STRUCTURAL ANALYSIS OF DRONE PROPELLER USING FINITE ELEMENT ANALYSIS

LORRAINE PIDANG JOHN

UNIVERSITI SAINS MALAYSIA

2021

STRUCTURAL ANALYSIS OF DRONE PROPELLER USING FINITE ELEMENT ANALYSIS

By:

LORRAINE PIDANG JOHN
(Matric No: 137823)

Supervisor:
Associate Professor Dr. Abdullah Aziz bin Saad

August 2021

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with honours degree in

BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed……………………………… (LORRAINE PIDANG JOHN)

Date………3/8/2021………………………

Statement 1: This journal is the result of my own investigation, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/ references are appended.

Signed……………………………… (LORRAINE PIDANG JOHN)

Date………3/8/2021………………………

Statement 2: I hereby give consent for my journal, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed……………………………… (LORRAINE PIDANG JOHN)

Date………3/8/2021………………………

ACKNOWLEDGEMENT

First and foremost, praises and thanks to the God, the Almighty, for His showers of blessings throughout my research work to complete the research successfully.

I would like to express my deep and sincere gratitude to my research supervisor, Associate Professor Dr Abdullah Aziz Bin Saad for giving me the opportunity to do research and providing invaluable guidance throughout this research. His dynamism, vision, sincerity, and motivation have deeply inspired me. He has taught me the methodology to carry out the research and to present the research works as clearly as possible. It was a great privilege and honour to work and study under his guidance. I am extremely grateful for what he has offered me. I would also like to thank him for his friendship, empathy, and great sense of humour. Adding to our current situation right now, it was a very tough time for most of us to finish up and complete this research, however I am very thankful and grateful for the countless effort and guidance he provides despite the outbreak situation. Not to forget, I would deliver my thanks to all the lecturers and technician staff of School of Mechanical Engineering that have helped throughout my 4 years studies and for giving me the chance to learn and accomplished my goals as a engineering student.

I am extremely grateful to my parents for their love, prayers, caring and sacrifices for educating and preparing me for my future. I am very much thankful to my brothers and my sisters for their love, understanding, prayers and continuing support to complete this research work. Also, I express my thanks to all my friends, for their support and valuable prayers.

Finally, my thanks go to all the people who have supported me to complete the research work directly or indirectly. Thanks for giving me the wisdom to never give up and keep me going.

Lorraine Pidang John

TABLE OF CONTENTS

LIST OF TABLES

Page

LIST OF FIGURES

Page

LIST OF SYMBOLS

ω	Angular velocity
mm	Millimeter
MPa	Megapascal
Pa	Pascal
g	Gram
N	Newton

LIST OF ABBREVIATIONS

UAV	Unmanned Aerial Vehicles
UAS	Unmanned Aircraft Systems
GPS	Global Positioning System
CFD	Computational Fluid Dynamics
RANS	Reynolds-average Navier’s Stokes
CAD	Computer-aided Design
FEA	Finite Element Analysis
APC	Armored personnel carrier
CFRP	Carbon fiber reinforced plastic
RPM	Revolutions per minute
FEM	Finite Element Method
CMM	Coordinate Measurement Machine
CWW	Counter-Clockwise
RC	Radio-controlled

LIST OF APPENDICES

ABSTRAK

Prestasi penerbangan quadcopter terutamanya dipengaruhi oleh kipas-kipasnya. Kelancaran Quadcopter terhalang oleh kipasnya yang besar dan lemah. Dalam tesis ini, tujuan utama kami adalah untuk menyiasat hubungan pekali tuju dan kelajuan putaran menggunakan analisis Computational Fluid Dynamics (CFD) dalam ANSYS Fluent. Analisis ini dilakukan menggunakan pemecah Navier-Stokes dengan model turbulensi Standard k-Omega bersama dengan pembetulan kelengkungan di mana ia dapat mengubah daya putaran menjadi pendorong linar, yang mendorong Quadcopter melalui ruang angkasa. Oleh itu, setiap kipas-kipasnya adalah unik dari segi diameter, nada, tujahan, dan, yang paling penting, bahan. Model propeller CAD telah direka dan diubahsuai menggunakan SolidWork Software ver2020. Seterusnya, reka bentuk kipas yang dibuat telah diimport ke *ANSYS Workbench Fluid Flow (Fluent)* dan *Static Structural ANSYS*. Data kemudian dipindahkan dari modul aliran bendalir ke analisis struktur statik untuk menilai ubah bentuk dan tekanan yang disebabkan oleh tekanan yang disebabkan oleh putaran kipas pada kelajuan 3000 RPM dan 6000 RPM. Selain itu, bahan yang digunakan untuk kipas sepanjang penyelidikan ini adalah *Carbon Fiber Reinforced Plastic* (CFRP) dan Aluminium Aloi. Ini disebabkan oleh ciri-ciri ringan dan kecekapan berprestasi tinggi semasa penerbangan. Pembahagian tekanan menunjukkan kawasan tekanan positif pada bahagian muka dan kawasan negatif pada bahagian belakang yang menghasilkan penjanaan tujahan. Taburan tegangan kipas-kipas ini disiasat, dan kemudian analisis struktur dilakukan dengan mendapatkan tegasan von-mises maksimum, regangan von-mises maksimum, dan total ubah bentuk kipas sebagai hasil kajian ini. Dari analisis tersebut, taburan tekanan meramalkan kawasan yang sangat tinggi di bahagian hub iaitu tengah dan menurun mengikut penambahan nilai radius kipas.

STRUCTURAL ANALYSIS OF DRONE PROPELLER USING FINITE ELEMENT ANALYSIS

ABSTRACT

The flight performance of a quadcopter is primarily influenced by its propellers. The maneuverability of the Quadcopter is hindered by its massive and feeble propeller. In this paper, our main goals were to investigate the relationship of thrust coefficient and rotational speed using Computational Fluid Dynamics (CFD) analysis in ANSYS Fluent. These analyses were carried out using a Navier-Stokes's solver with Standard k-Omega turbulence model along with curvature correction where it can convert rotational power into linear propulsion, which propels the Quadcopter through space. Moreover, every propeller was unique in terms of diameter, pitch, thrust, and, most crucially, material. The solid CAD model of propeller has been designed in SolidWork Software ver2020. The created propeller design has been imported to ANSYS workbench fluid flow (fluent) and ANSYS Static Structural. The data was then transferred from the fluid flow module to static structural analysis in order to evaluate the deformation and stress caused by the pressure induced by propeller rotation at 3000 RPM and 6000 RPM. Apart from that, the materials used for the propeller throughout this research were carbon fiber reinforced plastic (CFRP) and Aluminium alloy. This was due to their lightweight characteristics and high-performance efficiency during flight. The pressure distribution demonstrated a positive pressure region on the face section and a negative region on the back section that produces the thrust generation. The propeller stress distribution was investigated, and then structural analyses were done by getting the maximum von-mises stress, maximum von-mises strain, and total propeller deformation as a consequence of this study. From the analysis, the stress distribution predicted a highly concentrated region near the hub and decreasing with the growing value of the propeller radius.

CHAPTER 1

INTRODUCTION

1.1 Research Background

A drone is a type of unmanned aircraft. Drones are also known as unmanned aerial vehicles (UAVs) or unmanned aircraft systems (UASes). A drone is essentially a flying machine that can be remotely controlled or remotely controlled by utilising operating system flight plans in its integrated system works in conjunction with onboard sensors and GPS [1]. Commercial drone use has been more widespread over the past few years, and the used of drone technology have been expanded rapidly. The first fundamental distinction in drone utilization is between the drone itself (the platform) and the equipment attached to it (the payload)[2]. In this context, the drone is best viewed as a flying platform that may be configured to achieve a variety of objectives. These objectives can be met by using the appropriate payload. A camera, for example, can be mounted to a drone to make it appropriate for specific inspections as shown in Figure 1.1.

Figure 1.1 Example of Quadcopter Drone[3].

Previously, unmanned aerial vehicles (UAVs) were linked with the military, where they were employed for anti-aircraft target practise, information gathering, and, more controversially, as weapons platforms. Drones are also utilised in civilian activities such as search and rescue, observation, traffic monitoring, military surveillance, and firefighting, as well as personal and commercial drone photos, videography, agriculture, and even courier services.

Currently, in Malaysia, there will be a new project on upgrading the delivery services by using a drone service. AirAsia Digital had partnered with the Malaysian Global Innovation and Creativity Centre (MaGIC) for their project to launch the Urban Drone Delivery Sandbox[4]. Drone delivery can be expanded beyond e-commerce to carry critical or medical supplies to rural, distant, or natural disaster-affected areas, for example. Drone delivery can be expanded beyond e-commerce to carry critical or medical supplies to rural, distant, or natural disaster-affected areas, for example. The six-propeller drone somehow can only carry maximum 800 gram of load. Thus, as the trial stage is called a success, they might can increase the load to a maximum of 3 kilogram depending on the good feedback by the customers[5].

In addition, different drones may travel at different heights and distances. Close-range drones, which can go up to three miles, are commonly used by hobbyists. Unmanned aerial vehicles (UAVs) with a range of less than 30 miles are known as close-range UAVs. Short-range drones with a range of up to 90 miles are mostly used for espionage and information collection. Mid-range unmanned aerial vehicles (UAVs) have a 400-mile range and could be utilized for intelligence gathering, scientific research, and meteorological investigations. The longest-range drones are classified as “endurance” UAVs, and they can fly up to 3,000 feet in the air and have a 400-mile range [6].

Computational fluid dynamics (CFD) has been used extensively in analysing and constructing various air distribution systems. Many factors can influence the use of CFD to analyse air dispersion. The selection of an acceptable CFD method and a turbulence model are the most crucial elements. Recent advancements in CFD techniques and turbulence models hold tremendous promise for enhancing the accuracy of air distribution prediction in confined spaces[7].

As a result, designing extremely efficient UAVs is critical. The propeller has an impact on a UAV's efficiency. The propeller chosen must be capable of meeting the UAV's aerodynamic needs. The current approach to UAV propeller optimization relies on traditional parameters such as diameter, pitch, aerofoil, and blade angle. Propellers of unusual designs, such as slotted, serrated, and tubercle blades, are rarely utilised in UAVs. As a result, the purpose of this research is to study the conditions of this designed propeller blade under various circumstances utilising two different materials.

This research study is separated into two parts: aerodynamic analysis and static structural analysis. FLUENT, a commercial computational fluid dynamic (CFD) solver, is used to perform the flow simulation to obtain the thrust value generated by the propeller and overall efficiency of the propeller blade. Meanwhile, the programme the propeller blade's highest stress, strain, and maximum deformation were evaluated using ANSYS Static Structural.

1.2 Problem Statement

For the aircraft industries specifically in commercial use of unmanned aerial vehicles (UAV), weight reduction has always been a concern, mostly to minimize consumption but also enhance handling. The power consumption, their speeds and forces are critical aspects in the design of this unmanned aerial vehicles especially for the blades as it is crucial to know more on the requirements that will decides on the performance of the drones consequently on the vehicle's dynamics, and stability – which the results can be finalize and to be off to market. Therefore, the purpose of this project is to study structural performance of a drone propeller's blade under several conditions along with improving the performance of the drone itself. Thus, the ANSYS Workbench Software will be used as the main tool for the study.

1.3 Objectives

The specifics objectives of this research are:

1) To investigate the structural analysis occurs on the drone propeller's blade using different material conditions such as Carbon Fiber Reinforced Plastic (CFRP) and Aluminium Alloy.
2) To study the thrust coefficient and pressure distribution act on the propeller under different constraints where the rotational velocity used at 3000 RPM and 6000 RPM respectively.

1.4 Scope of Work

This project is mainly simulation-based, with the use of fluid flow analysis and structural analysis. At first, an existing design propeller blade was taken to be measured using Coordinate Measurement Machine (CMM). The entire data measurement was then utilised to create a solid CAD design of a drone propeller blade using SolidWorks software. However, many modifications to the measurement had been made in order to meet the design requirement. After finishing the solid propeller blade design, Finite Element Analysis will be conducted using FLUENT to obtain the pressure and thrust coefficient of propeller. This data will be transmitted to Static Structural in order to calculate the displacement, stresses, and strain acting on the propeller blade within specified parameters. Lastly, this thesis concludes with findings and suggestions for future research directions linked to the project's topic.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The purpose of this chapter is to provide an overview of an unmanned aerial vehicle's propeller, a definition of Computational Fluid Dynamics (CFD) study, propeller performance characteristics, software used in this project, and propeller history with innovation.

2.2 Review on the propeller

A comprehensive investigation into the traditional design of propeller blades focusing on fundamental properties of diameter, pitch, blade form and even chord length was conducted. As a consequence, sophisticated designs including serrated, slotted, tubercle, and adjustable structure may be employed to improve propeller blade design. Design analysis of these propeller blade can be seen in the work of B. Rutkay and J. Laliberte et al. [8] and Adkins et al. [9]. Material testing, manufacturing trials, and wind tunnel testing of the propellers under simulated flight circumstances were used to examine the potential of additive manufacturing (3D printing) in the fabrication of flightworthy propellers. The propeller performance produced nearly the predicted design thrust, but the efficiency and power consumptions could not be measured accurately with the current test setup.

The figurative findings are compared with experimental records for advanced precise composite (APC) slow flyer propeller blades to evaluate the difference in thrust coefficients, strength coefficients, and effectiveness, as reported by H.A. Kutty and P. Rajendran et al. [10]. During the study, they used unstructured tetrahedron meshing and a popular k- turbulence version. A. Seeni et al. [11] also investigated the k-ω turbulence model using various reference frames to integrate the propeller's rotational speed. The findings of the k-ω turbulence model are improved.

F. Md Ahmed, M. N. Zafar, and J. C. Mohanta et al. [12] focuses on the modelling and structural analysis of the frame of a quadcopter where AutoCAD 2016 software was used for modelling, and ANSYS 17.0 software was used for analysis of various parts. The collected results were compared to the ability of the quadcopter to sustain the loads created. They found that the proposed design is safe, as only minor deformations of the plates occurred.

Meanwhile, alternative approaches, such as M. N. K. Othman et al. [13]'s relation of mouth-ring application, can be utilised to assess the propeller's thrust generation. According to the results of their investigation, a bigger diameter propeller is used to enhance the thrust. The bigger the area of the propeller, the greater the velocity, and hence the greater the thrust. A larger diameter propeller also provided improved air flow across the blade's surface.

There are two approaches for determining the performance of a propeller blade: experimental and numerical. Based on the applicability of the analysis, both approaches have been widely used by numerous researchers. A. R. Nuranto, A. J. Fitroh, and H. Syamsudin et al. [14] performed aerodynamic load calculation using both Blade Element Theory (BET) and computational fluid dynamics (CFD) simulation. Different thrust forces were obtained with only 1.2 percent and 4.1 percent for two types of mesh.

Other approaches such as FEA is used in a variety of engineering domains, including structural analysis, fluid flow, heat transfer, and mass transfer. FEA can now frequently provide numerical solutions to even the most complex stress situations[15]. As a result, FEA is utilised to solve mathematical problems by discretizing the whole physical domain into a small number of finite elements. The use of FEA software (Ansys) helps reduce the number of prototypes required throughout the design and optimization phases.

Ahmad et al. [16] used Finite Element Analysis to analyse and compare the modal frequencies and maximum deformation of three distinct design propellers. Based on the results of the FEA simulation, it can be deduced that the Case-C type of propeller has a higher failure (resonance) frequency and a lower maximum deformation value than the other two designs, Case-A and Case-B; consequently, Case C is recommended for propeller design.

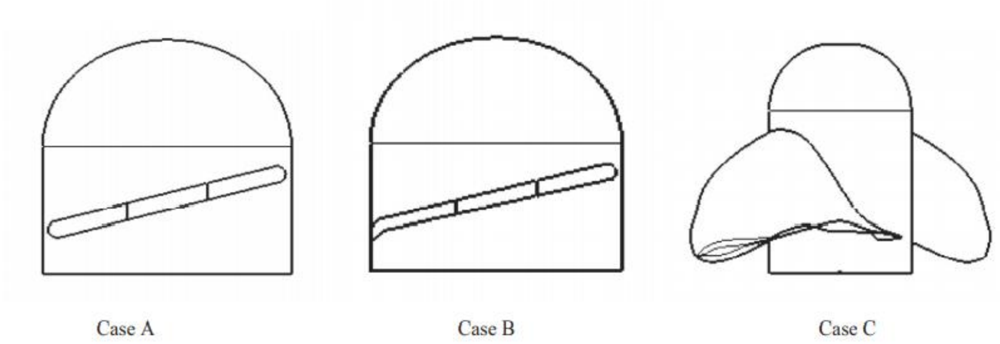

Figure 2.1 Case Type of Propeller[16]

Most significantly, several studies debated the strength of propeller blades. Not only must the blades be sufficiently resistant to endure extended periods of rigorous operation without failure or irreversible deformation, but elastic deflection under load must not modify the geometrical shape to the point where it alters the planned distribution of loads.

Kishore et al. [27] use ANSYS to compare the structural performance of two alternative materials for propeller blades. The von-mises gathered maximum stress, stress, and total deformation data. The results of the studies have been compared to the material's mechanical qualities. Using FEA, Yeo et al [28] evaluate a marine propeller blade's stress distribution. In the study, the pressure distribution along the blade is used to determine the maximum stress and deflection.

2.2.1 Basic Nomenclatures of Propeller in Terms of Performance.

Propellers are a type of fan that converts mechanical energy into kinetic energy of the fluid[17]. The thrust is generated by the fluid's momentum increase. The parameters of a propeller are listed below.

- **Blades**

The twisted fins or foils that emerge from the propeller hub are known as blades. The torque a propeller can provide is determined by the form of the blades and the speed at which they are pushed. The blade root is where the blade connects to the hub, while the blade tip is the blade's outermost edge at the furthest point from the propeller shaft.

- **Number of blades**

This parameter has a negligible impact on efficiency. A propeller with more blades will typically perform slightly better due to more evenly distributed power and thrust in its wake[18].

- **Diameter**

The diameter of the propeller has a significant impact on performance. Diameter is one of the most important geometric characteristics in determining how much power a propeller can absorb and deliver, and therefore how much thrust is available for propulsion. Usually the diameter is proportional to the efficiency of propeller, but in high speed larger propellers typically have higher efficiency because they catch more incoming fluid and thus distribute their power and thrust over a larger fluid volume[19].

- **Pitch**

A variable pitch propeller can raise the aircraft's maximum take-off weight and enhance hover power efficiency, especially if the load fluctuates between flights [20]. A propeller's blade pitch is an important component to consider. The pitch, which is analogous to the gears on a manual vehicle gearbox, determines how much air is cut by the blade. The pitch is set for initial acceleration (which is necessary for take-off), but it is fixed, which limits the speed in fixed blade pitch. However, you may change the blade pitch for maximum flying speed. To gradually build up enough speed to produce enough lift for take-off, this would need a very long runway. They are usually set at a compromise position where take-off is not maximised, and maximum speed is less than what is possible. As researchers studied the mechanics behind propeller power, fixed pitch designs slowed initial aeroplane speeds.

2.2.2 Materials on propeller

Propellers were initially constructed of wood, but since the 1920s, they have been manufactured of steel and aluminium, and since the 1970s, sophisticated composite materials. Wood and metal propellers are still widely designed and manufactured, despite the fact that newer composite materials offer excellent mechanical properties. Like manned aircraft, RC aircraft propellers are usually built of wood and composites, but they are also made of reinforced plastics because to their low cost, weight, and strength. This project's propeller is composed of Carbon Fiber Reinforced Plastic (CFRP) and Aluminum Alloy.

2.2.2(a) Wood

If the diameter, pitch, and form are the same, wood propellers are the lightest and offer the least load to an engine. They have a higher RPM capacity than a heavier propeller. Wood has a high strength-to-weight ratio and is unaffected by tiredness. The high internal friction, or hysteresis, of the wood contributes to its fatigue resistance. Hysteresis is so efficient at damping vibrations that fixed-pitch wood propellers don't need to pass many of the certification tests. Wooden propellers, on the other hand, are the most brittle.

2.2.2(b) Metal

Metal propellers are generally made of extruded and welded steel tubes or forged and machined aluminium slabs. On the other hand, metal propellers are not utilised for RC aircraft or for little unused aircraft (UAVs). This is due to the very high blade weight, which increases the aircraft empty weight and slows the motor response to changes in control speeds, compared to wood and composite propellers. They are also seldom utilised since RC aircraft have a far larger chance of disaster than bunker planes and are normally much closer both to the controller and the observers. If the propeller malfunctions during a flight, a metal blade's enhanced cinematic speed and knife-like shape may kill and injure if a person is hurt.

2.2.2(c) Composites and Plastic

Polymeric composite materials, such as carbon fibre, nylon, fibre glass, and stainless titanium, are used to make various UAV components, mainly propellers, due to their outstanding characteristics, such as high strength and light weight. Drone propellers are also made from such composites, which have polymer-based origins. When a conventional UAV is in flight, debris might collide with the fast-moving propeller blades, causing the UAV to malfunction. It is common knowledge that polymer-based composite materials offer benefits over other materials in terms of strength[21] .

In UAVs, composites offer a variety of advantages over metals like low weight, excellent corrosion resistance, reduced machining to a high fatigue strength, ability to manufacture tapered sections and elaborate contoured parts, the ability to orient reinforcement fibres towards maximum rigidity and strength with cocure or co-consolidation processes.

2.3 Propeller Performance

Due to slipstream and propeller wake effects, rotating propellers have a major impact on an aircraft's aerodynamics, stability, and control. The performance of the fitted propeller is also affected by wing upwash.[23]. As a result, the installed arrangement should maximise propeller efficiency while avoiding unfavourable effects on aircraft aerodynamics.[23]. The propeller changes the flow direction behind it by increasing air speed. The wing lift and drag will increase as dynamic pressure rises. The aeroplane stall is also delayed by the propeller slipstream. While this is a positive benefit, propeller stall can be unacceptably dangerous [24]. For example, sophisticated propellers employed in early C-130J designs prevented the inner wing from stalling [24], leading the stall to originate at the wing tips, resulting in the loss of roll control.

In addition, heavily charged propellers generate a propeller wake because of the strong tip vortices at the tips of the propeller blades. The propeller wake produces a substantial shift in the elevator and drag distribution across the whole wingspan when these propellers are positioned before the wing [24]. This might potentially lead to a disproportionate aircraft weight distribution.

Many of these UAVs have feature propellers that operate at 75 percent propeller pitch at the low Reynolds of 50,000 to 100,000 on the propeller chord. The propeller efficiency was measured under these conditions at the University of Illinois at Urbana-Champaign (UIUC). A total of 79 propellers were examined, the bulk of which were between 9 and 11 inches in diameter. During the testing, the propeller speed (RPM) was kept constant while the wind tunnel speed was changed to sweep through a variety of advance ratios until the windmill condition was reached (zero thrust). The efficiency of propellers ranged from a high of 0.65 (for an efficient propeller) to a low of 0.28 (for a less efficient propeller) (for an exceptionally poor propeller). As a result, the research findings suggest that choosing the right propeller for a UAV can make a big difference in how well it performs[25].

The majority of drones may work less than 10 metres per second at wind speeds. Therefore, current drones cannot be deployed successfully in windy situations. Drones cannot be used in very warm or cold locations since their operating temperatures usually range from -10°C to 45°C. Finally, a remote controller usually has a considerably lesser maximum transmission distance than the maximum flying range. The use of UAVs with more costly sensors and communication equipment can, however, solve this constraint.

Furthermore, flying times are often restricted by battery restrictions. The volume is dependent upon a variety of characteristics, including efficiency of the battery, battery size, amount of cargo, weather and topography and the autonomy or remote control of the drone in line of sight (LOS). The drone with a maximum claimed range of 35 kilometres is provided to customers within a 14km (8,7-mile) radius (assuming that is uses 80 percent of the theoretical range). Heavier weights decrease the range further [26].

2.4 Computational Fluid Dynamics (CFD) Solver

Computational Fluid Dynamics (CFD) is a useful tool for investigating aerodynamics in a variety of applications, including modelling small-scale spinning propellers. Various research exists in the literature, including CFD analysis and validation of quad-rotor UAV propeller aerodynamic properties.

T. Oktay and Y. Eraslan et al. [27] conducted computational fluid dynamics (CFD) analysis of a quad-rotor UAV propeller to investigate the link between airspeed and propeller thrust coefficient. With incompressible and turbulent flow assumptions, the overset mesh approach was applied. Their methods are analysed using a Navier-Stokes solver (Ansys Fluent v17.2) utilising the turbulence model k-Omega SST, with correction of curvature. The results shown that thrust coefficient decreased at higher airspeeds. Thus, the numerical analyses found were closer to experimental results at lower airspeeds.

Furthermore, P. P. Hector Guillermo, A. M. Victor Daniel, and G. G. Elvis Eduardo et al. [28] used Computation Fluid Dynamics to undertake a transient investigation of physical factors related with aerodynamic behaviour in an unmanned aerial vehicle (UAV) (CFD). The simulation is used for post-processing to investigate vortex, pressure, and turbulence kinetic energy (TKE). The Hexacopter's aerodynamic behaviour changes with time, and there is an effect when four and two blades are employed in its rotors.

Wen et al. [29] investigated the effects of flying speed, altitude, and nozzle spacing on droplet drifts and spray distributions on agricultural quad-rotor drones. Some computational and experimental evaluations were carried out on a plant protection quadcopter for this aim. The computational fluid dynamics (CFD) calculations, which were based on the lattice Boltzmann technique, included an examination of spray droplet motion in wake vortices, flow patterns in near and far wakes at various flight speeds, and hovering flight. The CFD findings were found to be in good agreement with the experimental wind tunnel experiments that they conducted. The major factors influencing spray dispersion are flying speed and altitude.

2.5 Numerical Software

2.5.1 ANSYS

ANSYS is the latest package for simulation in single and multiphysics and offers extended tools and capabilities to enable engineers to work efficiently with their jobs. ANSYS offers extensive ability, capacity expansion and interface to almost all design tools, including pro-engineers, AutoCAD and SolidWork. In addition, ANSYS offers the best solver technology, specialised meshing technologies for physics and computation fluid dynamics, combined physics to produce sophisticated simulations. Structural, thermal, fluid, sound and multi-physical problems may be addressed using ANSYS [30].

2.5.2 Workbench

ANSYS workbench is an easy-to-use platform that integrates ANSYS, Inc's range of sophisticated engineering simulation technology. It connects to major CAD systems in both directions. The Workbench environment is designed to boost engineering teams' productivity and simplicity of use. [31]

2.5.3 Fluent

ANSYS Fluent provides complete modelling capabilities for incompressible and compressible fluid flow, laminar and turbulent problems. Stable or transitory analyses can be performed. The ability to simulate complex geometries in ANSYS Fluent pairs a wide variety of mathematical models in transportation operations (such as heat transfer and chemical reactions). Several important characteristics are offered that make it easier for industrial machinery and processes to represent fluid flow and related transport phenomena. They include porous mediums, lumped parameters (fan and heat exchanger) and stream-specific flow and heat transfer [32].

2.5.4 Static Structural Analysis

Software of structural analysis ANSYS simulates each of the structural aspects of a product, including linear static analysis that only offers stresses or deformations, modal analysis that evaluates the characteristics of vibrations and advanced transient nonlinear phenomena that includes dynamic effects and complex conducts. It is used by engineers of all skill levels across many sectors to optimize designs, minimize the costs of physical testing, and reduce the need for recurrent prototyping. Thus, ANSYS structural analysis software performs finite element analysis (FEA) on structural mechanics problems to examine numerous design scenarios.[33]

CHAPTER 3

METHODOLOGY

3.1 Propeller Model

In this research project, the model used is a two-bladed propeller with 13 inches diameter and 4 inches pitch. This propeller was initially measured using a Coordinate Measuring Machine (CMM) to observe the length, diameter and pitch of the propeller shows in Figure 3.1 to Figure 3.3. Thus, all the measured value are stated in Appendix B.

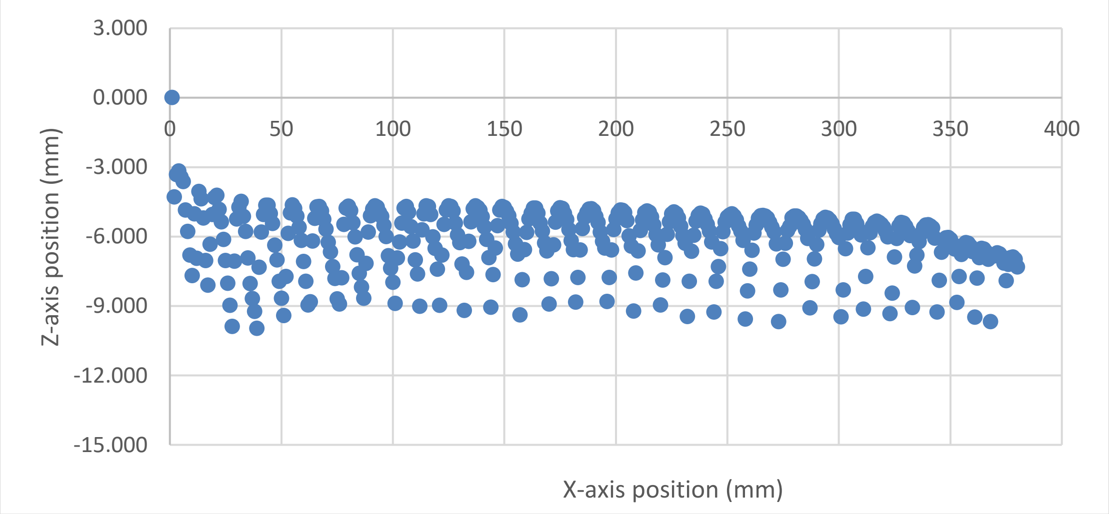

Figure 3.1 Side profile of propeller on CMM machine

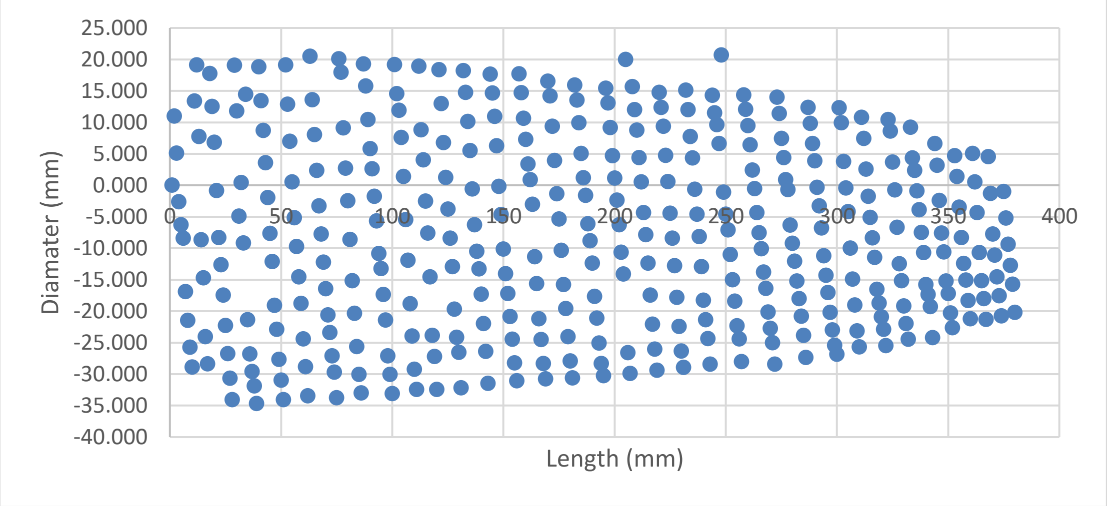

Figure 3.2 Top profile of propeller on CMM machine

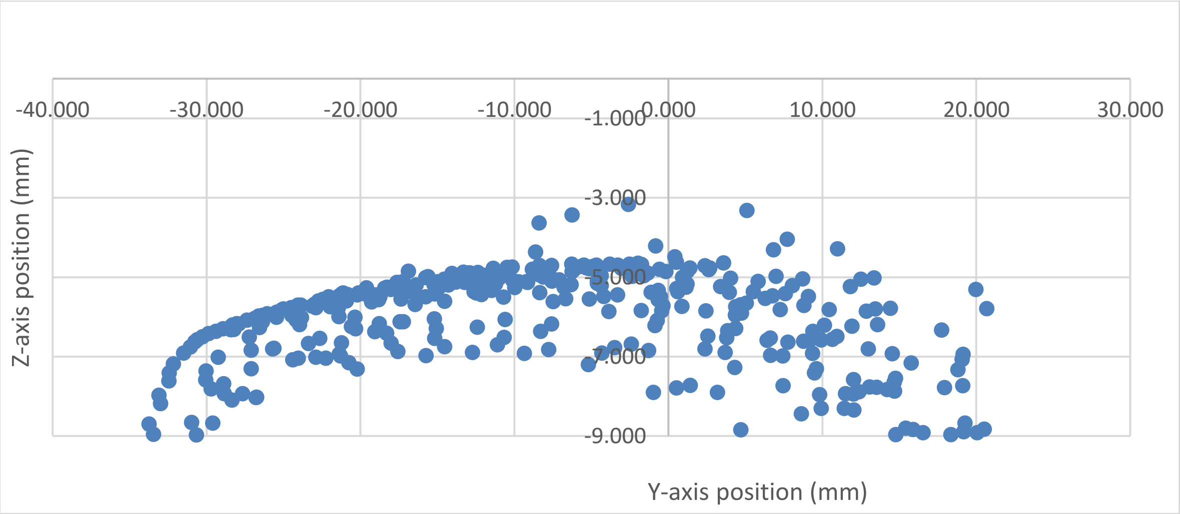

Figure 3.3 Side Profile Curve of propeller on CMM machine

The basic design was initially assessed numerically by utilising the dynamic computer fluid software ANSYS Fluent. In order to evaluate the number setup for forecasting the propellant performance, the numerical analytical results are compared to experimental data [34]. The propeller was then developed using the 2020 version of SolidWorks Software in Figures 3.4 and 3.5 along with the propeller size.

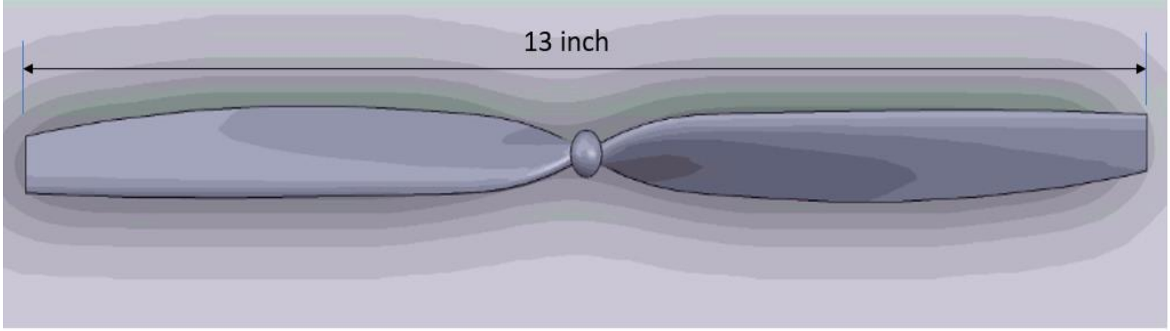

Figure 3.4 Top view of propeller on CAD

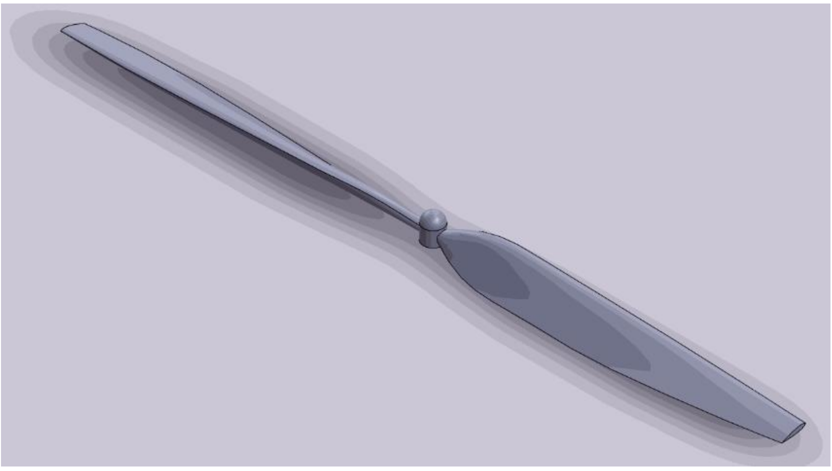

Figure 3.5 Isometric view of propeller on CAD

3.2 Computational Fluid Dynamics (CFD)

Figure 3.6 depicts a typical ANSYS Workbench configuration. First, an analysis in Fluent is performed to determine the pressure distribution of the propeller. After that, the meshing in Static Structural is completed, and the pressure loads from the Fluent are transferred to the structural analysis. Finally, for rotational speeds of 3000 RPM and 6000 RPM, the total deformation, maximum stress, and strain acting on the propeller will be determined.

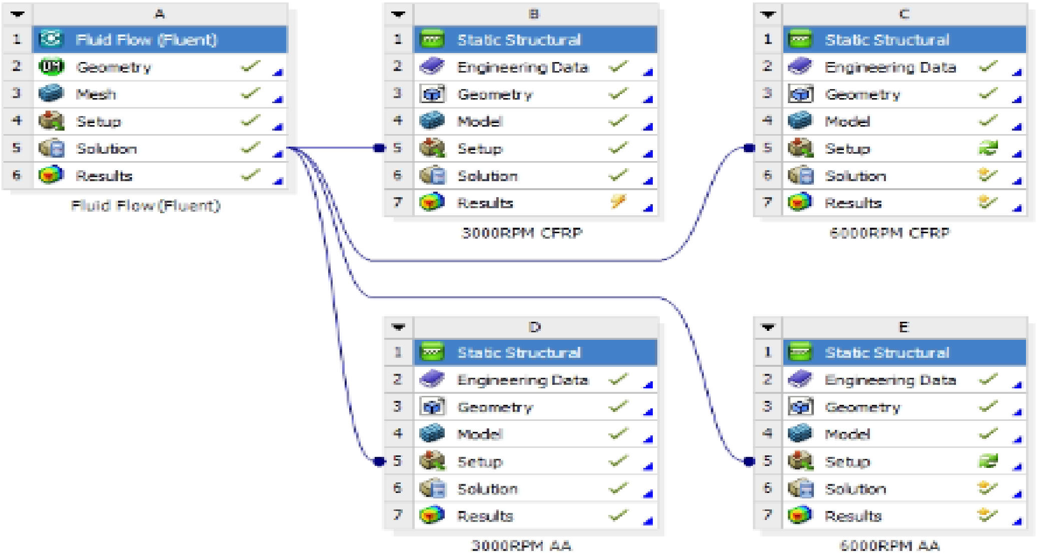

Figure 3.6 ANSYS Software Workbench

3.2.1 Computational domain

The computer domain is separated into two areas: rotating and stationary areas. The rotating area, which consists of a smaller cylinder with a full blade of the propeller, has a diameter of 0.4D and 1,5D.

Meanwhile, the stationary domain has a 2.7D upstream and 4.7D downstream imitation zone, in order to avoid recirculation of flow in the rotating region that would affect the analysis result. Figure 3.7 and Figure 3.8 define and display the domain. The dimension used were taken for the exists experimental analyses done by researchers[35].

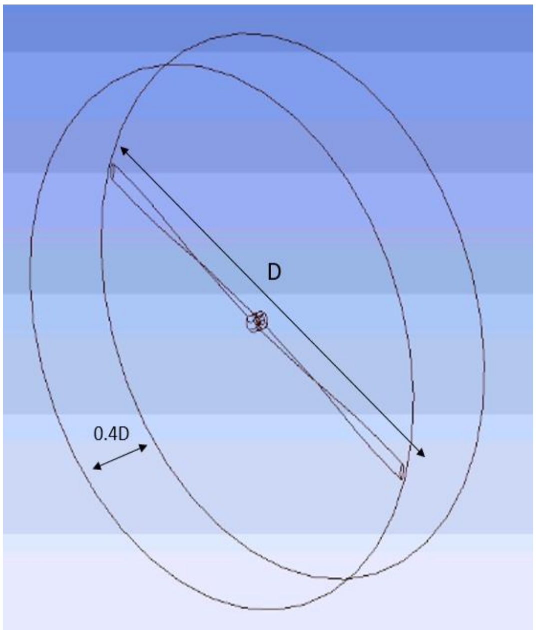

Figure 3.7 Isometric view of rotating domain

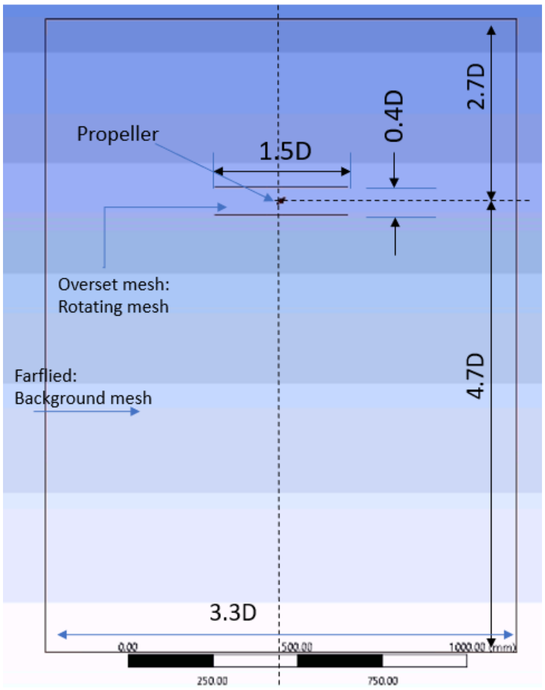

Figure 3.8 Side view of stationary domain

3.2.2 Meshing

In both the moving and stationary regions, the grid is completely tetrahedral unstructured meshing. The use of completely tetrahedral mesh is justified by the fact that the grids can discretize complicated geometry with minimal user interaction.

Furthermore, it takes less computing time and captures the boundary layer condition to enable a successful analysis. The meshing is improved on the blade and progressively enlarged to stationary area to better capture the boundary layer. This is done to ensure that the blade area concentrates more on the validity of the analysis, the convergence rate and the time required for calculating. To guarantee better mesh generation, the meshing was configured with a curvature advance size function and an element size function of 50mm.

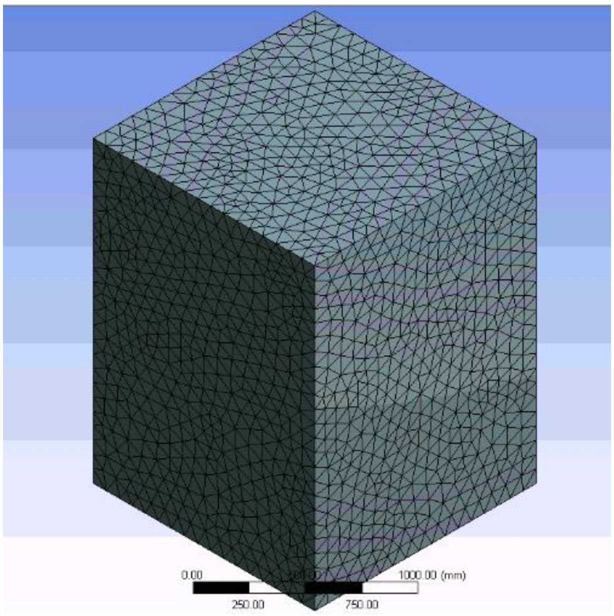

Figure 3.9 Stationary region view mesh

Figure 3.9 shows clearly meshing on the whole stationary domain. The number of nodes created by this meshing is 42,754 and 221,475 elements.

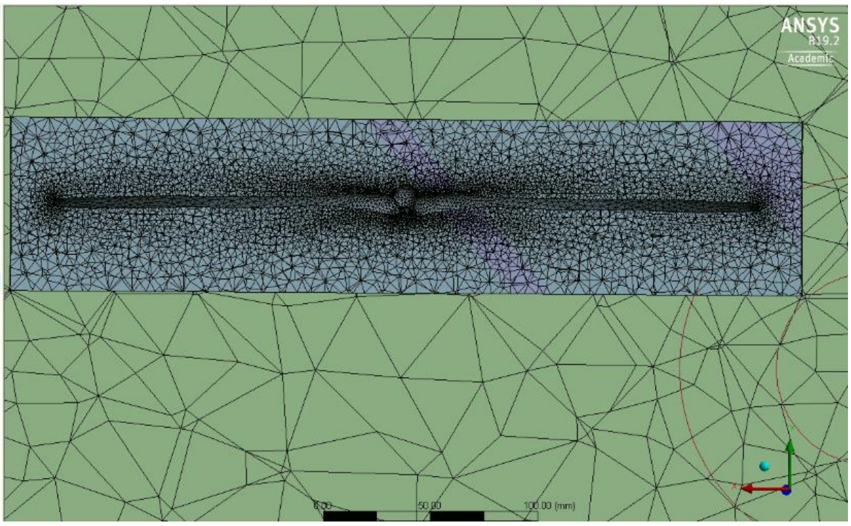

Figure 3.10 Cross-sectional view of Propeller

Figure 3.10 illustrates a zoomed area of the propeller model's mesh generation, which clearly demonstrates mesh generation in and around the propeller model.

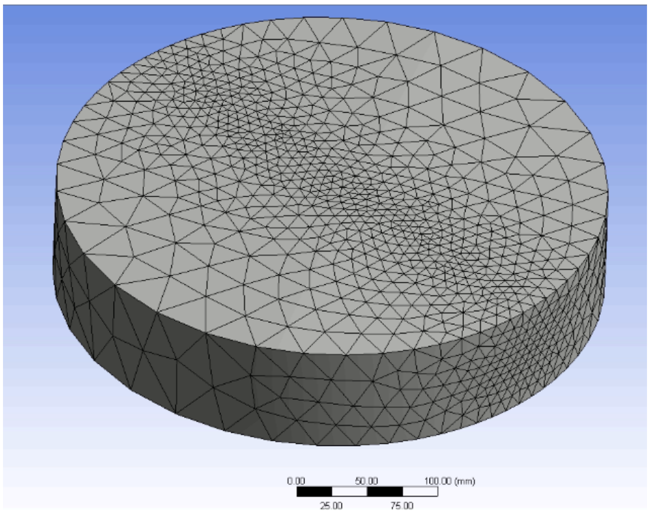

Figure 3.11 Rotating region view mesh

Figure 3.11 illustrates a successful tetrahedral mesh generation on the whole rotating domain including the meshing of propeller inside the domain.

The orthogonal quality, as measured by the mesh metric, may be viewed in this Fluent section. This have been done in order to see the mesh quality so that it will not affect the later results. It also can be one by looking at the skewness and also element quality as shown in Figure 3.12. This option determines the mesh quality by providing a scale between 0 and 1, with 1 being the best[36]. As shown in Table 3.1, this measure is based on the following scale:

Table 3.1 Spectra of Orthogonal Quality Mesh Metrics[37]

Unacceptable	Bad	Acceptable	Good	Very Good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

The number of items within each quality range is depicted in Figure 3.12. According to Table 3.1, 0.09 percent of the components are less than acceptable.

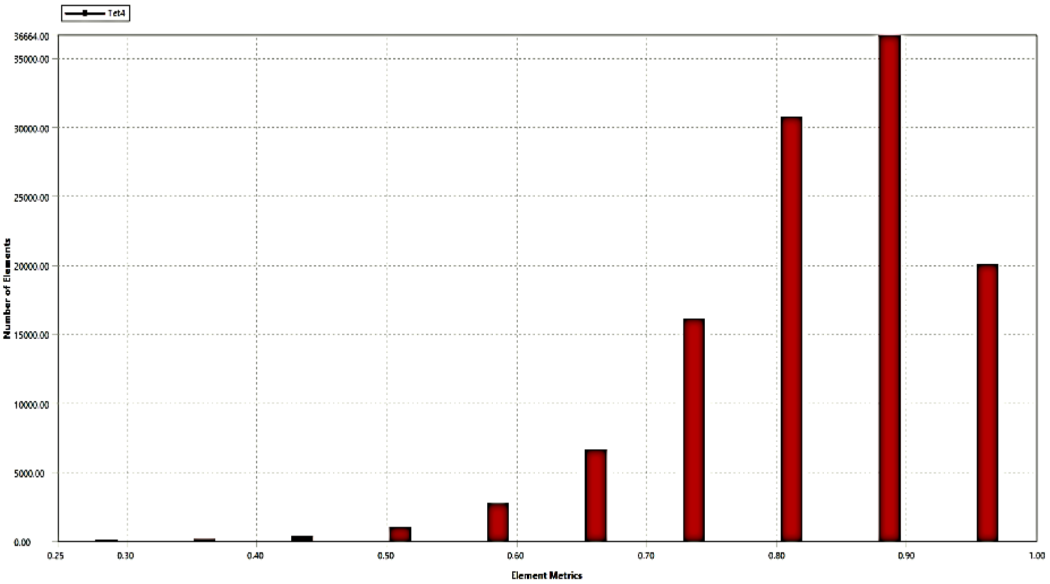

Figure 3.12 Element Quality

3.2.3 Boundary condition

The domain built around the propeller model was subjected to a symmetrical boundary constraint. The domain's inlet and outlet are located, with the inlet velocity and pressure visualized at the inlet geometry and outflow velocities, respectively. It is set as outflow at the outlet boundary condition. Outflow is chosen when no information about the exit flow, such as velocity or pressure, is available prior to the analysis.

Table 3.2 Boundary Condition parameters

Boundary condition	**Inputs**

Solver	Pressure based
Velocity Formulation	Absolute
Time	Transient
Model	Viscous (k-omega SST with curvature correction)
Material	Fluid(air), Solid (Aluminium)

Table 3.2 describes the boundary conditions applied to the propeller model and the rectangular domain of the propeller. In the domain of the model, the inlet and outlet sections are established before introducing boundary conditions. The velocity of air is increased as it travels from the domain's intake to its outlet. The propeller part is designed with a rotatory axis to provide rotation when hitting a target. This domain's rotation was accomplished using Multiple Reference Frames.

3.2.4 Post Processing Setup

The transient-based solver was chosen because it works well in incompressible and moderately compressible flows. Thus, there are fewer terms in the steady-state simulation, it is easier to convergence. These analyses were conducted fully using k-Omega SST turbulence model with curvature correction at fixed rotational speed of 3000 RPM and then followed by 6000 RPM. On the inlet flow domain, the inlet velocity is set as 10 m/s meanwhile turbulence intensity is set to be 1%.

In addition, a Semi-Implicit Method for Pressure-Linked Equations is used to create pressure–velocity coupling (SIMPLE). Momentum and pressure were calculated using the Second Order Upwind method. The gradients were calculated using the First Order Upwind for both Turbulent Kinetic Energy and Turbulent Dissipation Rate, as well as the Least Square Cell-based Algorithm. For this investigation, first order algorithms produced accurate findings. Throughout the experiment, 30 iterations per time step were employed. And, for each angular velocity, the solution stabilisation of the flow near to the rotor took roughly 500-time steps. When the variance in thrust was less than 5%, this stability was accomplished.[35]

3.3 Static Structural Analysis

ANSYS Static Structural Workbench was used in this computational simulation to study the structural analysis of the propeller blade. The sections that follow describe the procedure for determining von-Mises maximum stress, von-Mises maximum strain, and total deformation.

3.3.1 Engineering Data

For this investigation, carbon fibre-reinforced plastics (CFRP) and aluminium alloy are the materials utilised. The mechanical characteristics are taken from [38]. Table 3.3 gives information about the material characteristics.

Table 3.3 Material properties

Properties	**Carbon Fiber Reinforced Plastic (CFRP)**	**Aluminum Alloy**
Density (kg/m3)	1600	2770
Young's Modulus (MPa)	70000	71000
Poisson's ratio	0.3	0.33
Yield Stress at break (MPa)	250	280[39]

CFRP is categorized as composite material which also a lightweight material, corrosion resistance, requires low maintenance and it is easy to make compact design. These characteristics should be taken into consideration so that the propeller will not easily break. There has been the contraption in the aeronautical industry. CFRP material can be used for scenic photography from the raised position which requires the lightweight material. CFRP also presently the most common material used to build