CFD ANALYSIS OF PROPELLER BLADE DESIGN FOR QUADROTOR

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE AWARD OF THE DEGREE OF

MASTER OF TECHNOLOGY

IN
COMPUTATIONAL DESIGN
BY

YAMIKA PATEL
ROLL NO. - 2K15/CDN/16

UNDER THE GUIDANCE OF

Mr. K.Srinivas

DELHI TECHNOLOGICAL UNIVERSITY

DEPARTMENT OF MECHANICAL, PRODUCTION & INDUSTRIAL AND AUTOMOBILE ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY
BAWANA ROAD, DELHI-110042
JUNE 2016

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daulatpur, Bawana Road,

Delhi-110042

STUDENT'S DECLARATION

I, **Yamika Patel (2k15/CDN/16),** hereby certify that the work which is being presented in this thesis entitled **"CFD ANALYSIS OF PROPELLER BLADE DESIGN FOR QUADROTOR"** is submitted in the partial fulfilment of the requirement for degree of **Master of Technology (Computational Design)** in Department of Mechanical Engineering at **Delhi Technological University** is an authentic record of my own work carried out under the supervision of **Mr. K. Srinivas.** The matter embodied in this thesis has not been submitted in any other University/Institute for the award of Master of Technology/certificate. Also, it has not been directly copied from any source without giving its proper reference.

Signature of Student

CERTIFICATE

This is to certify that the report entitled, entitled **"CFD ANALYSIS OF PROPELLER BLADE DESIGN FOR QUADROTOR"** submitted by **YAMIKA PATEL (2k15/CDN/16)**in the partial fulfilment of the requirement for the award of **Master Of Technology Degree in the Computational Design at Delhi Technological University, Delhi** is an authentic work carried out by him under our supervision and guidance.

To the best of our knowledge the matter embodied in this report has not submitted to any other university/ institute for award of any degree.

(Signature of HOD)

Prof. R.S. Mishra

HOD Mechanical

(Signature of Supervisor)

Mr. K.SRINIVAS

Assistant Professor

ACKNOWLEDGEMENTS

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 extend my gratitude to **Prof. R. S. Mishra, Head,** Department of Mechanical Engineering, Delhi Technological University, for providing this opportunity to carry out the present thesis work.

I would like to express my deep and sincere gratitude to my research supervisor, **Ass. Prof. K.SRINIVAS,** Department of Mechanical Engineering, Delhi Technological University, 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. Without the wise advice and able guidance, it would have been impossible to complete the thesis in this manner.

I am also grateful to all the faculty members of the Mechanical Engineering Department for moulding me at correct time so that I can have a touch at final destination and to all my friends, for the help, moral support and encouragement; they had given to me during completion of dissertation work.

I am extremely grateful to my parents and family for their love, prayers, caring and sacrifices for educating and preparing me for my future.

YAMIKA PATEL
M. Tech. (COMPUTATIONAL DESIGN)
2K15/CDN/16

ABSTRACT

The enhancing applications of mini UAVs and MAVs are the reason of increasing keen interest in evaluation of the power of small scale Propellers. When comparing the operating condition of small scale propeller with the propeller of aircraft, it is shown that these small scale propellers operate on the lower value of Reynolds number. As performance characteristics of propeller changes by varying the Reynold's number, so it's difficult to predict the propeller performance at lower value of Reynold's number. So to analyse the effect of Reynold's number on UAVs propeller, study of 11 in. propeller is done on FLUENT on various operating condition. It is shown by the result that there is improvement in the performance of propeller on increasing the value of Reynolds number in terms of thrust coefficient and power coefficient. Performance is also calculated in terms of propeller efficiency.

Computational fluid dynamics (CFD) solver i.e. FLUENT is used to replicate the experimental result and this software has a great computational accuracy and has good initial prediction. Although result is mesh independent, coarse mesh refinement is used along with standard k-Ɛ viscous laminar model. The Multiple Reference Frame model was used to consider the rotation of the propeller towards its local reference frame.

Keywords: UAVs, CFD, k- Ɛ model, multiple reference frame.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1: INTRODUCTION

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. The chamber shape of the airfoil causes the airflow in front of the blade to travel at higher speed.

Based on Bernoulli's principle, due to the acceleration of the airflow, it causes a reduction of static pressure in front of the blade. Meanwhile, lower speed at the back of the propeller causes the propeller to experience higher static pressure. Thus, as the pressure is lower at the front, the aircraft is pulled forward due to the reaction force. 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.

Different methods are available to determine propeller performance, including experimental and numerical analysis. In the experimental method, the propeller blade is tested in a wind tunnel for both static and advancing flow conditions. Meanwhile, numerical analysis adopts three-dimensional computational fluid dynamics (CFD) simulation, utilizing the Reynolds-average Navier–Stokes (RANS) equation. CFD methods have become significant and highly useful tools for propeller design and analysis.

The current study aims to provide an extensive investigation of one method to determine the performance of a propeller blade operating at a low Reynolds number. The current study seeks to perform flow simulations of a propeller blade using CFD software (FLUENT) to determine the thrust coefficient, power coefficient, and efficiency with a variation of the advance ratio. This paper intends to propose a validated method and setup capable of estimating propeller performance using CFD Fluent, thus further research need not perform a CFD setup study and validation prior to a new propeller design analysis.

A propeller may be considered as a rotating wing that assemble Quadrotor aerial robot is an automatic system that is an remote-controlled VTOL (vertical take-off and landing) quadrotor. It can be controlled by varied the speed of the four rotors and there's no need of any mechanical linkages to vary the rotary wing pitch angles as compare to a traditional quadcopter.In Quadcopter commonly two pairs of identical fixed pitched propellers is used; two clockwise (CW) and two counter clockwise (CCW). By varying the speed of each rotor it is possible to produce a required total thrust; to locate for the centre of thrust both laterally and longitudinally; and to create a required total torque, or turning force.

Problem Statement:

- Effect of Reynolds number at different speed of propeller for a given velocity and thus evaluating coefficient of thrust, coefficient of power, and efficiency in order to optimize the results.
- Evolution of optimum operating condition i.e. rotational speed of propeller and wind velocity for the stable Quadcopter movement.
- Study of material for propeller.

1. **Quadcopter:**

Quadcopter is different from conventional helicopters, in terms of variable pitch. Conventional helicopter uses rotor which are able to vary the pitch of their blades dynamically as they rotate around the rotor hub. In the very beginning days of flight, 'quadrotors' were seen the only possible solution of the continuous problems in vertical flight, torque-induced control issues can be eliminated by counter-rotation and the relatively short blades are much easier to construct. These vehicles were among the primary successful heavier-than-air vertical take off and landing (VTOL) vehicles. Although early prototype suffered from poor performance, and latter prototypes needed an excessive amount of pilot work load, owing to poor stability augmentation and restricted management authority.

1.1.Recent developments

In the last few decades, small-scale unmanned aerial vehicles have been used for many applications. The need for craft with larger mobility and hovering ability has light-emitting diode to an increase in Quadcopter research. The four-rotor design permits Quadcopter to be comparatively straightforward in design howeverextremely reliableand manoeuvrable. Res earch is continuing to extend the abilities of Quadcopter by creating advances in multi-craft communication, surroundings exploration, and mobility. If these developing qualities will be combined, Quadcopter would be capable of advanced autonomous missions that are presently impossible with other vehicle.Some current programs include:

- The Bell Boeing Quad Tilt Rotor concept takes the fixed quadcopter concept further by combining it with the tilt rotor concept for a proposed C-130 sized military transport.
- Aero Quad and ArduCopter are open-source hardware and software projects based on Arduino for the DIY construction of Quadcopter.
- Parrot Ar.Drone is a small radio controlled Quadcopter with cameras attached to it built by Parrot SA, designed to be controllable by smart phones or tablet devices.
- Nixie is a small camera-equipped drone that can be worn as a wrist band.

Several camera drone projects have turned into high-profile commercial failures:

- Zano (drone) - a high-profile Kick-starter project to build a Quadcopter-camera drone, Zano failed after delivering only a small fraction of their orders in a partially non-functional state.
- Lily Camera - a start-up attempting to make a Quadcopter-camera drone, sued by the San Francisco District Attorney after they closed down without fulfilling any of their pre-orders.

1.2. Applications of Quadrotor

- **Aerospace:** Airlines and maintenance, repair, and operations contractors use UAVs for aircraft maintenance. In June 2015 Easy Jet began testing UAVs in the maintenance of their Airbus A320s and in July 2016 at the Farnborough Airshow, Airbus (manufacturer of the A320), demonstrated the use of UAVs for the visual inspection of an aircraft. However, some aircraft maintenance professionals remain wary of the technology and its ability to properly catch potential dangers.

- **Defense against UAVs:** The US armed forces have no defense against low-level UAV attack, but the Joint Integrated Air and Missile Defence Organization is working to repurpose existing systems. Two German companies are developing 40-kW lasers to damage UAVs. Other systems still include the Openwork Engineering Skywall and the Battelle Drone Defender.
- **Search and Rescue:**

  UAVs were used in search and rescue after hurricanes struck Louisiana and Texas in 2008. Predators, operating between 18,000 and 29,000 feet, performed search and rescue and damage assessment. Payloads were an optical sensor and a synthetic aperture radar. The latter can penetrate clouds, rainor fog and in daytime or night-time conditions, all in real time. Photos taken before and after the storm are compared and a computer highlights damage areas. Micro UAVs, such as the Aeryon Scout, have been used to perform search and rescue activities on a smaller scale, such as the search for missing persons. In 2014, a UAV helped locate an 82-year-old man who had been missing for three days. The UAV searched a 200-acre field and located the man in 20 minutes. UAVs have been tested as airborne lifeguards, locating distressed swimmers using thermal cameras and dropping life preservers to swimmers.

1.3. Components of UAV

Manned and unmanned aircraft of the similar type commonly have recognizably similar physical components. The main exceptions are the cockpit and environmental control system or life support systems. Some UAVs carry payloads (such as a camera) that weigh considerably less than an adult human, and as a result can be significantly smaller. However it carries heavy payloads, weaponized military, UAVs are lighter than their manned aerial vehicle.

- **Chasis :**

The skeleton of a Quadcopter is that the frame (chassis), some motors and propellers connected to the chassis. Quadcopter frames are available a range of sizes and weight ratings. Most have identical basic look – a imprecise X form. For hobbyists want to mount one thing with further weight like a camera, a sturdier frame rated for additional weight is usually recommended. However, adding a sturdier material generally creates additional weight itself, inflicting itself to require longer propellers and a stronger motor to make the elevation necessary to pull up the load. There's continuously a fragile balance played by the producer between flight speed, mobility, and flight time.

Fig 1 : Chasis

- **Motor:**

The next most important component of a quadrotor is the motor. Motors are rated in "Kv" units, which identify in to the number of revolutions per minute a motor can achieve when a 1V current is introduced to it unhindered. The higher the Kv, the faster the spin of motor. However, faster is not always better because a faster motor spin requires much more power from the battery, causing your flight times to decrease. Higher the revolution per minute, lower will be the life of motor over long period of run time.

Fig 2: Motor

- **Electronic Speed Controller :**

The most common and important components of the quadcopter are- chassis, motors and propellers for lift. But, a machine of this sophistication does have more to it than that. And this part is is an electronics component called an electronic speed control, or ESC. There's need for an ESC for each of the four motors of the quadrotor. An ESC provides the correct modulated current to the motors, that successively produce correct rates of spin for each carry and manoeuvring..

Fig 3: ESCs

- **Propeller :**

Propellers largely impact the speed at that the quadcopters fly, the load that they will carry, and also the speed at that they will maneuver. To impact these varied attributes you'll increase or decrease the length of the propellors and also the pitch of the propeller. The pitch is that the form and slant of the propellor.Longer propellors can do stronger lift at lower revolutions per minute than a shorter propeller, however take longer to hurry up and slow down. on the far side a certain size, they're literally unable to fly.

Fig 4: Propeller

- **Flight Controller :**

Brains of the quadcopter is known as the flight controller. The flight controller is basically the little computer which controls the craft, and interprets the signals the transceiver sends to guide the quadcopter. For the manufacturer of quadcopters, choosing a flight controller is more of a difficult choice in many ways, not unlike choosing from various PC processors in the same power range. Each have various options that each manufacturer wants and may or may not be customizable.

Fig 5: Flight Controller

- **Radio Receiver :**

Job of radio receiver is to send and receive the signal to the flight controller through the transmitter (the remote control). It consists of a component which connects to the flight controller, to receive signals, and a controller to transmit them. There are a lot of very slick receivers which work quite well with standard quadcopter flight controllers. A channel is a control input. If quadcopter had no channels, it would just hover in place. A minimum of 4 channels is required to get the quadcopter to move. 2 channels will be available for each stick on the transmitter. Each additional channel allows to add controls for accessories onto the transmitter.

Fig 6: Radio receiver

- **Battery and Battery Charger :**

To power the quadcopter there is the need of power source, which is typically a LiPo (Lithium Polymer) battery. LiPo batteries use a C rating, which stands forit's capacity to discharge. It has been normaly seen that LiPo battery have "20C". If it is written like 25C 4000mAh LiPo battery, it means that quadrotor can get a maximum of 25C * 4 = 100A (A standing for Amps). The power of the battery is usually dictated by the energy draw required from the ESCs. A lot of battery types can be fully discharaged, but the LiPos have a minimum voltage requirements, which if gone beyond can cause damage to the battery. In most cases it's 3.0 volts, but can vary from battery to battery. This is generally about 80 – 85% usage of your battery. Once past this mark, battery power drops fairly quickly.

Fig 7: LiPo Battery

- **Loop Principles:** UAVs employ open-loop, closed-loop or hybrid control architectures.
- **Open loop**: This type loop provides a positive control signal (faster, slower, left, right, up, down) without incorporating feedback from sensor data.
- **Closed loop**: This type loop in corporates sensor feedback to adjust behavior (reduce speed to reflect tailwind, move to altitude 300 feet). The PID controller is common. Sometimes, feed forward is employed, transferring the need to close the loop further.

- **Propeller Blade**:

Propeller is a machine element which transfers power by changing its rotation motion into net thrust force. It has a profile of aerofoil shape. This profile utilises Bernoulli's principle to leviate the aerial vehicle. As the fluid flows over the blade, a pressure difference in formed between the rear and front side of the blade. This causes a net force to develop in the direction opposite to the gravity.The function of the propellor is to change the mechanical energy from the motor (within the electrical system, the energy hold on within the battery) into thrust. So the propellor directly effects the aircraft's performance and drives the design of the system. Ideally the propellor are going to be designed such that the specified thrust is often produced whereas being driven by the motor and battery when both are in operating at their peak efficiencies, because of the character of fixed-pitch propellers this is often not forever possible.

1.4. Principle of design of Propeller:

A blade-element momentum method is employed to calculate the blade chord and twist distributions that minimize the required power given the tip speed, radius, blade number, advance ratio, section geometry, and desired thrust. This must be combined with additional methods to take structural, noise, and manufacturing considerations into account.

Several performance analysis methods are commonly employed in propeller design.The following paragraphs discuss, in brief, the methods most appropriate to the present design problem in approximate order of increasing fidelity.

- **Blade-Element Momentum (BEM) theory:**

BEM is commonly used due to its relative simplicity and computational efficiency, in the design of and analysis of propeller, in many cases without a large sacrifice in fidelity. Three-dimensional aerodynamic effects can be neglected and the blade is treated as a union of radial blade elements (strips).

Vortex theory assuming a rigid constant-pitch wake, ignoring wake contraction and roll-up, allows prediction of induced velocity vectors at each of these blade elements. Together with the rotational and free-stream velocities, the aerodynamic forces at each station can then be predicted from section data. A momentum loss function approximates tip losses. A significant advantage of this method is the ability to analytically derive the propeller geometry that results in maximum efficiency, obviating the need for an optimization framework unless other objectives must be simultaneously evaluated. However, BEM carries several drawbacks. While quite accurate at low to moderate loading at conventional flight speeds (i.e., advance ratios greater than around 0.5), it is less reliable at other operating conditions (including, notably, static conditions). Typical formulations do not capture any effects of blade sweep, rake, and more unusual geometries (for example, particularly high-solidity propellers). To avoid designing a propeller with a solidity higher than BEM theory applies to, the solidity must be artificially limited.

- **Vortex methods**

It compose a special case of the Euler equations, assuming an incompressible potential flow with the wake vorticity confined to a finite number of nominally helical vortex elements. (Compressibility corrections are often included.) Wake contraction and roll-up can be modelled, and viscous drag is often approximated by 2D methods. Vortex methods can be classified into prescribed and free wake methods. Prescribed wake methods, employing an a priori specification of the position of the vortex elements from experiments, are simple and computationally efficient but are limited by the availability of relevant experimental data. Because relevant experimental data was not available for the present conditions, prescribed wake methods are not practical for this analysis. Although free vortex wake methods require a large amount of computational time — about two orders of magnitude higher than BEM methods — they do not require an a priori specification of the vortex element positions. However, they do require various wake parameters (for example, the vortex strength and roll-up distance) to be set by statistical methods and the lack of relevant experimental data precludes the use of free wake methods as well.

- **Euler and Navier-Stokes methods**

These are the highest-fidelity analysis methods considered; these methods can model wake contraction and roll-up, compressibility effects, three-dimensional aerodynamic effects, and, with Navier-Stokes methods, three-dimensional boundary layers. However, a solution can easily require over four orders of magnitude more computational time than a BEM solution. Generation of the computational mesh for a propeller at static conditions, when the wake and tip vortices are in close proximity to the blades, is a complex and time-consuming endeavour, particularly if the code employs a structured mesh; unstructured meshes, while potentially far simpler to create, generally require a further order of magnitude of computational time. Additionally, there is no convenient method to employ these CFD methods to produce an optimum design; instead, the analysis must be coupled with some sort of optimization framework, further increasing the computational expense.

Therefore, a blade-element momentum method was chosen to prescribe propeller geometry with optimal propulsive efficiency. Although this choice abandons hopes of obtaining very accurate performance predictions, it facilitates a quick and robust determination of the propeller configuration.

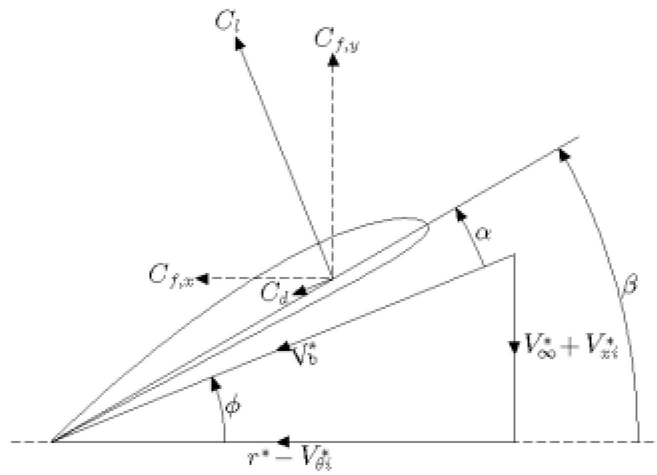

Fig 8: Flow geometry and force components for a blade element.

1.5. Propeller Design:

- **Propeller Operation:**

Blades of full scale propellers could be applied to the design of propellers for UAVs since the aerodynamic data for the airfoils is available at the correct Reynolds's number for the UAV flight regime.

In 1998 Lorenz prepared a thesis and gives detailed information about the design and testing of propellers through wind tunnel for the "Stationary High Altitude Relay Platform" (SHARP) aircraft. It was found that it had been possible to provide efficient propellers that functioned as designed; but no structural analysis was enclosed within the design program.

In 2012 a thesis presented by Wall demonstrates the design of optimized propellers for electrically power. Propellers are used since the earliest tries at flight to drive power-driven craft and can be the first different to turbofans for the predictable future. Their operation is explained by the classical momentum theory**.** Classical momentum theory assumption that thrust is made attributable to a distinction within the static pressure across an infinitely thin mechanism disc. It's more assumed that:

1. The velocity and pressure over the disc are constant and uniform;
2. The air flows straight through the disc while not rotation;
3. The flow over the propeller is forced to a well outlined stream tube; and
4. The flow is incompressible.

- **Design Methods:**

➢ **Early Propeller Design Approaches:**

Before the end of the 2nd World War propellers, were designed using an empirical approach. Propellers, that were found to possess excellent performance characteristics throughout wind tunnel and flight testing, were scaled and changed as needed to suit a selected UAV, RC craft and its engine. Example of this type of approach is registered in "National Advisory Committee of Aeronautics" (NACA) Technical Note, which permits for a propeller to be designed for a lightweight craft by calculating two coefficients,

The first coefficient J, is the advance ratio of a propeller and is defined as:

$$J = \frac{V}{n \times D}$$

Where V = aircraft speed

n = revolutions per second

D = propeller diameter.

The second coefficient is defined as speed power coefficient:

$$\sqrt{\frac{\rho V^5}{P n^2}}$$

Where, P = power

ρ = air density

The pitch of the propeller, or the theoretical ratio of its onward motion per revolution, was constant for the whole blade, in theory leads to a constant angle of attack for every section of the propeller blade. The speed power coefficient and advance ratio are still used for the preliminary fabrication of propellers for craft if the propeller is to be form on an existing style. This approach powered by internal combustion engines. Borst analysed that the methods used for designing of the red UAVs. Wall's approach was extensively used through the 2nd world war during pre-war propeller blade styles were scaled and used on a range of wartime craft.

- **Traditional Methods of Design:**

Theories existed during pre-war for the design of optimum propellers, beginning with Betz's theory in 1919. Betz's theory demonstrates that the foremost efficient propeller can minimize induced power losses by having one and the other a constant induced velocity and a constant wake pitch in the propeller's slipstream.

- **UAV Propeller Design:**

A report published by Borst authored in 1978 demonstrated that the design of UAV propellers ranges from 24" -36" in diameter and optimized not only the propeller blade to suit the performance of the motor but also included optimization of aerofoil. Based on simulations, the optimized propellers performed better than the baseline propeller and it was concluded that there were definite differences in blade plan form between climb and cruise propellers. However in this approach structural strength or propeller performance had not considered in other phases of flight.

- **Airfoils for Propellers:**
- **Propeller Airfoil Series:**

Now days a number of airfoils and airfoil series have been used for propeller blades. Before World War 2 most of the propellers designed used airfoils with a lower flat surface similar as the Clark Y airfoil and, to a lesser extent, the RAF. One of the major cause for their use is that the "lower flat surface allowed for measurements of blade angle to be made during manufacturing and maintenance”. Both airfoils do have suitable aerodynamic attributes such as a high maximum lift coefficient and benign stall characteristics. However, testing by NACA showed that in compressible flow the Clark Y did not function well which lead to the use of the two “classic” propeller airfoil families the” NACA 16” series and the “NACA 65” series. It should be noted that the use of the ‘Clark Y’ did not end with the introduction of these airfoil families as modern propellers are still designed using them.

In the mid late 1930s early 1940s both the “NACA 16 and 65” airfoil families were introduced and this series of airfoil offer less drag in ‘compressible flow regimes. The advantage of 65 series airfoils is that it has the drag bucket or a reduced drag over a narrow range of lift coefficients because of laminar flow around the airfoil. This is helpful, especially for a fixed pitch propeller, as it increases the range of advance ratios the propeller can perform near its maximum efficiency.

In the mid late 1970s “ARA-D” series is introduced, which was created by the “Aircraft Research Association”. ARA-D airfoils provide a larger lift coefficient for better take off performance and have a thicker form designed for a stiffer, easier to manufacture blade. These airfoils were used for the composite propeller blades flown on a range of transport craft propellers like the Bombardier.

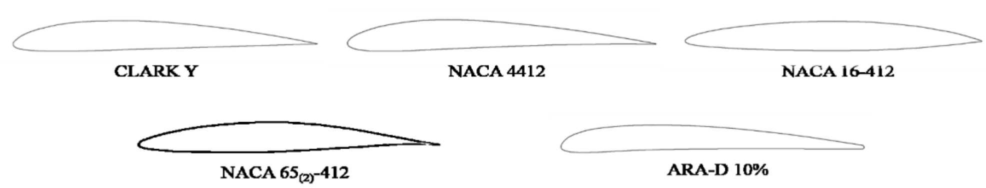

Fig 9: Airfoils commonly used for propeller blades

For an airfoil having a low thickness-to-chord ratio (t/c) is used near the tip of the propeller blade to obtain a thin cross-section and for its higher critical Mach number. The thickness of the blade then increases with the span as is required for adequate structural strength and stiffness, until the root where the blade becomes very thick. For a 'fixed-pitch propeller' to avoid stress concentrations, the increment in thickness is required to evenly blend the blade into the hub.

In the early 1930s, studies of 'National Advisory Committee of Aeronautics' (NACA), showed that if the profile drag and zero-lift, the lift-curve slope ,camber and camber position of an airfoil are held constant, Angle of Attack changes proportionally to the thickness-chord ratio.

- **Airfoil Data:**

The aerodynamic property of the "NACA 44xx series", as well as other propeller airfoils such as "NACA for the 16- and 65- series", is extensively available. Three different methods for gathering sufficient airfoil data at low Reynolds's numbers for design purposes were examined as discussed below.

- **Correction Factors:**

In the 1970s airfoil data available for propeller design at very low Reynolds's numbers. Due to this reason correction factors were developed by Jacobs and Sherman, based on trends allowing for aerodynamic data obtained at a higher Reynolds's number to be scaled down for lower Reynolds numbers.

- **Use of XFOIL for Predicting the Lift and Drag Characteristics of Airfoils at Low-Reynolds Numbers:**

XFOIL is one of the most widely used tools for evaluating airfoils, specially at low Reynolds's numbers. XFOIL is a CFD program using panel methods. It was created in the beginning of 1980s specifically for low Reynolds number airfoil data for the MIT Daedalus human powered aircraft. On account of its creation it has continually been upgraded and validated by forming it a very mature and widely used analysis tool. When it is possible to interface XFOIL with MATLAB, Data was collected from XFOIL for several airfoils and a MATLAB function was written to interpolate the tabulated data for an airfoil within the range of thicknesses and Reynolds's numbers. For instance, if the properties of a NACA 4414 airfoil at a Reynolds's number of 200,300 are required, they would be calculated by the interpolation of the properties of the NACA 4412 and NACA 4415 airfoils at a Reynolds numbers of 75,000 and 300,000. This method works efficiently well with tabulated drag predictions being close to those predicted by XFOIL of the actual drag polar of the airfoil.

- **Predicting the Lift and Drag Characteristics of Airfoils by Hybrid Approach at Low-Reynolds Numbers:**

There was some problem to assume that the properties of the airfoil at Reynolds's numbers below 75,000 would not change further. This is happening because below a critical Reynolds's number (in the range from10^4 to 10^6) the flow will remain laminar and the drag will increase due to the effect of viscosity, and above this critical Reynolds's number the drag will decrease as the flow becomes turbulent. To reimburse for this drawback of XFOIL, a correction factor was used.

The correction factors were meant to correct data from a Reynolds's number of 9 x 10^6 to a Reynold's number of 50,000-300,000. The original factors did not work well and that's why new ones were created using the same technique and using the data for the NACA 4409, 4412, and 4418 airfoils.

- **Design and Testing Requirements for UAV Propellers:**

The key applicable necessities relevant to the design and testing square measure summarized below:

- **Design:**

1. The propeller might not have options or characteristics that will build it unsafe for the uses in normal operational conditions.

2. The quality and sturdiness of materials utilized in the propellor should be established on the basis of expertise, tests, or both and account for environmental conditions expected in commission. The design values of properties of materials

should be appropriately associated with the foremost adverse properties declared within the material specification for applicable conditions expected in service.

3. The utmost stresses developed within the propeller might not exceed values acceptable to the administrator considering the particular type of construction and also the most severe operational conditions.

- **Testing:**

1. The hub and root of the blade should be tested for an hour at a load corresponding to double the utmost centrifugal load to that the propeller would be subjected throughout operation at the utmost rated rotational speed without proof of failure, malfunction, or permanent deformation that will lead to a significant or risky propeller effect.

2. A fatigue evaluation of the propeller must be conducted to analyze the hazardous propeller effects due to fatigue will be evaded throughout the intended operational life of the propeller on either the intended airplane. Fatigue limits must be setup by tests, or analysis based on tests, for the propeller taking into account.

3. Endurance tests of the propeller system must be made on a representative engine without any proof of failure or malfunction.

- **Other Design Requirements :**

Aside from meeting regulatory requirements, propellers need to be able to function in their operation environment for their intended lifespan. A few relevant design criteria were identified as:

Survivability & Reliability: Minor impact damage from the bottom, stones, and alternative scrap mustn't have an effect on the continuing safe flight for a minimum of that mission. The maximum allowable stresses ought to be chosen considering service damage and producing defects.

Inspect ability and Maintainability: The pre-flight supervision of the blade should consist of nothing more than cleaning the propeller and analysing for signs of damage. It should be easy to replace in the field.

1.6 Materials and Manufacturing:

- **Description of Materials for Propellers:**

Propellers were initially made of wood but since the 1920s it have been made from steel and aluminium and since the 1970s it have been made from advanced composite materials. Now day's latest composite materials have excellent mechanical properties.

1 Wood: It has some characteristics that create it a most suitable material for propellers however it has some vital drawbacks. It offers a high strength-to-weight ratio and its ultimate strength is not significantly affected by fatigue, it is due to its high internal friction, or hysteresis, which is so effective at damping vibrations.

2 Metal: Early in the 1920's, Metal propellers were developed as an alternative to wood propellers. Due to the comparatively high weight of the blades, it is not used for RC aircraft or small UAVs.

3 Composites and Plastics: As a recent advancement in aviation, composite materials are seen and it has a greater advantage over other propeller material. Development of "Vertical Take-Off and Landing" (VTOL) aircraft, required propellers that had the capability of producing high amounts of thrust and had strong enough to cope with the high harmonic forces that the blades would experience when the propellers were changing its orientation from a vertical to a horizontal and vice-versa. Composite blades eliminate the shape restriction enforced by wood or metal, leading to thinner and more efficient propellers and permit excellent aviation performance.

4 Reinforced Plastics: APC Propellers is one in of the most important makers of propellers for SUAV and RC craft is, that produces injection-moulded propellers made up of a fibreglass-reinforced nylon. APC instead uses comparatively long fibre glass strands that stay oriented along the span of the blade. This provides a higher load path to the hub, leading to a stiffer mechanical device blade a lot of capable of withstanding vibrations caused by the motor or engine or attributable to aero elastic effects. Bolstered plastics supply a high strength-to-weight quantitative relation that may be a terribly fascinating quality for a material used for the manufacturing of propeller blade.

- **Propeller Performance :**

Movement of aircraft through air is achieved by a force known as Thrust. There are various kinds of propulsion systems which can develop thrust in many ways, though it generated as a consequence of Newton's Third Law. Propeller is one of the major sources of the propulsion system in any of the aerial component and hence its function is to move the aircraft through the air. It consists of two or more blades connected together through a hub. The hub helps to attach the blades to the engine shaft.

The blades are made in the shape of an airfoil like wing of an aircraft. When the engine rotates the propeller blades, the blades produce lift. This lift is called thrust and moves the aircraft forward. Most aircraft have propellers that pull the aircraft through the air. These are called tractor propellers. Some aircraft have propellers that push the aircraft. These are called pusher propellers.

Description:

Leading Edge of the airfoil is also known as the cutting edge that slices into the air. As the leading edge cuts the air, air flows over the blade face and the camber side.

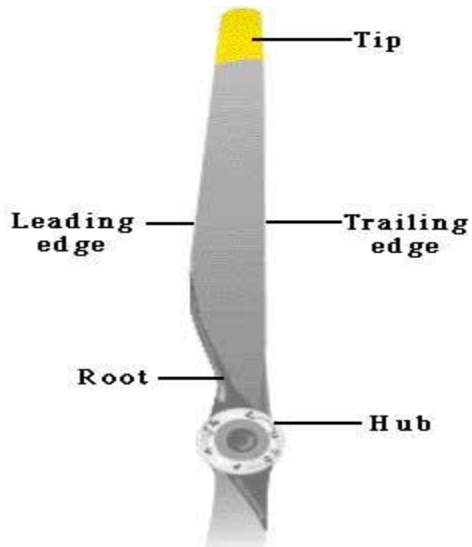

Fig10: Leading Edge of the blade

Blade Face is the surface on the propeller blade that conform to the lower surface of an airfoil or flat side, ones called Blade Face.

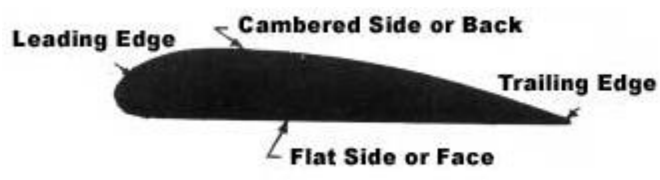

Fig 11: Cross section of a propeller blade

Blade Back / Thrust Face: The curved surface of the airfoil is known as the blade face.

Blade Shank (root) is the section of the blade which is nearest to the hub.

Blade Tip is the peripheral point of the blade, farthest point from the hub.

Blade Angle: It is the angle form in between the chord of a propeller or rotor blade and a plane perpendicular to the axis of rotation, its value is changing along the span and reducing from root of the blade to tip because of twisting of the blade. The blade angle along the length of the blade is not the same. The reason for placing the blade element sections at different angles is because the various sections of the blade travel at different speed. “Thrust is produced by the propeller attached to the engine drive shaft. While the propeller is rotating in flight, each section of the blade has a motion that combines the forward motion of the aircraft with circular movement of the propeller. The slower the speed, the steeper the angle of attack must be to generate lift. Therefore, the shape of the propeller's airfoil (cross section) must change from the center to the tips. The changing shape of the airfoil (cross section) across the blade results in the twisting shape of the propeller”.

Blade Element is the airfoil sections joined side by side to form the blade airfoil. These elements are placed at different angles in rotation of the plane of rotation.

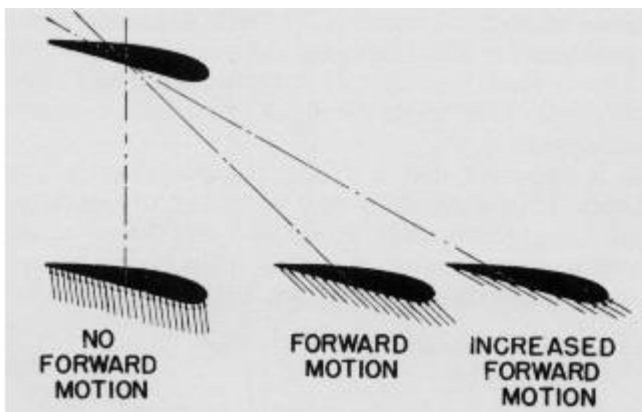

Fig 12: Relative Wind with respect to propeller blade

Relative Wind is the air that strikes and pass airfoil is driven through the air over the airfoil as the.

Angle of Attack: Angle of Attack is the angle between the chord of the element and the relative wind.

Pitch refers to the distance a spiral threaded object moves forward in one revolution. As a wood screw moves forward when turned in wood, same with the propeller move forward when turn in the air.

Forces and stresses acting on a propeller in flight:

The **forces** acting on a propeller in flight are :

1. **Thrust** is the air force on the propeller which is parallel to the direction of advance and induces bending stress in the propeller.

2. **Centrifugal force** is caused by rotation of the propeller and tends to throw the blade out from the centre.

3. **Torsion or Twisting forces** in the blade itself, caused by the resultant of air forces which tend to twist the blades toward a lower blade angle.

CHAPTER 2: LITERATURE REVIEW

1. **Pradeep Shetty et al. (2016) [1]** This paper predict and correlate the mass flow rate of propeller type axial fan used in condenser unit using Computational Fluid Dynamics (CFD) technique. Computation is done for the axial fan speed of 820 rpm for the steady state condition using moving reference frame approach. The flow rate is correlated with the test results to validate the CFD modelling approach. The correlation level found closer with tested results, hence which will help to improve the futuristic model during conceptual design itself.

2. **Brian Rutkay et al. (2016) [2]** the aim of this research was to develop a process for the designing and manufacturing of propellers for small UAV. This aim was achieved by creating a computer program to design a propeller that meets user-defined aircraft performance requirements within the limitations of the electric motor capacity, user-selected materials, and manufacturing techniques. The feasibility of 3D printing technique in making flight worthy propellers was checked through material testing, manufacturing trials, and by testing the propellers in a wind tunnel. The results shows that the propeller performance generated approximates the predicted design thrust but the efficiency and power consumption could not be accurately measured.

3. ***Prof. A.V. Javir et al. (2015) [3]** this paper focuses on the aerodynamic effects of quad copter. It addresses all the aspects of quad copter ranging from mechanical design to electronics used. It provides backup to the selection of different components with the help of various formulas from research papers. It also gives clear results with respect to weight of components and their corresponding costs. Along with this, finite element analysis is done on the frame so as to sustain the loads generated in the vehicle and concluded that small deformation occurred on the centre plates are safe and within the limit.

4. **Karna S. Patel et al. (2014) [4]** In this paper the lift and the drag forces are obtained using CFD which are also experimentally obtained using wind tunnel testing. The wind tunnel experiment is a laborious one and also costs more than CFD technique for the same problem. Thus first the data was obtained through analytical method then it cross checked by experimental testing. The analysis of the two dimensional subsonic flow over a NACA 0012 airfoil at various angles of attack and operating at a Reynolds number of 3×E06 is presented. The CFD simulation results show close agreement with those of the experiments, thus suggesting a reliable alternative to experimental method in determining drag and lift.

5. **Mark W. Mueller et al. (2014) [5]** Here the analysis of quadrotor is done under a condition that each propeller gets out of service respectively. This paper has sequentially tried to establish the modalities/capabilities of the quadrotor in terms of its height, position, time of flight etc with limited no of propeller in operation, where one, two and three propellers get down and out. . The stability is analysed using the deviation or the tilt in the primary axis with respect to the stable position control axis, along with the changes in altitude are used to design the quadrocopter with proper validation from experimental data.

6. **S. M. Mahbobur Rahman et al. (2014)** [6] in this paper study of fluid flow characteristics for the flow over a propeller used in vertical takeoff and landing (VTOL) radio controlled (RC) aircrafts. Simulation investigation has been conducted through SolidWorks flow simulation using a propeller model. Design can be modified

7. **Shlok Agarwal et al. (2014) [7]** In this paper structural design and manufacturing of twin rotor AVs has been detailed down. Individual component parts have been thoroughly describe and studied which would help

in development of trirotor type unmanned aerial vehicle. Here the cost and dimension constraints are the main design feature that would provide for practical applications. Thrust vector is directly explained and illustrated. On the basis of rolling pitching and yawing a stable rotor design and propeller orientation has been proposed for the trirotor system.

8. **Mr. Kalpesh N. Shah et al. (2014) [8]** in this paper payload capacity of UAVs is the area of research. Additional payloads on the accounts of design improvement lead to more functionality in a UAV. In order to do such, different basic design structures of UAVs were compared to developed enhanced payload capacities and choose a quadrotor. On rough estimate a quadrotor has a pay load capacity around 4 kg which gives it additional features for military and other applications. Calculation of aerodynamic drag-lift forces has been done. The stresses generated in the body of quadrotor is not more than 22.5 MPa in "Box" sectional chassis and 22.2 MPa in "C" Sectional chassis as per the practical testing. To validate these results, the stress analysis of two types of body of quadrotor ("Box" and "C" section) is carried out in ANSYS static structural solver. As per the Static Structural solver the stress generated are 15.4 MPa and 16.97 MPa in "Box" and "C" sectional chassis respectively.

9. **S. Subhas et al. (2012) [9]** the paper predominantly focuses on CFD analysis of the propeller, for say ship propeller is used here for CFD. Cavitations and fluid dynamics solution are the parameter are the under consideration. Fluent solution has been implemented here such that the local pressure reduces below surrounding pressure in order to simulate cavitation result. The CFD result is used to establish the design criteria only after experimental validation of the result. The optimization of vibration in flow domain and burrs is done using the above analysis. Variant boundary conditions, fluid computational domain

and grid refinement is considered for improvement in mesh generation methods.

10. **John B. Brandt et al. (2011) [10]** In this paper a full scale research on propeller has been done which has earlier been done for aircraft but with growing needs of UAVs, it has highly become imperative to perform such analysis on it too. UAVs use propeller that operate under conditions of low Reynolds number ranging from 5000 to 100000 based on the propeller chord at 75% propeller blade station. Experiments were done at university of Illinois to evaluate the efficiency of propeller under these conditions. 9 to 11 inches was the diameter range for set of 79 propellers under the test. In test conditions at constant rpm. of propeller varied velocities of wind tunnel were used to sweep over a vast range of advanced ratios until a zero thrust condition was reached i.e. windmill state. To study the effect of variable low Reynolds number over the propeller, a range of 1500 to 7500 rpm of propeller revolution, depending upon the nature of propeller was the velocity set used. The results of efficiency varied from 0.68 for best condition to 0.28 under worst possible scenario, implicated that propeller design has a huge impact over the aircraft and UAVs operations.

11. **W.Shawn Westmoreland et al. (2008) [11]** in this paper propeller modelling using CFD is done to simulate fluid flow condition over it. Research with the support NLOS-T (Non Line Of Sight Transportation) is published here. The NLOS-T is a conceptual vehicle that will be canister launched, deploy wings and control surface, and then fly to a destination within approximately 15-20 minute of the launch point. The scientific investigation was undertaken over the propeller i.e. spinning geometry of UAV, excluding the rest of air frame structure.

12. **Dr K.C. Wong et al. (2006) [12]** This paper includes the instrumentation of off-the-shelf Remote-Controlled flight platforms, the design development and operation of flight research platforms, innovative airframe concepts, of Micro Air Vehicles (MAVs), and exploring commercial applications for UAV.

13. **Bruno Herisse et al. (1998) [13]** In this paper a controller of nonlinear nature to control flight and touchdown operations of a vertical take-off and landing unmanned aerial vehicle is presented. The VTOL is a rigid body equipped with only a camera and IMU circuit which moves over textured flat plane. Stability of hovering flight and automatic landing of the unmanned aerial vehicle using optical flow feedback system is achieved. Results show that the proposed control strategy is a successful one.

14. **AbdellahMokhtari et al. (2000) [14]** In this paper, a nonlinear quadrotor unmanned aerial vehicle is combined with a feedback linearised controller. The most unfavourable case of control is analysed by introducing Actuator saturation and by constrain on state output. Parameter uncertainties and external disturbances causes performance issues of a controller which is shown through a simulation study. The results shows that the system becomes robust depending upon the selection of weighted functions.

15. **Peter G. Ifju et al. () []** This research shows the effects of windy environment and unsteady aerodynamics over the micro air vehicle. The introduction of flexible wing on this vehicle improves its flight stability and durability and its performance in adverse weather conditions. The flexible wing has a operating range wingspan of 18 inches to 5 inches. In addition with the concept of flexible wing, aerodynamic assessment, and flight data analysis, fabrication method are also shown in this research.

16. **T. A. Maitre et al. (1991) []** Circulation around **liftin**g body is modelled through a non linear function in this paper. The circulation potential is well defined inside the body and evaluation is made for the flow over the body. This method is well suited for the flow analyses over the propeller of UAVs which are consider to be extremely thin. This method is applied over non cavitation fluid flow condition in a marine propeller. Here the researcher deeply tried to solve the problem pertaining to Joukowski condition, problem of propeller mesh, wake region analysis and the influence of wake region over the hub is numerically analysed.

17. **Jonathon Bell et al.** This project has undertaken the task of designing and constructing a test rig for the purpose of experimental analysis of SUAVs coaxial rotor system. The focus is led on the importance to highlight aero mechanical components and variable that dictates the co-axial flight performance with the aim of optimizing the propulsion system to be used for HALO coaxial SUAVs manufactured by Middlesex University. The chief contribution of the paper is to design and optimize the co-axial configuration with respect to motor and propeller variation. Inter rotor spacing has been detailed out with the help of H/D ratio that is in between 0.41 to 0.65.

18. **Pierre Jean Bristeaue et al** Here different models of quadrotor UAV have been studied. Aerodynamic effects of the propellers and their interaction with rigid body motion of the UAV has been modelled. The main assumptions are the twisting of the propellers in such a way that the local angle of attack is constant along the blades in stationary flight and, secondly the local induced velocity is invariable along the blade, these conditions are used to optimize the hovering rotor and hence conclude that the dynamics of the UAV is prominently dependent on the flexibility propeller design (location of centre of gravity) thereby playing important role in designing close loop controller**.**

19. **S. NorouziGhazbi et al. (2016) [3]**This paper reviews and gives an overview of various works done on quadrotor with dynamic modelling and control features

CHAPTER 3: METHODOLOGY

FLUENT is part of the ANSYS is one of the module, is a sturdy software for "computational fluid dynamics". It helps in achieving the optimization of one's product's performance in a fast manner. This software confirms fast modelling capabilities and provides more optimized results as compared to any other software's due to its multi physics applications. Fluent software of ANSYS has wide capabilities of physical modelling required for flow of a model, transfer of heat, turbulence and for industrial use- ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from flow of blood to semiconductor manufacturing, and from design of the rooms to plants of waste water treatment. Fluent captures a wide range that may include models like aero-acoustics, and modelling the in-line cylinder combustion, turbo-machinery etc.

3.1 Types of FLUENT solvers:

- Based on pressure
- Based on density

Pressure based solvers:

In PBS, the key variables are "momentum and pressure". Continuity equation is rearranged to derive the algorithms for pressure velocity coupling.

Pressure based solvers has two type of algorithm:-

- Segregated Solver: It sequentially solves for 'momentum and pressure'.
- Pressure Based Coupled Solver (PBCS): It simultaneously solves for 'momentum and pressure'.

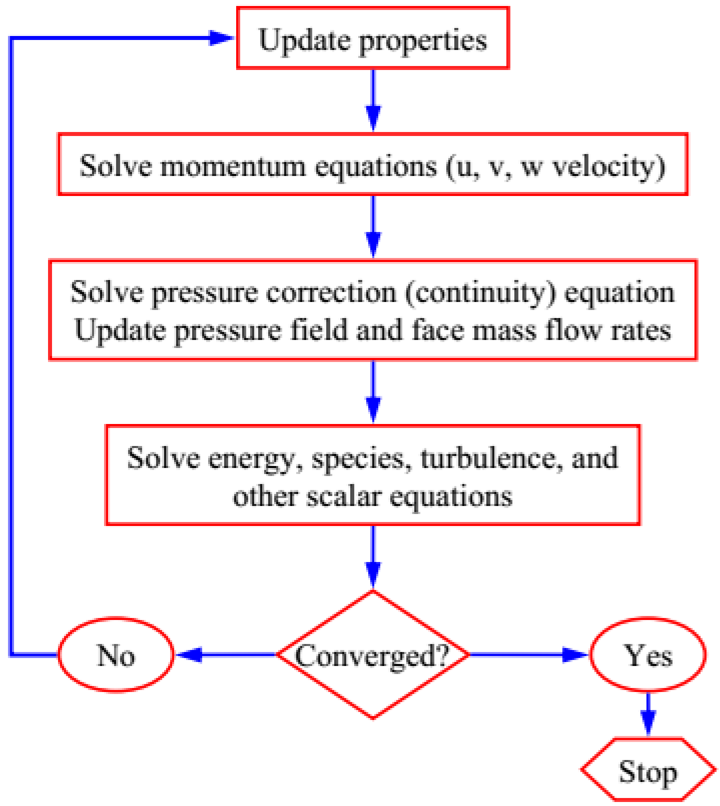

Fig 13: Flow Chart for Pressure based Coupled solver

Density based solvers:

It solves continuity equations, energy, and equations of momentum, and if needed, its vector form can also be solved. Pressure can be derived from equation of state. Extra scalar equations can be solved in a separate manner.

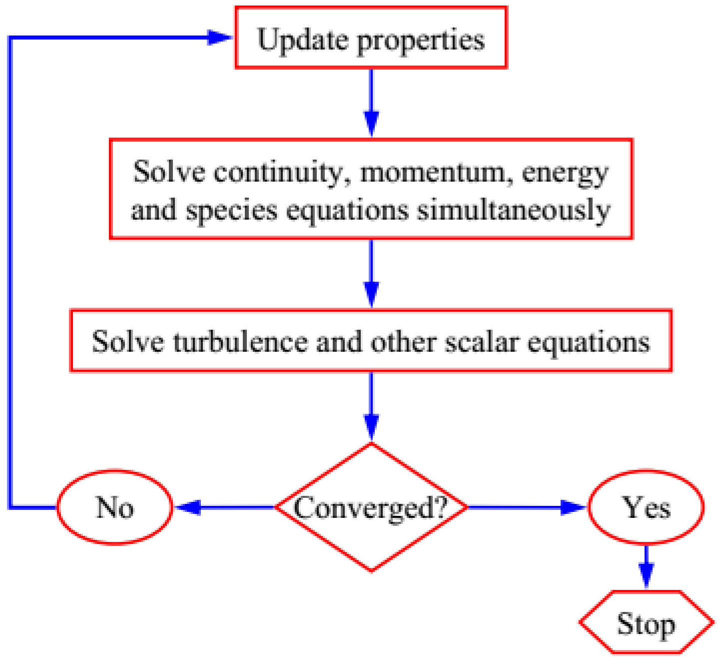

Fig 14: Flow chart for DBS

Density based solver also have the capability to solve the equation either by implicit approach or explicit:-

1.) Implicit: Utilizes single point "Gauss Seidel method/ symmetric block Gauss-Seidel / ILU(Incomplete Lower Upper decomposition)" method to find a solution for variables.
2.) Explicit: Utilizes multi-procedure "Runge-Kutta explicit time integration method".

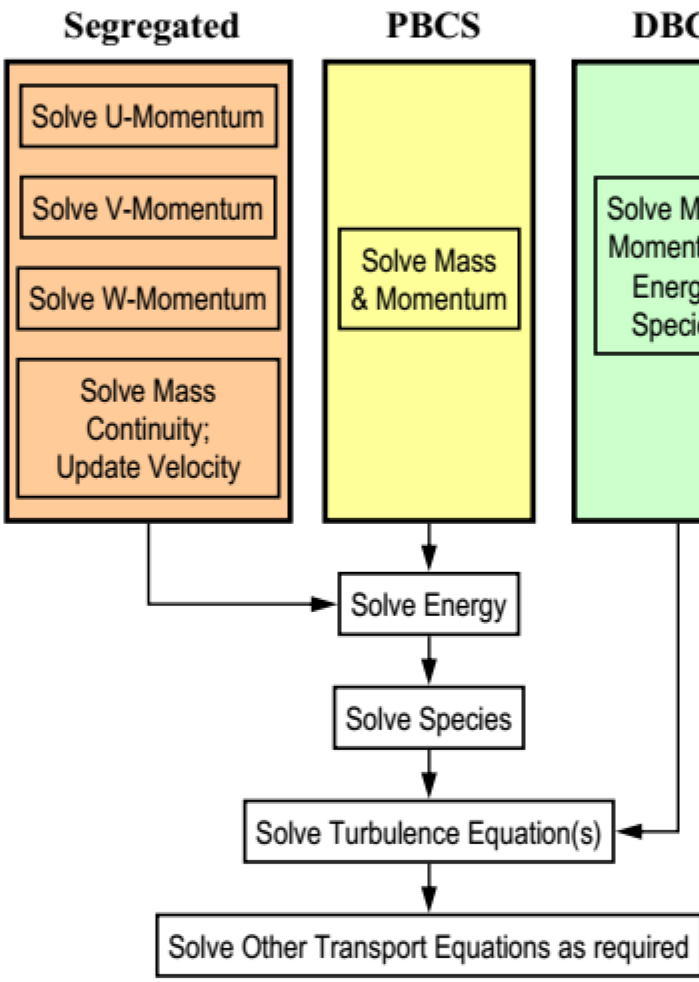

Fig 15: Flow Chart of the Available Solver

3.2 Selection of FLUENT solver

Pressure based solver (PBS) is applied where flow regions varies from low speed 'incompressible flow' to high-speed 'compressible flow'.

a.) It requires less storage capacity.

b.) Has the flexibility while solving the solution.

PBCS for single phase flows gives superior results as compared to PBSS.

1) Multiphase (Eulerian) and periodic mass-flow as well as NITA ("Non-Iterative Time Advancement") cases cannot be solved with this.
2) PBCS consumes 1.5–2 times more space than the PBSS.

(DBCS) also known as density based coupled solver is applied to places where there is requirement of a strong coupling, or interrelation between density, energy, and momentum. For instance, combustion along with excessive speed compressible flow, hypersonic flows, interactions of shocks. The Implicit solution approach is normally preferred over explicit approach, which has constrained on time step size .The explicit approach is utilized where the flow characteristic on time scale is of the similar order as that of the acoustic time scale. (e.g.: propagation of higher value of Mach number shock waves).

3.3 Discretization (Interpolation Methods):

Field variables (occupy the place at cell centres) must be interpolated to the faces of the control volumes.

$$\frac{\partial(\rho\phi)}{\partial t}V + \sum_{f}^{N_{\text{faces}}} \rho_f \mathbf{V}_f \phi_f \cdot \mathbf{A}_f = \sum_{f}^{N_{\text{faces}}} \Gamma_\phi \nabla\phi_f \cdot \mathbf{A}_f + S_\phi V$$

Interpolation techniques for the convection term:

- First-Order Upwind –it gives accurate result for the first order differential equation and Easier to converge.
- Power Law – higher accuracy than first-order. Used for lower values of Reynolds’s number less than 10^5.
- Second-Order Upwind – It is necessary with triangular/tetrahedral mesh structure or when there is misalignment of flow with grid; it leads to slower convergence.
- ‘Monotone Upstream-Centred Schemes for Conservation Laws’ (MUSCL) – used for unstructured mesh quality; gives higher accuracy in secondary flows, vortices, forces, etc.
- ‘Quadratic Upwind Interpolation’ (QUICK) – Applied in case of quadrilateral/hexagonal as well as hybrid mesh structure, also useful in rotating/swirling flows.

3.4 Interpolation Methods (Gradients)

Solution variable of gradients are needed so as to calculate Diffusive flux, derivatives of velocity and for Discretization scheme of higher order.

$$\frac{\partial(\rho\phi)}{\partial t}V + \sum_{f}^{N_{faces}} \rho_f \mathbf{V}_f \phi_f \cdot \mathbf{A}_f = \sum_{f}^{N_{faces}} \Gamma_\phi \nabla\phi_f \cdot \mathbf{A}_f + S_\phi V$$

To calculate the solution variable of gradient at every centre of cell following three techniques are used:

- Green-Gauss Cell-Based :
- Green-Gauss Node-Based: Gives more accurate result, reduces the case of false diffusion. It is recommended for triangular and tetrahedral mesh structure.
- Least Square Cell-Based : IT is mainly used for polyhedral mesh structure and has the similar accuracy as node based gradient

Gradient uses Taylor series expansion to calculate the solution variable at every face of the component.

Interpolation Methods for Face Pressure:

Following ways are available to calculate the cell face pressure while using pressure based segregated solver.

Standard: It is the default scheme; and reduced the accuracy for flows exhibiting large surface-normal pressure gradients near boundaries (but should not be used when steep pressure changes are present in the flow – PRESTO scheme should be used instead.)

PRESTO: Mainly used for swirling flows. It used in case of fan model, porous medium etc. Steep pressure gradient involves in the flow.

Linear: When there is difficulty in converging the solution, while using other option, then this option is used.

Second Order: Only used when the flow is compressible. It is not used with the porous model, fan model or multiphase model.

Body Force Weighted: When the body force is very high for example: high value of Rayleigh number for natural convection.

3.5 Pressure-Velocity Coupling:

In the PBCS, Pressure-velocity coupling is a type of numerical algorithm used to solve the momentum equation and equation of continuity for resolving the equation of pressure. Following type of algorithm are available with FLUENT:

1.) SIMPLE: It stands for "*Semi-Implicit Method for Pressure-Linked Equations*". Algorithm is default and Robust
2.) SIMPLEC: It stand for "*Semi-Implicit Method for Pressure-Linked Equations Consistent*": For simple problem it gives more accurate result.
3.) PISO: It stands for "Pressure-Implicit with Splitting of Operators". IT is useful for unsteady flow cases and for higher skewness mesh cell.
4.) FSM: It stands for "Fractional Step Method". Identical characteristics like PISO. IT is used for "NITA scheme".

3.6 Initialization:

Before calculating the solution, all the variable of solution must be initialized to start the iterative procedure. Some realialistic guess is done to make the solution stable and to converge the solution rapidly. One of the initialization techniques is "Full Multi Grid Initialization".
In "Full Multi Grid Initialization; TUI command is given for FMG initialization which give accurate initialization for the flow field. FMG is computationally faster and inexpensive as well. It is used for both pressure and density based solver in steady state condition. "FMG initialization is also very useful in turbulence flow problem having large pressure gradient and velocity on larger domain area.

3.7 Case Check:

Case check is a service in Fluent, which check the common setup error and guide everyone to select the models and parameters case. It checks for grid size, selection of model, initial boundary condition, properties of material and setting of solver.

3.8 Convergence:

For converging the iteration following condition should be satisfied:

- All the conservation equation like momentum, energy etc. should follow the specified tolerance and it will not change with successive iterations.
- Overall energy, mass and momentum balance are carried out.

Residual history is used for monitoring convergence. Qualitative convergence attained when residuals is decreased by the 3 order of magnitude. In PBS residuals must be reduced to 10^{-6}. The residual values reached within the specified tolerance limit shown by residual plot.

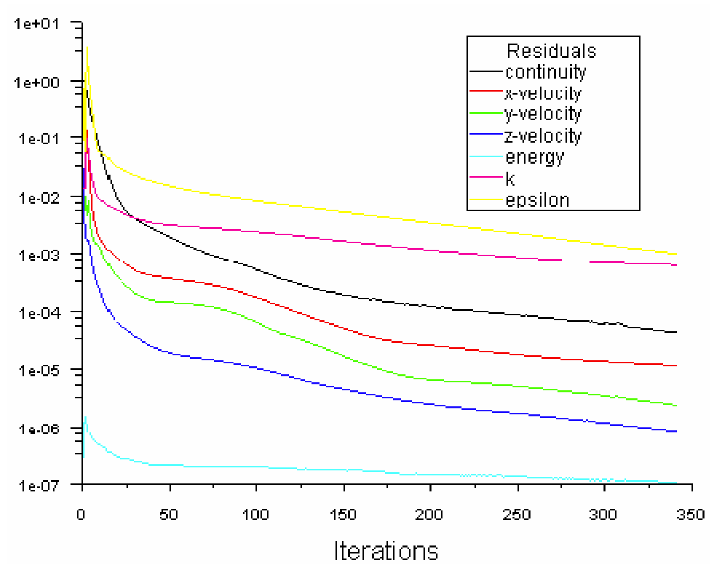

Fig 16: Residual plot

In convergence monitor along with residual plot we can also monitor moment, lift and drag. Sometimes monitors showed that solution is converged its value is changing, this shows that solution is still not converged. In such cases ones should decrease the value of convergence criterion in the residual monitor's panels and then again continue the iteration till the solution is converged.

Poor quality of mesh and incorrect solver setting cause's numerical instability which result in poor problem solution. For instance diverging the residual gives misleading results. To overcome this problem initial solution should be computed by using "1st order Discretization scheme". In PBCS, under-relaxation factor should be decreased and for DBS, Courant number should be decreased. Refining of mesh is also done for the mesh having larger skewness or aspect ratio.

Relaxation Factor: It is denoted by α and helps to stabilize the process in case of PBS. Its value should keep by default while starting the calculation.

Courant Number: Even in the case of steady state, transient term involved in DBS. It is used to define time step size.

3.9 Solution Accuracy:

Most of the time it happened that converged solution is not accurate. In such cases ones should inspect the procedure and evaluate the solution by using some principle and data available.

- Always use the "second-order upwind Discretization" for final result.
- Solution should be "Grid –Independent". Adaption is used to modify the independency of grid.
- Solution accuracy mainly depends on mesh quality. Higher cell skewness and aspect ratio should be minimized. Sufficient mesh density is required to solve the feature of flow, because interpolation errors decrease with smaller cell size.

3.10 Summary:

Solution process for both the pressure-based and density-based solvers is almost similar.

- Calculation will be continued until the solution is converged.
- Second order solution is preferred.
- Refining the mesh structure and recalculating it till grid independent solution is achieved.
- Every solver has a tool to converge and improve the stability of the result.
- Solution accuracy will also depend on the boundary condition specified by ones.

3.11. Propeller Model

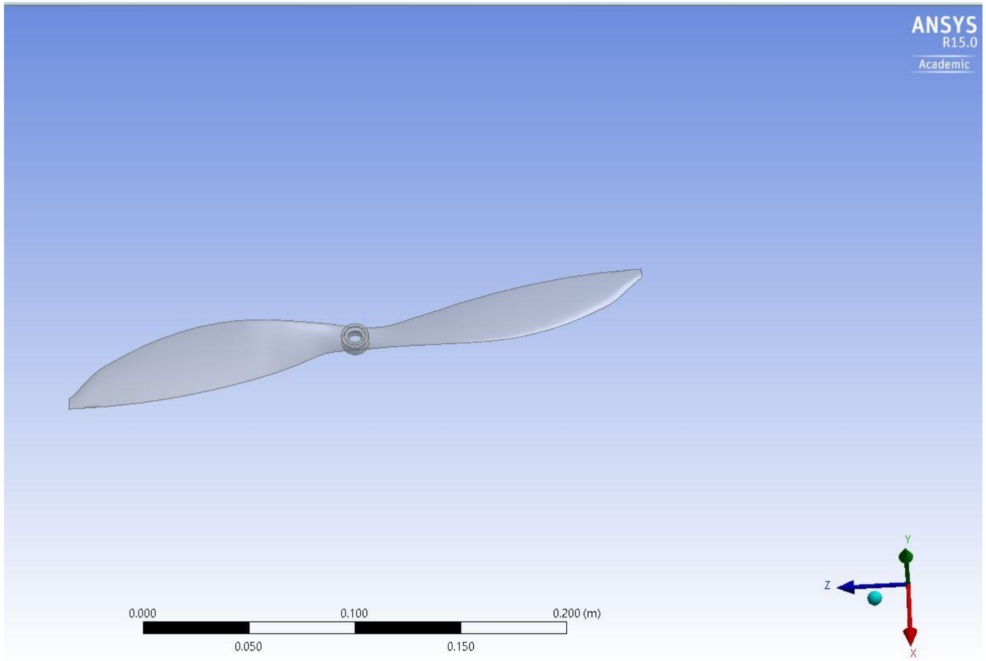

Fig 17: Propeller Model

- Numerical Setup :

The setup used in the numerical analysis is further explained in this section.

3.12. Flow domain:

The numerical simulation has been done in this study were performed using the software "ANSYS FLUENT15.0" commercial CFD solver. The "Multiple Reference Frame model" (MRF) approach was used to numerically conclude the flow around the propeller. The domain is specify and depicted in Figure 2a.The domain is then break into a global stationary and rotating region, called the rotating domain. The rotating domain is indicated by a smaller cylinder fully enclosing the blades and the hub i.e. propeller.

Enclosure	Enclosure1
Shape	Cylinder
Cylinder Alignment	Y-Axis
Number of Planes	0
Cushion	Non-uniform
FD1, cushion Radius(>0)	0.006m
FD1,cushionRadius(>0),+veDirection	0.06
FD1,cushionRadius(>0),-ve Direction	0.06

Table1: Details of enclosure1

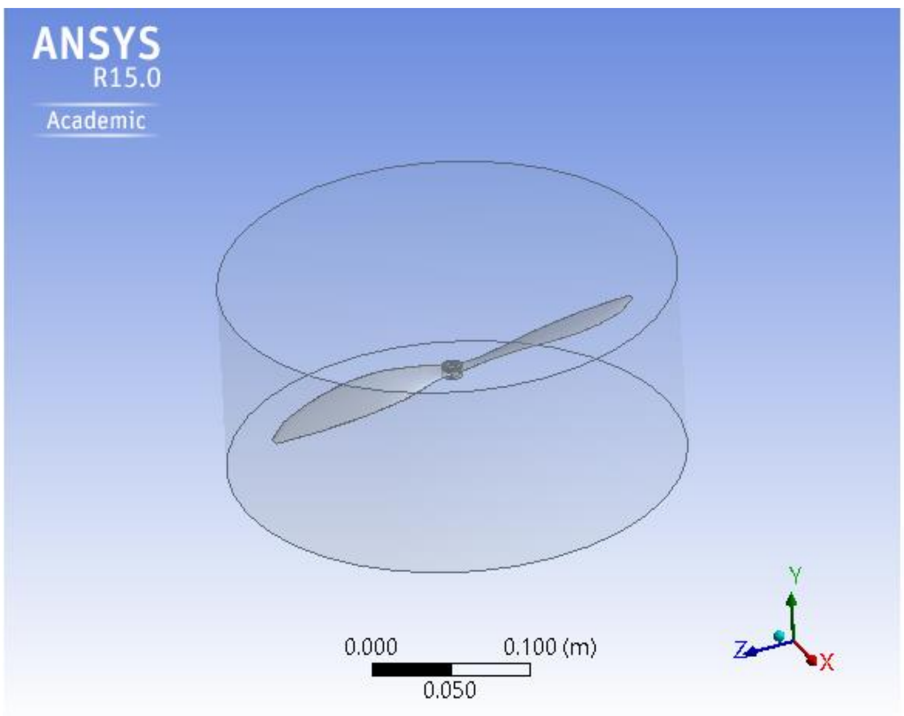

Fig 18: Enclosure1

Enclosure	Enclosure2
Shape	Box
Number of Planes	0
Cushion	Non-Uniform
FD1,Cushion +X Value (>0)	2
FD1,Cushion +Y Value (>0)	1.5
FD1,Cushion +Z Value (>0)	1.5
FD1,Cushion -X Value (>0)	2
FD1,Cushion -Y Value (>0)	1.5
FD1,Cushion -Z Value (>0)	1.5

Table2: Details of enclosure2

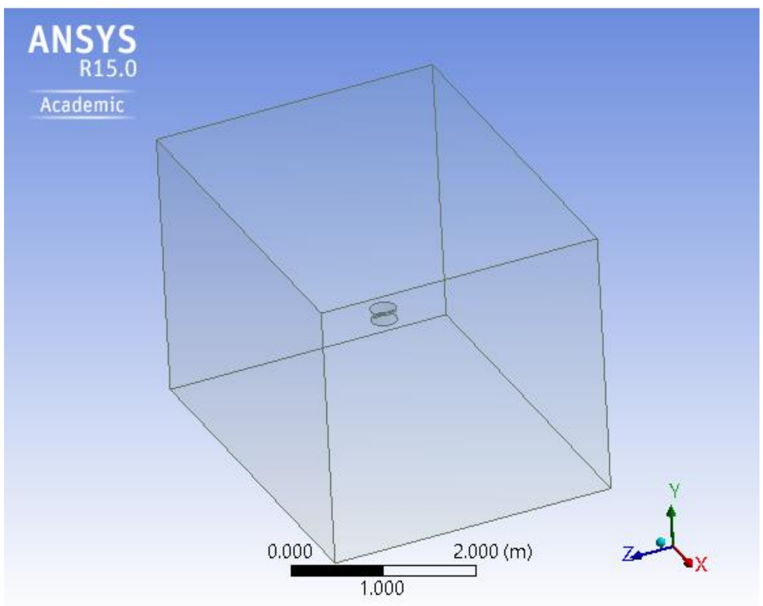

Fig 19: Enclosure2

The inlet, outlet, and outer boundaries of the stationary region are placed far enough from the propeller to prevent the full development of the upstream and downstream flow from affecting the results of the analysis. The inlet and outlet boundaries are located 1.5”, both upstream and downstream of the origin of the propeller. For the rotating domain, the enclosure is set to be 1.1D and 0.4D.Proper selection of the flow domain upstream and downstream distance is very important to prevent recirculation of the flow that will cause convergence problems.

3.13. Mesh generation

A mesh tool in FLUENT 15.0 was used to develop the grid. The matrix is vital on the grounds that it gives the individual presentation of the geometry. The nature of the mesh structure straightforwardly impacted the rate of convergences, the execution gotten from the numerical investigation, and computational time required for the simulation. For the present analysis, the size of the cells of mesh was produced to be in a small range along the rotating blade and to step by step increment toward the stationary area. The enhancement of the accuracy of the results across the interface is done by ensuring the enough grid refinements.

The matrix, in a stationary and rotating space, is completely tetrahedral and unstructured. The refinements on the basis of the avocations that unstructured tetrahedral matrices have the abilities to distinguish complex geometries with quick and least user intercession. Table 2 demonstrates the information for grid cell development. Adjustment of the mesh to the physical arrangement, that is refinement of grid or coarsening, is easy to fulfil on an unstructured over an organized matrix.

Use advance size function	Curvature
Relevance Centre	Fine
Curvature Normal Angle	Default (18.0^0)
Min Size	Default (9.122e-004 m)
Max Face Size	Default(9.122e-002 m)
Max Size	Default (0.182440 m)
Growth Rate	Default (1.20)
Minimum Edge Length	6.4681e-004

Table3: Fine Mesh Details

Figure 20 below shows the surface mesh of the propeller blade having coarse mesh, where Figure 21 shows the medium mesh, Figure 22 shows fine mesh,. Table 3 shows the numbers of cells, nodes and faces for the respective meshes. The cells sizes near the blade wall are smaller and increase towards the outer boundary.

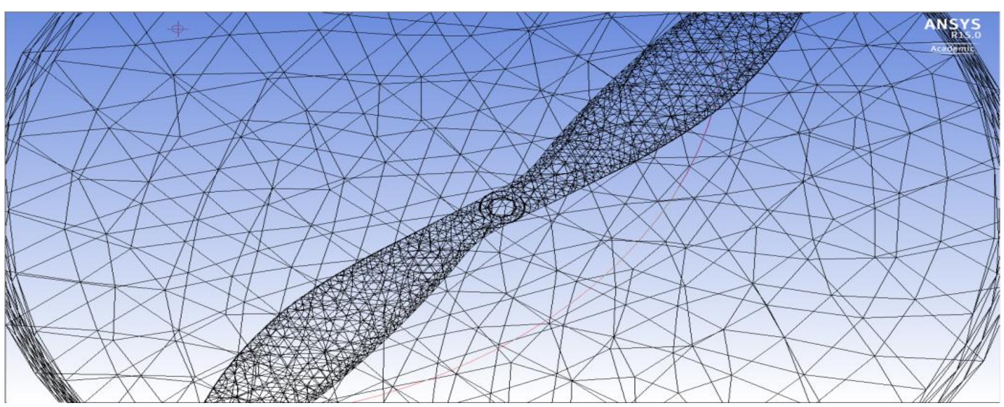

Fig 20: Surface mesh of the propeller blade. The cells sizes near the blade wall are smaller and increase towards the outer boundary. a) Coarse Mesh

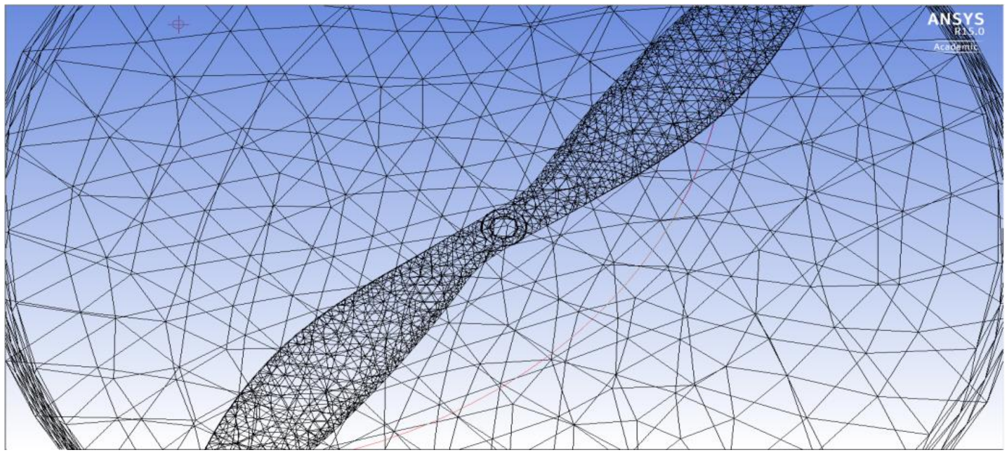

Fig 21: Medium Mesh

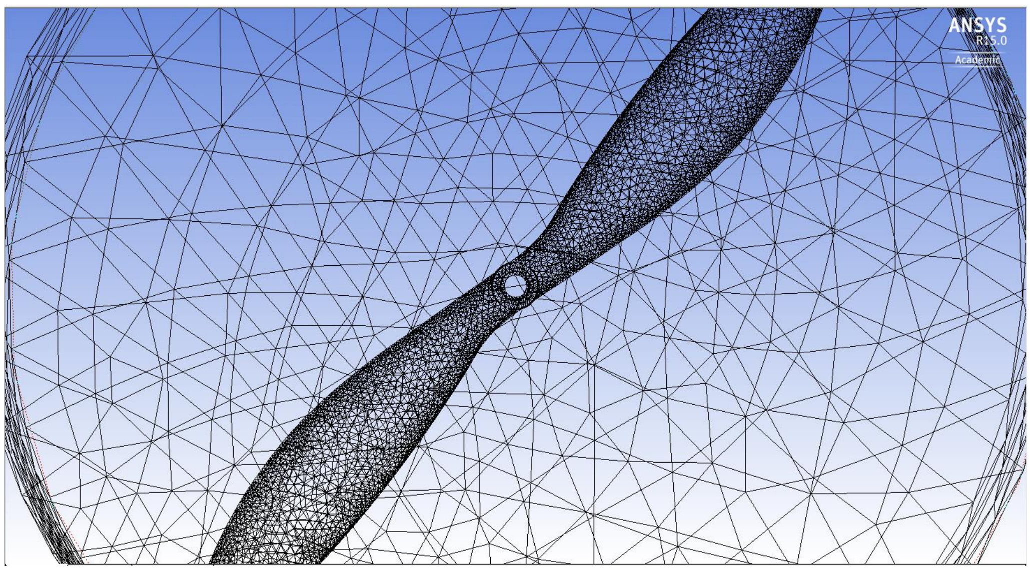

Fig 22: Fine Mesh

3.14. Boundary Condition

For determining the input conditions and the output conditions of a model, boundary conditions are defined. There are the differential equations which define the entire region or a part of the region. To solve these differential equations, we require some conditions which are satisfied in a region. These conditions are known as boundary conditions.

In this project work, the input conditions for the inlet and rotating body were defined. The range for the turbulent intensity is set in between 1% to 5%. At the inlet, for the input velocity, turbulent intensity is 1% and at the outlet, for pressure, this intensity is

5%. The reason to select this range of turbulent intensity is that at the entry the turbulence present in the air is very small, in the course of flow; the obstruction present in the path increases the turbulence intensity. The value of 3.7 % is taken as the default value for the turbulence intensity. This value is enough for the normal turbulence present in the circular inlet.

The turbulent viscosity ratio is taken in the range of 1-10. This range is beneficial for analyzing the external flows. For analyzing the internal flows, the value for the turbulent viscosity ratio is in the range of 10-100.

The simulation of the model was done on the CFD module of the ANSYS software. The flow conditions were presented on the table 1 given below. The rotational speed of the propeller is fixed at the 3008 rpm. At the inlet, boundary conditions are specified that is the free stream speed is defined with the intensity of turbulence of 0.1%. The value of the intensity is obtained from the experimental measurement done on the wind tunnel. The position from where the flow leaves the flow domain was taken as the point of determining the outlet boundary conditions. The condition of no-slip is applied on the surrounding walls. To incorporate the rotational speed in the domain region engulfing the propeller blade, MRF is assigned to the domain. This approach is very effective in the cases where there is interaction between the fixed and moving blades. The discrete speeds i.e. translational and rotational were assigned in the individual zones. To allow the one zone flow variables to be used in the adjacent flow zone, the interface between the two zones would undergo the local frame transformation. The rotating frame of reference is given the speed of 3008 rpm. The walls of the propeller domain and the hub were also provided with the same rotational speed as that of the propeller blade i.e. 3008 rpm. This would make the relative speed between the adjacent zones equal to the zero.

For determining the momentum and pressure developed in the model, the second order upwind scheme was used. Similarly for determining the turbulent kinetic energy and dissipation rate, the first order upwind scheme was used. Gradients were defined by least square cell algorithm. Accurate results were obtained by the first order algorithms.

3.15. Numerical Method Verification and Validation

The prediction of the flow over the propeller blades was determined by analyzing the effect of the grid and turbulence model. This was also done to validate the method used in this project work. The influence of the grid on the performance of a propeller was determined by performing the grid resolution study. A total of five grids were produced which were based on the geometry generation. These five grids were named as standard grid, coarse grid, mid size grid, mid-fine grid and fine grid. The number of cells generated by these five grids was 0.38×106, 1.06 × 106, 2 × 106, 3 × 106 and 4 × 106 respectively.

By default FLUENT workbench produces the standard mesh. It has small number of elements. One can refine the grid manually to attain higher quality of mesh. The number of cells can further be increased by changing the meshing parameter like minimum sizing. To increase the number of cells, minimum sizing value is reduced and vice versa. Computational timing to converge the solution and numerical accuracy becomes the base for the selection of optimum mesh size for the analyses. Three different turbulence models were also tested. These models were named as K-epsilon (k-Ɛ), standard k-ω and SST k-ω. The model which proved to be the most suitable for predicting the performance of the propeller was selected for the analyzing the propeller blades.

Table: 4 Grid for the whole Domain including

Mesh	Elements	Nodes
Coarse	322345	94651
Medium	572753	100392
Fine	941332	166929

3.16. Solution Control

Model Used:

Model	k-episilon (2 eqn)
k-episilon Model	Realizable
Near Wall Treatment	Non Equilibrium Wall Function
Model Constant	C2-Episilon: 1.9 TKE Prandtl Number: 1 TDR Prandtl Number: 1.2

3.16.1. Reference Values

Compute From	Inlet	
Reference Values	Area(m^3)	0.006251230
	Density(kg/m^3)	1.225
	Enthalpy()	0
	Pressure (Pascal)	0
	Temperature(K)	288.16
	Velocity(m/s)	8
	Viscosity(kg/m-s)	1.7894e-05
	Ratio of Specific Heats	1.4
Reference Zone	Rotating Body	

3.16.2. Solution

- **Solution Methods:**

Pressure velocity coupling

Scheme	Coupled

Spatial Discretization

Gradient	Least Square Cell Based
Pressure	Standard

Momentum	First Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

- **Solution Controls**

Courant Number	50	
Explicit Relaxation Factors	Momentum 0.25 Pressure 0.25	
Under Relaxation Factors	Density	1
	Body Force	1
	Turbulent Kinetic Energy	0.8
	Turbulent Dissipation Rate	0.8
	Turbulent Viscosity	0.8

- **Solution Initialization:**

Initialization Method	Standard Initialization
Reference Frame	Absolute
Initial Gauge Pressure (Pascal)	0
X Velocity (m/s)	8
Y Velocity (m/s)	0
Z Velocity (m/s)	0
Turbulent Kinetic Energy (m2/s2)	0.0096
Turbulent Dissipation Rate (m2/s3)	0.05678238

CHAPTER 4

RESULT AND DISCUSSION

Initializing the solution by means of standard initialization and iterating the calculation by 1000 times, values of thrust and torque were obtained. On changing the boundary condition, one gets the different set of values for thrust and torque, these set of data is utilized to predict the behaviour of output data whenever the input changes.

(a) Velocity Contour

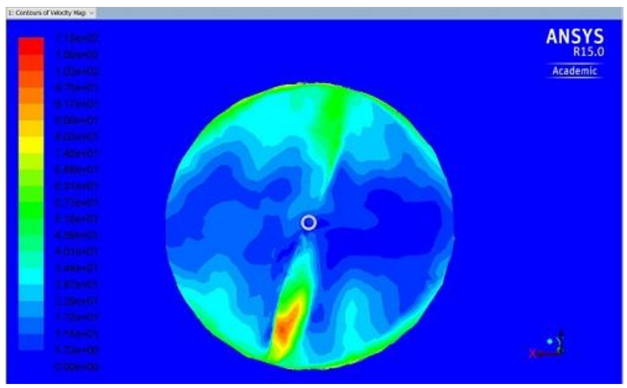

Figure: 23

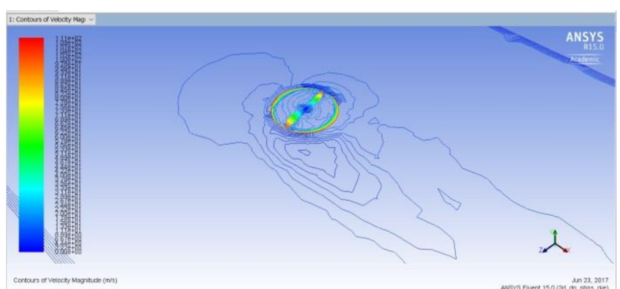

Figure: 24

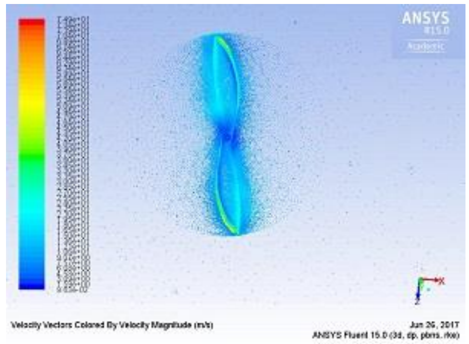

Figure: 25

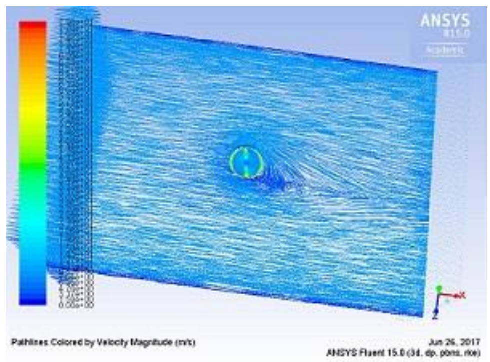

Figure: 26

Figure 23-26: Different Velocity countour on changing the boundary condition, shown by pathline @ air velocity 2m/s, 5m/s, 8m/s, 11m/s

(b) Pressure Contour

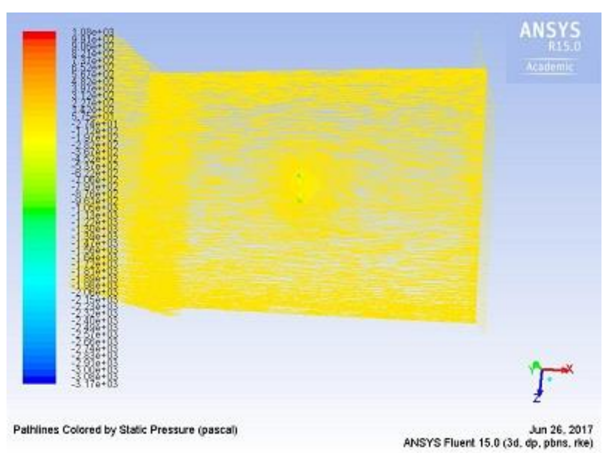

Figure: 27

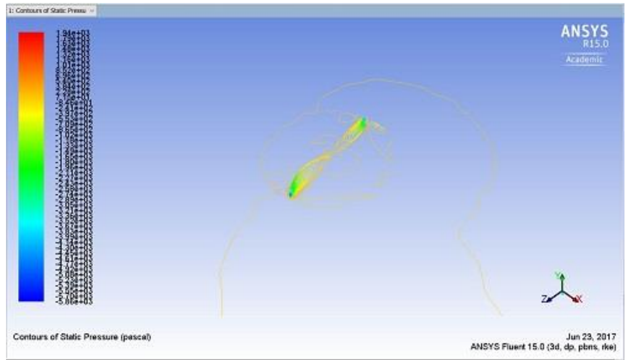

Figure: 28

Figure 27-28 shows the Different Pressure contour at a given boundary condition

For the given boundary condition and rotating the propeller at fixed RPM and varrying the flow condition following simulation result were made. Summary of individual simulation has been discussed:

CASE 1: @4000 RPM, and air velocity 4m/s, 6m/s, 8m/s, 9m/s, 12m/s

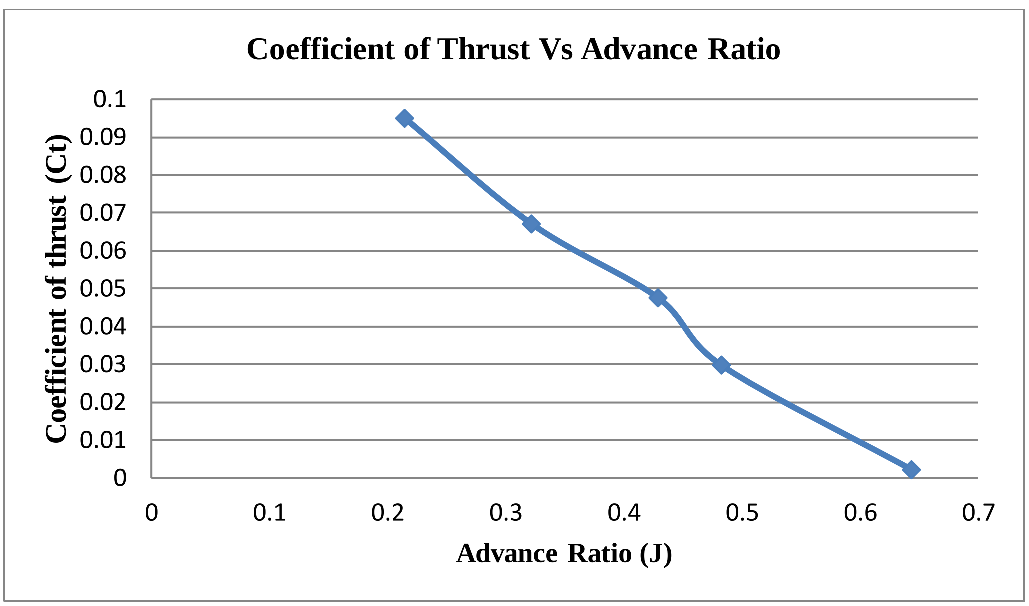

Figure 29: Coefficient of Thrust Vs Advance Ratio

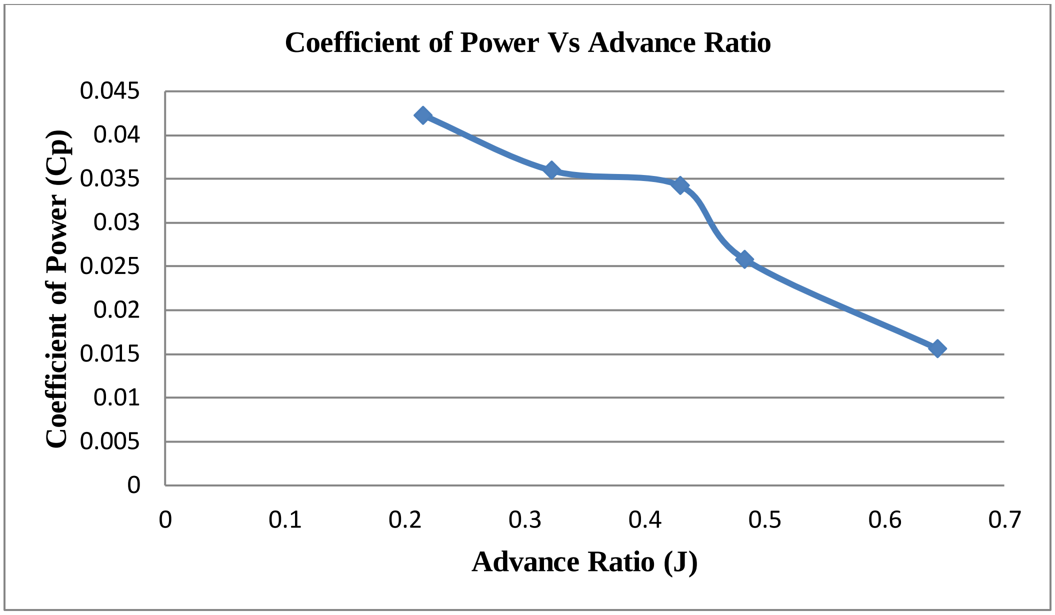

Figure 30: Coefficient of Power Vs Advance Ratio

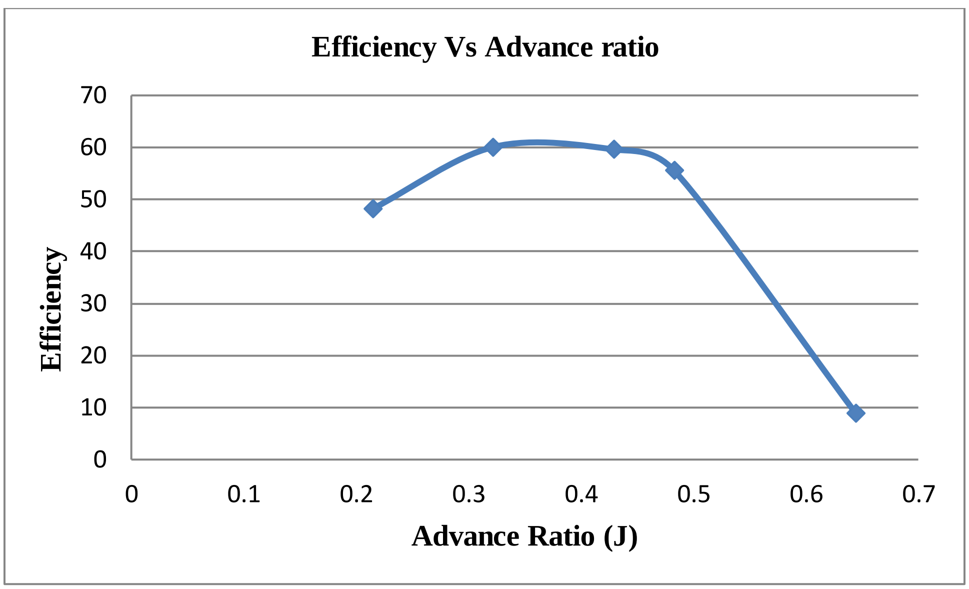

Figure 31: Efficiency Vs Advance ratio

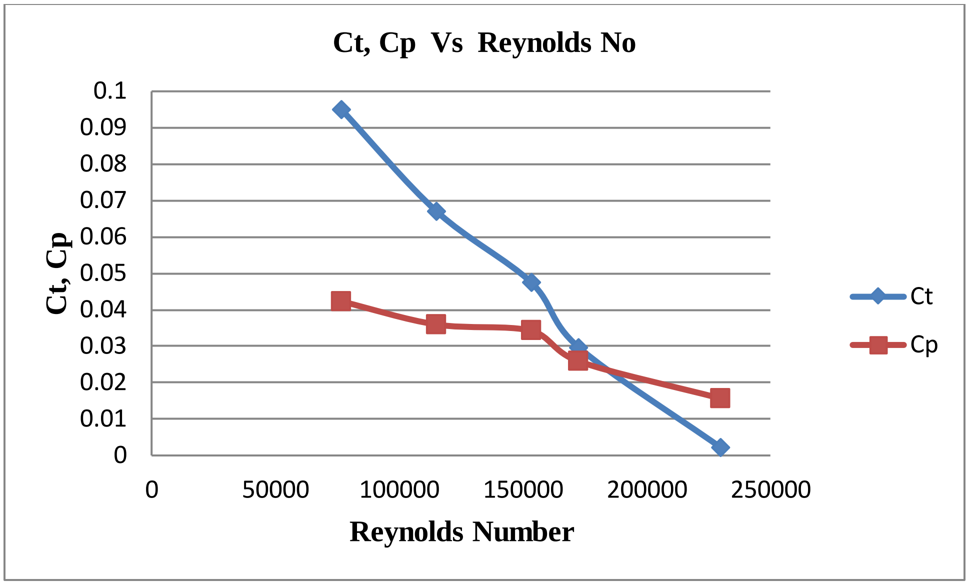

Figure 32: C_t, C_p Vs Reynolds Number

CASE 2: @5000 RPM, and air velocity 4m/s, 6m/s, 8m/s, 9m/s, 12m/s

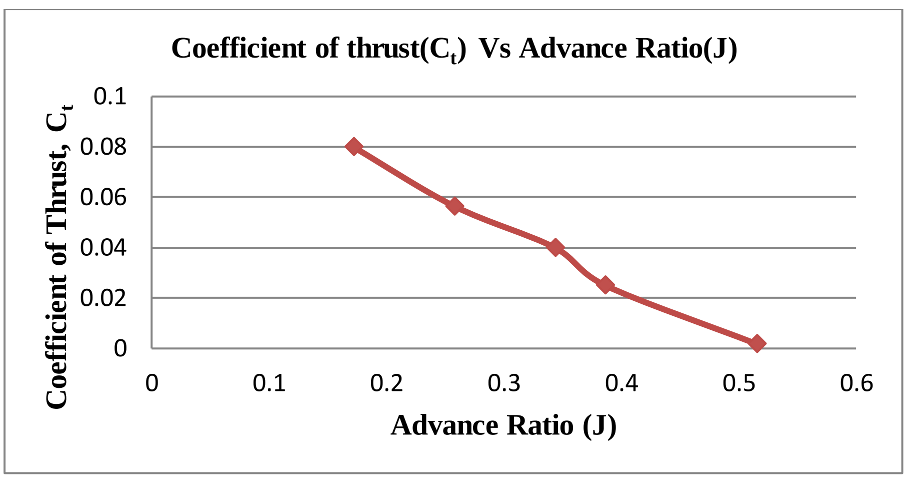

Figure 33: Coefficient of thrust(C_t) Vs Advance Ratio(J)

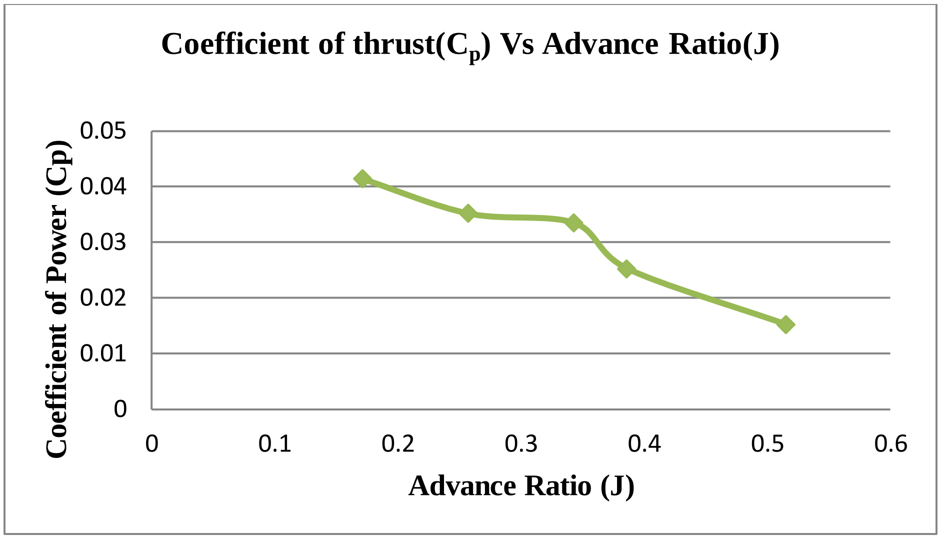

Figure 34: Coefficient of thrust(C_p) Vs Advance Ratio(J)

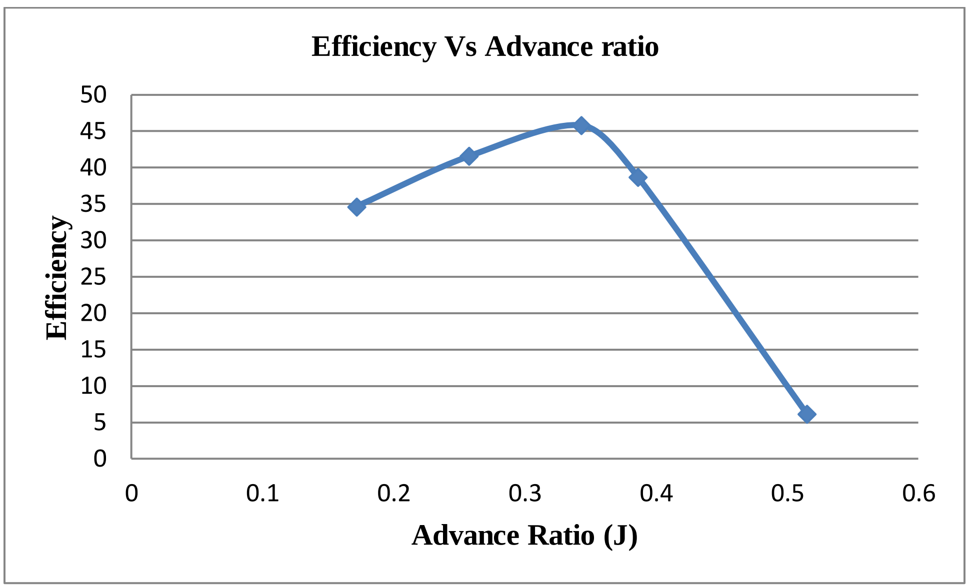

Figure 35: Efficiency Vs Advance ratio

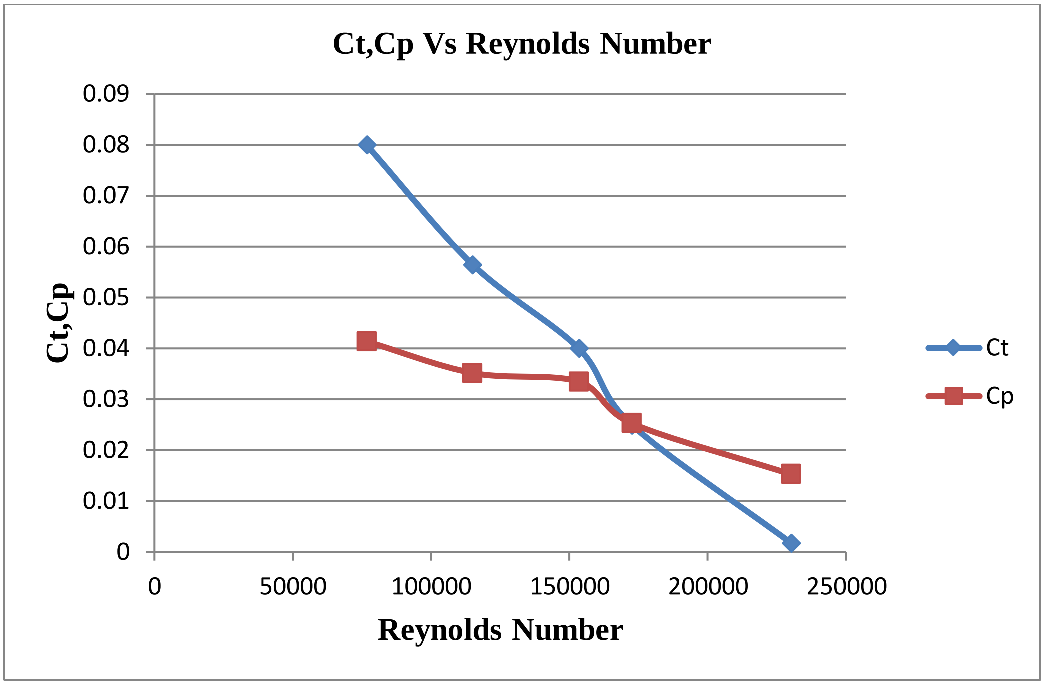

Figure 36: C_t,C_p Vs Reynolds Number

CASE 3: @6000 RPM, and air velocity 4m/s, 6m/s, 8m/s, 9m/s, 12m/s

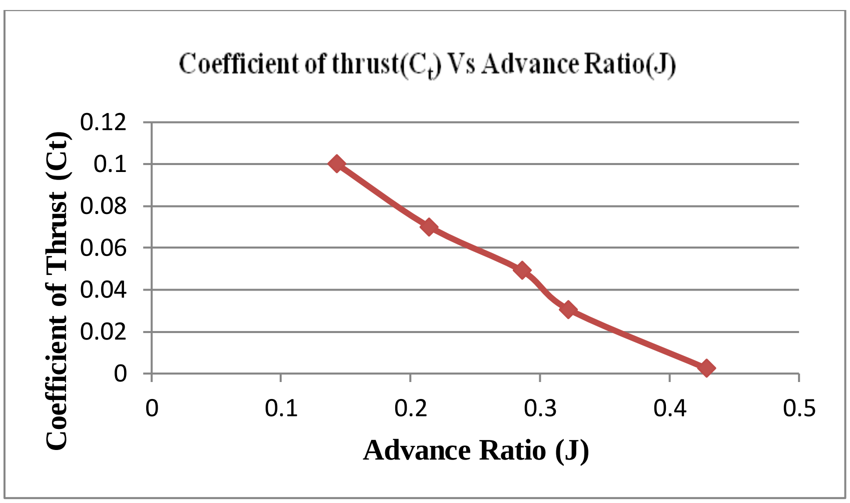

Figure 37: Coefficient of Thrust Vs Advance Ratio

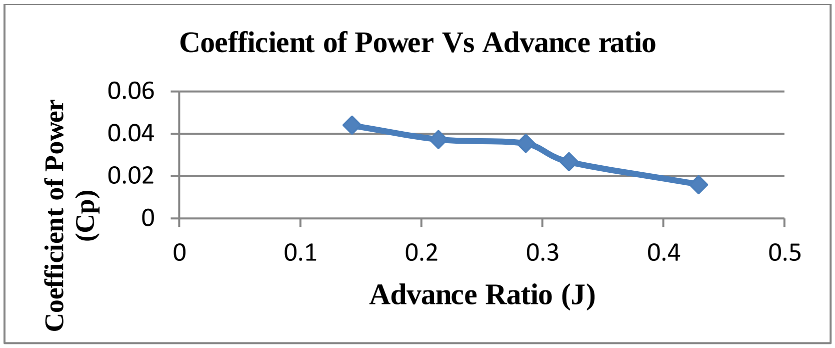

Figure 38: Coefficient of Power Vs Advance Ratio

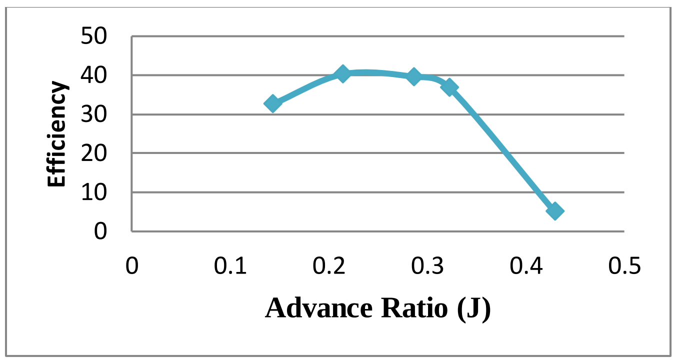

Figure 39: Efficiency Vs Advance Ratio

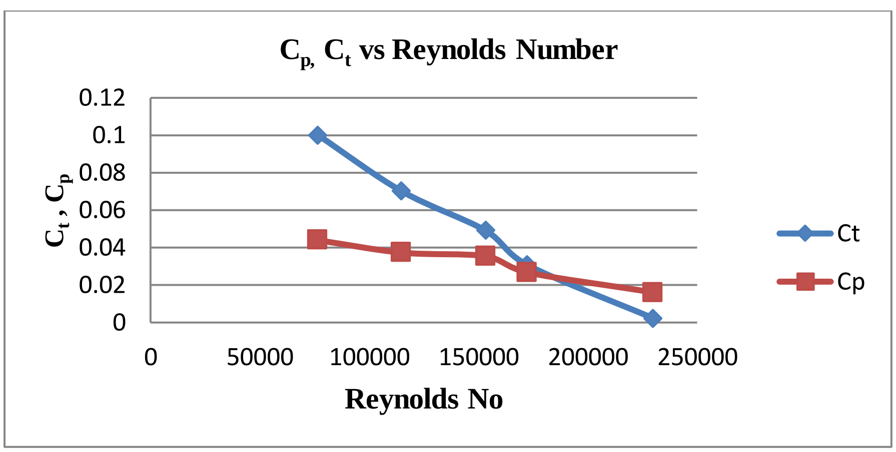

Figure 40: C_t,C_p Vs Reynolds No

CASE 4: @7000 RPM, and air velocity 4m/s, 6m/s, 8m/s, 9m/s, 12m/s

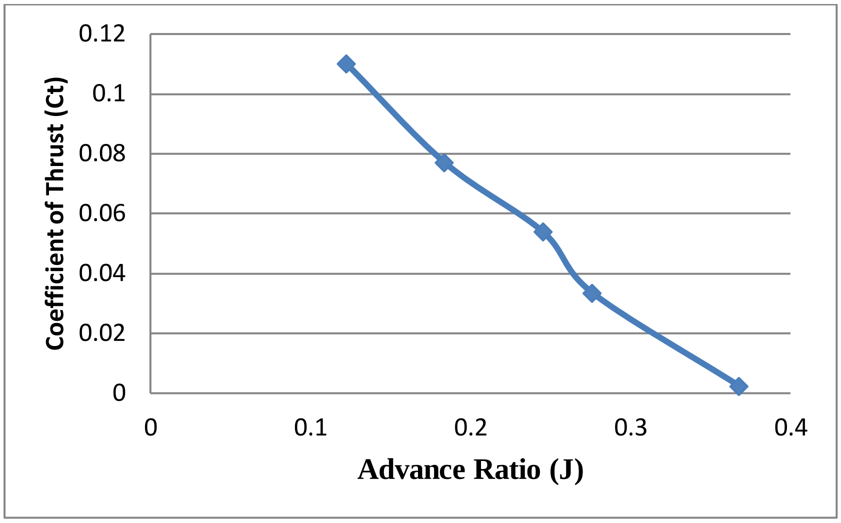

Figure 41: Coefficient of thrust Vs Advance Ratio

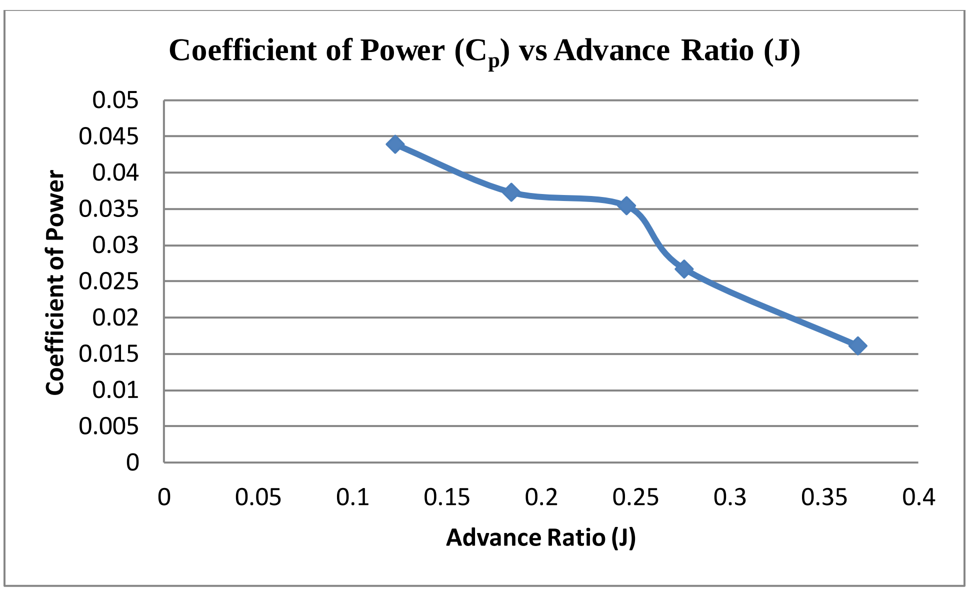

Figure 42: Coefficient of Power vs Advance Ratio

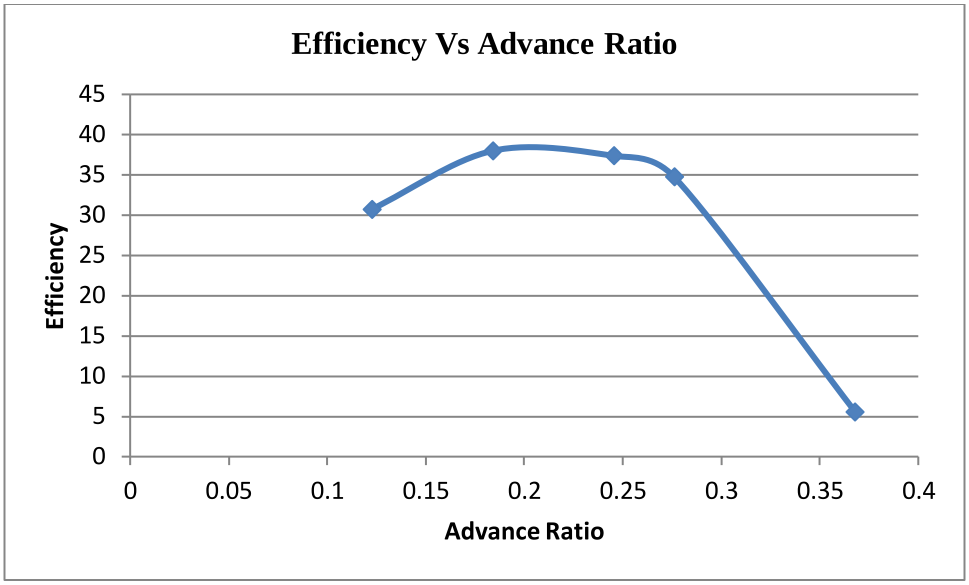

Figure 43: Efficiency vs Advance Ratio

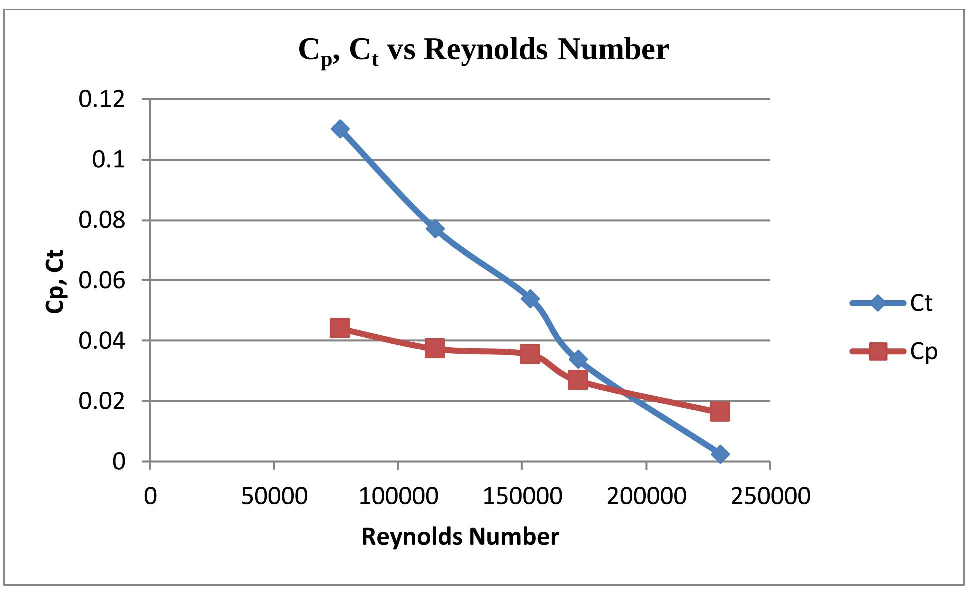

Figure 44: C_p, C_t vs Reynolds Number

CASE 5: @8000 RPM, and air velocity 4m/s, 6m/s, 8m/s, 9m/s, 12m/s

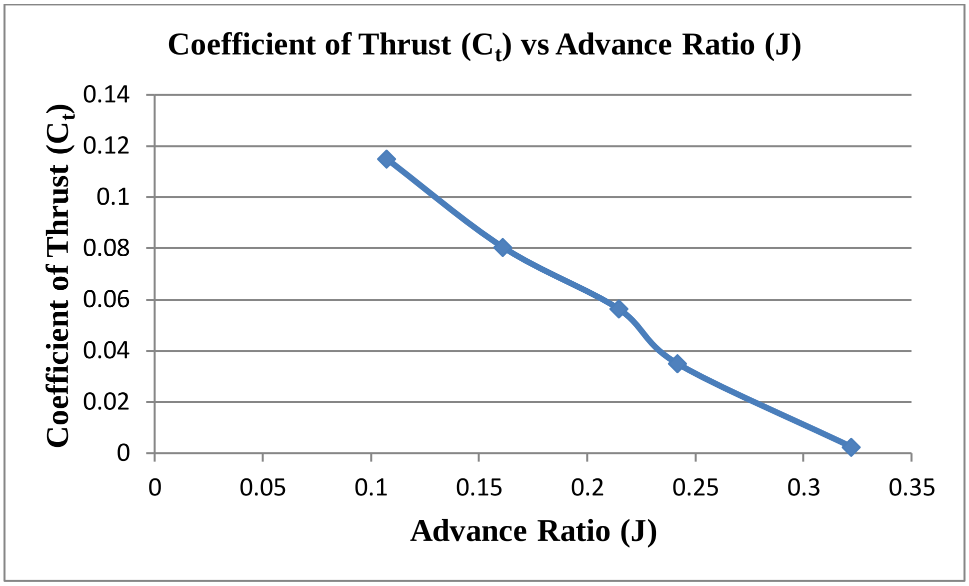

Figure 45: Coefficient of Thrust vs Advance Ratio

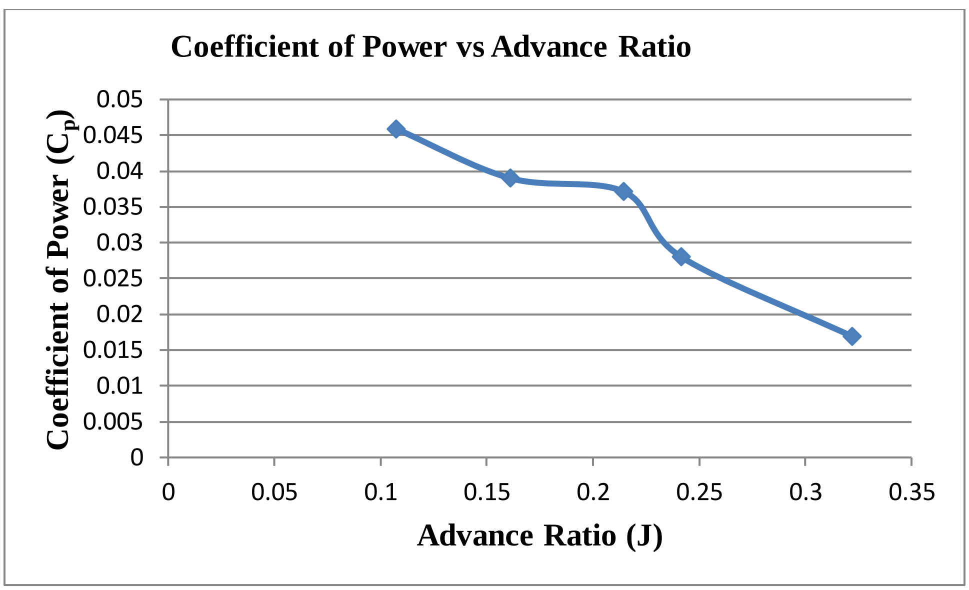

Figure 46: Coefficient of Power vs Advance ratio

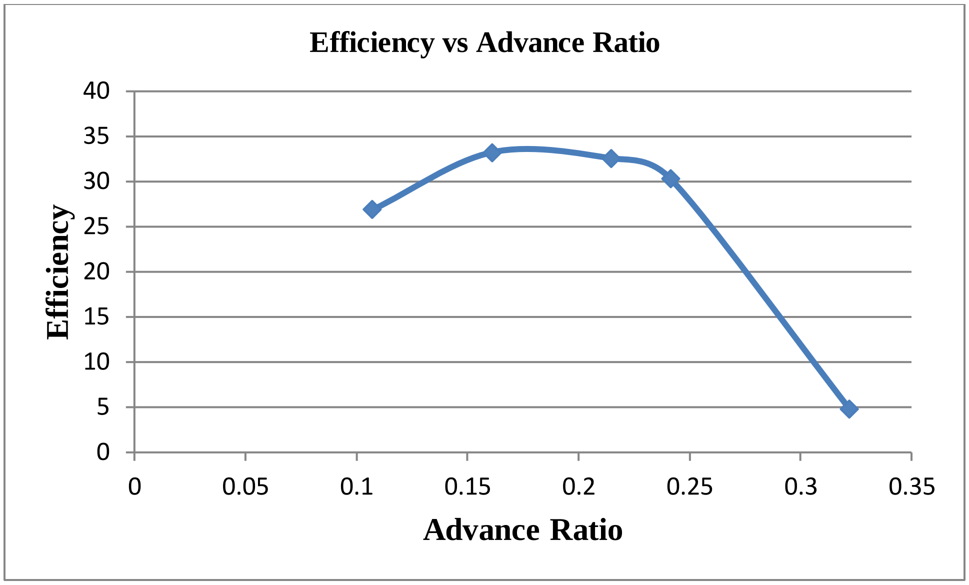

Figure 47: Efficiency vs Advance Ratio

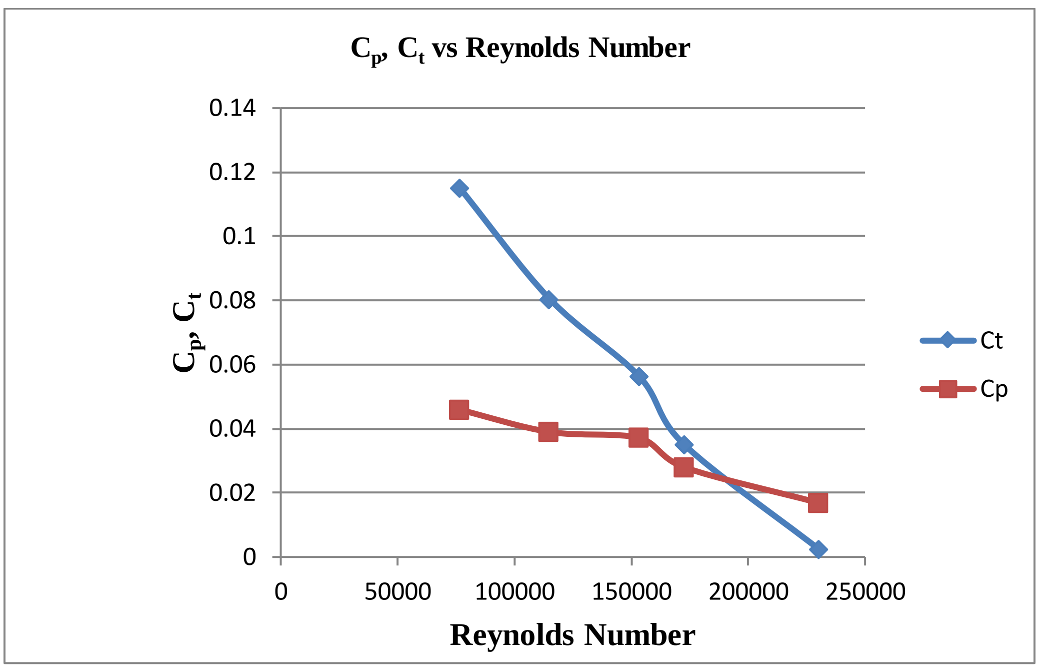

Figure 48: $C_{p,}C_t$ vs Reynolds Number

After studing data of all the five cases, it is conluded thet the coefficient of thrust increases, for the same velocity of air, with increase in the rotational speed of propeller. But on the other hand, the coefficient of power decreases, upto certain limit, with the increase in the rotational speed of the propeller, for the same velocity of air.

However the efficiency of the propeller increases with increase in the advance ratio and after reaching its maximum value, it decreases with the increase in the advance ratio.

CHAPTER 5

CONCLUSION

The small-scale nature of propellers used for small UAVs and MAVs lead to difficulty in predicting their behaviour. Part of this difficulty is from the large dependence the performance has on the propeller Reynolds number. To help measure this Reynolds number dependence, static and advancing flow performance data were measured on 11 in. pusher propeller on various boundary condition. As the Reynolds number increases, the thrust coefficient increases, the power coefficient decreases or both. The change in thrust and power coefficient is due to the increase in the lift coefficient and decrease of drag coefficient with increasing Reynolds number.

The results for the thrust coefficient showed a slight under-prediction for a low advance ratio, whereas the result for the power coefficient shows both an under-prediction for a low advance ratio and an over-prediction for a higher advance ratio. Efficiency was high throughout the advance ratio, with only a slight over-prediction on the higher advance ratio. Overall results showed a reliable capability to predict the performance of a low-speed, low Reynolds number small-scale propeller.

CFD simulation used in this study successfully replicated the experimental results which may significantly improve computational accuracy. The commercial CFD code such as FLUENT was found to be reliable in providing good initial prediction, thus it can replace experimental analysis, which is time-consuming and more expensive.

REFERENCE

[1] Rutkay Brian, “Design and manufacture of propellers for small unmanned aerial vehicles”, Department of Mechanical and Aerospace Engineering, Carleton University, 3135 Mackenzie Building, 1125 Colonel By Drive, Ottawa, Ontario, Canada K1S 5B6

[2] S. Patel Karana, B. Patel Saumil, “CFD Analysis of an Aerofoil”, International Journal of Engineering Research, Volume No.3, Issue No.3, pp : 154-158,2014.

[3] W. Mueller Mark, “Stability and control of a quadrocopter despite the complete loss of one, two, or three propellers”, International Conference on Robotics & Automation (ICRA) Hong Kong Convention and Exhibition Center, 2014 IEEE

[4] S. M. Mahbobur Rahman, “Simulation investigation on flow characteristics for the flow over a propeller used in VTOL RC aircrafts”, International Conference on Mechanical, Industrial and Energy Engineering 2014.

[5] Agarwal Shlok, “Design, Construction And Structure Analysis Of Twin rotor UAV”, International Journal of Instrumentation and Control Systems (IJICS) Vol.4, No.1, January 2014

[6] Kalpesh N. Shah, "Quadrotor – An Unmanned Aerial Vehicle", 2014 IJEDR , Volume 2, Issue 1, ISSN: 2321-9939

[7] Subhas.S, “CFD Analysis of a Propeller Flow and Cavitation”,2012 , Visakhapatnam International Journal of Computer Applications (0975 – 8887)

[8] Brandt John B., “Modeling of Flow Around a Marine Propeller Using a Potential Based Method”,2011, University of Illinois at Urbana-Champaign, Urbana, IL 61801, USA

[9] W. Shawn Westmoreland, Robert W. Tramel, and Jennie Barber, “Modeling Propeller Flow-Fields Using CFD”, 46th AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-402, 7-10 January 2008, Reno, Nevad.

[10] Wong K.C, “UAV Design Activities in a University Environment” School of Aerospace, Mechanical and Mechatronic Engineering University of Sydney NSW 2006

[11] Herisse Bruno, “Hovering and vertical landing control of a VTOL unmanned aerial vehicle using optical flow”, International Conference on Intelligent Robots and Systems”, ANR-06-ROB-0007, 2008

[12] Abdellah Mokhtari,” Robust feedback linearization and GH_{∞} controller for a quadrotor unmanned aerial vehicle”, journal of electrical engineering, vol. 57, no. 1, 2006, 20–27.

[13] Peter G. Ifju, “Flexible-Wing-Based Micro Air Vehicles”, AIAA 2002-0705

[14] Maitre T. A., “Modeling of Flow Around a Marine Propeller Using a PotentialBased Method”, Journal of Ship Research, Vol. 35, No. 2, June 1991, pp. 114-126

[15] Bell Jonathon, “Optimizing Performance Variables for Small Unmanned Aerial Vehicle Co-Axial Rotor Systems”, Middlesex University, School of Engineering and Information Sciences, Trent Park Campus, Bramley Road, London N14 4YZ, United Kingdom

[16] Bristeaue Pierre Jean, “The role of propeller aerodynamics in the model of quadroter UAV”, European control conference 2009

[17] S. Norouzi Ghazbi, “Quadrotor Unmanned aerial vehicles: A review” International journal on smart sensing and Intelligent Systems vol 9. No.1 March 2016.

[18] A.V.Javir,” Design, Analysis and Fabrication of Quadcopter”, Journal of The International Association of Advanced Technology and Science, Vol. 16 | MARCH 2015

[19] A. A. Johnson and T. E. Tezduyar, “Advanced mesh generation and update methods for 3D flow simulation”, Computational Mechanics 23(1999) 130-143, © Springer-Verlag 1999.

[20] Anderson, J. D. (1991), “Fundamentals of Aerodynamics”, McGraw-Hill Companies, 2nd

[21] Anderson John D (1995), Computational Fluid Dynamics, Basics with Applications, McGraw Hill Publications, ISBN 0-07- 113210-4.

[22] Shetty Pradeep, “computational fluid dynamics simulation of propeller fan”, IJESRT, October 2016 IJSS7

[23] Deters, R.W.; Krishnan, G.K.A.; Selig, M.S. Reynolds number effects on the performance of Small-scale propellers. In Proceedings of the 32nd AIAA Applied Aerodynamics Conference, Atlanta, GA, USA, 16–20 June 2014; pp. 1–43.

[24] Morgado, J.; Silvestre, M.Â.R.; Páscoa, J.C. Validation of new formulations for propeller Analysis. J. Propuls. Power **2015**, 31, 467–477

[25] ANSYS,Inc.ANSYSFluentUser'sGuide.Availableonline:http://159.226.251.229/videoplayer/ANSYS%20Fluent%20Users%20Guide.pdf?ichuri=4430682d62b326e212655a3feab3426a&ichstart=0&ichend=0&ichkey=174502892375046359245 5&ichtype=1&ichdiskid=1&ichunit=1 (accessed on 15 January 2017).

[26] ANSYS, Inc. Modeling Turbulent Flows, Introductory Fluent Training. Availableonline:http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf(accessed on 15 January 2017).

[27] A.V Javir, Design Analysis and Fabrication of Quadrotor, Journal of International Association of Advanced technology and Science ISSN-3347-4482, Vol-16 March 2015