Computational Fluid Dynamics simulations of vertical axis wind turbine. The author concluded the results in contrary with the previous study by stating that the SST turbulence model variants produced results close to the experimental results. (Abdolrahim Rezaeiha, 2019) used Seven eddy viscosity turbulence models which are Spallart-Allmaras (SA), RNG k-ε, Realizable k-ε, SST k-ω, SST k-ω with additional intermittency transition model (SSTI), k-kl-ω and Transition SST (TSST) k-ω models are tested based on three sets of experiments on Vertical Axis Wind Turbine with different geometrical settings and operational conditions. The software involved for the numerical simulation is ANSYS Fluent.

(Fu et al., 2018) specifically states that the AKN k-ε turbulence model has higher accuracy. This article discusses the effect of turbulence models on the Computational Fluid Dynamics simulation on the aerodynamics of a full-scaled NASCAR Gen 6 Cup racecar (Fu et al., 2018). The turbulence models used are Realizable k-ε, AKN k-ε and SST k-ω. Furthermore, it is stated that the SST k-ω turbulence model offers an advantage in terms of the prediction of multiple horse shoe vortex cores on the deck line. In contrary to (Abdolrahim Rezaeiha, 2019), this study also stated that realizable k-ε turbulence model showed the worst accuracy in the terms of capturing appropriate vortex dominated flow.

2.6 Summary Table of Literature Review

Airfoil profile		
Study Name	Airfoil Used	Result
(Panigrahi and Mishra, 2014)	NACA 747A315 Eppler 420 Eppler 544 Eppler 855 FX74 CL5 140 NACA 64(3)-418	Best airfoil NACA 747A315 because it offers highest C_L/C_D
(Wang and Zhao, 2019)	NACA 8H12	Optimized NACA 8H12 had higher L/D and higher Thrust than its original design.
Blade Profile		
Study Name	Procedure	Result
(Maizi et al., 2018)	Tip blade altered: Reference tip, Shark tip, Original tip	Shark tip gave best performance in terms of acoustics by Reducing sound by 7%
(Liu et al., 2017)	Optimization of Purdue model blade	Increase in blade efficiency
(Jinsoo Cho, 1998)	Untwisted and twisted blade performance comparison	Twisted blade produced 1.9% higher power and 7.8% higher thrust than untwisted blade
Slot Shape		
Study Name	Shape	Result
(Ni et al., 2019)	Shape constructed using two circles	Airfoil Lift and L/D ratio increases
(Belamadi et al., 2016)	Straight diagonal slot, creating passage between upper and lower surface of airfoil	Efficiency is inversely proportional to slot size
(Rong et al., 2015)	Unique wave shape	At low flow rate the vortex formation reduced and a uniform surface flow field is formed.
Blade Number		
Study Name	Blade Number	Results
(Ahmadi Asl et al., 2017)	No. of blades: 2 blades, 3blades 4 blades	The RPM of the rotor reduces 10% - 12.5% for adding each blade.
(Singh and Nestmann, 2011)	No. of blades: 5 blades, 6 blades	The flow guidance improved for increasing number of blades. However, efficiency reduces when flow guidance improves.
(J. A. Lieser, 1997)	2 blades, 4 blades, 6 blades	The 6 bladed fan has the best acoustic performance
Reynolds Number		
Study Name	Reynolds Number	Results
(Bartl et al., 2019)	8 Reynolds number ranging from 0.5×10^5 to 6.0×10^5	The C_L increased at Re = 4.0×10^5 compared to at Re = 0.7×10^5. The C_D reduced at

		Re = 2.0 x 10^5 compared to Re = 0.5 x 10^5
(McTavish et al., 2013)	Ranging from 3620 to 31400	Wake expansion for 3 blades propeller increases. Wake expansion reduced 30% to 50% for 2 bladed propeller. Thrust coefficient reduce when Reynolds number reduced.
(Zanforlin and Deluca, 2018)	Ranging from 2.20 x 10^5 to 1.63 x 10^7	The L/D ratio increases within the Re range from 1.0 x 10^6 to 1.63 x 10^7.
Mesh Independency Study		
Study Name	Mesh resolution	Result
(Almohammadi et al., 2013)	7 different mesh resolutions	Higher mesh resolutions give less errors
(Wang et al., 2019)	Mesh resolution: 5.5 million, 13.8 million, 20.1 million	Error between computational and experimental results reduced when mesh resolution increase.
(Scuro et al., 2018)	Up to 3.4 million cells	Simulation results reached independency after the mesh resolution of 3.4 million
Mesh Shape		
Study Name	Mesh Shape	Result
(Li et al., 2017)	Hexahedral and tetrahedral	Hexahedral gives better accuracy compared to Tetrahedral. However, Tetrahedral can be used for complex shaped domains
(Rupak Biswas 1998)	Hexahedral and tetrahedral	Tetrahedral mesh had higher elements compared to hexahedral. Hexahedral gives better accuracy in terms of results.
(Bahramian, 2019)	Hexahedral and tetrahedral	Hexahedral mesh results are more accurate compared to tetrahedral
Turbulence Model		
Study Name	Turbulence Model	Results
(Ayadi et al., 2018)	Standard k-ε, Transition-k-kl-ω, RNG k-ε, Realizable k-ε, Transition-SST models	k-ε is the best turbulence model to its high accuracy results
(Abdolrahim Rezaeiha, 2019)	Spallart-Allmaras (SA),RNG k-ε, Realizable k-ε, SST k-ω , SST k-ω with additional intermittency transition model (SSTI), k-kl-ω, Transition SST (TSST) k-ω models	SST variant turbulence models produce result nearly similar to the experimental results compared to other turbulence models.
(Fu et al., 2018)	Realizable k-ε, AKN k-ε, SST k-ω	Realizable k-ε showed the worst accuracy while AKN k-ε had the best accuracy

CHAPTER 3: METHODOLOGY

3.1 Propeller Model

The propeller model chosen for the basis of this simulation is the 10 x 7 inches 2 bladed APC Slowflyer illustrated in figure 3.1.1 and 3.1.2 due to its availability of data. The APC Slowflyer is well made for speeds at low Reynolds number due to its design of its blade composed of 2 airfoils, the Eppler E63 and Clark Y. The Eppler E63 airfoil will be replaced with 7 different airfoils with different aerodynamic performance as shown in table 3.1.1. Figure 3.1.3 and 3.1.4 illustrates the graph of lift coefficient and the graph of lift coefficient over drag coefficient for 8 airfoils respectively.

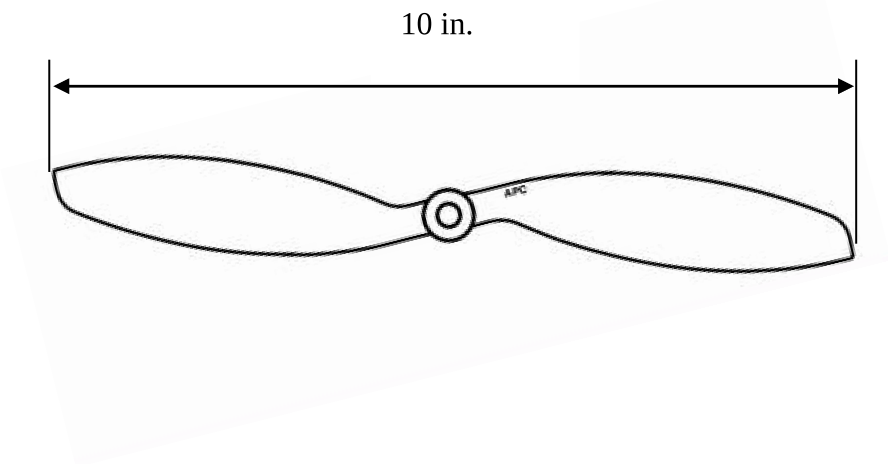


Figure 3.1.1: Front view of APC Slowflyer propeller

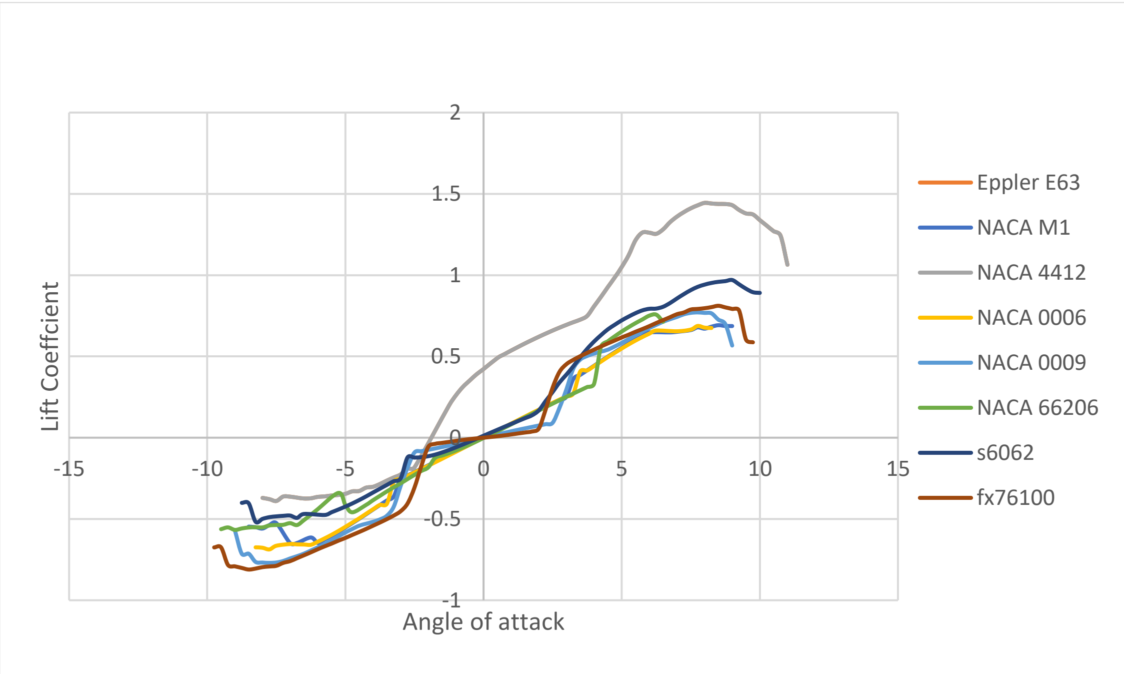


Figure 3.1.2: Graph of lift coefficient for eight airfoil types

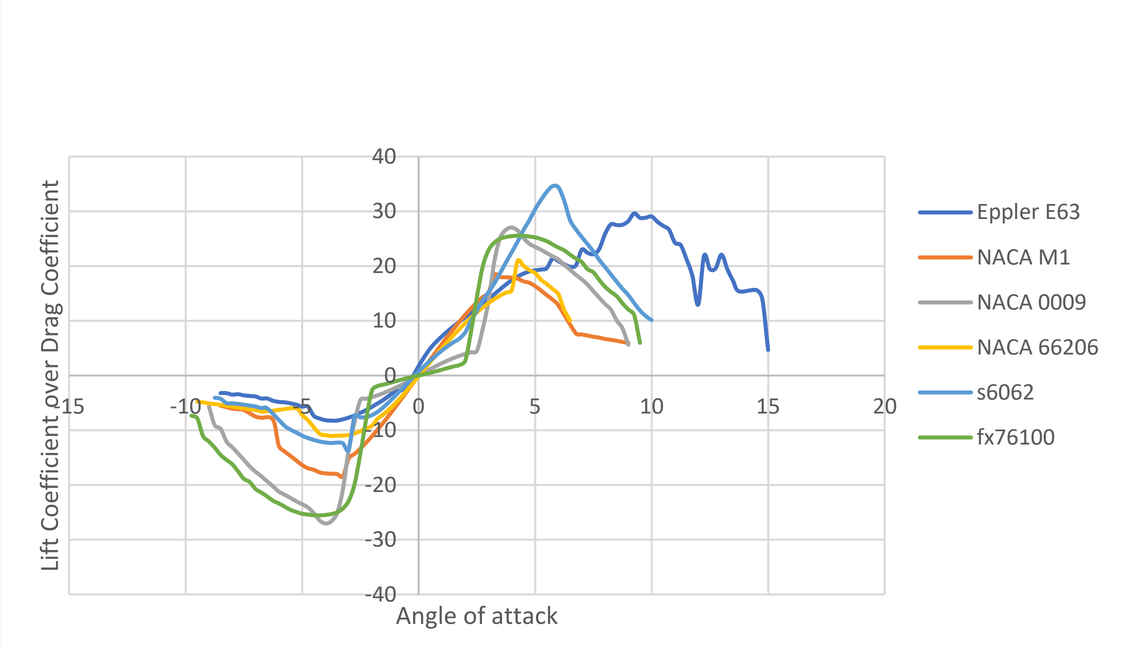


Figure 3.1.3: Graph of lift coefficient over drag coefficient for eight airfoil types

Based on the objectives of the paper, eight different slotted propellers are design including the APC Slowflyer propeller are designed using Catia V5. The remaining seven propellers are design by swapping the Eppler E63 airfoil with seven airfoil as shown in table 3.1.1 with each propeller designed while maintaining the APC Slowflyer's geometrical shape. Implementing the slotted airfoil design in to the into each of the propeller, there will sixteen propellers altogether designed. Each airfoil chosen is based on eight different aerodynamic property combinations as shown in table 3.1.1.

Table 3.1.1: Category of airfoil for baseline propeller

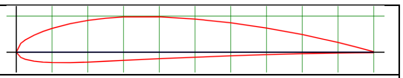
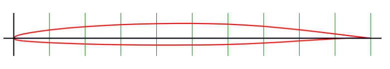
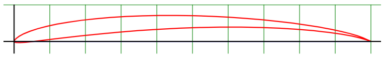
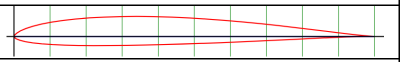
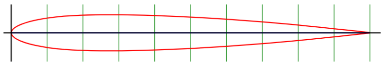
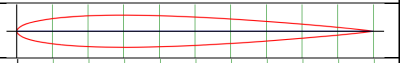
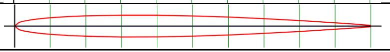
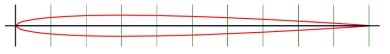
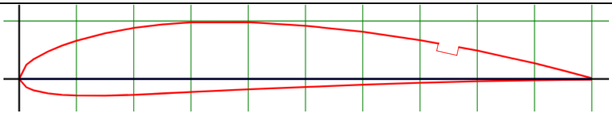
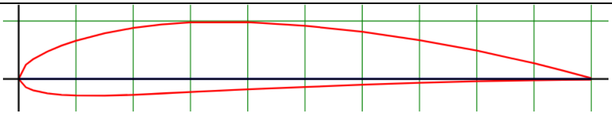
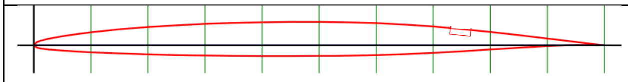
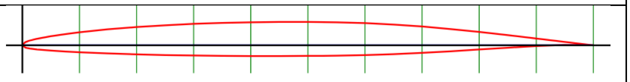
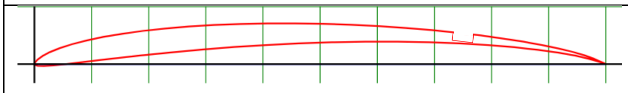
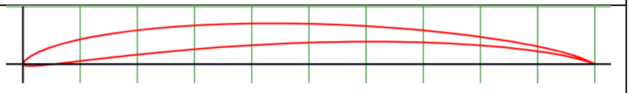
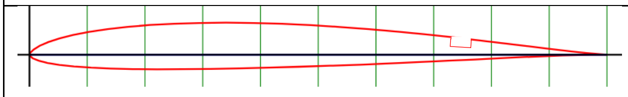
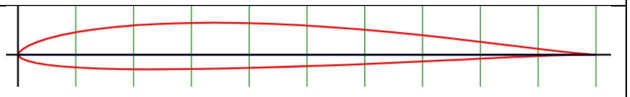
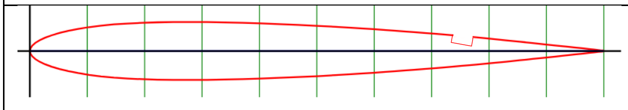
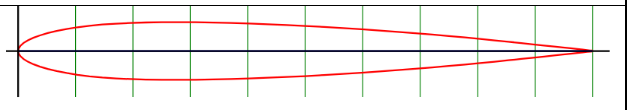
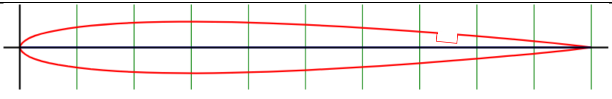
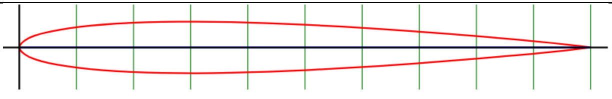
Asymmetrical	High Reynolds Number	High Lift	NACA 4412	
		Low drag	NACA 66206	
	Low Reynolds Number	High Lift	Eppler E63 (APC Slowflyer)	
		Low drag	s6062	
Symmetrical	High Reynolds Number	High Lift	Wortmann fx76-100	
		Low drag	NACA 0009	
	Low Reynolds Number	High Lift	NACA M1	
		Low drag	NACA 0006	

Table 3.1.2 : Airfoil of slotted and baseline propeller

Slotted	baseline
	
	
	
	
	
	

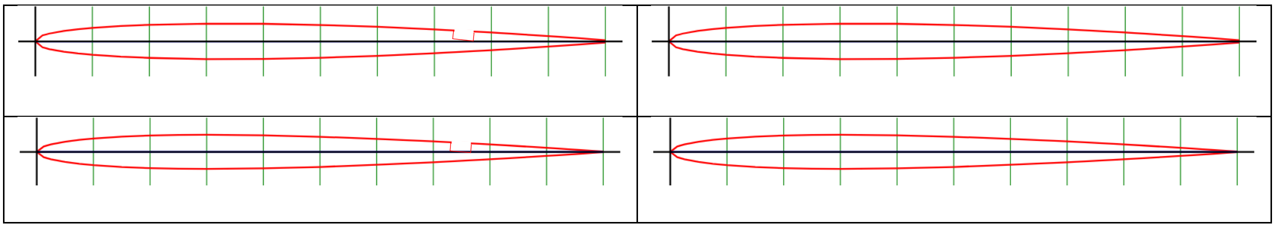


Table 3.1.3 : Airfoil of slotted and baseline propeller

The slot design selected in this study is the square shaped design situated on top of the airfoil, at 75% of the chord length as shown in Table 3.1.2. The dimension of the slot is 0.16mm x 0.836 the same for all the propellers as illustrated in Table 3.1.2. The slot in the propeller will appear as a groove on along the propeller blade. Slotted design has been implemented on the airfoil in order to create the flow separation thus slowing down the airflow velocity along the airfoil. The reduction in airflow velocity over the airfoil of the propeller will have a significant effect on the propeller's thrust, power and efficiency. Based on the objective of this study, the result of the simulation on the slotted and baseline propeller design will be compared and airfoil that has the most positive effect due to the implementation of the slotted designed airfoil will be selected and justified.

The thrust coefficient (K_T), power coefficient (K_P), torque coefficient (K_Q) and the efficiency (η) of propeller is clearly presented in equation (1),(2),(3) and (4) respectively are three parameters that need to be analyzed in order to determine the slotted and the baseline propellers' performance. From the equations, T (N) is Thrust, P (N.ms^{-1}) is power, Q (N.m) is torque, n (rps) is revolutions per second, D (m) is the propeller diameter and ρ(kgm^{-3}) is the density of fluid.

$$K_T = \frac{T}{\rho n^2 D^4} \qquad (1)$$