



The Netherlands Press

Journal of Airline Operations and Aviation Management

Article

PRESSURE AND VELOCITY VARIATION IN REMOTE-CONTROLLED PLANE USING CFD ANALYSIS

Gurunadh Velidi

Program Leader and Associate Professor,
School of Engineering,
University of Petroleum and Energy Studies
Energy Acres, Bidholi, via, Prem Nagar, Uttarakhand, India

Orchid Id: <https://orcid.org/0000-0003-4024-247X>

E-mail: gvssvelidi@ddn.upes.ac.in

Abstract.

Aircraft are significant because they transport people and commodities through a wide range of terrains in a short amount of time. Newton's third law states that thrust is created by fly motors due to the response powers provided by fume gases. In this paper, computational fluid dynamics (CFD) analysis of an aeroplane is used to create an ideal design. Prior to the time-consuming production of aircraft, qualitative and quantitative wing characterisation gives important information for verifying wing choices and design. Each wing was independently subjected to CFD study at various velocities. Each wing's lift, drag, pressure, and velocity were measured, and a comparison analysis revealed how little adjustments to the wing's pressure, drag, velocity lift may have a significant impact. Each wing's lift, drag, pressure, and velocity were measured, and a comparison analysis revealed how small adjustments to the wing's overall flow characteristics improved the overall flow characteristics.

Keywords: Lift, Drag, Weight, Thrust, Remote controlled plane, pressure, velocity.

Journal of Airline Operations and Aviation Management Volume 1 Issue 1

Received Date: 11 May 2022

Accepted Date: 06 June 2022

Published Date: 25 July 2022

1. Introduction

Remote controlled plane has all the dynamic characteristics of an aircraft. RC planes are used to find the missing things by fixing camera with it. Many Organisations such as Government or military or scientists they use this aircrafts for weather readings, drones and finding enemies and many uses instead of travelling by human and this decreases the human effort and it also solves many problems[1]. The basic components of the RC Plane are

- Power plant
- Landing gears
- Aileron
- Tail
- Elevator
- The fuselage
- Vertical stabilizer
- The wings
- Flaps
- Rudder
- Horizontal stabilizer

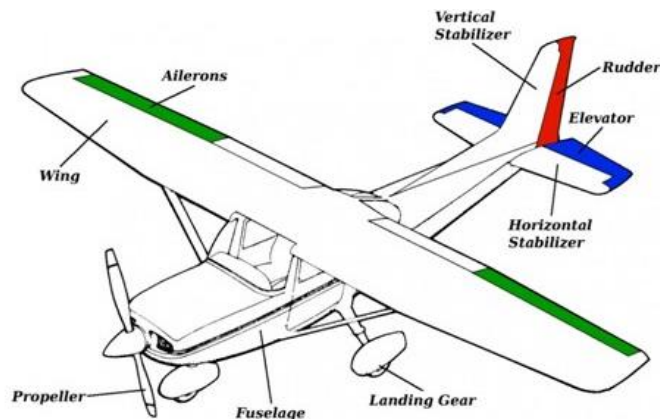


Figure 1: Components of RC plane[2]

Elevators, Rudder and ailerons are the control surfaces of an RC Aircraft. The stabilizers are Horizontal and Vertical. These RC Planes are controlled with the help of joysticks. The Four forces exerted on the RC Plane are Lift, weight, Thrust, Drag[3]. Thrust is generated for moving of a plane in the forward direction through air. Lift is obtained by the pressure difference of the upper and lower surface[4].

If the thrust force is greater than drag, there will be an acceleration in the plane. If the weight is greater than the lift Plane falls down. If lift is greater than the weight Plane moves upward or raises. If drag is greater than the Thrust force, there will be deceleration in the Plane[5].

Design of an RC plane:

RC Plane is designed by using CATIA V5 software and design is symmetrical. Many functions in the CATIA are used and create a geometry of RC plane and it is saved in the '.iges' format. This can be loaded in the Ansys software with the help of import tool in Ansys [6]. By opening the saved folder and selecting .iges files the design can be uploaded in the Ansys workbench . It can be observed with Design modler. After design of RC Plane a boundary should be created such that the air flows through. This is for the simulation pupose. The shape of the closed surface is cuboid and it should be more than 10 times the sizee of the plane[7].

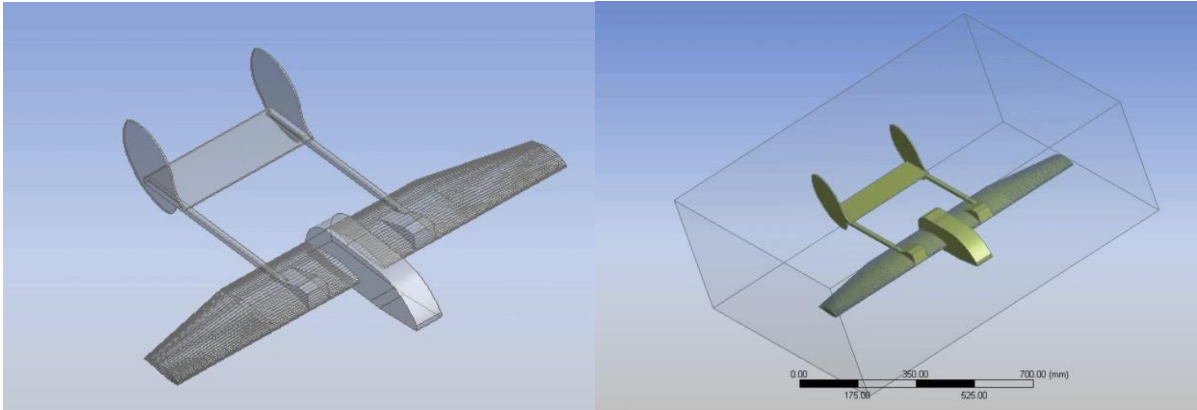


Figure 2: Geometry of RC Plane

Meshing the surface:

For meshing the cuboid Fine meshing should be selected for meshing. The Quality for smoothing is high. Inflation is program controlled. The size of the mesh is 5 mm. Then by clicking Generate mesh option a mesh of 5 mm is generated as shown in the figure 3. For naming the surfaces the front part of the RC plane is named as inlet and the rear part is named as outlet. And entire medium is named as Air. Before closing the mesh update mesh.

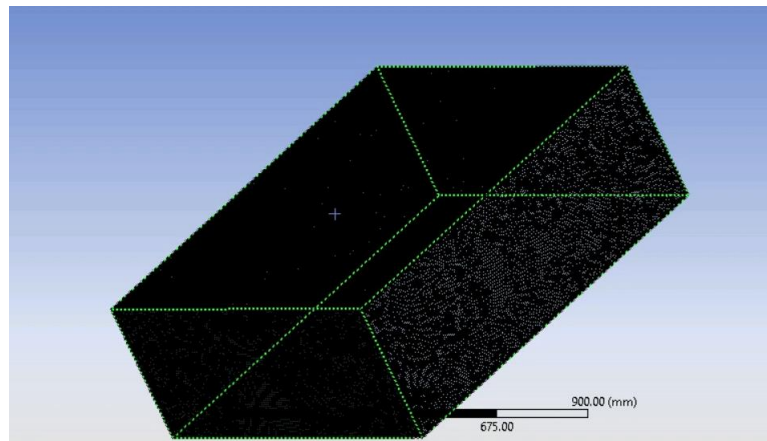


Figure 3: Meshing

Setup:

For setup tool 3D dimension is taken and Double precision and the solvers are 4. For the simulation process in the Ansys fluent, some parameters to be set in setup. The solver in the setup is set as pressure based and the velocity formulation is absolute.

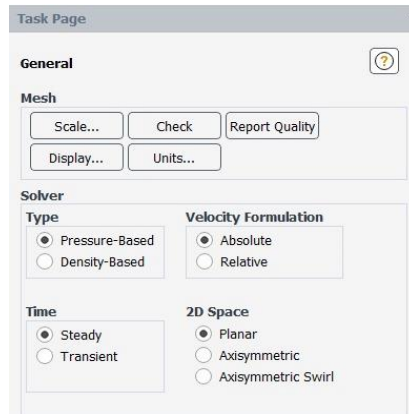


Figure 4: General setup

The viscous model used here is Realizable k-epsilon (2 equation) and the model constants used are shown in Figure 5. The original motive for using the k-epsilon model was to find alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows[8]. The first transported variable is the turbulent kinetic energy(k). The second transported variable is the rate of dissipation of turbulent kinetic energy(epsilon).

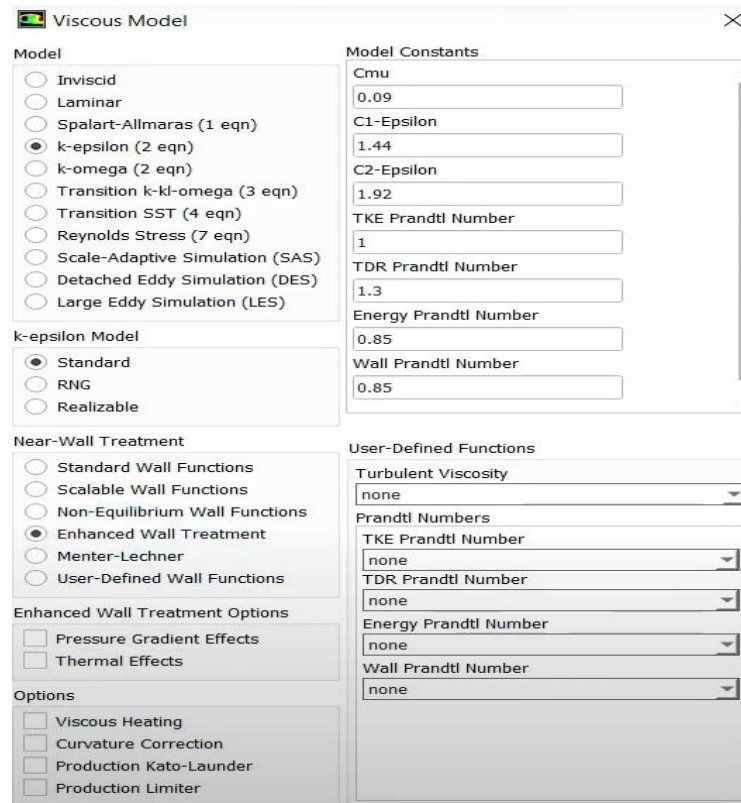


Figure 5: Viscous Model

Materials Used:

Air is used as a fluid in the enclosure. The properties of the material are specific Heat of 1006.43 J/Kg-K and Thermal Conductivity of 0.0242 w/m-K and viscosity of 1.784×10^{-5} Kg/m-s[9].

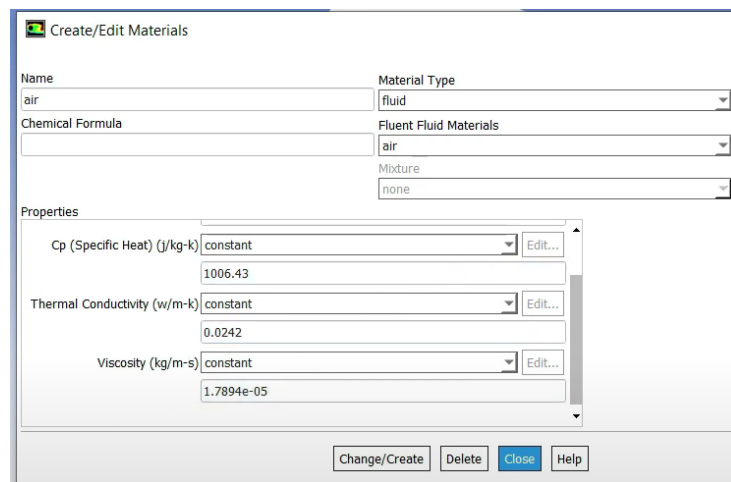


Figure 6: Properties of Air

Boundary conditions:

The inlet velocity of air is taken as 15 m/s and the turbulent intensity is 5% and turbulent viscosity ratio is 10 conditions and values are shown in the figure 7. The thermal zone condition is taken as 313 K.

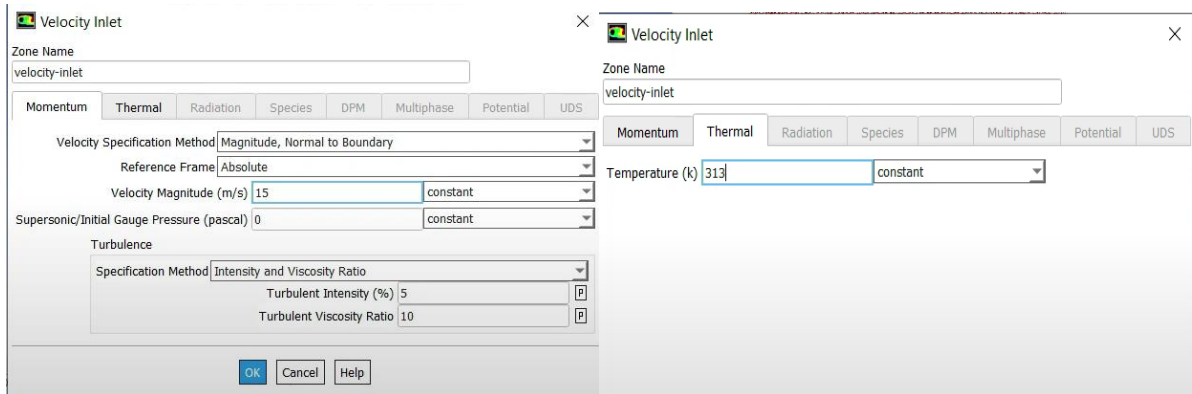


Figure 7: velocity inlet momentum condition

The outlet pressure conditions are taken as the gauge pressure of 0 and turbulent conditions are same as the inlet conditions as shown in figure 9. These conditions are backflow pressure conditions.

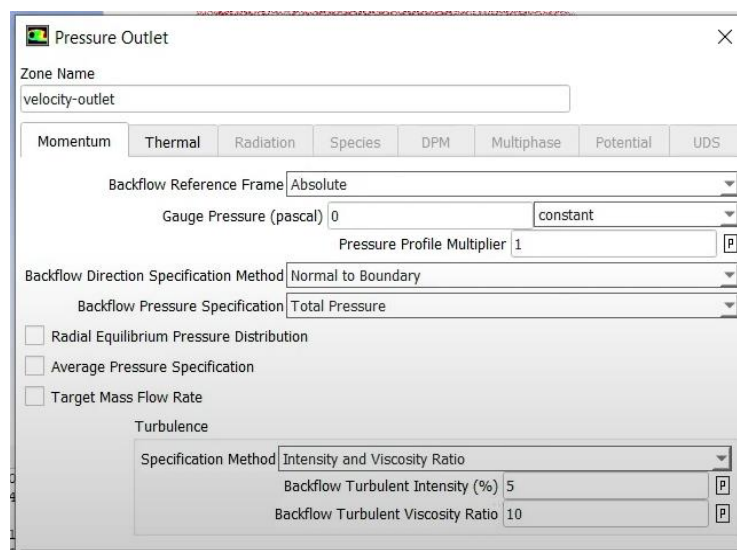


Figure 8: Pressure outlet

The reference values may be found in the table below. This section will allow you to choose the technique for computing the variable parameters.

2. Results and Discussions:

In the Fluent, there are two forms of discretization: temporal and spatial. The former refers to the discretization of time-related concepts, while the latter refers to the discretization of space-related phrases. The pressure-based solver is linked the continuity and momentum equations to solve these equations, as mentioned in the previous chapter, therefore the Coupled scheme is employed in the Pressure-Velocity Coupling[10]. The Pressure term is discretized spatially using the Body Force Weighted scheme, which is an optimal scheme for buoyancy driven flows. The Second Order Upwind technique is used to decrease the truncation error in the computation of Momentum, Turbulent Kinetic Energy, and Energy equations. Figure depicts the entire solution arrangement.

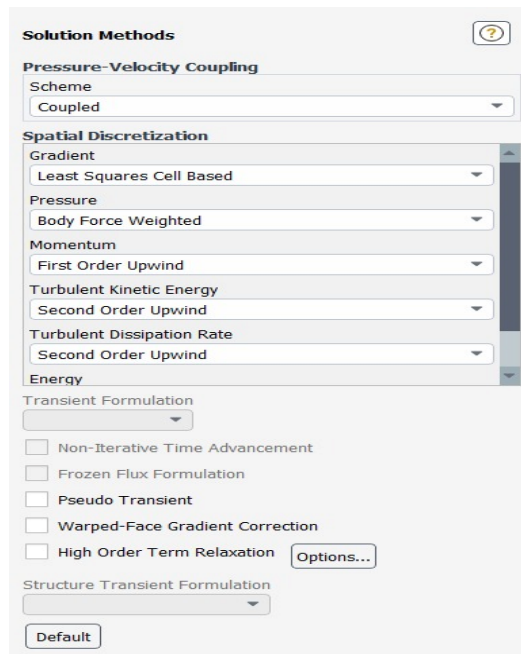


Figure 9: Solution Methods

By using the hybrid initialization setup and setting up the reporting intervals. Calculation is initialized. The definitions to be reported are Facet maximum pressure and velocity. The lift and drag in the walls are also reported.

For the k-epsilon method the residuals obtained are x,y,z velocities and energy k and epsilon. After 50 iterations the solution gets converged and the graph obtained is shown below.

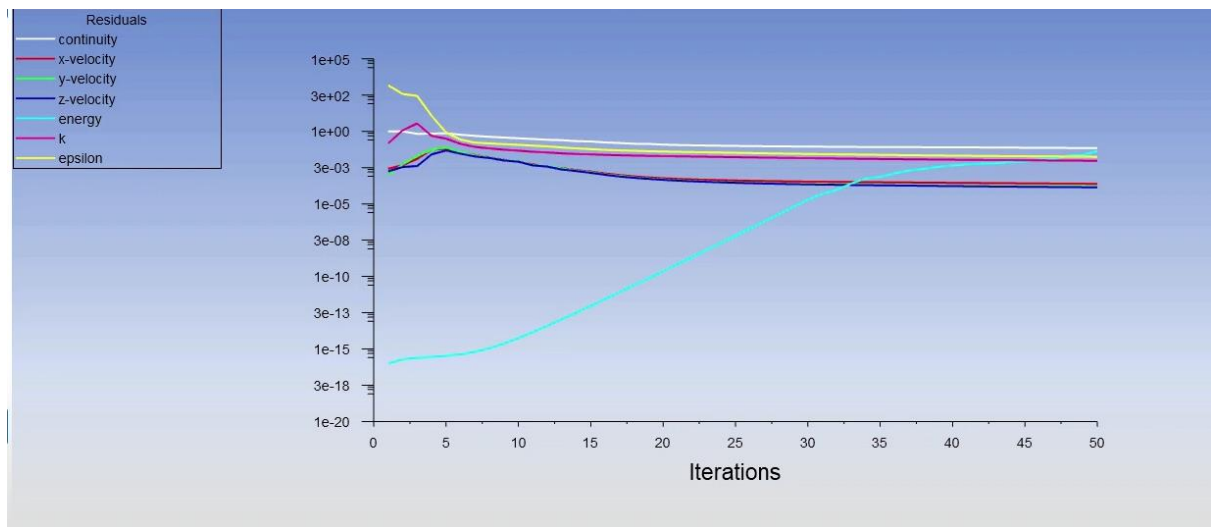


Figure 10: Residuals

In the figure 11 the static pressure in pascals versus position is plotted and the static pressure is increasing at 6.33m from the inlet. The nature of pressure can be observed in the graph.

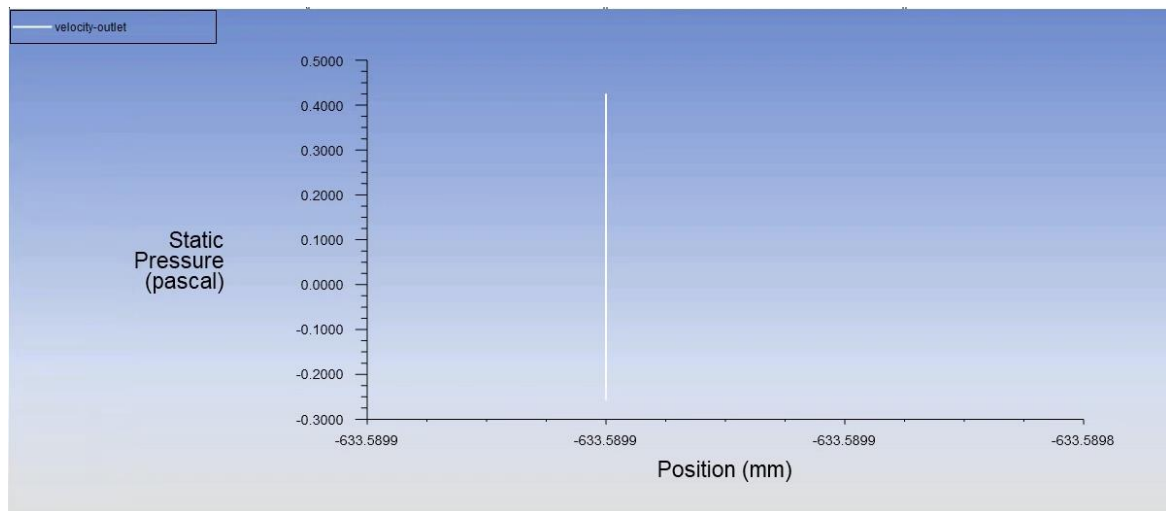


Figure 11: Static pressure vs distance

The drag on the wall and the pressure outlet is reported and the graph obtained is shown in figure 12. The outlet pressure is strictly decreasing as the air flows from high pressure to low pressure when air starts hitting the airfoil the air gets contacted with upper and lower surfaces of the airfoil. And reaches the tailing edge so from the graph, the pressure outlet is decrease hence the air reaches the tailing edge. And drag on the wall is 0.

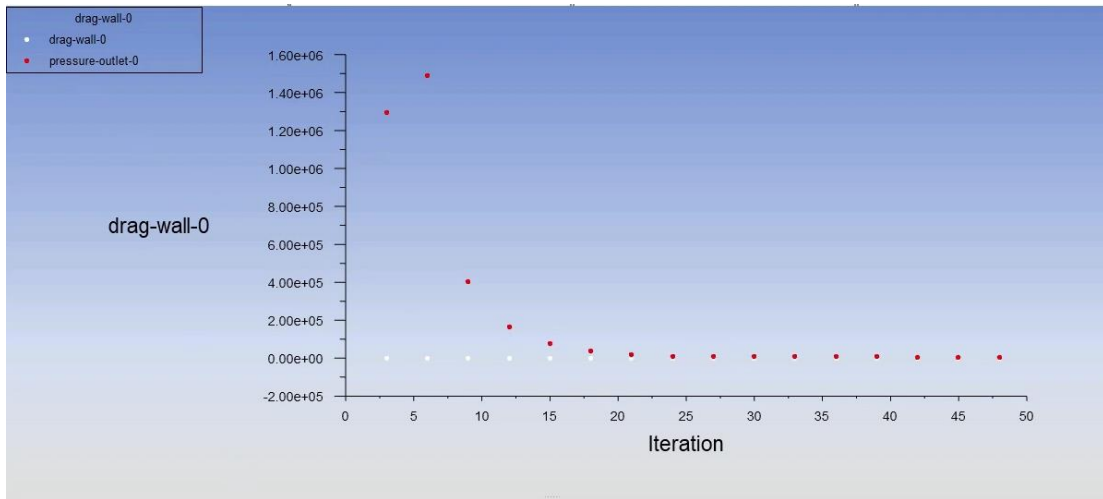


Figure 12: drag on wall

As moving to the CFD post the contours and streamline are observed. At first, the pressure contour is shown in figure 13. The pressure in RC plane is moderate and more pressure is on the airfoil edges this is because of the high pressure wind hits the airfoil and moves to the tailing edge as pressure reduces.

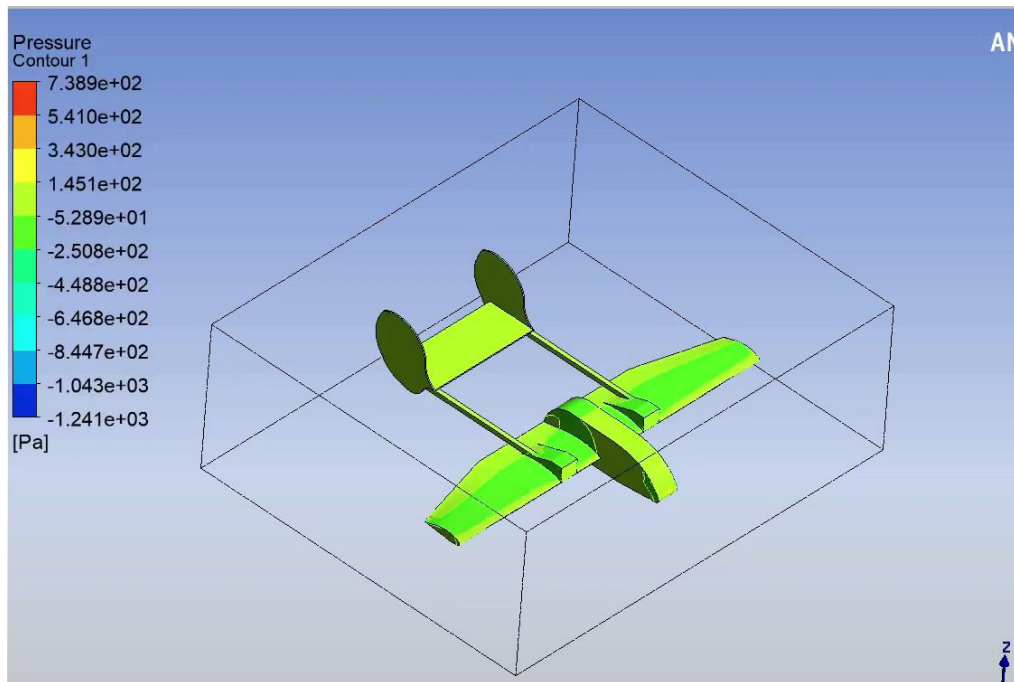
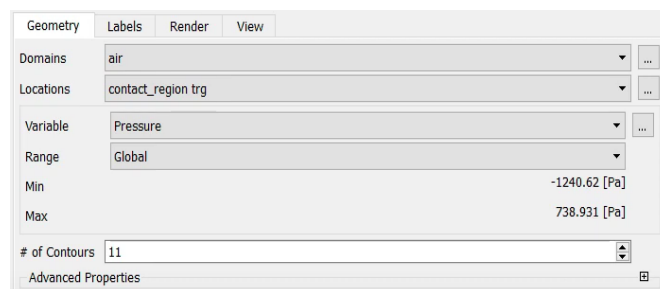


Figure 13: Pressure contour

Maximum pressure is 738.931 pascals and minimum pressure is -1240.62 pascals.



By observing the velocity contour in figure 14. The velocity of air at the tip of the plane is high and as going to the edge or stagnation point there is reduction in velocity of air as it strikes the end by using the Bernoulli's equation tailing

edge velocity can be obtained and that is lower than the velocity of air at inlet. Maximum and minimum velocities are 10.8112 m/s and -53.1794 m/s

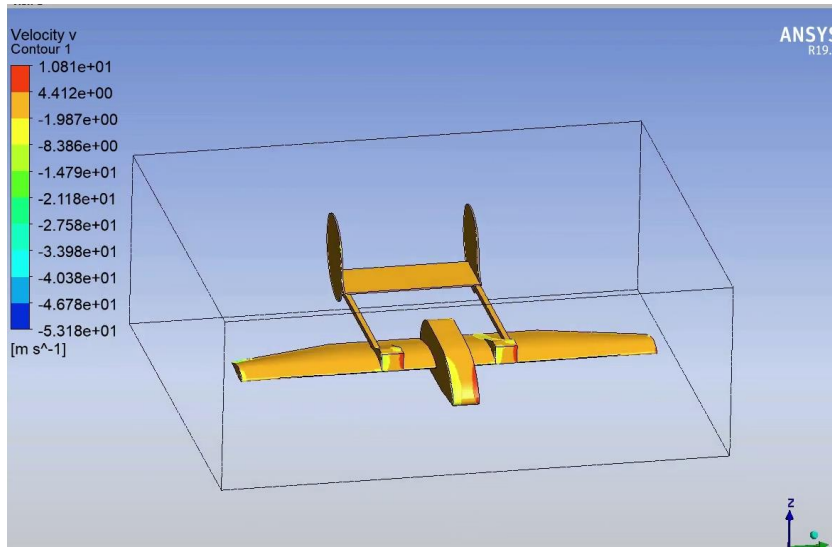
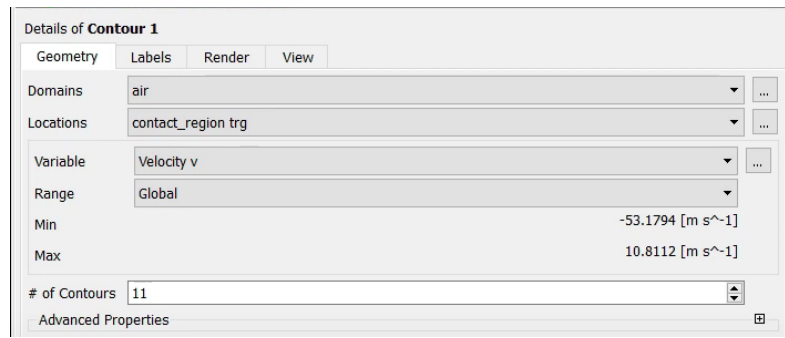


Figure 14: velocity contour

In the figure 15 the air streamlines can be observed as the air going from inlet and moving towards the outlet the pressure difference is created at the ends of the airfoil and at the plane edge.

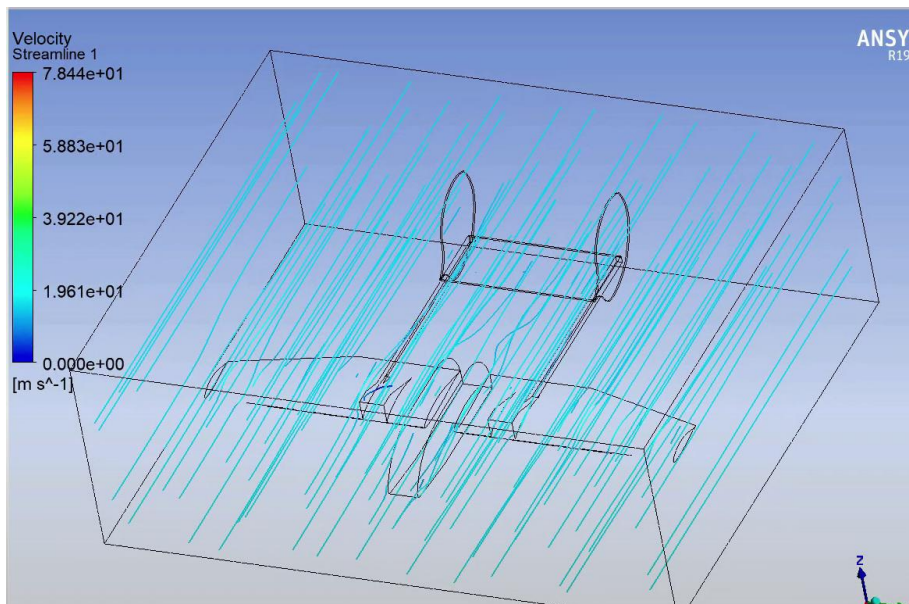


Figure 15: Velocity streamline

3. Conclusion:

RC planes are fast becoming a frequent feature in many applications as inexpensive hardware becomes more widely available and diversified, and practically all of its connected subsystems are functional, and operating costs and complexity fall dramatically. The majority of the study has focused on the k-epsilon model. For linear and non-linear analysis, the Finite Element Analysis approach was used. The performance of the radio-controlled aircraft may be improved by optimising the design parameters, according to the findings. To improve performance, the design should first be analysed, and then characteristics such as the length of the RC plane, the angle of the airfoil, and the length of the airfoil may be altered to optimise the design.

References

- [1] Understanding flight by David f Anderson 2e MC GRAW HILL PUBLICATIONS
- [2] Fundamentals of flight volume 1 basic aerodynamics by LALIT GUPTA & DR.O.P. SHARMA HIMALAYAN BOOK PUBLICATIONS NEW DELHI
- [3] Fundamentals of aerodynamics by JOHN D. ANDERSON JR 5 TH edition MC GRAW HILL PUBLICATIONS
- [4] Introduction to flight by JOHN D. ANDERSON JR. 6 TH edition MC GRAW HILL PUBLICATIONS
- [5] Wirachman Wisnoe, Rizal Effendy Mohd Nasir, Wahyu Kuntjoro, and Aman Mohd Ihsan Mamat, “Wind Tunnel Experiments and CFD Analysis of Blended Wing Body (BWB) Unmanned Aerial Vehicle (UAV) at Mach 0.1 and Mach 0.3”, 13th International Conference on Aerospace Sciences & Aviation Technology, ASAT- 13, May 26 – 28, 2009.
- [6] William Runge, “Characterizing the Aerodynamics of a UAV Wing”, Intermediate Fluid Mechanics, Prof. Luo, 12/18/2008.
- [7] Benjamin Sweeten, “CFD analysis of UAVs using VORSTAB,FLUENT, and Advanced Aircraft Analysis software”,Aerospace Engineering, Graduate Faculty of the University of Kansas, April 27, 2010.
- [8] Tholudin Mat Lazim, Shabudin Mat, Huong Yu Saint, “Computational Fluid Dynamic Simulation (CFD) and Experimental Study on Wing-external Store Aerodynamic Interference of a Subsonic Fighter Aircraft”, Acta Polytechnica Vol. 44 No. 2/2004.
- [9] L. A. Schiavetta, O. J. Boelens, W. Fritz, “Analysis of Transonic Flow on a Slender Delta Wing Using CFD”, American Institute of Aeronautics and Astronautics.
- [10] M. Dinesh, Premkumar P S, J.S.Rao, C. Senthilkumar, “Performance Analysis of Winglets Using CFD Simulation”, Altair Technology Conference, 2013.
- [11] M. A Azlin, C.F Mat Taib, S. Kasolang and F.H Muhammad, “CFD Analysis of Winglets at Low Subsonic Flow”, Proceedings of the World Congress on Engineering 2011 Vol I, July 6 - 8, 2011.
- [12] Cheolwan Kim, Jindeog Chung, “Aerodynamic Analysis of Tilt-Rotor Unmanned Aerial Vehicle with Computational Fluid Dynamics”, Journal of Mechanical Science and Technology (KSME Int. J.), Vol. 20, No.4, pp. 561-568, 2006