# Design and Computational Fluid Dynamic Simulation of Micro Air Vehicle Propeller

Mr.D.Lakshmanan, D.Prem, R.Ragul ,D.Ragupathi,

Assistant Professor of Aeronautical Department, Bannari Amman Institute of Technology,Erode-638401,Tamilnadu,India.

Department of Aeronautical Engineering,Bannari Amman Institute of Technology,Erode-638401,Tamilnadu,India.

Department of Aeronautical Engineering,Bannari Amman Institute of Technology,Erode-638401,Tamilnadu,India.

Department of Aeronautical Engineering,Bannari Amman Institute of Technology,Erode-638401,Tamilnadu,India.

Email: lakshmanand@bitsathy.ac.in,prem.ae16@bitsathy.ac.in,ragul.ae16@bitsathy.ac.in,ragupathi.ae16@bitsathy.ac.in

***Abstract*— This paper deals with a model of Micro Air Vehicle (MAV) propeller which is designed and its thrust performance investigation. This paper tries to increase the thrust of the propeller which derive on the basis of simple analytical relationships and are examine on the set of results from CFD simulation. The thrust and lift which are called aerodynamic coefficients can increased when a considerable reduction or changes in drag. The result of the paper is propose with the relationship between analytical and simulation results of thrust. In addition to this, the paper also provides the 3d streamline flow over the propeller and lift, drag performance through the graphical representation.**

*Keywords: MAV propeller, Thrust, Computational Fluid* Dynamics

## 1) INTRODUCTION

### A) *Introduction to UAV*

In 2008, the TU DELFT University in Netherlands have developed the miniature ornithopter with only a camera for surveillance process. This measures only 10 cm and it not tested successfully. Later, many researchers implemented the idea of airplane and aircrafts to reduce the size to very micro and improve their performance and made use for spying in initial stage. UAV-Unmanned Aerial Vehicle are the aircrafts without human as a pilot on board. It is a component of unmanned aircraft system in which it consists of ground control system and communication system. These are mostly used for the missions where the place is too dirty, dull and dangerous for humans which are originated in military applications, but also UAV's are expanded to scientific, commercial, aerial photography, geography, product deliveries and extra which leads to idea the weather, agricultural and geological information and data without our presence these vehicles are called Unmanned Aerial Vehicle(UAV), Manned Aerial Vehicle, Micro and Macro Aerial Vehicle.

### *B)* Introduction to MAV

MAV-Micro Aerial/Air Vehicle are one of the form of adapted vehicle from UAV's which has been classified as a miniature unmanned aerial vehicle. These have the size restrictions of maximum limit when it exceed, it comes under UAV. The function of these aerial vehicles can also autonomous. Modern vehicles of these types are as small as 5 cm. The development of these air vehicles are driven for military purposes, research, commercial and etc. With an aircraft of insect size, it can be used for surveillance in hazardous environment and also for aerial photography. The usage of these small vehicles get increased know-a-days than the big commercial aircrafts. These vehicles are having the advantages of easy controlling and auto performances, high efficiency, less time operations, no commercial damages and etc. Thus, the usage and performances of these vehicles are increased, the condition to increase the efficiencies such as thrust, lift or any other operations has arrived.

### *C)* Propellers for MAV

Since the invention of aircraft by Wright Brothers in 1903, the propulsion methodologies have been periodically evolved with considerable changes in the conventional methods for aerial vehicles. Though the evolution of propulsion systems began during early 90s, the contributions of many scientists and researchers helps in improvement of the efficiencies and performance of the propeller-based propulsion systems. At the same time, some of the researchers have identified that low speed engine propeller-based propulsion systems has better efficiency in compare with many other propulsion systems. This leads to the massive researches on propeller-based propulsion systems with a constant aim to improve their efficiencies and performances. The studies on the propeller- based propulsion system are very worthy at the steady state cases for Reynold's numbers greater than 350,000. To increase the thrust performance of the propeller-based MAV's. Many of the studies were concluded that there is increase in lift-curve slope followed by significant delay in stall and increase in maximum lift due to propeller induced flow. This study is carried out for inclination and fixed position of the propeller. Some of facts revealed that increase in lift-to- drag ratio for propeller inboard upward rotation at constant power settings and increase in lift coefficient of propeller. Since the influence of position and inclination of propeller on the performance was understood without considering the influence of dynamics of flow over the propeller. The merits and demerits of propeller wash were identified through a study, which was carried out for tractor and pusher configurations propeller. Though both the configurations revealed the same behavior with increased lift coefficient, due to propeller induced flow over the wing, but pusher configuration had upper hand as the effect of propeller induced flow was more intense. Also, its performance was influenced by propeller position. Further the study of pusher configuration of propeller had immense effect on the regular propeller. The study of interaction between aspect ratio and propeller for the range of incidence of Horizontal Micro Aerial Vehicles are revealed from the configuration of results by enhancing the relationship between the results of analytical and CFD simulation. With the aspect ratio to consideration, the flow effects over propeller were studied.

### D )Thrust

Thrust is a force over a system which pushes/accelerates mass in one direction which produces forward force to move in opposite direction. This can be explained through Isaac Newton's Second and Third Law of Motion.

### *E)*Computational Fluid Dynamics

CFD is a branch of fluid mechanics that uses data structures and numerical analysis to solve problems that induce fluid flows. In this process, the computers are used to perform simulations through software and calculations through iterations of the free stream flow of fluid. Validation of such software is typically performed using wind tunnel. The analytical analysis of particular problem can be used for comparison.

This paper involves three sections, the first section is estimation result through analytical part, the second section refers the Computational Fluid Dynamics simulation using Ansys Fluent software and the third section includes the relation between the first and second section.

## 2)Design of Propeller for MAV's

The design of MAV propeller are designed on the basis of aerodynamic rules such as the selection of airfoil section and the proper dimensions of the airfoil section and propeller model. The main principle of our design is to increase the thrust efficiency of the vehicle by increase the wingspan of the propeller model of the base paper [13]. This is concluded from that the lift and thrust is very much related to the wingspan of the airfoil and propeller. Our design principle is based on propeller geometry optimization for specific static condition to maximize the thrust and power for operation is minimized. The thrust estimation of the propeller is determined from general thrust equation.

### *A)* Airfoil Selection

Airfoils are uncountable which consists a high database for researchers to select the apt airfoil required for the applications. In order to find the airfoil section, many analysis are carried on three different airfoils namely, NACA0012, Selig-1223 and Clark-Y. These are the mainly used airfoils for propelled in many MAV's and UAV's. Our requirement is that the airfoil should have better maneuverability and controllability because of the propeller's flat bottom. In those above mentioned airfoils, the Clark-Y and NACA0012 are the most used sections of all the airfoil sections for the MAV propellers.

| Parameters | NACA0012 | Selig-1223 | Clark-Y |
|---|---|---|---|
| Cl max | 0.83 | 1.76 | 1.35 |
| (L/D)max | 38 at 13° | 63 at 13° | 76 at 13° |
| Stall | 13 | 13 | 14° |

**Table.1**

From table.1, the Clark-Y airfoil which is one of the type of symmetrical airfoil whose value of Cl max is 1.35 which is higher than both NACA0012 and Selig-1223, (L/D)max is equal at 13° which also maximum than those airfoil sections and stalling is occur at the angle of 14°. Thus, the Clark-Y airfoil section is selected for the propeller design.

| % | 0 | 2.5 | 5 | 10 | 20 | 30 | 40 | 50 | 60 | 70 | 80 | 90 | 100 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Upper | 3.5 | 6.5 | 7.9 | 9.6 | 11.4 | 11.7 | 11.4 | 10.5 | 9.2 | 7.4 | 5.2 | 2.8 | 0.1 |
| Lower | 3.5 | 1.5 | 0.9 | 0.4 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

**Table.2**

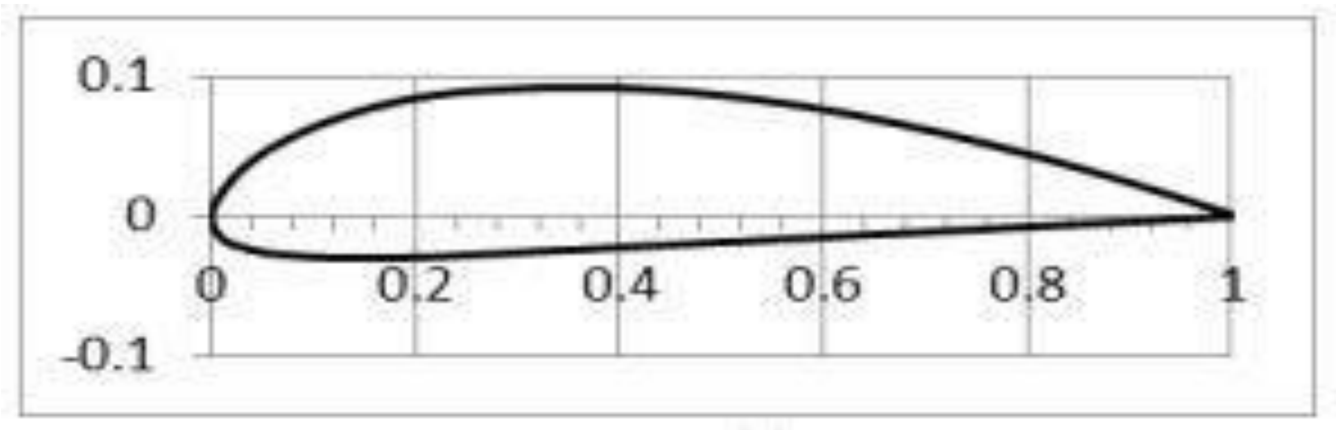


**Figure.1**

The above table.2 and figure.1 explains the co-ordinates of the airfoil section called Clark-Y which is choose for the design of the propeller model. The co-ordinates also known as nomenclature of the airfoil section. The Clark-Y airfoil section is one of the symmetric airfoils and have many advantages than any other airfoil used for MAV propeller.

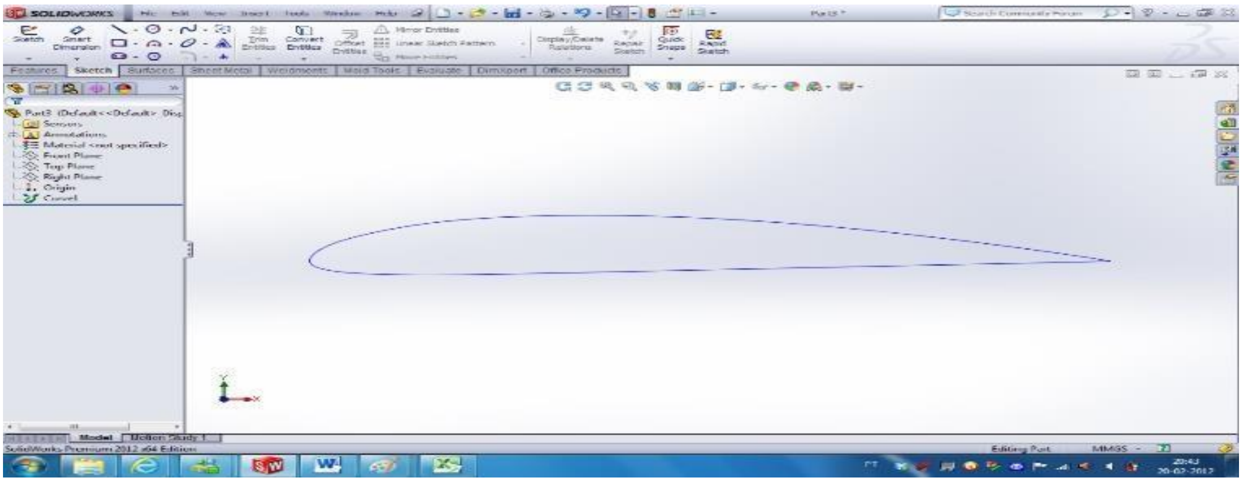


**Figure.2**

The figure.2 shows the Clark-Y airfoil section is imported to the Solidworks. This importing of airfoil section is done to design the propeller model.

### B)Propeller Design using Solidworks

The three-dimensional two-blade propeller model with an airfoil cross-section of Clark-Y section is created by using the Solidworks 2019 software. This software is very easy learning higher than both NACA0012 and Selig-1223, (L/D)max is equal to 76 at 13° which also maximum than those airfoil sections and the stalling is occur at the angle of 14°. Thus, the Clark-Y airfoil section is selected for the propeller design.

**Table.2**

| Description | Dimensions |
|---|---|
| Airfoil Section | Clark-Y |
| Span | 500 mm |
| Chord | 200 mm |
| Thickness | 0.119 c |
| Camber | 0.43 c |

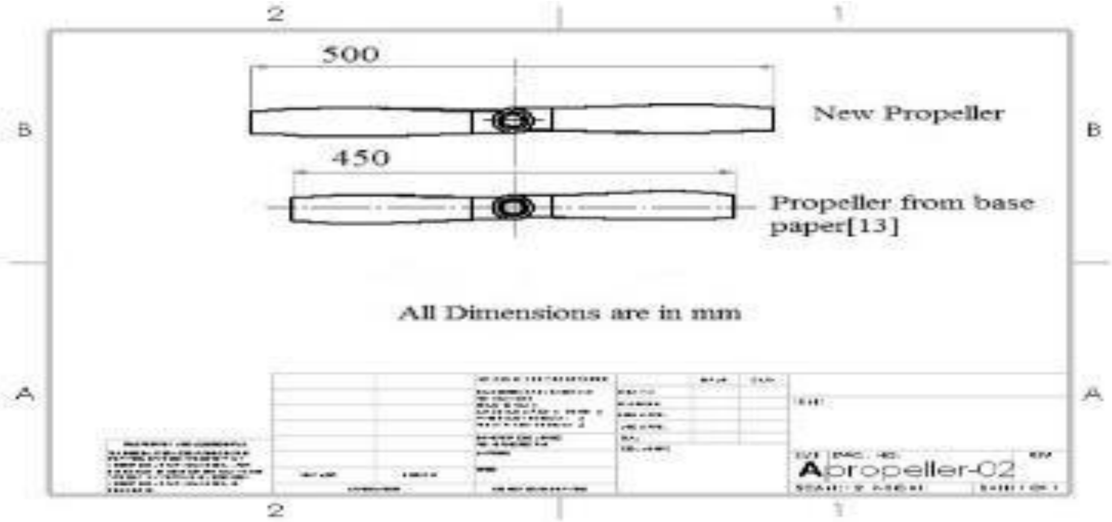



**Figure.3**

The above table.2 and figure.3 clearly shown the dimensions of the propeller. The wingspan of the airfoil and propeller is 500mm. The length of the chord and camber is 200mm and 0.43c respectively while the thickness is 0.119c. After the model of propeller is designed in Solidworks software, save the propeller model in IGES format which helps in importing the geometry in Ansys fluent.

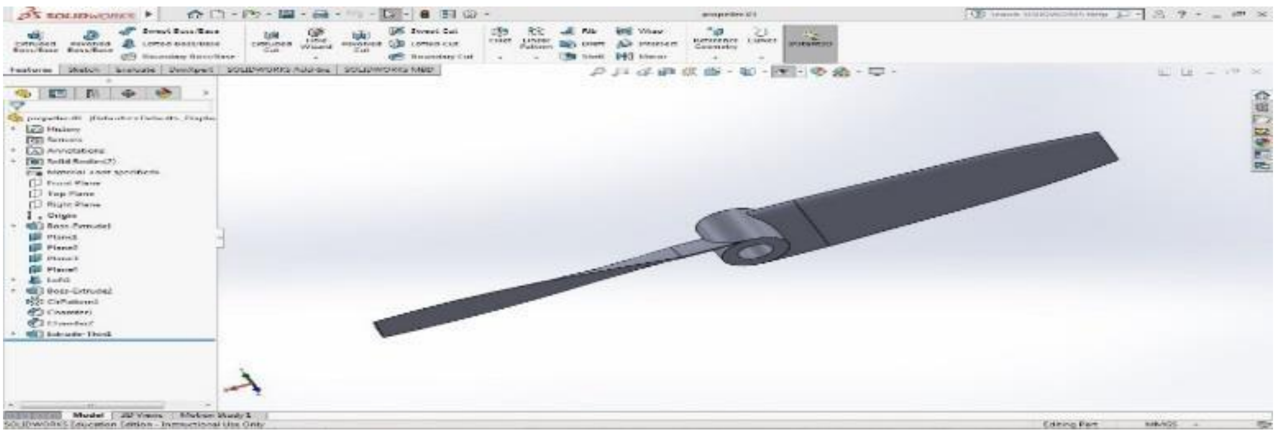


**Figure.4**

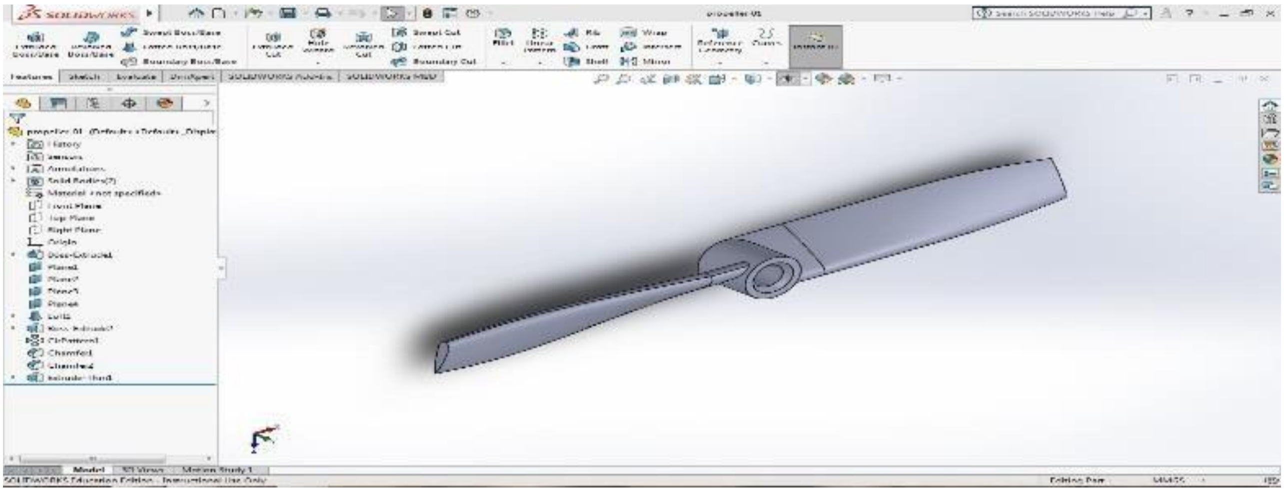


**Figure.5**

The figure.4 and figure.5 shows the full model of proposed propeller design drawn in Solidworks using the airfoil section of Clark-Y whose co-ordinates and nomenclature are shown in table.1, table.2 and figure.1. The propeller model is drawn with the dimensions from the table.2 and figure.3.

**3)Estimation of Propeller Static Thrust**

General two blade propeller thrust equation F was derived by Gabriel Staples is used in this study for calculation thrust for different RPM. This Propeller Static Thrust equation is very complex and limited to manufacturers formulations. In addition to this, the equation has 90 percent and above accuracy for two blade propeller and 90 percent accuracy for three blade propeller and 85 percent for five blade propeller.

Consider that the general two blade propeller static thrust obtained, equation F is mentioned below,equation F is mentioned below,

$$F = 0.00000004392399 \times rpm \times [d^{3.5} \times \sqrt{p}] \times [0.000423333 \times rpm \times p - v]$$

- Where, F=Thrust
  - rpm=revolution per minute
  - d=Diameter of propeller
  - p=pitch of propeller in inches
  - v=velocity in m/sec

### 4)Computational Fluid Dynamics

CFD is a branch of fluid mechanics that uses data structures and numerical analysis to solve problems that induce fluid flows.In this process, the computers are used to perform simulations through software and calculations through iterations of the free stream flow of fluid. Validation of such software is typically performed using wind tunnel. The analytical analysis of particular problem can be used for comparison. Here, the simulation of proposed propeller is done using Ansys 18.1 (fluent). Treatment of engineering problems involves three parts namely, creation of model, solving problems and analyzation of result. Finite Element programs like Ansys are also divided into three processors. They are preprocessor, solution processor and postprocessor.

### A)Preprocessor

The preprocessor is the first and main part of the CFD which involves the importing of geometry using IGES format or creating the model and editing the model. These geometries can be used to create other geometries by making use of Boolean operations. While creating the geometry, the key idea are simplify generating the element mesh. Every element has to be assigned by a particular material and are defined by its material constant and then mesh generation taking place.

### B)Postprocessor

Boundary conditions are usually applied over an element or on nodes. The loads and any other inputs in Ansys may also be edited from the preprocessor. The solutions of the problems can only be, if the problem is defined. The visualization of results are done by plotting the shape of stress or geometry. The list of results are found with file printouts and tabular listings. During the analysis process, the user will communicate with Ansys using a graphical user interface.

### *C)*Simulation of Proposed Propeller on CFD

The model of two blade propeller was modelled in the Solidworks software. Then, comes to simulation process which is done with the help of Ansys software especially Fluent. First step is to import the IGES file of propeller in the geometry section and update the geometry file. This simulation part involves three main section namely creation of domain and rotatory axis, mesh generation and fixing boundary conditions for flow over the object. Followed these processes, we can obtain the required result of thrust for various RPM which will represented in Graph form very easy to identify.

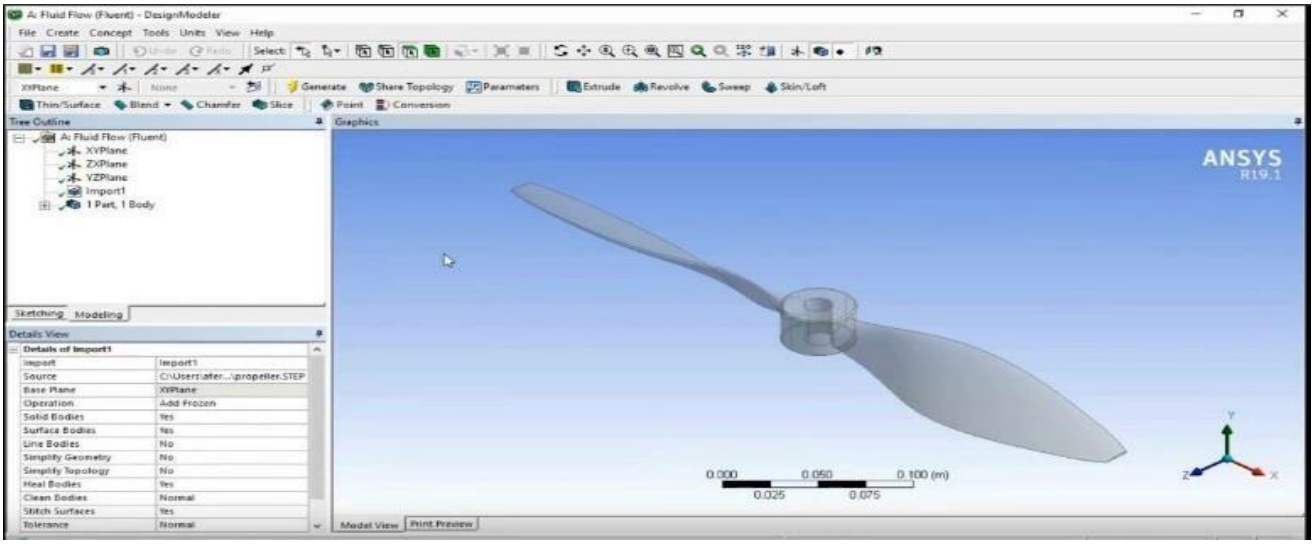



**Figure.6**

The figure.6 is the propeller model imported to Ansys which is done by the IGES format of the propeller designed using Solidworks. After import of geometry, click generate option to display the model which is shown in above figure. In this geometrical import of Ansys, we can edit the geometry. Then, the enclosures and Boolean are created to make the model ready for mesh generation.

## *D)*Mesh Generation

The generation of mesh is followed by the creation of rotatory axis for the propeller to rotate and creation of rectangular for the propeller. The creation of computational flow (CFD) domain is important, which is very much helps in providing input with a symmetrical condition. In this condition, we have created two meshes one is close to propeller and another one is far away the propeller were generated to visualize flow near and away from the propeller. Close to the wall surfaces, a fine mesh was used. The outer boundaries were relatively coarser but fine enough to ensure proper 3D streamline flow over the propeller blades. Tetrahedral elements were used with a mesh count of approximately 15, 00,000. Velocity was specified as the inlet boundary and pressure was specified at the outlet boundary. Generated mesh and the boundary conditions. The unstructured CFD mesh for airflow domain (enclosure) is developed consists of tetrahedral, pyramidal, hexahedral, and/or prismatic elements with inflation layers. The inflation layer was well applied especially for mesh detailing near each wing boundaries.

Twelve layers of mesh inflation were well developed on the wing wall with the transition ratio and growth rate at 0.77 and 2.2 respectively. The tetrahedral mesh of optimized mesh (≈500,000 elements) with inflation layers is created.

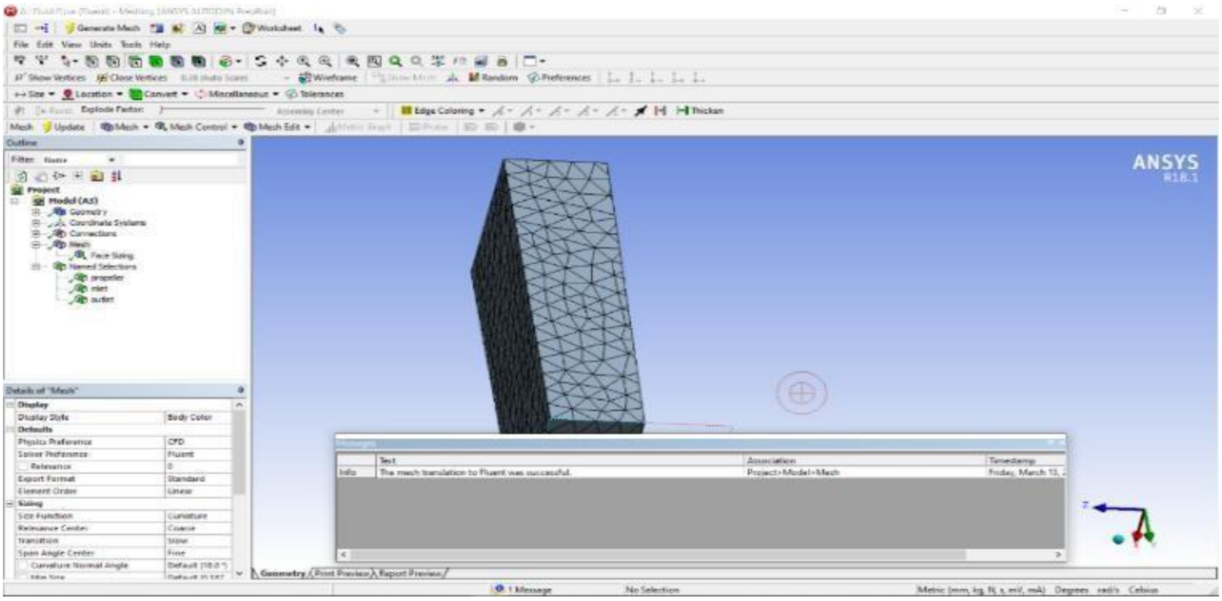


Figure.7

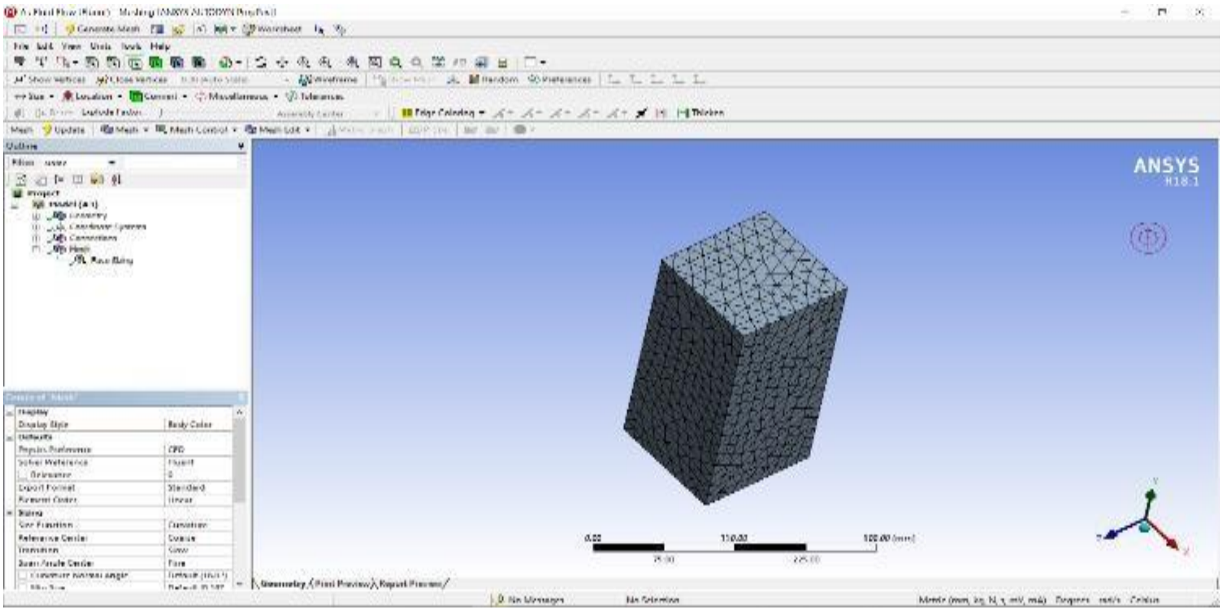


Figure.8

The figure.7 shows that the tetrahedral mesh is successfully done to the whole domain and propeller model inside it. The figure.8 shows the mesh generation of the domain. This mesh clearly separate the whole model into small tetrahedral parts to take each and every sections has to be in consideration while the determinations of the results for each velocity of air through the iteration process.

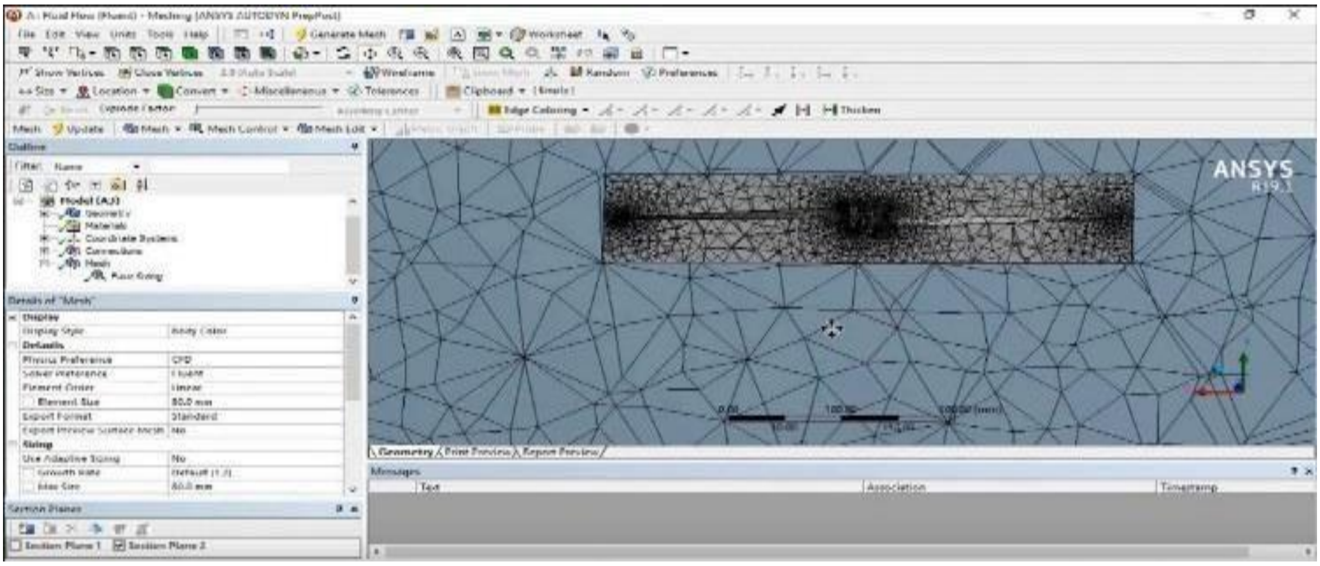


**Figure.9**

The figure.9 shows the zoomed section of mesh generation of the propeller model which clearly show the mesh generation in and around the propeller model.

**E)Boundary Condition**

The symmetrical boundary condition applied on the domain created around the propeller model. The inlet and outlet of the domain are located and the inlet velocity is given at the inlet geometry and pressure is visualized at the outlet velocities. The inlet flow velocity of the fluid which is choose as air was specified at the inlet which is equivalent to Re=1.7*10^5. The thrust has to find for different velocities by changing the inlet velocity at the inlet of the domain. Zero pressure boundary condition is implemented at the outlet to ensure airflow continuities. The symmetrical wall and side walls imposed as symmetrical and slip surface boundary conditions. Non-slip boundary surface imposed on wing surface and automatic wall function is fully employed to solve the flow viscous effect.

| Boundary Conditions | Inputs |
|---|---|
| Solver | Pressure Based |
| Velocity Formulation | Absolute |
| Time | Transient |
| Model | Viscous(reliable k-epsilon, scalable wall function) |
| Material | Fluid(air), Solid(Aluminium) |

**Table.3**

The table.3 explains the boundary conditions given to propeller model and rectangular domain of the propeller. Before the entering of boundary conditions, the inlet and outlet sections are determined in the domain of the model. The velocity of air is get pass from inlet to the outlet of the domain. The propeller section is created with the rotatory axis to get rotation while the hitting of air on the propeller blade.

## 5)Result and Discussion

### A)Velocity Contour

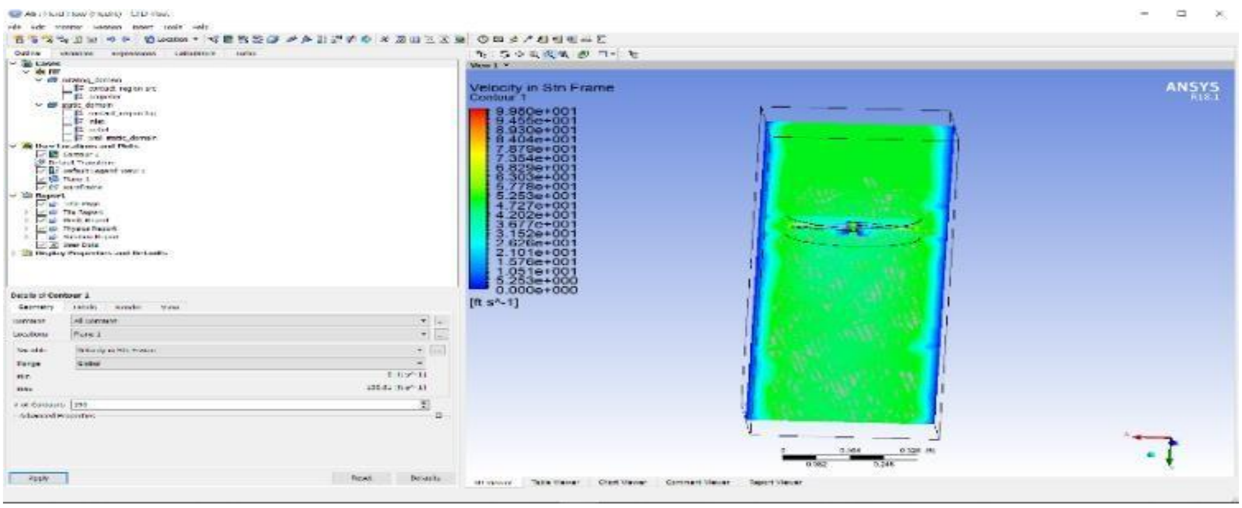



**Figure.10**

Figure.10 represents the velocity of fluid flow over the surface of the surface of the propeller. The image shows that the flow after the surface is turbulence and the flow velocity over the design is at the safe point and so the design is safer according to the analysis that have been done over the propeller.

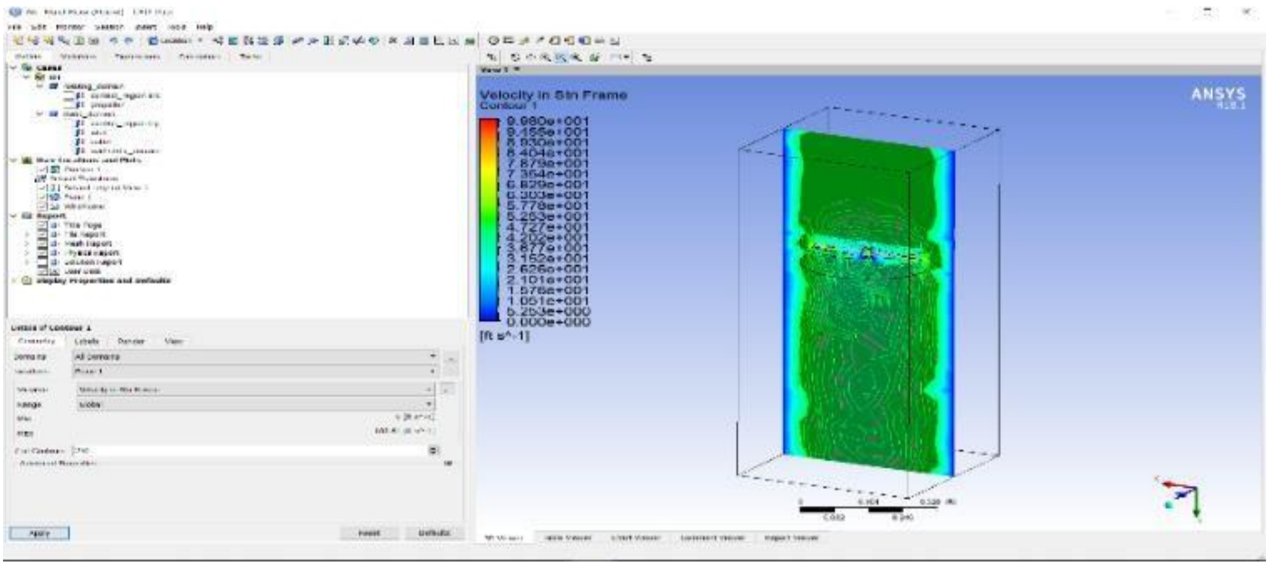



**Figure.11**

The above figure.11 shows the flow of fluid over the propeller and the velocity changes occurred during the fluid flow. The lines in the above image shows the velocity disturbances over the proposed propeller and the domain created around the propeller model due to fluid flow,

### *B)*Pressure Contour

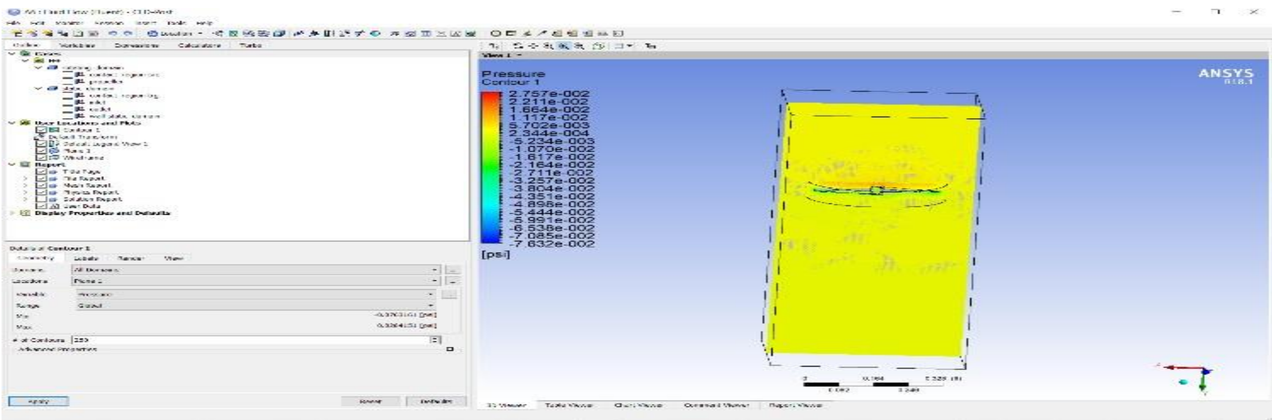



**Figure.12**

The figure.12 represents the pressure difference and pressure distribution over the propeller surfaces from the atmosphere while certain amount of velocity of air hitting the propeller. The pressure over the propeller is below the maximum pressure that the propeller can withstand.

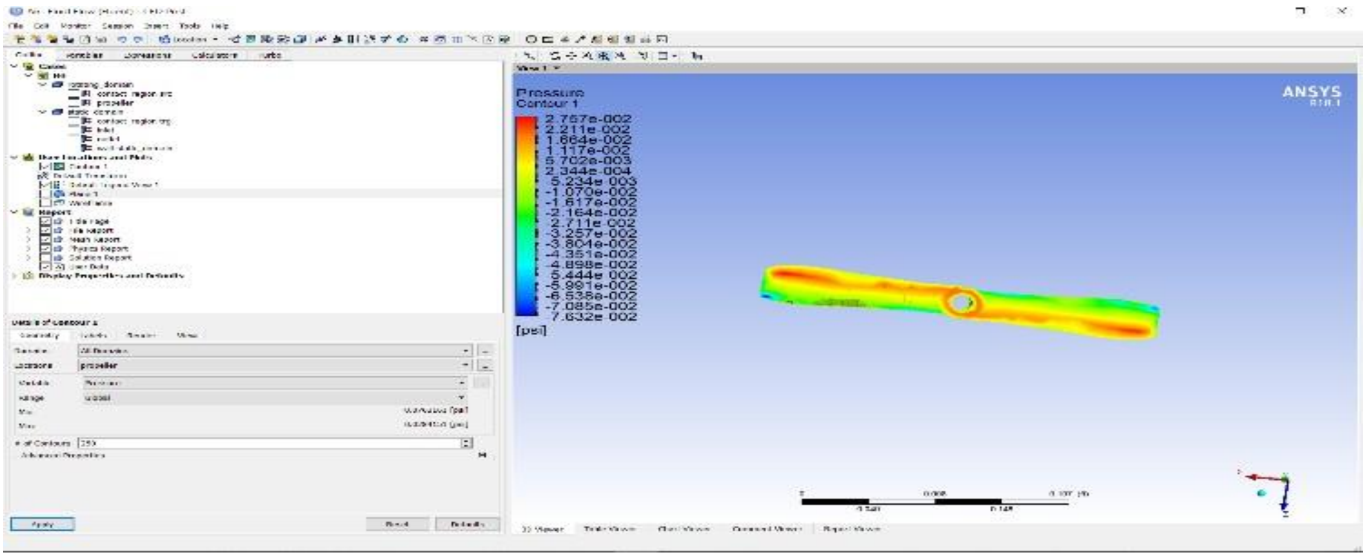



**Figure.13**

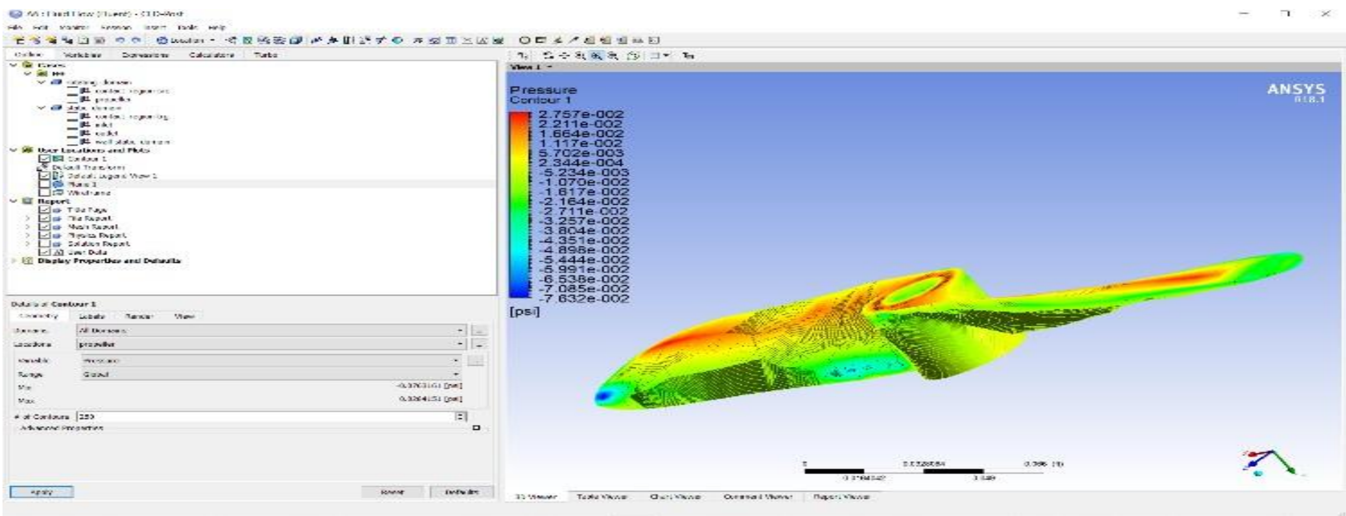



**Figure.14**

The figure.13 and 14 shows the pressure acting over the propeller surface and the color differences represents the pressure acting at each points of the propeller. The average pressure over the surface is below the maximum pressure that the propeller can withstand. The fig.14 shows the closer view of the pressure contour over the surface of the propeller for the clear visualization of the part. The pressure contour is nothing but is also known as pressure distribution over the propeller blade. The color combination on the propeller explains that the yellow and green color regions have very less pressure contribution and the small red color region have high pressure contributed area, but this region is only small in area. Thus, the pressure distribution over the propeller through the velocity of air are clearly explained.

### *C)*3-Dimensional Streamline Flow over the Proposed propeller

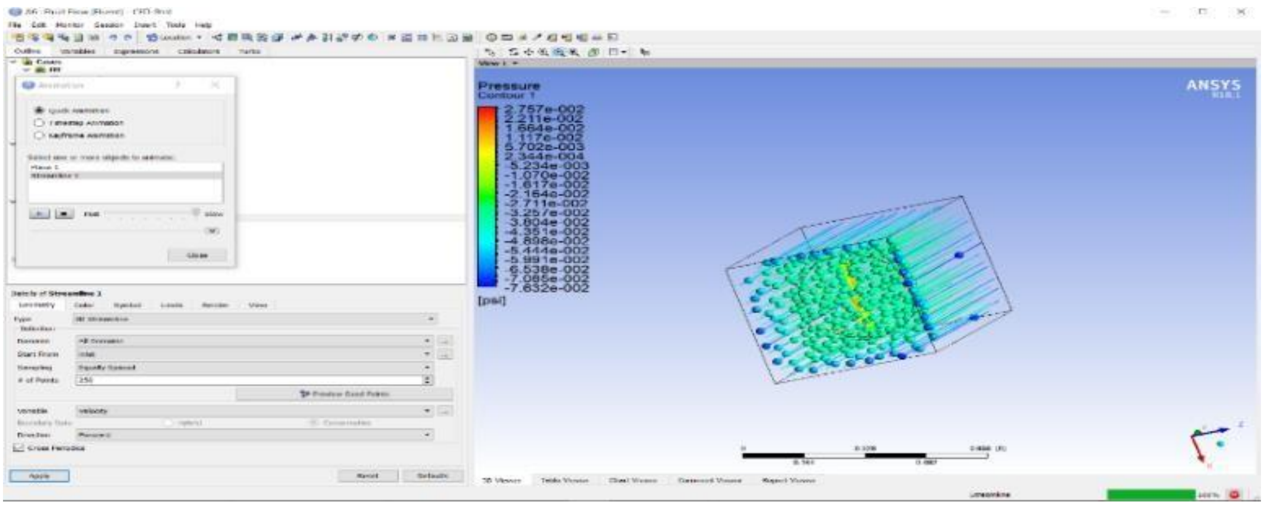



**Figure.15**

The figure.15 represents the streamlined flow of fluid over the surface and blade of the propeller. The flow of air over propeller and the distribution of air are clearly shown through the streamline flow over the propeller. The rotation of the propeller through the turbulence flow and the changes of turbulence flow because of the rotation of propeller are clearly shown through this streamline flow diagram. The result of the animated view of the fluid flow and also represents whether the flow is under safe condition. The green color in the above image represents that pressure difference and pressure distribution and flow are safer. The blue color balls refer that the regions does not affected by the rotation of propeller and the region have the continuous flow of air.

### *D)*Graphical Representation of Lift and Drag vs RPM

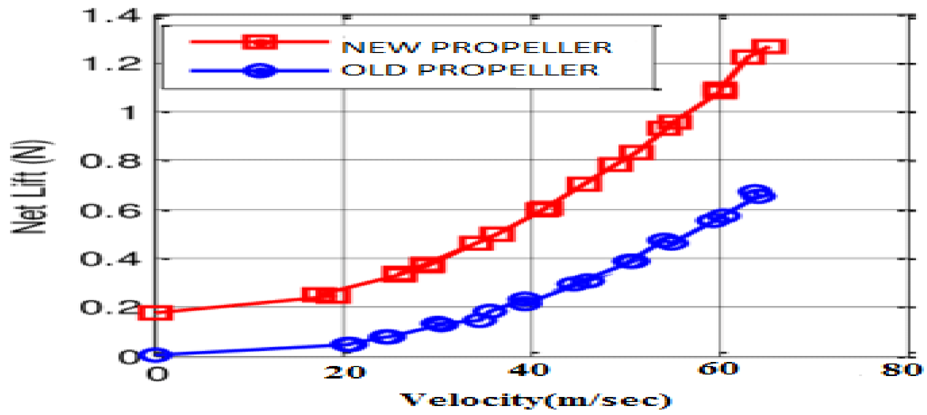
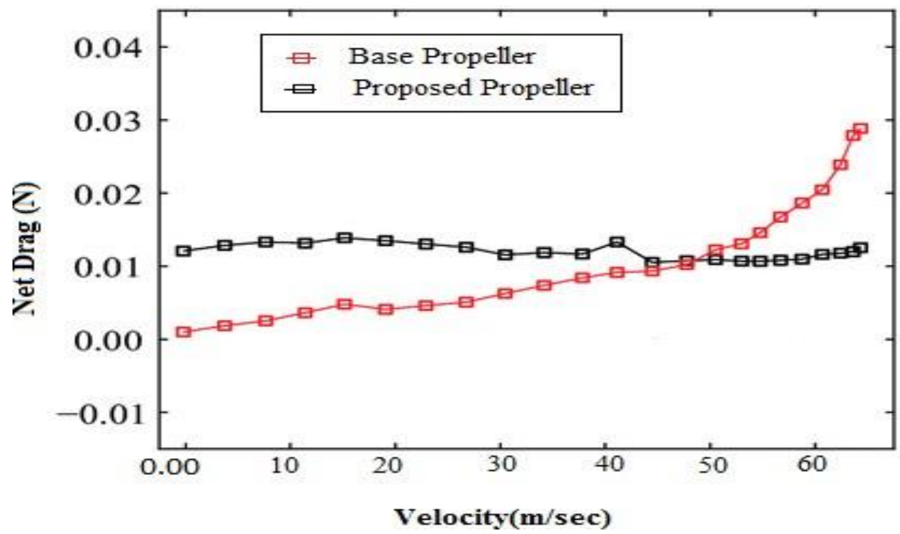
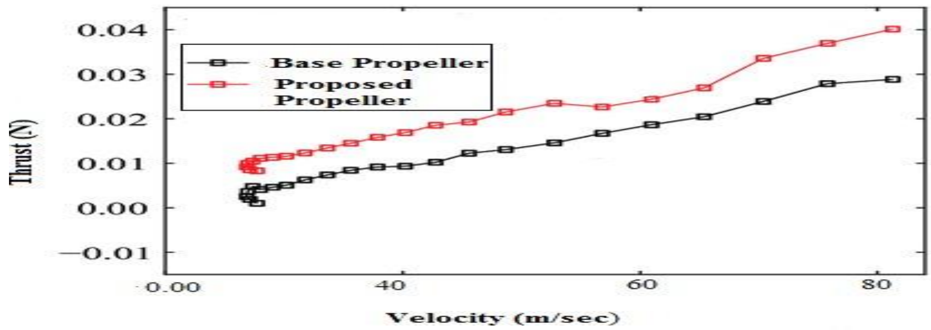



**Figure.17**

The figure.17 shows the differences in lift between the base propeller and the proposed propeller. The propeller which have proposed produced more lift than the base propeller from reference paper.

**Figure.18**





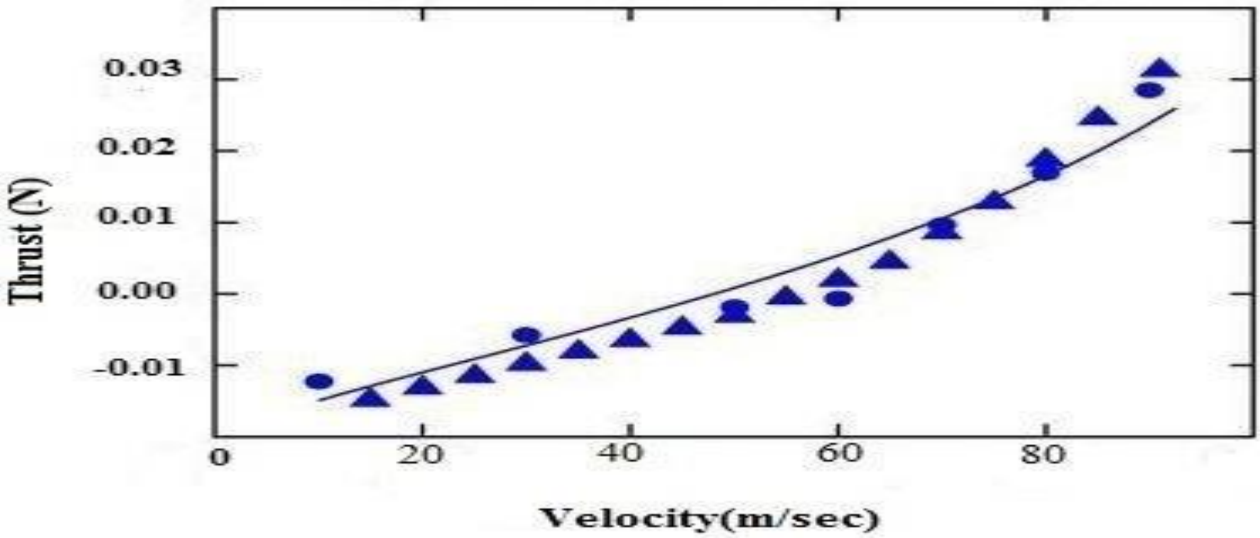




Figure.18 shows that propeller which is proposed in the reference paper attains more drag along with increase in velocity of the propeller. The propeller which proposed attain the drag-coefficient remains constant and at certain point it reduces with increase in velocity of the propeller.

**E)Estimated Result through Analytical Process**

Using the above formula of general thrust equation, the force for different and various RPM are calculated analytically for the designed two blade propeller. These values are plotted in graph which is represented below,

**Figure.19**

The figure.19 shows that the increase in thrust along with increase in velocity of the propeller. By constantly increasing thevelocity of the propeller, the thrust increases.

**F)Result of CFD Simulation Process**

From the above simulation and analysis process using Ansys Fluent, the thrust force for various RPM are calculated for the designed two blade propeller. These values are plotted in graph for the easy approach which is represented below,

**Figure.20**

The figure.20 represents the comparison of thrust and velocity differences between the propeller from the reference paper and the proposed propeller.

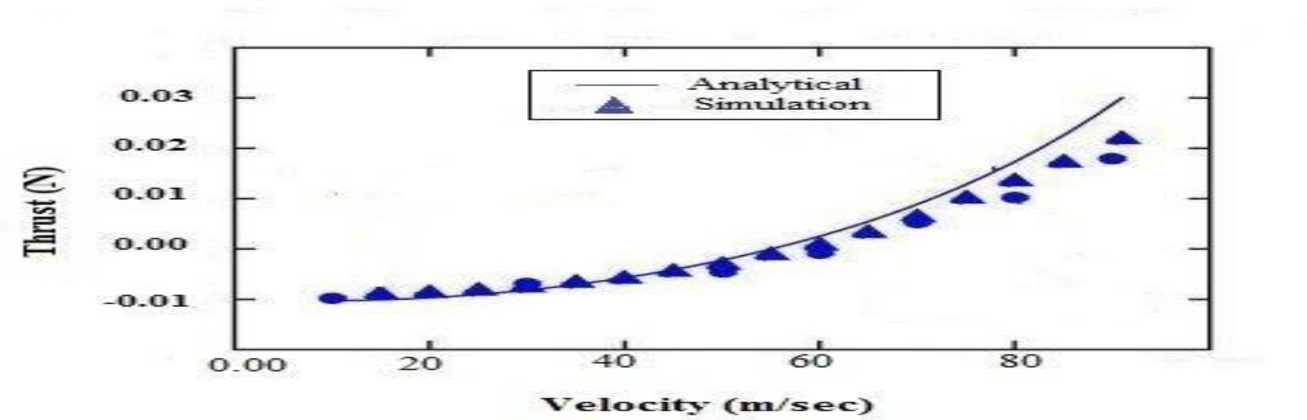



**Figure.21**

The line in the graph of figure.21 refers the analytical calculations result and the triangle shapes refers the simulations result. There is only very small and minute deviations between the two graphs.The thrust force in the analytical process starts at 1000 rpm as same as the simulation process and attain the rapid increase at 5000 rpm are also same for both analytical and simulation but have slight changes only at 6000 rpm. This slight cannot attain big disadvantages so that the designed propeller with changes in wingspan gives more efficiency.

## 6)Conclusion

A series of analytical and simulation tests were carried out with various velocities in rpm using the Thrust equation and Ansys Fluent software respectively. Then, the results of these tests were compared with the previous results from the journals. As same as in the manufacturing of blades and propeller using aluminium for commercial aircrafts, this same idea can be useful for UAV's and MAV's too. Because the aluminium based propeller are hardly used for micro vehicles. With making the propeller structure with aluminium metal, we get propeller structure stronger than the carbon fibre materials, wood and etc. And with change in wingspan and chord length of the propeller, we can obtain more thrust force. This is concluded with the relationship between the results of analytical procedure carried out using general thrust equation and simulation test using Ansys Fluent.

## 7)References

[1] Gavin K, Ananda, Robert W. Deters and Michael S. Selig"Propeller- Induced Flow Effects on Wings of Varying Aspect Ratio at Low Reynolds Numbers", AIAA Journal, ISBN-9781624102882.

[2] L.L.M. Veldhuis, "Review of propeller-wing aerodynamic interference", ICAS 2004, 24th International Congress of the Aeronautical Sciences.

[3]Catalano, F. M, "On the Effect of an Isolated Propeller Slipstream on Wing Aerodynamic Characteristics", ActaPolytechnica, Vol. 44, No. 3, 2004, pp. 8-14.

[4]J. B. Barlow, A. Pope, and W. H. Rae, “Low Speed Wind Tunnel Testing”, 3rd ed. (Wiley-Interscience, 1999).

[5]Kwanchai Chinwicharnam, David Gomez Ariza, Jean-Marc Moschetta, Chinnapat Thipyopas, "Aerodynamic Characteristics of a Low Aspect Ratio Wing and Propeller Interaction for a Tilt- Body MAV", International Journal of Micro Air Vehicles, Vol. 5 No. 4, pp. 245-260.

[6]Brian J. Gamble, Mark F. Reeder, "Experimental analysis of propeller interactions with a flexible wing micro-air-vehicle", Journal of Aircraft, Vol. 46, No. 1 (2009), pp. 65-73.

[7]Alex M Stoll, "Comparison of CFD and Experimental Results of the LEAP Tech Distributed Electric Propulsion Blown Wing", 15th AIAA Aviation Technology, Integration, and Operations Conference, AIAA AVIATION Forum, (AIAA 2015-3188).

[8]Kline, S. J, and F. A. McClintock. "Describing Uncertainties in Single-Sample Experiments", Mechanical Engineering, Vol. 75, No. 1, January 1953: 3-8.11

[9]S. Majumdar, “Pressure based Navier Stokes solver for three- dimensional flow in hydrodynamics and low speed aerodynamics application”, Proc. 3rd Asian CFD Conference, Bangalore, 1:137–146, 1998.

[10] B. N. Rajani and S. Majumdar, “Numerical simulation of turbulent flow past a circular cylinder”, NAL PD CF 0805, 2008.

[11]B. N. Rajani and S. Majumdar, “Large Eddy Simulation of flow past circular cylinder in the lower subcritical regime (Re = 3900)”, NAL PD CF 1004, 2010.

[12]S. Srinivasan, K. Bhaskar, R. Supreeth “Experimental Aerodynamic Investigation of an UAV wing with Wing Mounted Propellers” International Journal of Recent Technology and Engineering(IJRTE) ISSN:2277-3878, Volume-8, Issue-1C, May 2019

[13]Arivoli Durai “Experimental Investigation of Lift and Drag Characteristics of a Typical MAV under Propeller Induced Flow” Experimental Aerodynamics Division,National Aerospace Laboratories, Bangalore.