Numerical Studies on the Effect of Span wise Position of Maximum Blade Chord on the Aerodynamic Performance of an MAV Propeller

Sreenath.N[#1], Mr.Kalyan Yadav.S[#2], Mr.Subbaramu. S[*3]

[1]*M.Tech, East point college of engineering and technology, Bengluru, India*
[2]*Assistant Professor, East point college of engineering and technology, Bengluru, India*
[3]*Assistant Professor, MSRSAS, Bengaluru, India*

sreenathn051@gmail.com[#1]
skalyanyadav@gmail.com[#2]
subbaramu@msrsas.org[*3]

***Abstract* - In recent times the aerospace industry has taken increasing interest in micro air vehicle (MAV) technology. Growing interest in micro-air-vehicles has created the need for improved understanding of the relevant aerodynamics. The very challenging task is, design of propeller for low Reynolds number applications with proper matching of propeller characteristics with that of small, light weight electric motors with appreciable increase in efficiency. The flight regime of micro-aircraft poses numerous challenges for aerodynamic analysis and design, but little experimental or computational work exists for aerodynamic surfaces operating at ultra-low Reynolds numbers. This paper involves the performance evaluation of a mav propeller through cfd analysis. The steady state flow analysis was carried out using the commercial cfd solver fluent14 for obtaining results, plots and contours. The cfd results were validated using experimental values of the literature. Later the baseline geometry of micro propeller was modified by changing the position of maximum chord in span wise position and then compared in terms of efficiency, torque coefficient and thrust coefficient against advance ratio with base line.**

Key words: Micro air vehicle, low Reynolds number, span wise position of max chord, thrust coefficient, torque coefficient, efficiency.

I. INTRODUCTION

Technological feasibility follows from advances in several micro-technologies, including the rapid evolution of micro-electromechanical systems, also known as MEMS. These systems combine micro electronics components with comparably-sized mechanical elements of varying complexity to achieve useful, and often unique functionality (e.g. Integrated systems of sensors, actuators and processors). In many cases, these devices are produced with established micro fabrication techniques, providing a high degree of optimism for eventual low-cost production potential. Other maturing micro systems such as tiny ccd-array cameras, equally small infra-red sensors and chip-sized hazardous substance detectors, have been catalytic in providing the motivation for like-sized delivery platforms.

These continuous miniaturization of electrical and mechanical systems (Mechatronics) over the last decade, have catalyzed the development of Unmanned Aerial Vehicles (UAV's) and the growing interest in Micro-Aerial Vehicles (MAV's). These developments have increased the number of radio controlled (R/C) airplane hobbyists and R/C airplane designers. The hobby of flying R/C airplanes has turned into a professional sport, encouraging the hobbyist to modify their current designs. The design changes and fascination created by these MAV's have challenged the students in the universities to aim for improvements.

II. METHODOLOGY

A suitable propeller of known geometry and performance was selected from the literature [3] for CFD validation studies. The geometrical model (cad model) of the baseline micro propeller and flow domain will be prepared using solid works software. Latter the flow domain and propeller was discritised using pre-processor gambit. Then numerical simulations were carried out using the commercially available CFD solver FLUENT14.

The obtained cfd results were validated with the experimental results. Then the baseline geometry of micro propeller was modified by varying the location of maximum blade chord along the span to generate few design variants. Geometric modeling and CFD simulation will be carried out for each of the design variants using same software's. The performance of the different micro propeller design was compared in terms of efficiency, torque coefficient and thrust coefficient against advance ratio. An optimum location of maximum chord was arrived at based on propeller performance.
The following formulae is used to calculate the performance of a propeller.

Advance ratio $$J = \frac{V_\infty}{ND_p}$$

Coefficient of thrust $$C_T = \frac{T}{\rho n^2 d^4}$$

Torque coefficient $$C_Q = \frac{Q}{\rho n^2 D^5}$$

efficiency of propeller $$\eta_{pr} = \frac{C_T}{C_P} J$$

III. MODEL CONSTRUCTION AND SOLUTION

A. *Propeller design specifications*

The baseline propeller geometry was selected from the published literature on MAV propeller [3] the base line propeller was designed according to the requirements. In base line propeller the NACA16 K series- profile is used

Table 1: Design parameters of the base line

Propeller parameters	value
Propeller Diameter, D	355.6 mm
Pitch, P	152.4 mm, Fixed
P/D	0.454
Speed, N	8932.5
Static thrust	300 gmf
No of blades	2
Tip speed	20m/s
Advance ratio	0.4
Propulsive Efficiency	67%

Using the coordinates from ref. [3] the cad models was designed in a commercial cad software solid works. The propeller shaft diameter of 24mm was modelled with rotated parabolic ends with length of 24mm. Isometric and side view of the propeller is shown in the figure 1.a&b.The blade design was achieved by importing total 11 sections of airfoil coordinates from Microsoft excel into solid works and each profile was twisted as per twist distribution. Solid works is an ideal program for creating the 3d model and it is possible to export files directly into gambit software for meshing, and for analysis in fluent software

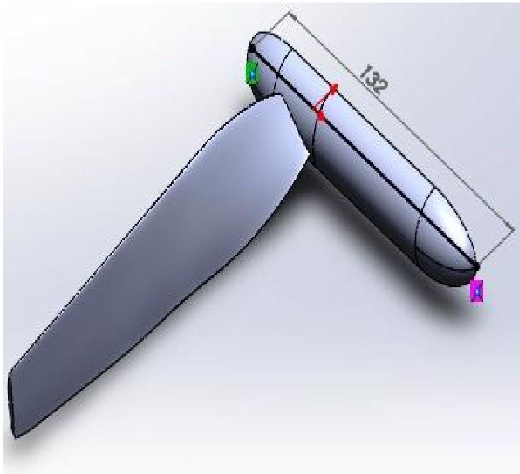

Fig 1.a: Isometric view of base line cad model

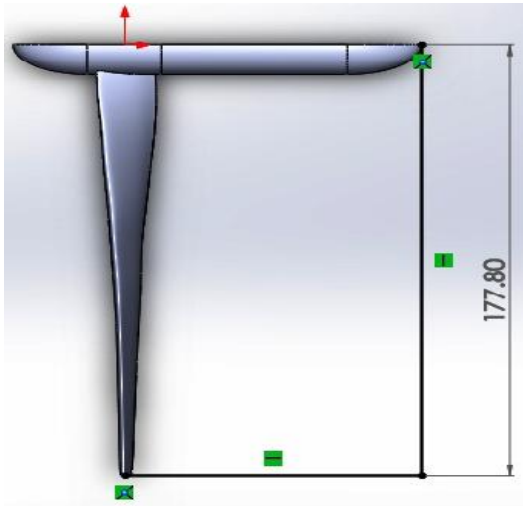

Fig 1.b: Side view of base line cad model

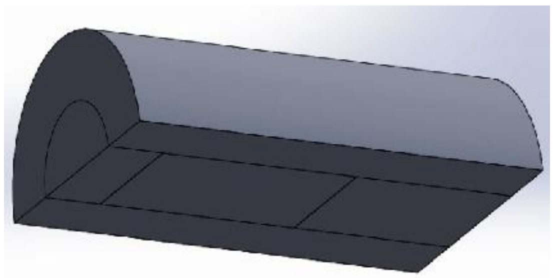

Fig 2: Flow domain around the propeller

IV. DISCRITISED FLOW DOMAIN AND BOUNDRY CONDITIONS

The flow domain (fig 2) was created around the propeller, and extending upstream and downstream to an appropriate distance in solid works software. Since the flow around and through propeller is steady and the two blades are placed symmetrically at an angular of 180 degrees, hence the flow domain was modeled with only one blade. The effect of other blade was taken care of by imposing periodic boundary condition on the other side. It is recognized that the cfd model using a moving cylindrical reference frame containing propeller geometry and connecting to a fluid interface is an accurate and effective method for determining propeller performance. The propeller was placed inside two cylindrical flow domains in order to create the moving reference frame model, required to simulate the flow around a propeller. The first inner cylindrical domain around the propeller was of diameter two times the diameter of propeller (2d), forming the rotating zone and the second larger outer cylindrical domain was of diameter five times diameter of propeller (5d), forming the stationary zone.

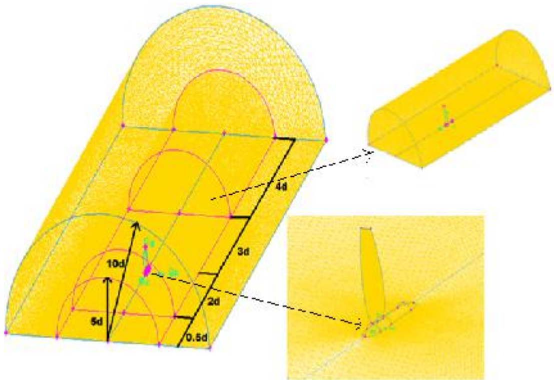

Fig 3: Discritised flow domain with propeller

The domain geometry was imported into gambit meshing software. Then different identifiers were created for the inlet, outlet, shaft, the blade tips, blade pressure and suction sides, periodic plane and outer wall. Entire flow domain is splitted in to two regions, inner and outer domain, so inner domain is as shown in the figure is created with finer mesh the grid of outer domain is coarser than inner domain. Tetrahedral elements are used. The propeller surfaces are discritised with triangular elements as shown in the figure 3.

A. Boundary conditions

In order to simulate the flow relevant boundary conditions are specified as shown in figure 4. The boundary conditions were imposed using the pre-processor Gambit 2.3 and fluent 14 was chosen as the solver. The propeller blade and hub surfaces were modeled as wall boundary conditions. The bottom plane was modeled as rotational periodic boundary condition. The outer cylindrical surfaces were specified as stationery wall. A constant free-stream velocity boundary condition was specified at the inlet boundaries. On the exit boundary, the static pressure was set to a constant value zero. The fluid is opted as moving reference frame, the rotational speeds are applied to the fluid, and the blades are considered to have relative motion with respect to the fluid. Performance analysis was carried out by varying the propeller rotational speed. Standard k-ε turbulence model was used and all simulations were converged to a residual value of 10^{-5}. The lift forces on the suction and pressure surfaces of the propeller were closely monitored during each iteration to ensure convergence.

In fluent software the material properties of the fluids used in the simulation can be defined by the user. In the present studies the fluid is air and since the air around the propeller is incompressible so the properties of the air were treated as constant. The air properties, like density (1.225kg/m^3) and viscosity (1.7894e-05 kg/m-s), are defined.

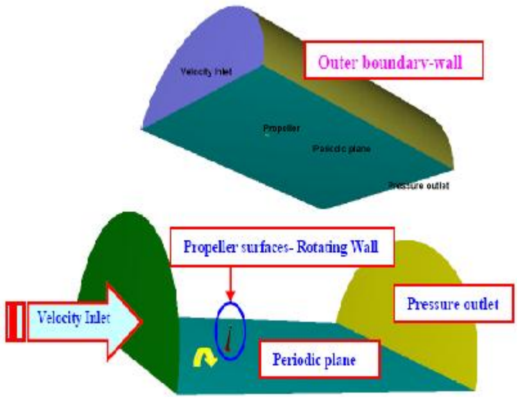

Fig 4: Boundary conditions

V. GRID INDEPENDENCE STUDY

Grid convergence studies were carried out on the propeller geometry chosen to validate the CFD procedure employed to extract the propeller performance characteristics. . In the present study, the flow domain is initially discritised with tetrahedral elements and they are later converted to polyhedral cells using a special option available in the CFD solver Fluent 14. With this option being chosen, the grid size was brought down from a grid size of 12.5, 21and 50 lacks tetrahedral cells to 9,75,000, 4,71,000 and 2,45,000 polyhedral cells respectively.

Thrust coefficients values for all three grids are plotted against advance ratio as shown in the Figure 5.

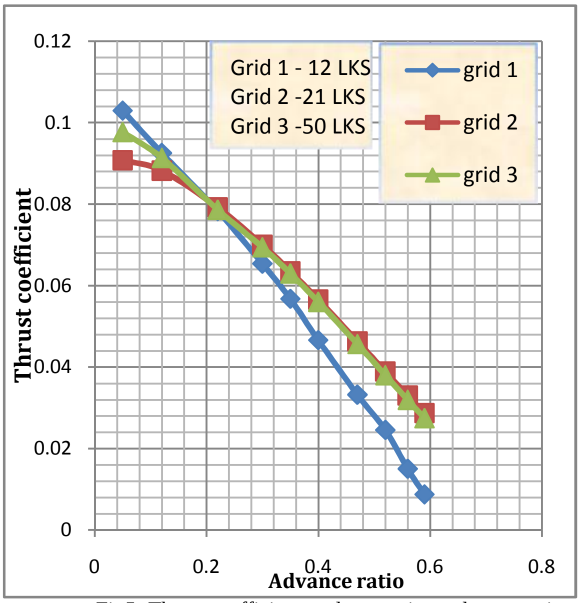

Fig5: Thrust coefficients values against advance ratio

It is observed that the results from Grid-2 and Grid-3 are close to each other. Therefore, Grid-2 was considered for further analysis.

VI. VALIDATION AND DISCUSSION OF RESULTS

for the evaluation of performance of the propeller the c

The propeller geometry and performance data given in published literature [3] was used to validate the CFD simulation. The propeller geometry was created, Boundary conditions and solver setting were applied in FLUENT 14.0 Analysis was carried out with **Grid-2**, chosen from grid independence study, for different inlet velocities ranging from 2m/s to 35m/s (different advance ratio J) to get the thrust values for each velocity inlet. The variation of efficiency with advance ratio **J** is compared with the data from experimental and CFD results [1] and as shown in Figure6.

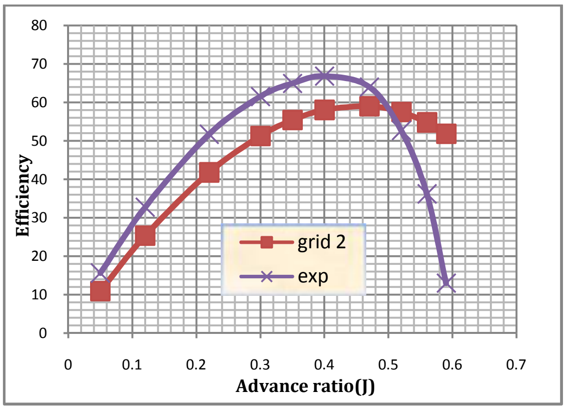

Fig 6: Comparison of efficiency between literature experiment and CFD simulation results

A good trend is found in CFD results. It is observed that the CFD simulation results are very close to the experimental values. A good agreement is found between CFD simulations and experimental data throughout the range of advance ratio from 0.1 to 0.5 with respect to efficiency and this provides confidence in present CFD simulations.

A. Design variants

TABLE: *2* Geometric parameters of base line and design variants

Design Variant	Percentage Of position	Max Chord Position, mm	Max Chord length , mm	Blade Setting Angle at R=75%, deg
Baseline design		r=49.784	32.004	64.03
Design Case-I	50%	r=88.90	32.004	74.74
Design Case-II	60%	r=106.68	32.004	77.19
Design Case-III	75%	r=133.35	32.004	79.69

To obtain design variants the base line propeller was modified by shifting the position of the maximum blade chord along span wise. In the base line propeller geometry the maximum chord position was at r=49.78mm. Three design variants were created by varying the maximum chord position at r=88.9mm (design variant case -I), r=106.68mm (design variant case-II), r=133.35mm (design variant case -III) Figure7 shows the base line propeller and its new design variants.

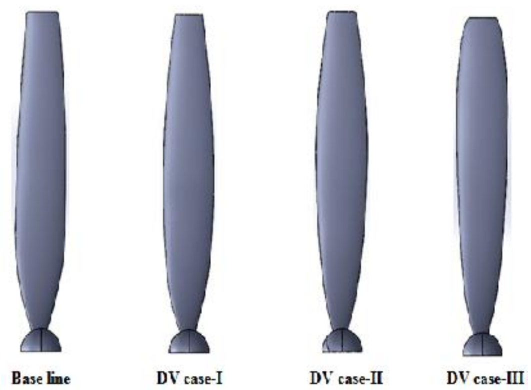

Figure: 7 Propeller designs with design variants

B. Design Variant- Analysis

For the new design variants computational analysis were done to analysis the performance of them in terms of torque coefficient, thrust coefficients and efficiency against different advance ratios.

The performance coefficients for original base line propeller and the three design variants Case-I, II and II are compared in Figure 8, 9 &10.

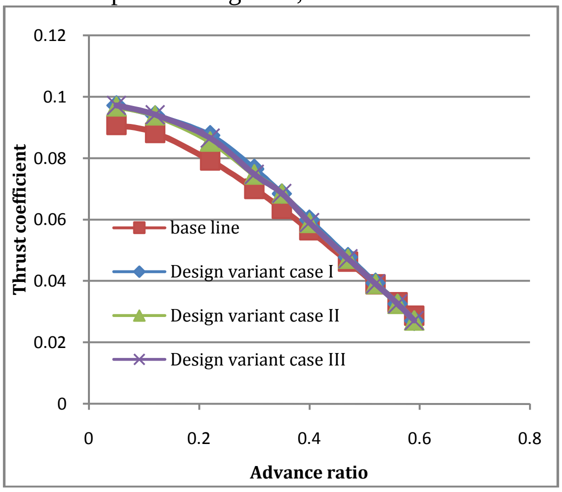

Fig:8 Comparison of thrust coefficient with J for different cases

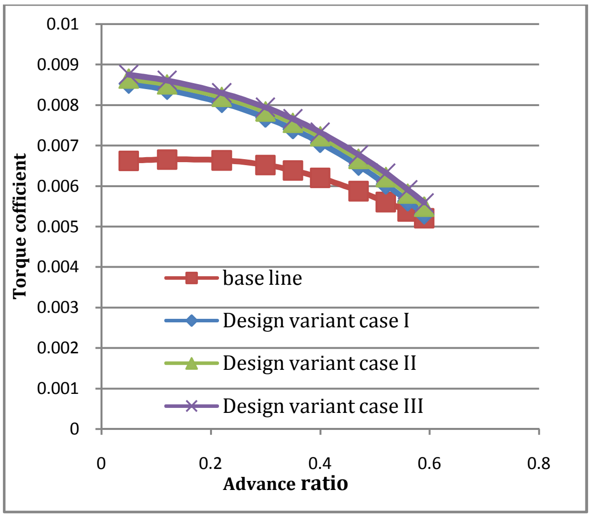

Fig:9 Comparison of torque coefficient with J for different cases

The performance coefficients for base line and design variants at peak efficiency(J=0.4, speed=8932rpm, V=21.15m/s) were plotted and compared. From the performance plots we can say that there is change in the torque, thrust and efficiency when compared with the baseline. There was slight increase in thrust and torque by changing the maximum chord position to 60% and 75% from 28% of the total length of the span. But there was decrease in efficiency of the design variant when compared to base line.

Though there was increase in the torque and thrust coefficients when compared to base line but there is no appreciable change in between the design variant cases.

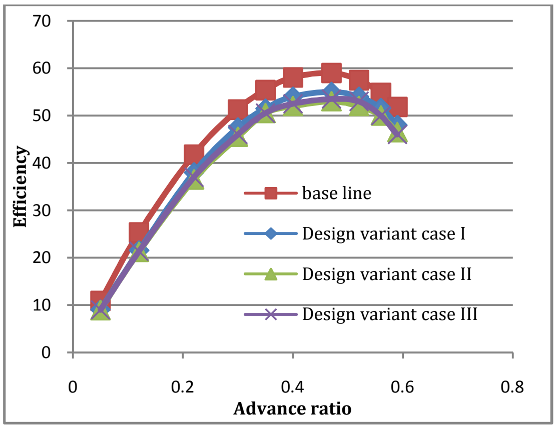

Fig:10 Comparison of Efficiency with J for different cases

Above figure shows the efficiency plots for different cases. The base line peak efficiency was 59 % at an advance ratio (j) of 0.4 and speed of 8900rpm. After shifting the maximum chord to 50%, 60% and 75% of total span of the blade, the obtained peak efficiency was 55%, 52% and 53.5% respectively, where decrement in efficiency was found.

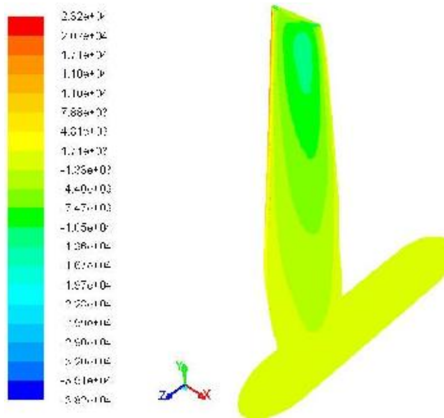

Fig11:Static Pressure Contours for Base Line and Design Variant Case 1

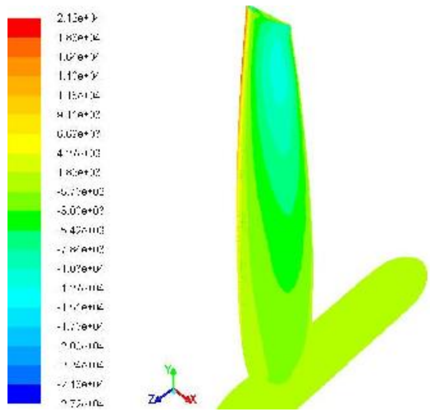

Fig12: Static pressure contours for base line and design variant case 1

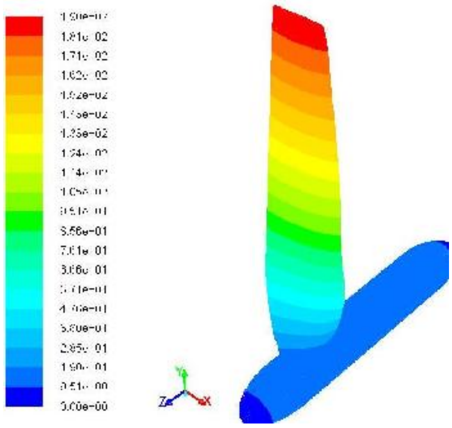

Fig: 12 Velocity contours for base line

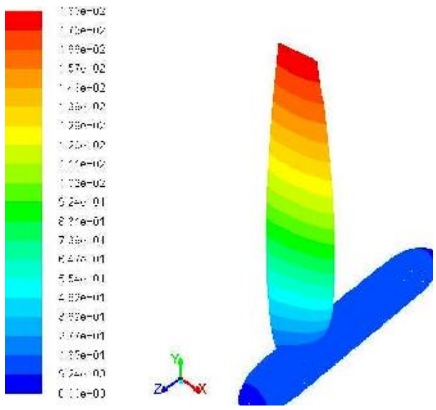

Fig13: Velocity contours for design variant

VII. CONCLUSION

Numerical studies were performed on the propeller by changing the position of maximum chord to different locations along the length of the span. Then the design variants were carried out by changing the position of the max chord. In the first case the max chord was shifted to 50% of the total length of the span. The obtained peak efficiency was 55%. In the second case, where max chord at 60% the peak efficiency was 52% and in the third case, max chord at 75 % of total length of span the peak efficiency was 53.5%.It is observed that there is no significant change in the propeller performance by shifting the position of the max chord along the length of the span, where the efficiency of the design variant cases was almost equal to base line design. Thrust coefficient is more for all 50%, 60% and 75% design cases when compared with base line and hence the new design thrust is increased.

Also for all the design variant cases the torque is more than base line.

Even though thrust is more, efficiency is decreased because of increase in torque.

VIII. REFERENCES

1. Peter J. Kunz, 2003, Phd, Stanford University **"Aerodynamics and design for ultra low Reynolds number flight".**
2. Prathapanayaka R, Vinod Kumar N, Krishnamurthy S J, CSIR-National Aerospace Laboratories, Bangalore**.**2011." **Design, analysis, fabrication and testing of mini propeller for mav's.**
3. Joshua T.Nelson ,2009, Batchelor Of Art , Bethany College. **"Practical modifications for a low Reynolds number Propeller applications".**
4. Adam Smedresman, Derrick Yeo And Weishyy,2011, Department of Aerospace Engineering, University of Michigan, Ann Arbor, Michigan, **"Design, fabrication analysis and testing of a mav propeller".**
5. Francis Barnhart, Michael Cuipa and Daniel Stefanis**,**2004**. "MAV design with low Reynolds number airfoils".**
6. Ryan Turner, 2010 SEIT, Aeronautical Thesis & Practical Experience.**"Design and optimisation of a propeller for a mav"**
7. Mihaela Amoraritei naval hydrodynamic department, university of Galati "**Application of CFD in analysis of conventional propeller and azimuth thrusters"**
8. Craig D Paxton, Peter J Gryn, Erisak. Hines, 2003, Dept of Mechanical Engineering, University of Miami **"High efficiency forward swept propeller at low speed"**
9. Yakov Sh. Flaxman, 2007, Moscow Institute of physics and Technology, Russia **"Experimental investigation of propellers at low Reynolds number"**
10. Ian P.Tracy,2011. Massachusetts Institute Of Technology. "**Propeller Design And Analysis For A Small, Autonomous UAV"**
11. Jurij Sodja, 2007, University of Ljubljana." **Turbulence models in CFD"**
12. Balaji, J. V. (2008). Unpublished Master Thesis, UK: Coventry University."**Design and performance evaluation of a propeller for micro- air vehicle application".**
13. Monal Pankaj Merchant, Wichita state university,2004 **"propeller performance measurement for low Reynolds number UAV applications".**

14. Sagar Ranjanagi ,MSRSAS, Peenya Banglore, 2013 **"Parametric studies on micro propeller of MAV application through numerical simulation"**
15. John B Brandt and Meichel S Selig ,2011, University Of Illinois , USA **"Propeller performance data at low Reynolds number"**
16. Mario Heene, Master of Science, Stockholm,Sweden 2012 **,"Aerodynmic propeller model for load analysis"**

