Constant Pitch Propeller Design for Low Subsonic Airplane

Arif Ahmed Sohel[1], Md. Abdus Shamir Talukder[2] and Arif Ahmed Sohel[3]

1

Received: 13 December 2013 Accepted: 31 December 2013 Published: 15 January 2014

Abstract

Constant pitch propeller whose blade angle is fixed with respect to hub is suitable for low speed airplane. The objectives of this thesis are to determine thrust, torque and the performance of the low subsonic airplane propeller. The thesis has been carried out by the two different approaches- Analytical and Computational Fluid Dynamic (CFD) simulation. The Analytical Method using ?Propeller Blade Element Theory? is one of the most effective methodologies available for determining the thrust and torque produced by the propeller. On the other hand, the Computational Fluid Dynamic simulation has been used to simulate and capture the performance of the propeller. Through both methodologies, the propeller performance was analyzed and compared. The results obtained were tabulated and the corresponding graphs were plotted. Both methods suggest that the thrust and torque change with the change in relative velocity. Besides, there are several factors that contribute to the variations in results of the Analytical method and CFD Simulation.

Index terms— CFD, constant pitch, blade element theory, advance ratio, airfoil, coefficients of thrust, torque, power, etc.

1 Introduction

n aircraft propeller is one of the most common important parts in the aircraft. It is an airfoil section designed to generate the aerodynamic forces. The propeller provides the necessary force i.e. thrust to push the aircraft through the air. Thrust is the component of the aerodynamic force that is parallel to the axis of rotation of the propeller. A propeller achieves a specified level of thrust by giving a relatively small acceleration to a relatively large mass of air. Maximizing thrust while minimizing the torque necessary to turn a propeller has becomes one of the most important aspects of good propeller design. The power that must be supplied by the engine is the multiplication of the torque required to turn the propeller and the angular velocity. The thrust developed by the propeller multiplied by the airspeed of the aircraft is called the propulsive power. The aircraft is propelled forward against the airframe drag by this power. The ratio of the propulsive power to the brake power for a propeller gives the propulsive efficiency which is one of the important measures of propeller performance. The thrust developed by the propeller when the aircraft is stationery is called the static thrust. This thrust is important for a propeller to produces high static thrust in order to accelerate the aircraft during takeoff.

In order to design an aircraft propeller, Computational Fluid Dynamic (CFD) analysis has been done to simulate the fluid flow over a body to solve and analyze the aerodynamic properties of a body. Moreover, analytical approach has been used to compare the result gathered from the CFD simulation.

In the time of rapid industrial development along with the current competition in the market has required companies to improve products and create more innovation compared to other products. Aviation industry is also on that trend, and the propeller performance plays an important role. A good propeller should meet the requirements of the flight and push the aircraft with high performance on the aerodynamic characteristics.

For studying these characteristics of the propeller, the numerical study seems to be the best way to accomplish that goal. In particular, the application of CFD simulation software to calculate the propeller aerodynamics is not so new study now and it has become popular methods. Today it becomes most powerful compared to other methods as well as experience or the other traditional methods. The present works focus on numerical simulation of the propeller at different rotating speed. Specific studies were conducted with the flight speed change, a fixed number J from 0 to 1. From these results such as pressure field, we can evaluate the performance of the propeller in the range of the flight velocity.

By using the mathematical models with considered assumptions, simulation results allow the predictions on propeller performance. The simulation has helped save time, effort and cost a lot for the design and manufacture of propellers.

2 II.

3 Physical Model

Propeller blade has different airfoil shape; the shape varies quite a lot from root to tip. In practice a large number of different airfoils are used to make up one propeller blade but we designed our propeller simply taking only one airfoil along the blades. There are hundreds of airfoils suitable for this application but we think it is better to make a choice of an airfoil containing flat bottom surface. We are going to select NACA 4412 (Fig. **??**) flat bottom airfoil used in a lot of propeller designs in the past, and is also used today for simple designs.

Figure **??** : NACA4412 airfoil [14] In this design one blade has been considered which consists of 10 sections each having different airfoil shape, and representing different chord length & thickness.

The front plane has been taken as the reference plane in the Solid works part file and 10 more planes were created which are equally spaced. Plane 1 was at 2.4 inch far from reference plane and other 9 more planes were placed successively maintaining 2.4 inch distance between two consecutive planes up to 24 inch. The 2.4 inch difference between reference and 1st plane was kept for propeller hub. Airfoil data points from text file are imported to draw profile curves on each of the 10 planes by "curve through XYZ points" commands. These curves are converted into sketches for every slice of the blade sections. The airfoil sections are placed on a line such a way that the line goes through the center of gravity of each section. Because, unbalanced exists when the center of mass does not coincide with the center of rotation, when the mass center axis is different to its running center axis. The centrifugal force due to the unbalance causes the device to vibrate. This vibration causes wear to the bearings, creates unnecessary noise, and can result in complete failure. "Mass properties" command of the "Evaluate" tab was used to determine these gravity centers of the sections. This will keep the propeller vibration free during rotation due to the balanced blade mass distribution.

Then the section profiles are rotated in the design geometric angles of the respective sections that will ensure the design pitch for the rotating propeller. "Loft" feature was used to create a solid body by lofting from root to tip of the section's profile sketches. Automatic linear twist distribution is generated in the blade according to the geometric angles given in the profiles. Tip of the blade is rounded by using dome features.

4 Figure 3 : propeller blade with sliced sections

Propeller hub is inserted in the origin of the part drawing which looks like a cylinder, and this is done by using the "boss extrude" command on a circular profile of 5.1inch diameter. After an array operation of the blade about the origin, blades are added with the hub. "Fillet" feature is used in the joints between the hub and blades. A pointy dome is created on the hub to reduce the drag.

5 Mathematical Model

Development of a rational propeller theory begins with the work of Rankine and Froude with their interest in moving propulsion, but the fundamental principle is the same for water and air. They developed the fundamental momentum relation governing a propulsive device in fluid medium.

Stefan Drzewiecki however, was the first to rigorously examine and apply Blade Element Theory (BET). He performed his work between 1892 and 1920. BET is very similar to the Strip Theory for fixed wing aerodynamics. The blade is assumed to be composed of numerous, miniscule strips that are connected from tip to tip. The lift and drag are estimated at the strip using the 2-D airfoil characteristics of the section.

Also, the local flow characteristics are accounted for in terms of climb speed, inflow velocity, and angular velocity. The section lift and drag may be calculated and integrated over the blade span. BET is a very useful tool for the engineer. He or she may perform a fairly detailed local analysis of the rotor in a short amount of time. The first important task is the selection of an airfoil to design a propeller. The selected airfoil must maintain the aerodynamic characteristics that will provide lift and drag required for the thrust (propulsive force for airplane) and torque (available torque from engine shaft to rotate propeller).

As the maximum operating altitude ranges up to 25,000 feet and for smaller aircraft with propellers and normally aspirated engines, the service ceiling (maximum altitude where 100 feet per minute climb can be maintained) ranges 12,000-14,000 feet, we choose 10,000 feet as the design altitude for the airplane that can be propelled by our propeller. Followings are the design characteristics:

6 Numerical Procedure a) Computational Design

The computational design is comprised of a frame of 2.5m×9.5m×2.5m. Now the boundary conditions are assigned as; 'Surface A' is the 'Velocity Inlet' of 50m/s uniform velocity, 'Surface B' is the 'Pressure Opening' of 70600pa'Surface C, D, E&F' are considered as the 'Ideal Wall' In computational domain of flow simulation computational mesh is rectangular everywhere, so the mesh cells' sides are orthogonal to the specified axes of the Cartesian coordinate system and are not fitted to the solid/fluid interface. As a result, the near-walls of mesh cell are cut by the solid/fluid interface. Nevertheless, due to special measures, the mass is treated properly in these cells named partial.

At first a basic mesh is constructed. For that, the computational domain is divided into slices by the basic mesh planes, which are evidently orthogonal to the axes of the Cartesian coordinate system. The basic mesh is determined only by the computational domain and does not depend on the solid/fluid interfaces.

Then, the basic mesh cells intersecting with the solid/fluid interface are split uniformly into smaller cells in order to capture the solid/fluid interface with mesh cells of the specified size (with respect to the basic mesh cells). The following procedure is employed each of the basic mesh cells intersecting with the solid/fluid interface is split uniformly into 8 child cells; each of the child cells intersecting with the interface is in turn split into 8cells of next level, and so on, until the specified cell size is attained. Finally, the mesh obtained with these procedures is refined in the computational domain to satisfy the so-called narrow channel criterion: for each cell lying at the solid/fluid interface, the number of the mesh cells (including the partial cells) lying in the fluid region along the line normal to the solid/fluid interface and starting from the center of this cell must not be less than the criterion value. Otherwise each of the mesh cells on this line is split into 8 child cells. As a result of all these meshing procedures, a locally refined rectangular computational mesh is obtained and used then for solving the governing equations on it.

After completion of mesh generation, flow Simulation solves the Navier-Stokes equations (formulations of mass, momentum and energy conservation laws for fluid flows) with the finite volume (FV) method on a spatially rectangular computational mesh designed in the Cartesian coordinate system. We have plotted dCT/dx vs X &dCQ/dx vs X in the Fig. 14 to get the equations for dT/dx and dQ/dx. Therefore,

7 ????

By analyzing the curve of the gradient of the elemental thrust and torque coefficient plotted against span wise direction of a blade (root to tip distance), it is observed that maximum thrust and maximum torque cannot be obtained for the same element. For the maximum thrust, the torque obtained is somewhat less than the maximum torque and the thrust obtained is less than its maximum value when the torque is maximum. The maximum thrust and the maximum torque typically occur at the outer half of the propeller blade and the lower half contributes little to the thrust and torque. So, one need to be much careful while designing the propeller that the thrust needs to be distributed in a manner so that together the blended propeller would give the necessary thrust which is required for flying the aircraft. Highest design RPM for our propeller is 4525 which limits Mach number to .85 at the tip of our propeller. So, it is in the transonic range 0.8-1.2, but it is desirable to drop the Mach no. to subsonic range when RPM decrease because of inconsistency of the supplied power to the propeller. In propeller design prospect, Mach no is tried to keep below .85 to avoid the shockwave due to the sonic flow around the propeller. From graph it is evident that highest thrust and power coefficient occur at lower advance ratio and efficiency is highest at higher advance ratio. For fixed blade propeller the pitch is fixed, angle of attack will decrease as the forward velocity of aircraft increases. Although this will result in an efficiency increase initially, further increasing the velocity will make the angle of attack zero and the Propeller will not be able to generate thrust which is avoidable by using variable pitch propeller. In the figure cruise point will be somewhere nears the point of highest efficiency, for which we will lose some thrust and power, but there still enough thrust and torque to propel the aircraft.

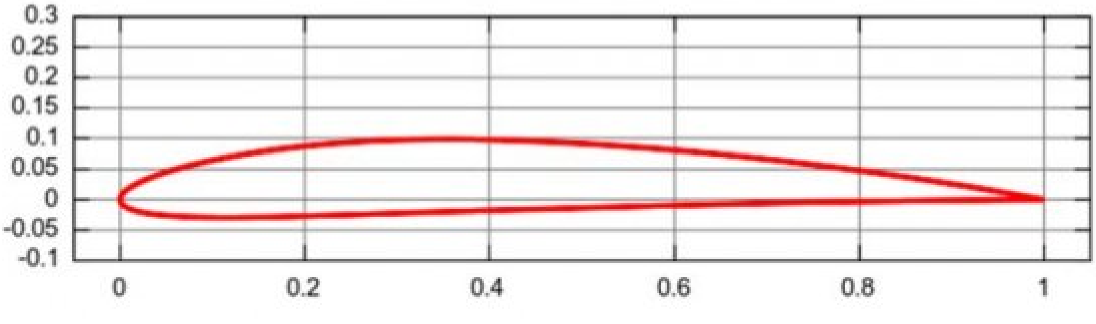
The performance graph of our propeller that we have obtained is similar to the characteristics graph of the propeller which is represented in Fig. 24.

8 Conclusion

From CFD analysis the thrust and torque for our propeller are 1007N and 311.19Nm which are 22.3% and 21.3% less than the theoretical calculated values. At 4525 RPM the propeller will provide maximum efficiency if the aircraft speed is 92ms-1 and there are 47% and 33.33% reduction in thrust and torque respectively than the maximum value that can be obtained by this propeller. [1]

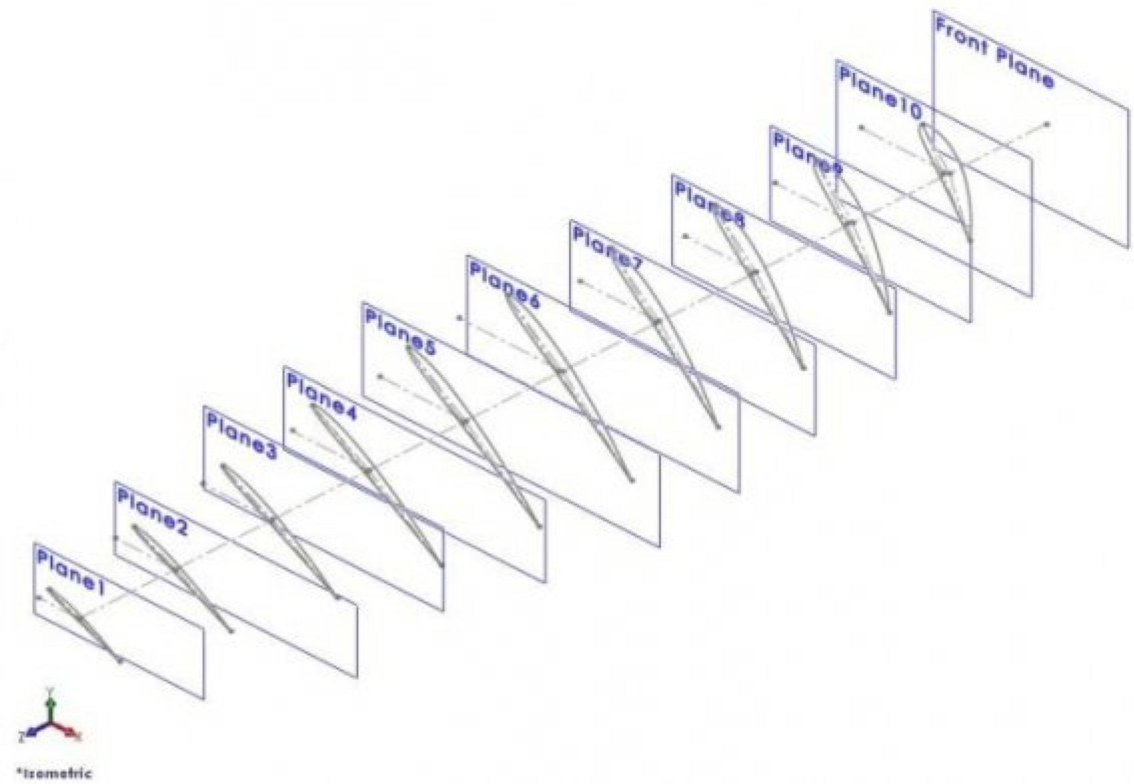
2

Figure 1: Fig. 2



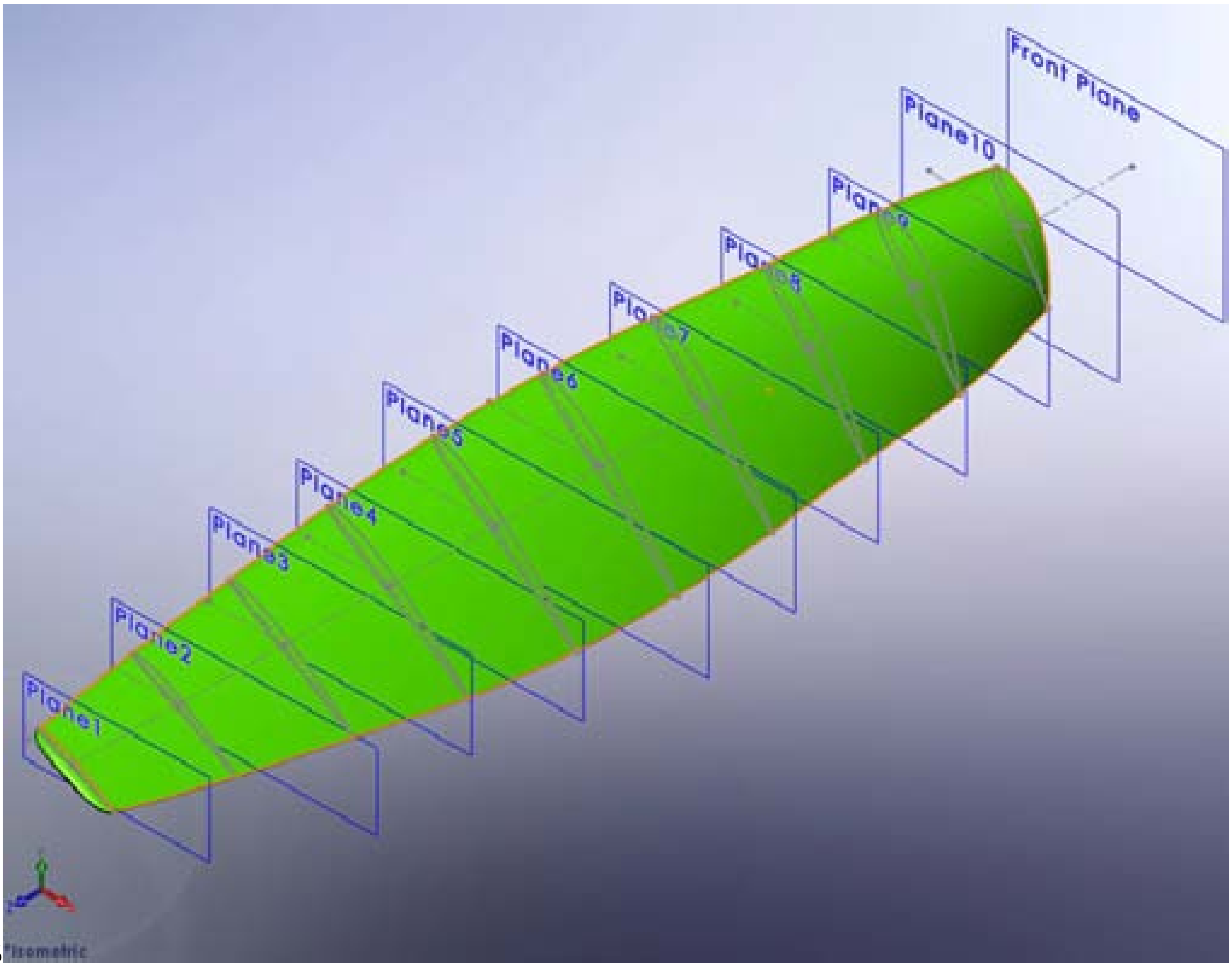
2

Figure 2: Figure 2 :



4

Figure 3: Figure 4 :



5

Figure 4: Figure 5 :

6212

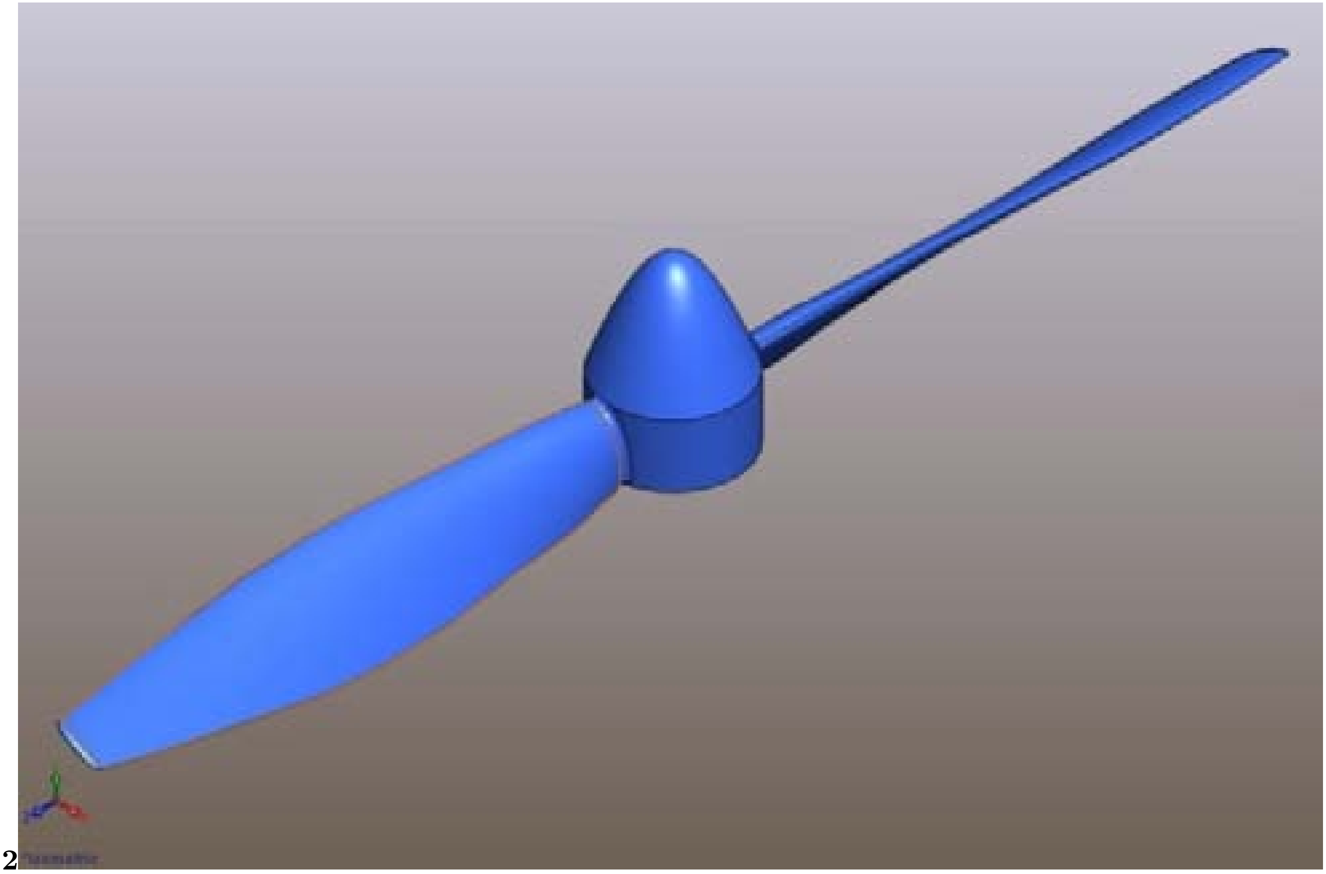


Figure 5: Figure 6 : 2 , 1 0;(2)

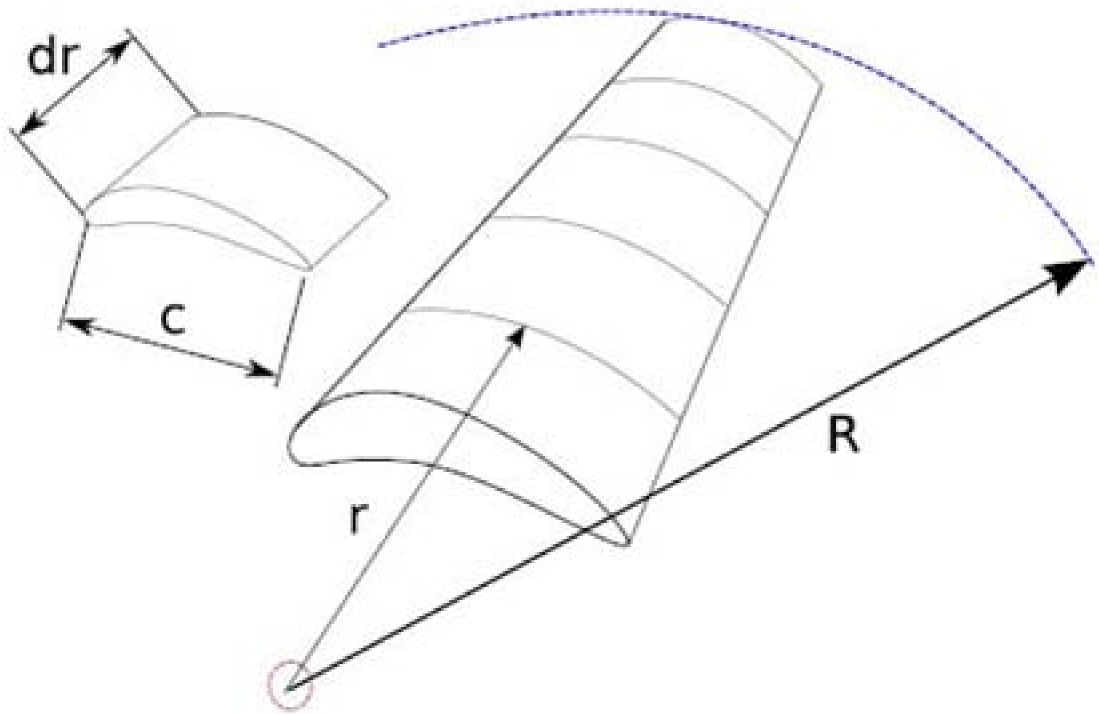


Figure 6:

7

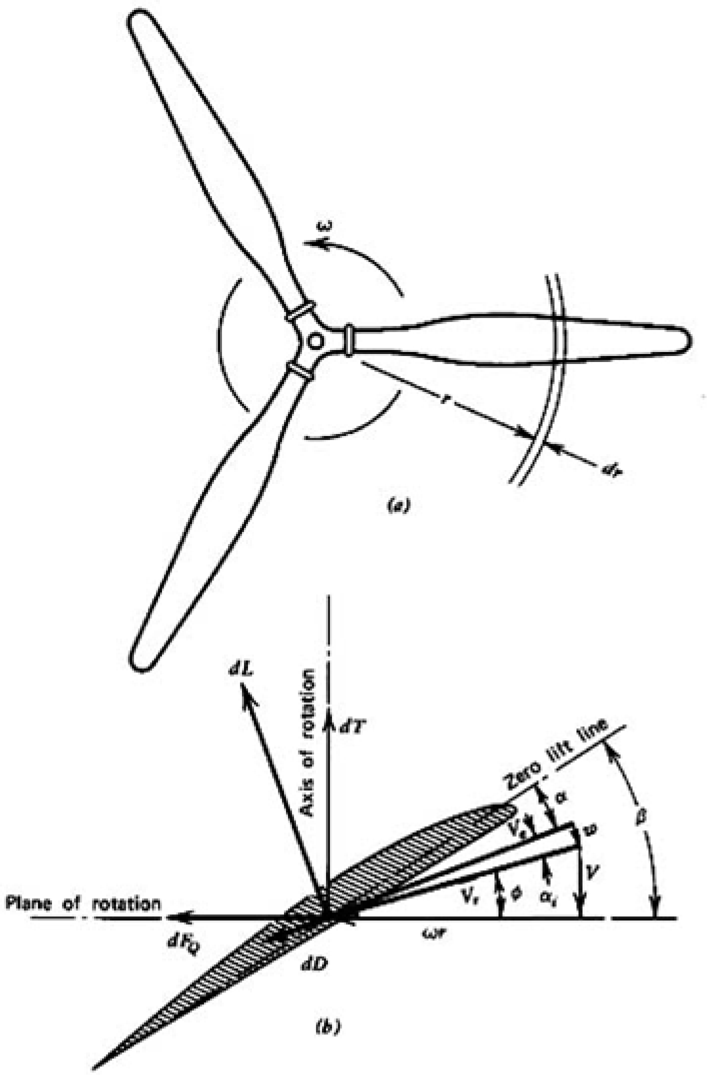


Figure 7: Figure 7 :

8

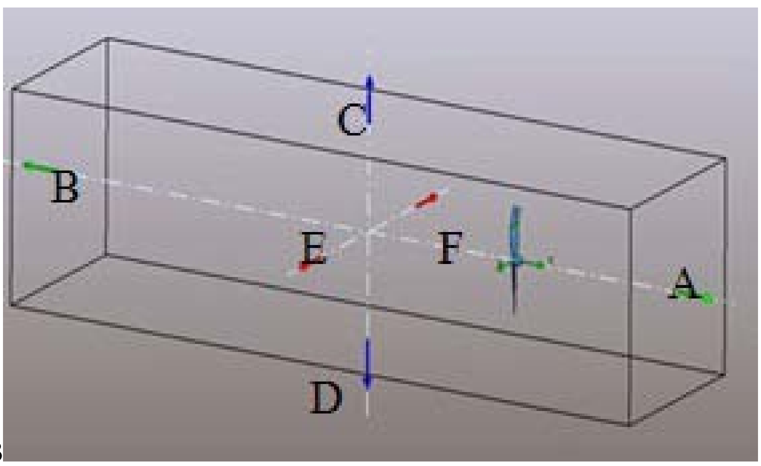
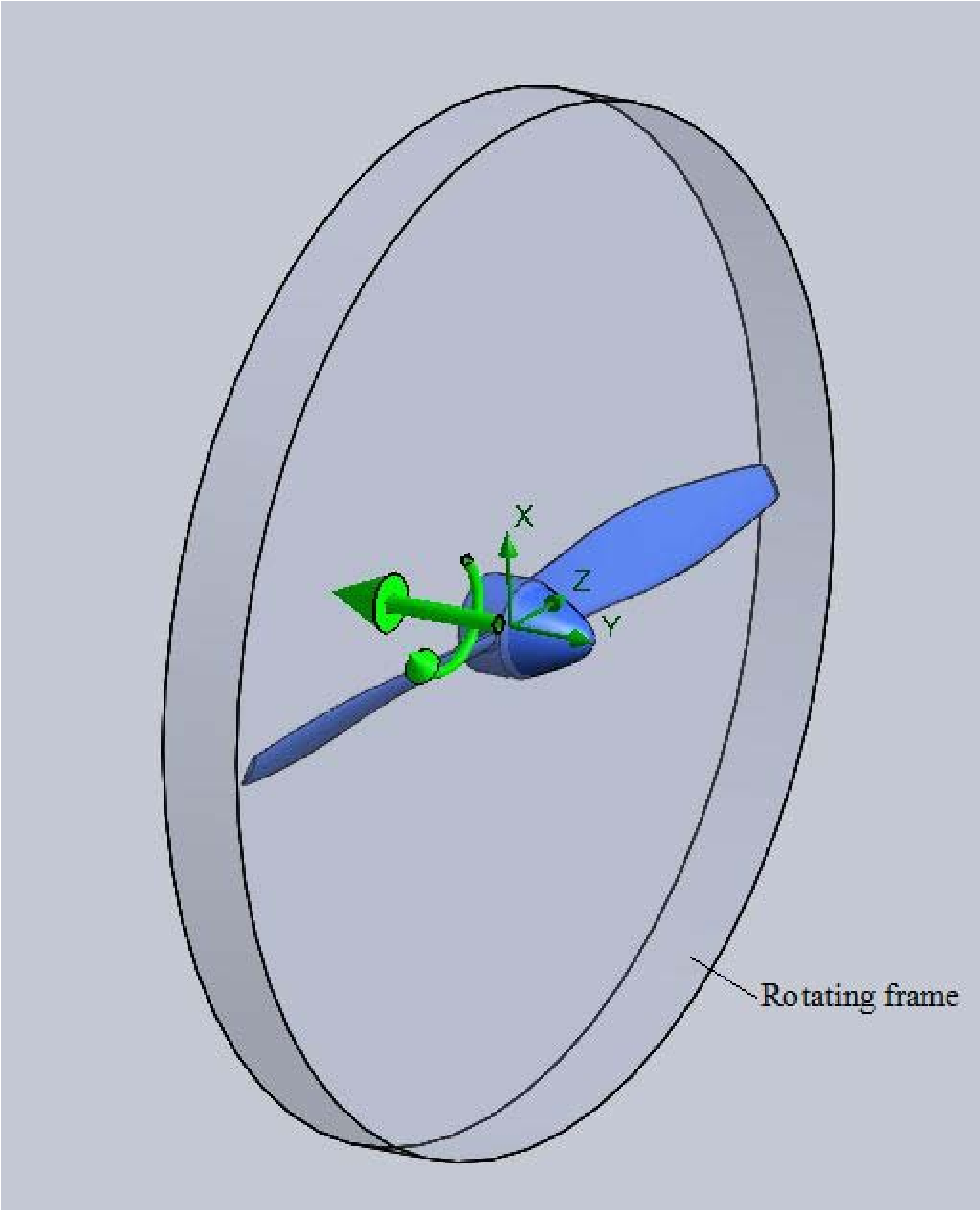


Figure 8: Figure 8 :



9

Figure 9: Figure 9 :

10

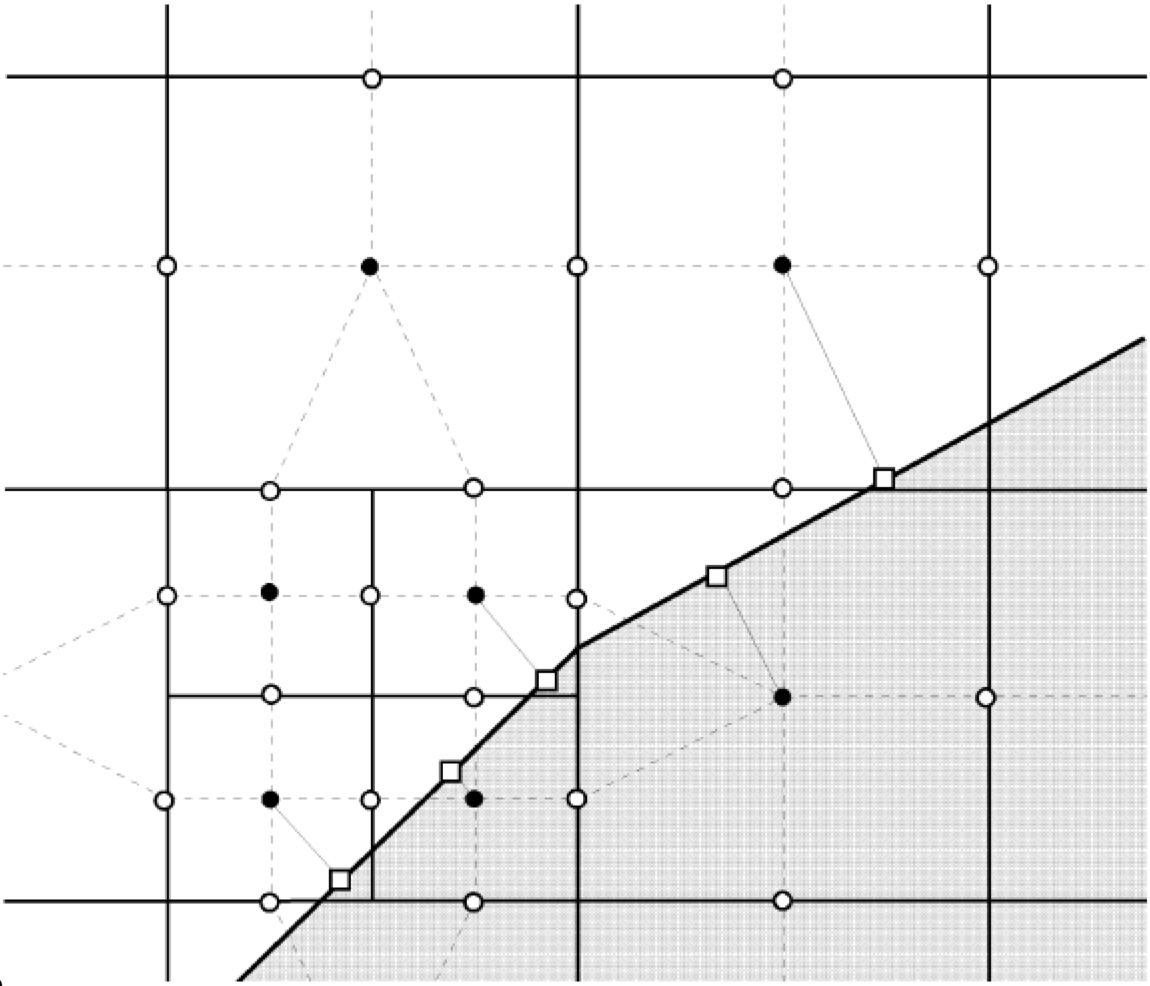


Figure 10: Figure 10 :

13

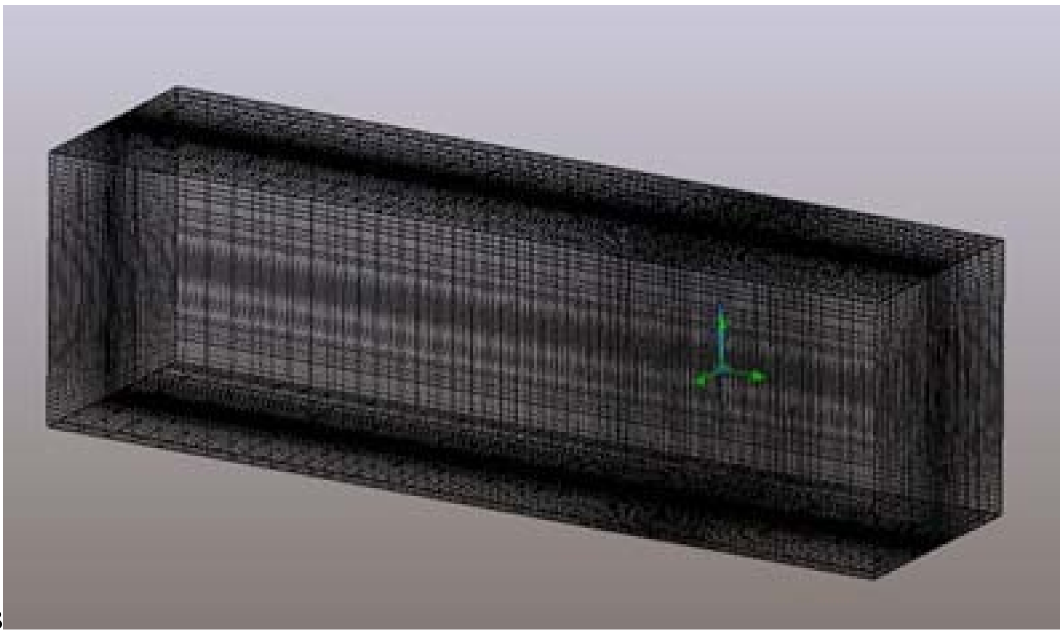
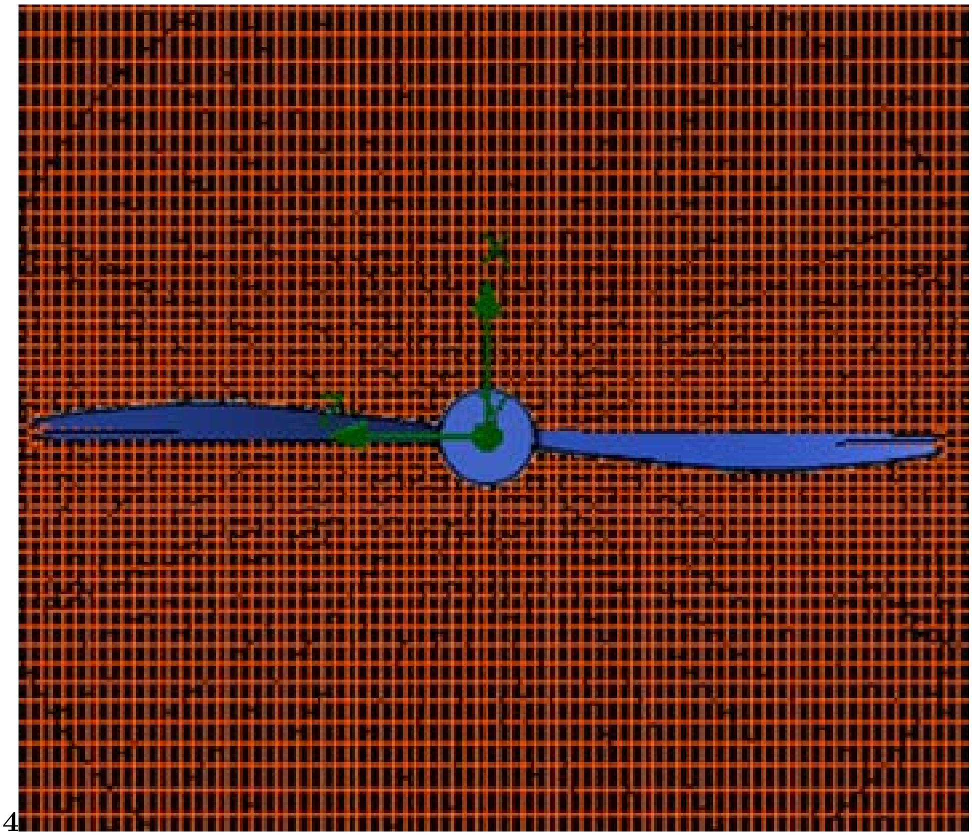


Figure 11: Figure 13 :



14

Figure 12: Figure 14 :

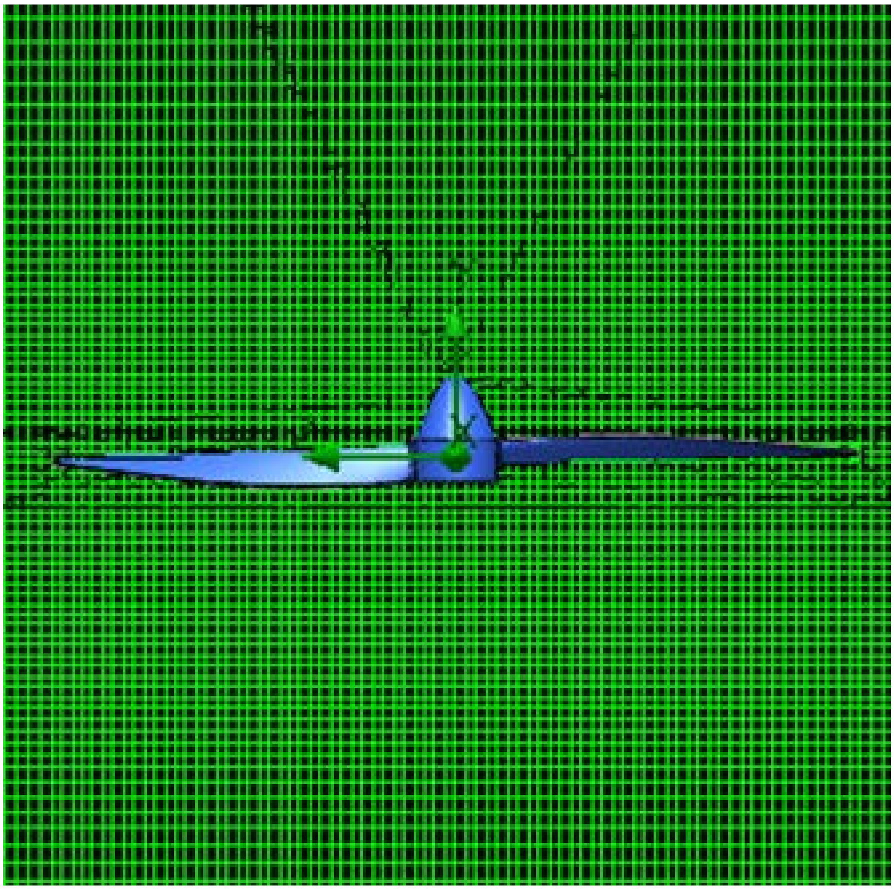
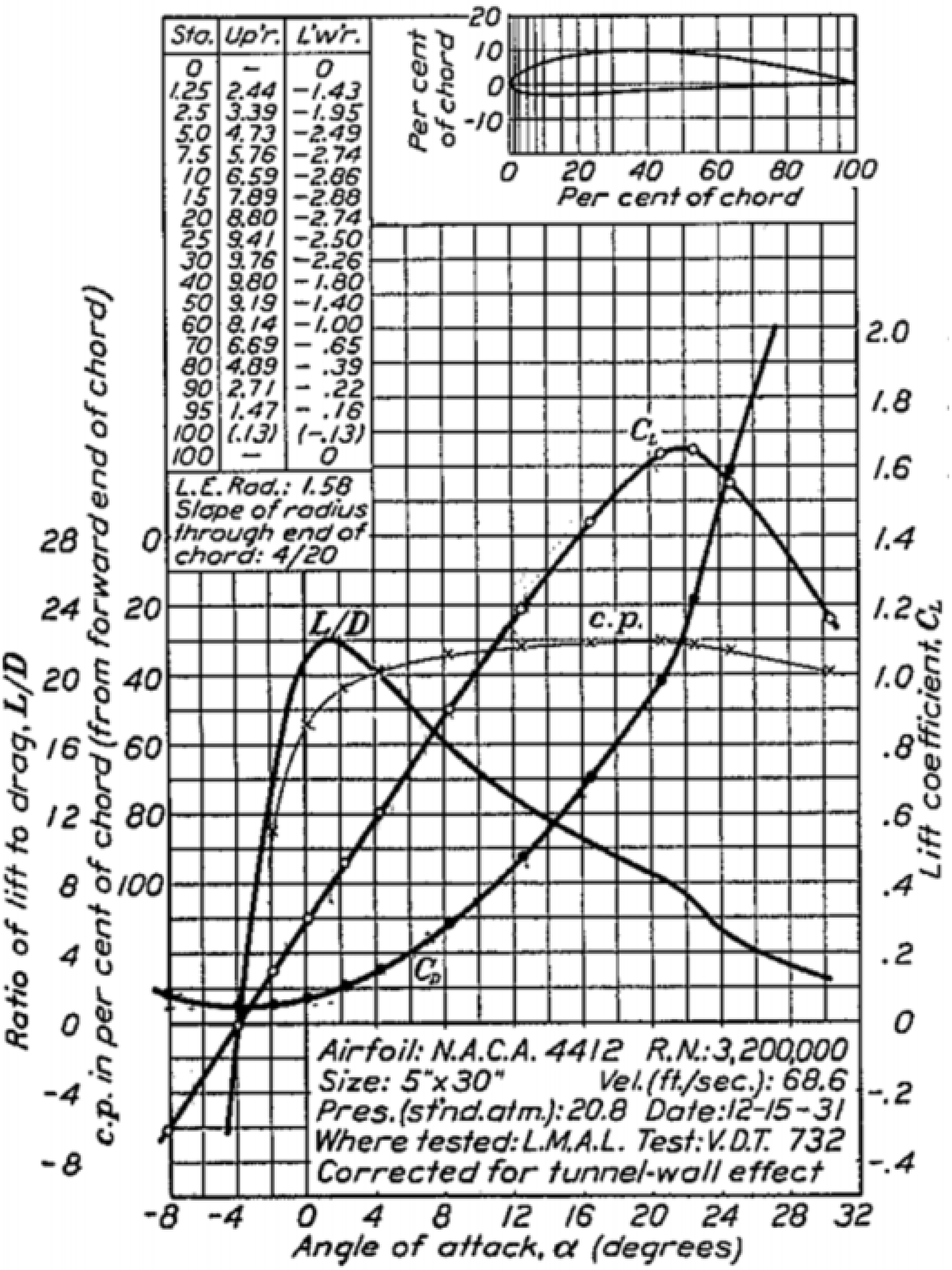


Figure 13:



15

Figure 14: Figure 15 :

17

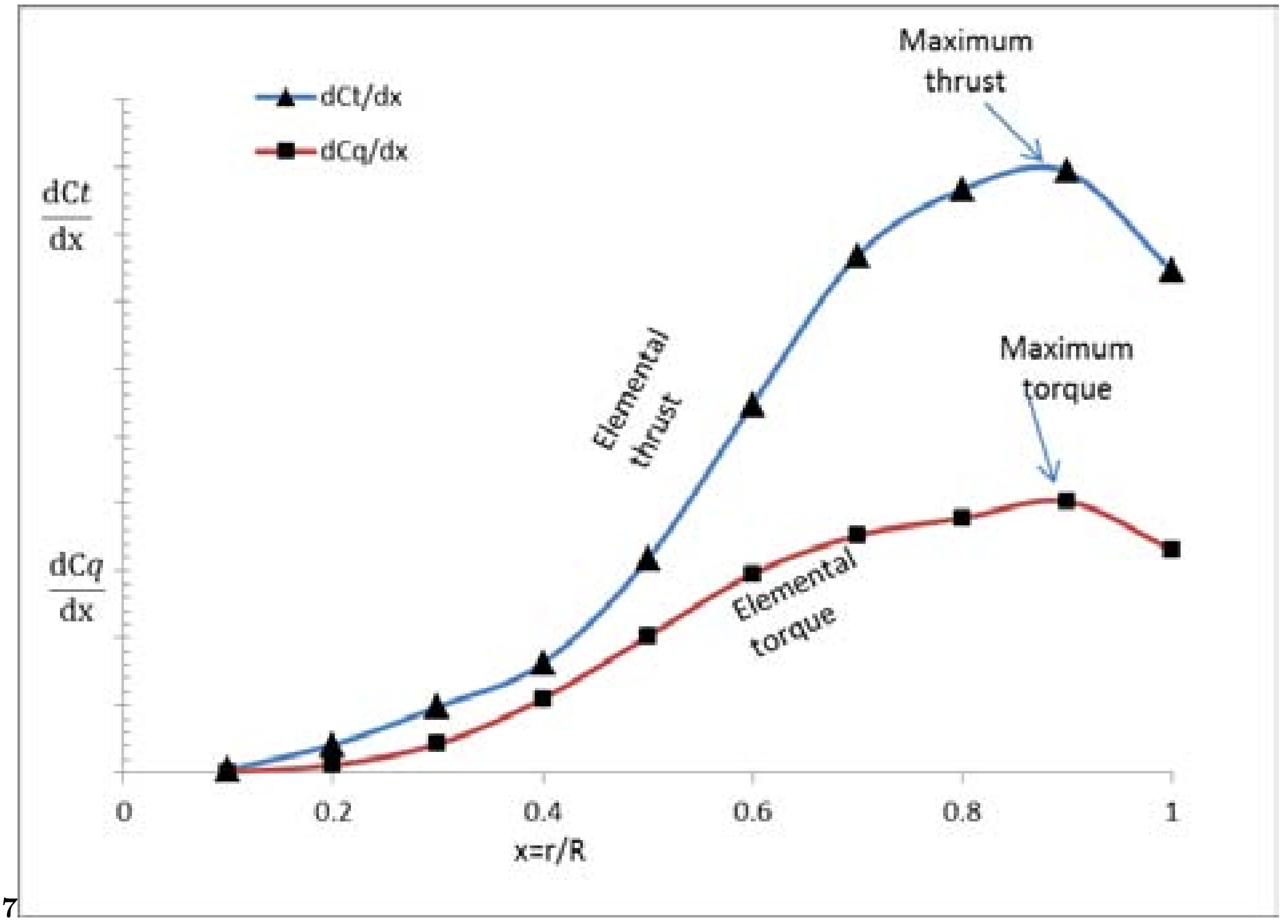


Figure 15: Figure 17 :

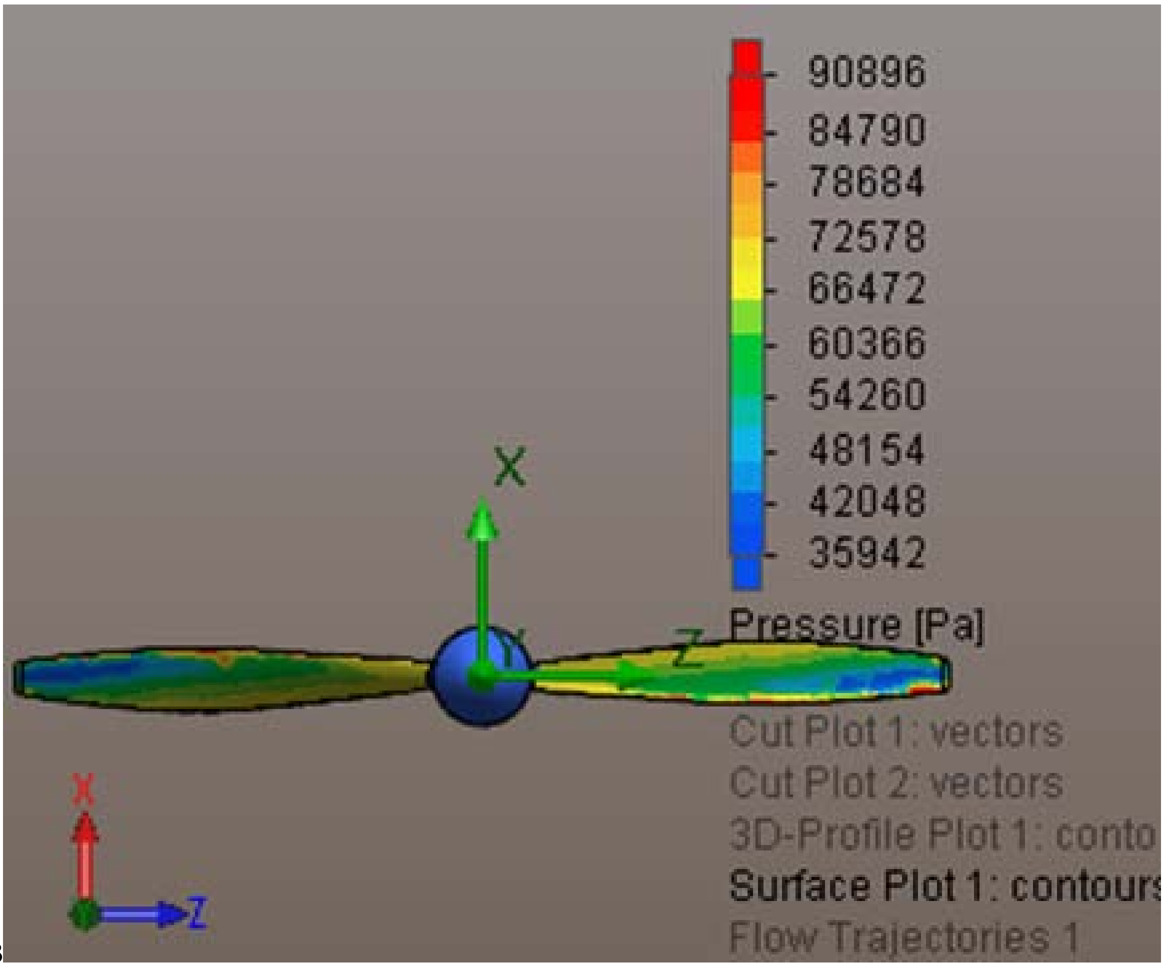
18 

Figure 16: Figure 18 :

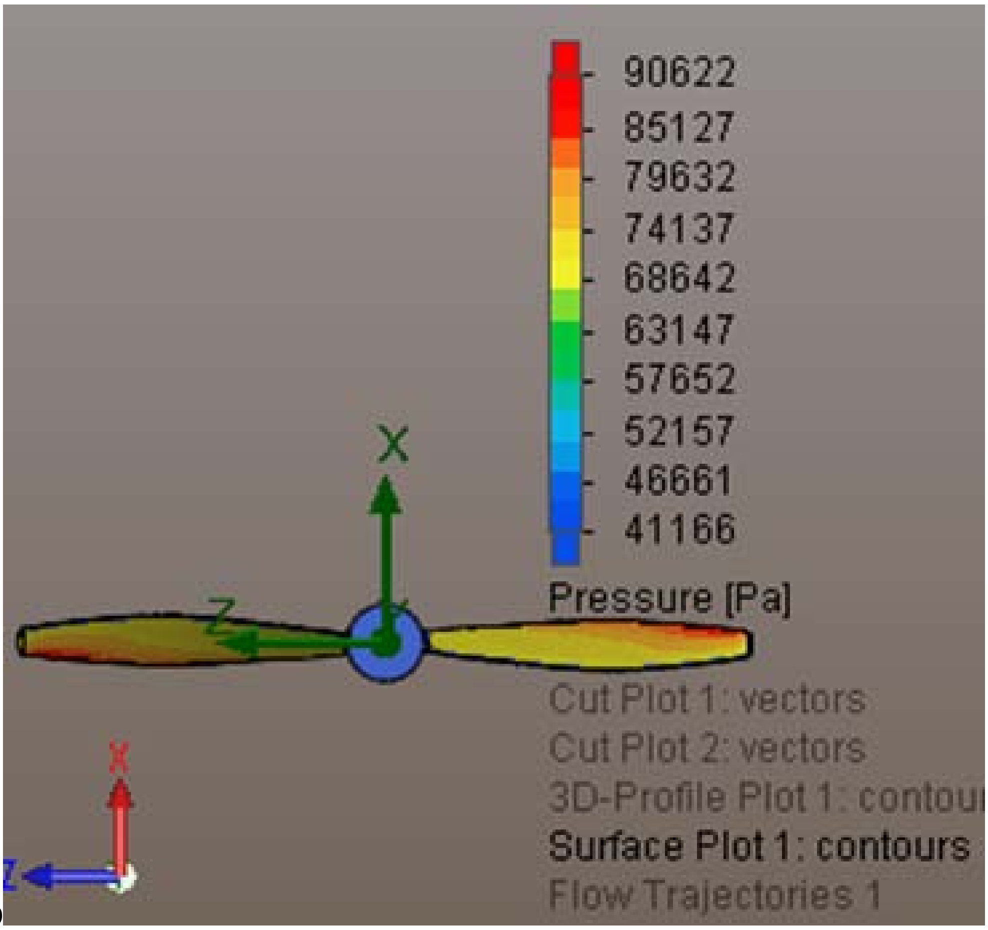
19 

Figure 17: Figure 19 :

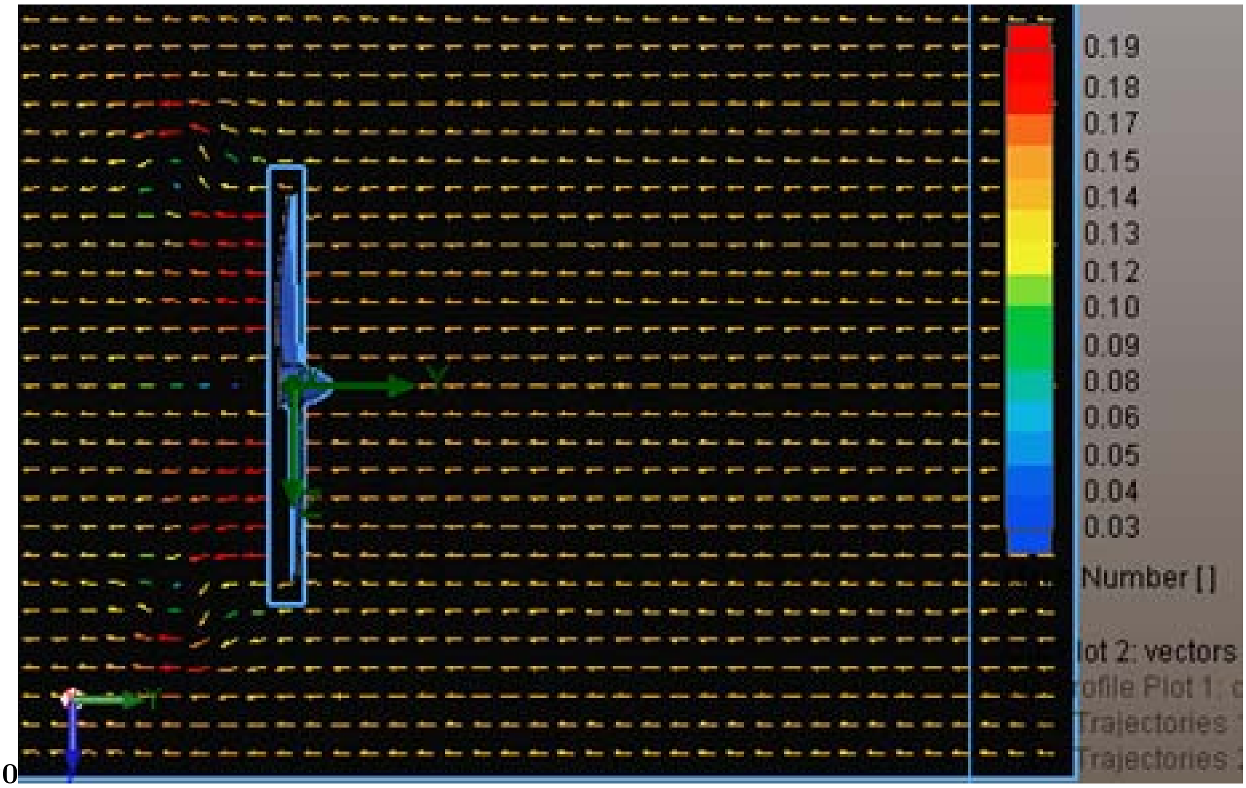
20 

Figure 18: Figure 20 :

23

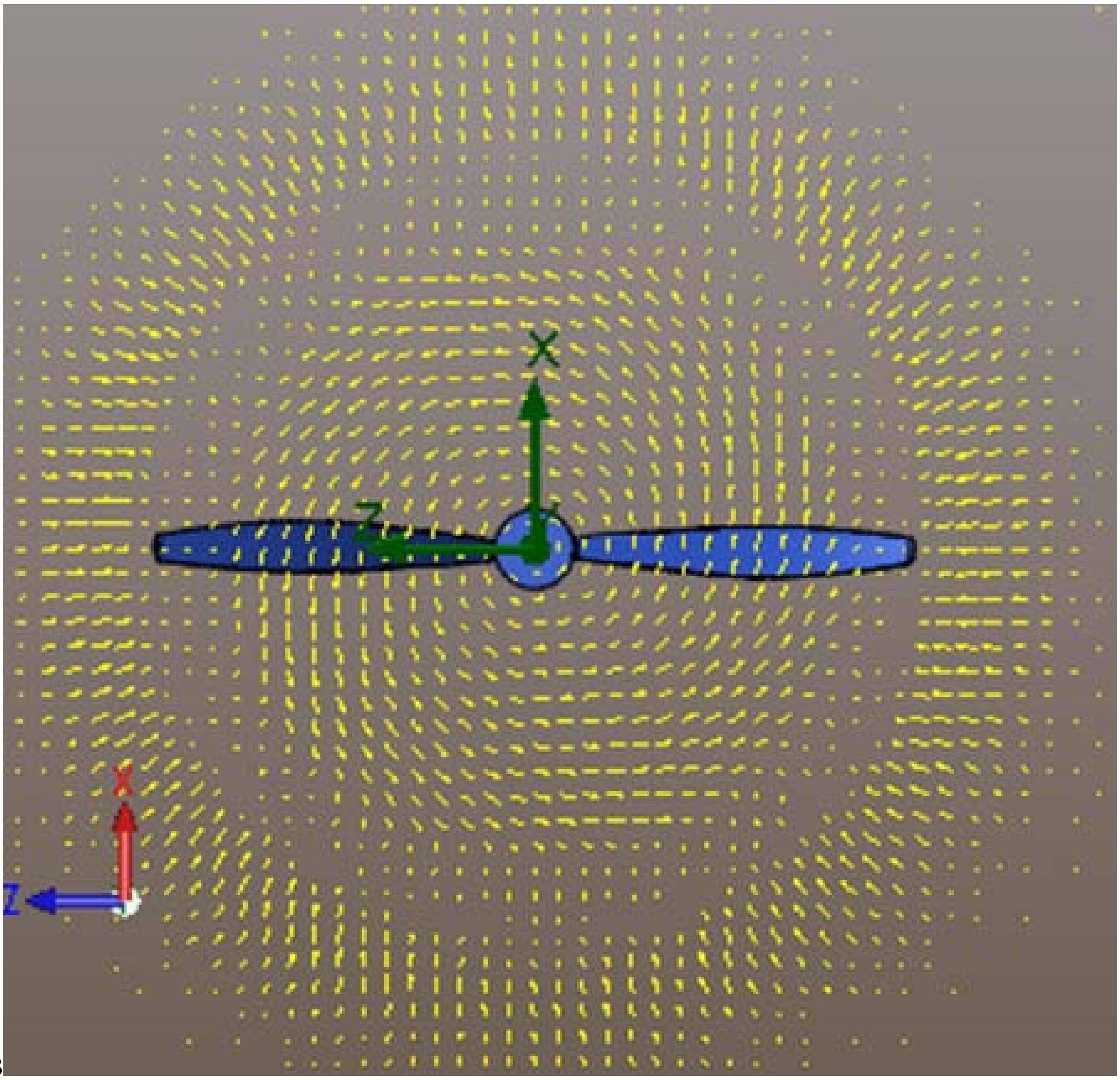


Figure 19: Figure 23 :

24

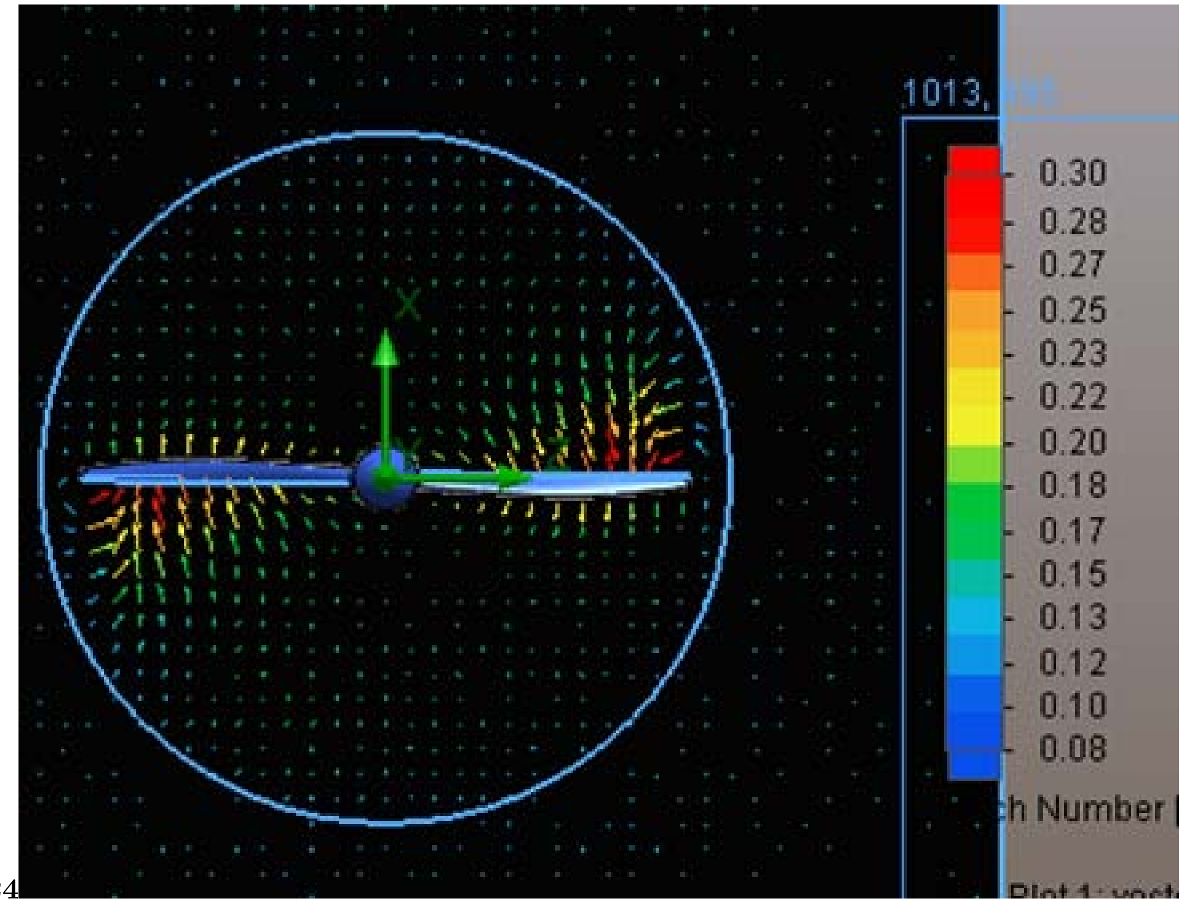


Figure 20: Figure 24 :

[Shethal and Kodiyattu (ed.) ()] *Article on Compressibility: Selection of propeller airfoil section (Prandtl-Glauert rule*, Thomas Shethal , Kodiyattu . 306. 12. Joe Supercool (ed.) 2010. San Jose State University (Design of a propeller for downwind faster than the wind vehicle)

[Mises] *Blade Element Theory, Theory of Flight By Richard. Prof. Bhakskar Roy, Video lectures on propeller*, Von Mises . Dept. of Aerospace Engineering, IIT, Bombay

[by Unified Engineering Lecture notes, MIT. Section 11] *by Unified Engineering Lecture notes, MIT. Section 11*,

[Melbourne] *Dynamic Balancing, Industrial Standards*, Australia Melbourne . (International Standards)

[Magdalena et al. ()] *Examining the Effects of Wind Farms on Array Efficiency and Regional Meteorology, University -Presented at Wind power*, R V Magdalena , Sta , Mark Z Maria , Jacobson . 2007.

[Federal Aviation Regulations, Amendment 23, FAR 23 ()] *Federal Aviation Regulations, Amendment 23, FAR 23*, 1527.

[Flow Simulation 2012 Technical Reference 1-49 of simulation software package] *Flow Simulation 2012 Technical Reference 1-49 of simulation software package*,

[Anderson ()] *Fundamental of Aerodynamics [SI Units*, J D AndersonJr . 2010. New Delhi: Tata McGraw Hill. (Fifth edition)

[Government Printing Office ()] *Government Printing Office*, 1976. U.S; Washington. U.S. Standard Atmosphere

[Fox et al. ()] *Introduction to Fluid Mechanics*, R W Fox , A T Mcdonald , P J Pritchard . 2009. Singapore: John Wiley and Sons Inc. (Fifth Edition)

[Mccormick ()] B W Mccormick . *Aerodynamics Aeronautics and Flight Mechanics*, (New York) 1979. Wiley.

[NACA report] no.389. *NACA report*,

[Merchant ()] *Propeller performance measurement for low Reynolds number unmanned aerial vehicle applications*, Monal Pankaj Merchant . 2004. B. S., Wichita State University