

Aerodynamic analysis over Unmanned Aerial Vehicle (UAV) using CFD

K Sreelakshmi¹ **Kota KSR Jagadeeswar**²

¹Assistant Professor, PG Student Aeronautical Engineering, MLR Institute of technology, Hyderabad, Telangana, India

¹sree.krishtipati@gmail.com.

Abstract. The origination of innovates in UAV (Unmanned Aerial Vehicle) has been created since a numerous years back. UAV has its own particular points of interest, for e xa mple, modernizing and self-governing without an on board pilot by using electronic network and computerized control which keep view on vehicle doesn't have. Hence, there is no risk of life and is easy to keep view on air vehicle. W ith the help of necessities and improvements in innovates, UAV is far reaching and utilizing in industries, defence and common civilians work as it can be controlled remotely. Surveillance for fire fighting and continuous catastrophes, suspect monitoring and system assaults are the exa mples of such UAV system. This paper discusses the aerodynamics behavior of a baseline design and analysis of a UAV's (Unmanned Aerial Vehic le Systems). Investigation is displayed in steady state 3D computational fluid elements (CFD) at Mach 0.1 and 0.19. In this technique for investigation, Lift, Drag, Lift Coefficient (CL) and Drag Coefficient (CD) are measured and looked at separate Mach numbers. Pressure forms and velocity shapes are plotted and the Turbulence zone is anticipated from streamline view, both removed from CFD examination.

Keywords : Unmanned Aerial Vehicle , Computational Fluid Dynamics, Ranger TUA V, CL, CD, ANSYSI CEMCFD.

1. Introduction

UAV is abbreviated as the Unmanned Aerial vehicle, which is a flying machine with absence of pilot on control desk. UAVs remotely controlled flying machines (control done from ground control station) and can be used for various missions which include surveillance and system assault parts. For the need of this paper, and to differentiated UAV from missile, a UAV has ability of self control, sustain and driven by jet engine. Moreover, a journey rocket can be treated as a UAV, however is deal with independently the premise that the vehicle is the weapon. The abbreviate UAS (Unmanned Aerial System) shows the robust system that insists ground control system and more other parts that an actual system involves



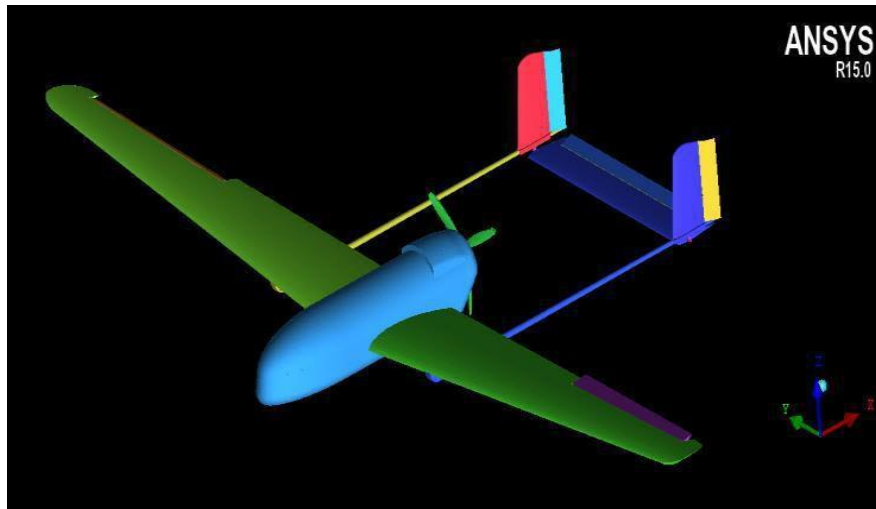


Figure1. Ranger UAV model

International Civil Aviation Organization (ICAO) characterizes this unmanned flying Machine classifies into 2 sorts:-

- Autonomous airplane – right now thought to be unacceptable for direction because of lawful and obligation issues
- Remotely guided air ship – subject to common direction under ICAO and under the important national flying power

1.1 Purpose of project

This paper discusses the aerodynamics behaviour of a baseline design and analysis of a UAV's (Unmanned Aerial Vehicle Systems). Investigation is displayed in steady state 3D computational fluid elements (CFD) at Mach 0.1 and 0.19. In this technique for investigation, Lift, Drag, Lift Coefficient (CL) and Drag Coefficient (CD) are measured and looked at separate Mach numbers. Pressure forms and velocity shapes are plotted and the Turbulence zone is anticipated from streamline view, both removed from CFD examination.

1.2 Problem description

Computational liquid flow (CFD) investigation of an air ship assumes a critical part in giving a perfect configuration of the airplane. The subjective and quantitative portrayal of the outline gives valuable data to check the determination and configuration before the tedious creation of flying machine.

Besides the applications of the UAV domain is lacking effectiveness in manufacturability, because of design constraints and its size. It is in need that the UAVs should be energy, cost effective and it should meet the desired performance targets as well.

Hence the work is supposed to concentrate on the UAV fuselage, wings and ailerons considering designable and non design areas in it with suitable load application.

2. Geometry selections and Modelling

2.1 Geometry selections

a) Ranger Tactical UAV

RANGER is the main strategic UAV framework overall affirmed to fly in non military personnel airspace and in addition over populated zones. Its design and some of its technology is based on Scout UAV system by Israel Aerospace Industries (IAI). The RANGER UAV uses a compact hydraulic launcher for takeoff. Due to a modular payload, the system can be adapted to a wide range of civilian and military missions. A skid based landing system enables UAV to land nearly anywhere, on grass or on concrete runways, on snow or ice. In reference to the manufacturer, RANGER is the only tactical UAV system worldwide certified to fly in civilian airspace as well as populated areas. The RANGER system is serviced with the Swiss Air Force under the designation ADS-95

Table1 General and Performance attributes of RANGER UAV

TYPE: TACTICAL UAV	
Dimension	
Length	4,60m(15.13ft)
Height	1,13(3.71ft)
Wingspan	5,70m(18.73ft)
Weight	
Maximum take-off weight	285kg(628lb)
Payload	45kg(99lb)
Propulsion	
Engine type	2cylinder 2 stroke
Performance	31,5kW
Flight Performance	
Speed	240 km/h max.
Service ceiling	Upto 18'000 ft
Range	upto 180 km
Endurance	Upto 9 hours

2.2 Mathematical model

Analysis is performed with the ANSYS CFX solver, allowing to the turbulence and Navier stroke equation approaches for 100 iterations. In this study an upstream velocity of 34.3m/s & 65.17m/s is specified at the inlet boundary. All other boundary conditions remain the same as well. In order to determine the necessary loads, the Mach no, temperature and velocity of the UAV is to be studied and the lift and drag calculation is performed.

Mach number: The Mach number (M or Ma) is a dimensionless quantity gives the ratio off-low velocity past over the boundary in the domain to the speed of the sound.

$$M = V/a$$

Where

M= Mach number = Flow velocity over boundaries and a= Speed of the sound

$$a = \sqrt{\gamma RT} = 340 \text{ m/s}$$

Where

γ is the density of air 1.4

R is the gas consistent 287J/kgK

T is the temperature of air (25°C) in K

The drag force acting on a body in fluid flow can be expressed as $D = 1/2 CD \rho V^2 A$(1)

Where,

D=drag force (N), CD = drag co-efficient, ρ = density of fluid (kg/m³), V = flow velocity (m/s), A = body area (m²)

Lifting Force

The lifting force acting on a body in a fluid flow can be expressed as $L = 1/2 CL \rho V^2 A$ (2)

Where, L = lifting force (N), CL=lifting co-efficient, ρ =density of fluid (kg/m³), V=flow velocity (m/s), A = body area (m²)

Calculation

for co-efficient of Drag & Lift

For fuselage with velocity-34.3 m/s at Mach no 0.1

$$\text{Drag Co-efficient } (C_D) = \frac{2 \times F_D}{\rho \times V^2 \times A} = \frac{2 \times 14.41}{1.4 \times 34.3^2 \times 1.04}$$

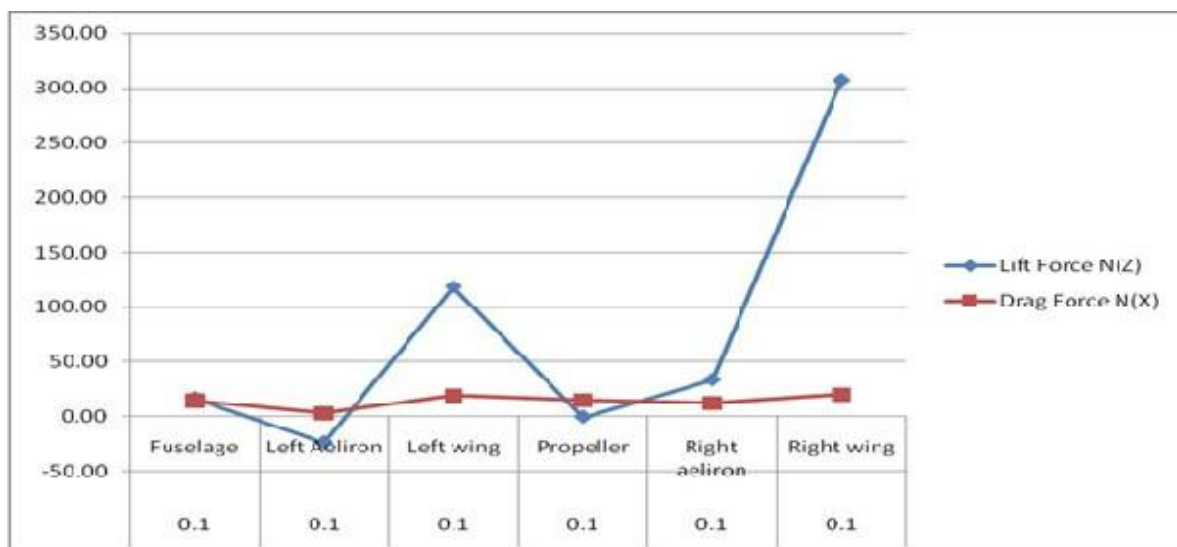
$$\text{Drag co-efficient } (C_D) = 0.0168$$

$$\text{Lift Co-efficient } (C_L) = \frac{2 \times F_L}{\rho \times V^2 \times A} = \frac{2 \times 16.76}{1.4 \times 34.3^2 \times 1.04}$$

$$\text{Lift co-efficient } (C_L) = 0.0195$$

Table2 forces on different parts at Mach No 0.1

Location	Drag Force N(X)	Side Force N(Y)	Lift Force N(Z)	Area m ²	Velocity m/s	Density kg/m ³	Drag Co-efficient(X)	Lift Co-efficient(Z)
Fuselage	14.41	0.39	16.76	1.04	34.30	1.40	0.0168	0.0195
Left Aileron	3.00	-2.00	-23.45	0.1414	34.30	1.40	0.0257	-0.2013
Left wing	19.39	7.87	118.49	1.38	34.30	1.40	0.0170	0.1042
Propeller	14.75	-0.01	-0.27	0.07	34.30	1.40	0.2558	-4.6835
Right Aileron	12.52	-2.28	33.61	0.1414	34.30	1.40	0.1075	0.2886
Right wing	20.02	-23.22	306.80	1.3809	34.30	1.40	0.0176	0.9780



Graph1. Mach no 0.1 v/s force on different parts

For Drag & Lift co-efficient on fuselage with velocity-65.17m/s at Mach no 0.19

$$\text{Drag co-efficient } (C_D) = \frac{2 \times F_D}{\rho \times V^2 \times A}$$

$$= \frac{2 \times 50.69}{1.4 \times 65.17^2 \times 1.04}$$

Drag co-efficient (C_D) = 0.0163

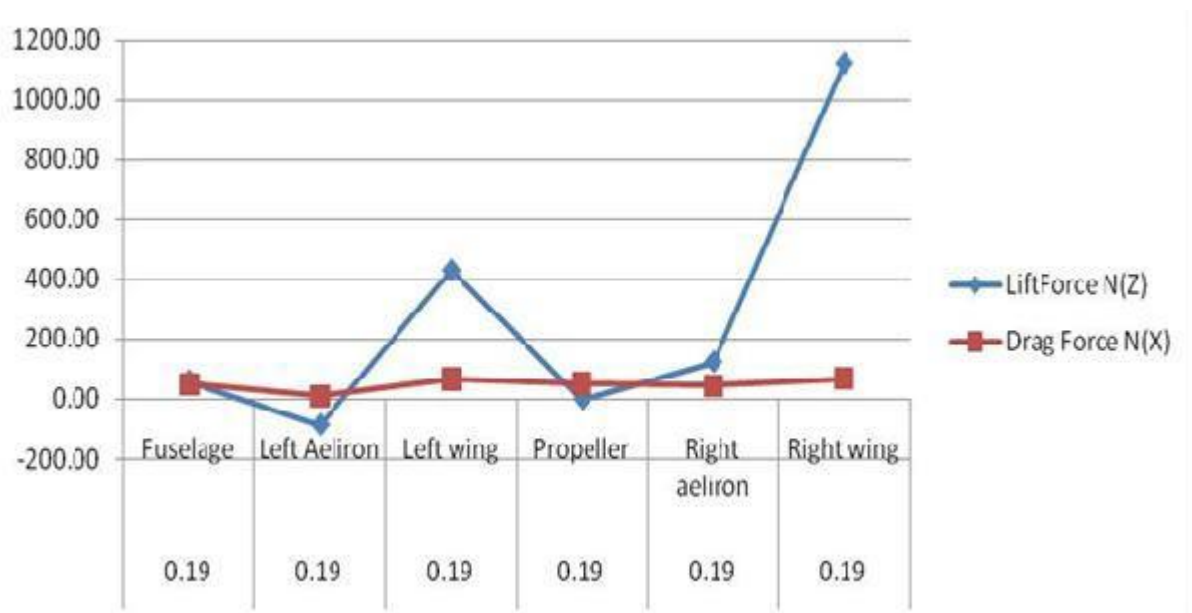
$$\text{Lift Co-efficient } (C_L) = \frac{2 \times F_L}{\rho \times V^2 \times A}$$

$$= \frac{2 \times 60.88}{1.4 \times 65.17^2 \times 1.04}$$

Lift co-efficient (C_L) = 0.0196

Table3 Forces on different parts at a Mach no 0.19

Loc ation	Drag Force N(X)	Side Force N(Y)	Lift Force N(Z)	Area m ²	Velo city m/s	Density kg/m ³	Drag Co-efficient(X)	Lift Co-efficient(Z)
Fuselage	50.69	1.49	60.88	1.04	65.17	1.40	0.0163	0.0196
Left Aileron	10.92	-7.35	-86.35	0.1414	65.17	1.40	0.0259	-0.2054
Left wing	66.72	28.58	430.38	1.38	65.17	1.40	0.0162	0.1049
Propell er	53.66	-0.05	-1.02	0.07	65.17	1.40	0.2578	-0.0049
Right Aileron	45.57	-8.44	123.70	0.1414	65.17	1.40	0.1084	0.2942
Right wing	69.09	-84.91	1121.70	1.3809	65.17	1.40	0.0168	0.2732



Graph2 Mach no 0.19 v/s force on different parts

3. CFD approach

Computational fluid dynamics is the branch of fluid dynamics that utilizes numerical strategies and calculations to examine the issues. It is a PC based instrument for reproducing the conduct of frameworks including liquid stream, heat exchange, and other related physical procedures. It works by explaining the conditions of liquid stream (in an exceptional structure) over a locale of enthusiasm, with determined conditions on the limit of that area.

3.1 Methodology of using CFD

The liquid continuum is discretized: i.e., field variables are approximated by their qualities at a limited number of hubs. The condition of movements is discretized: i.e., approximated regarding values at the hubs. Differential or Integral Equation → Algebraic Equation

The arrangement of Algebraic conditions is understood to give values at the hubs.

3.2 Steps involved in CFD process

The procedure of playing out a solitary CFD reproduction is part into 4-segments:

1. Creating the Geometry of a model/Mesh involves geometry parameters and domain shape and size.
2. Defining the Physics of Model includes flow properties, heat transfer and boundary condition.
3. Solving the CFD Problem involves iterations and numerical scheme.
4. Visualizing the Results in the Post-processor involves contours and streamlines.

4. External flow analysis using CFD

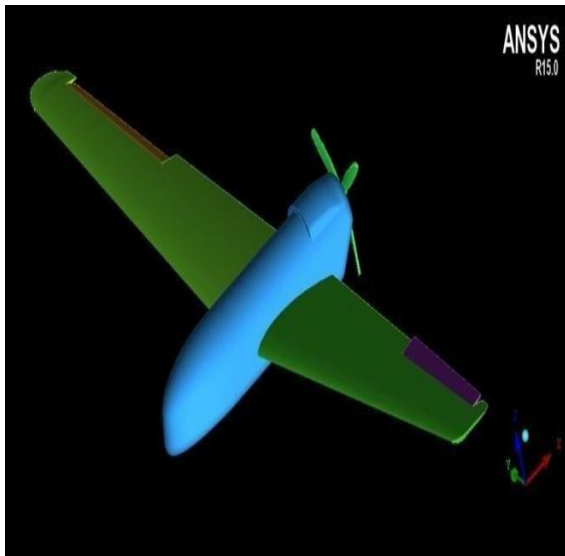
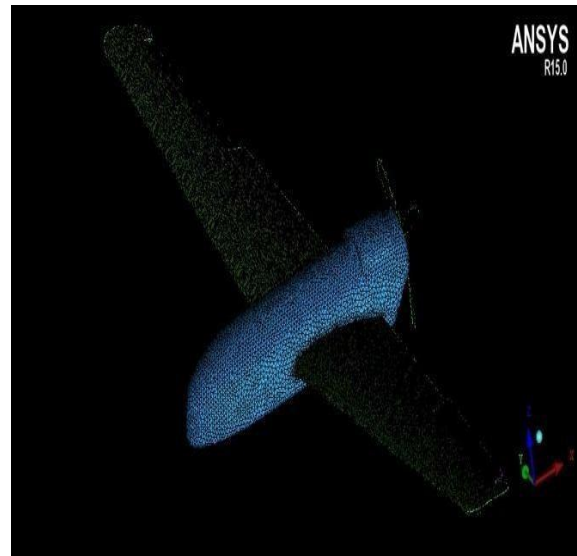
In order to calculate the pressure distributions on the aircraft, a CFD model of the UAV was developed. This was done simply by model to ICEM CFD platform which is the pre-process or for the flow analysis.

4.1 ANSYS ICEM CFD

ANSYS ICEM CFD is a popular proprietary software package used for CAD and mesh generation. It can create structured, unstructured, multi-block, and hybrid grids with different cell geometries. ANSYS ICEM CFD is meant to mesh a geometry already created using other dedicated CAD packages. Therefore, the geometry modelling features are primarily meant to 'clean-up' an imported CAD model.

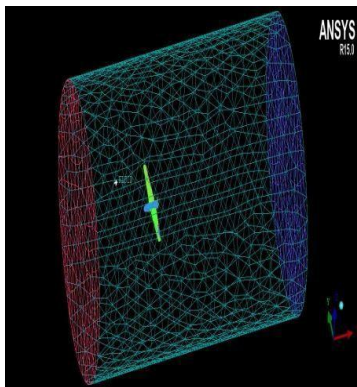
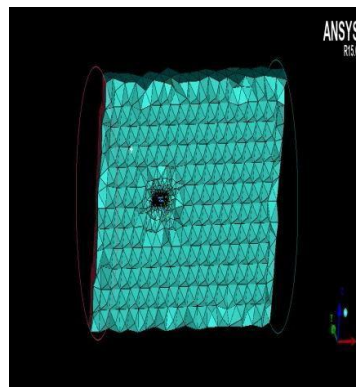
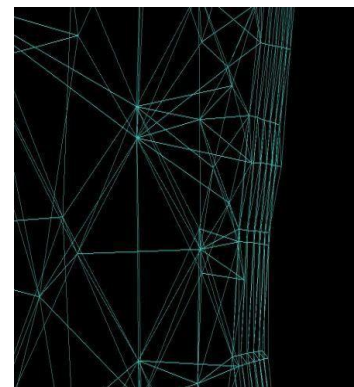
4.2 Creating Geometry and Flow Domain

The ANSYS workbench stage gives better bi-directional associations than all significant CAD frameworks, capable geometry change and creation instruments with ANSYS Design modeler, propelled fitting advancements in ANSYS coinciding, and simple move and customize exchange of information and results to share between applications

**Figure2.** Geometry of model**Figure3.** Mesh Model

4.3 Mesh Generation

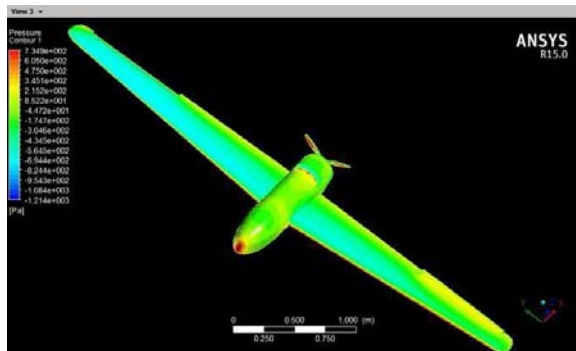
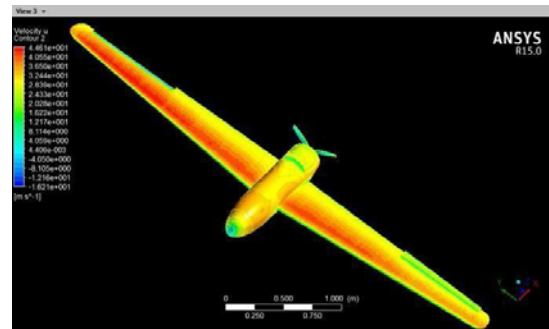
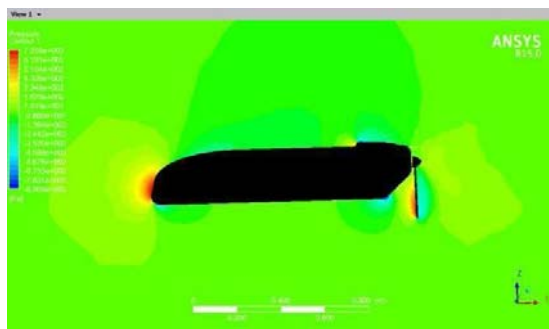
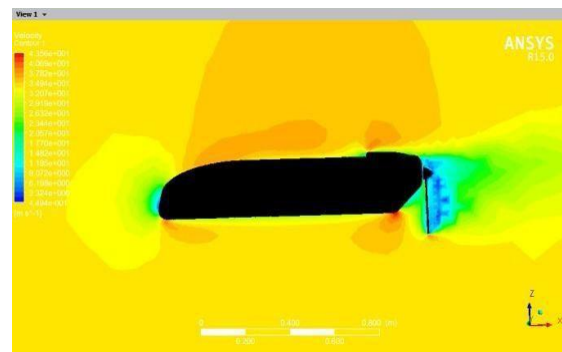
The UAV is meshed at its leading edges like wing, fuselage, horizontal and vertical tail for fine surface mesh size of 2 and mesh scale factor 0.1. And then complete mesh of size 12 and mesh scale factor of 1. The extent of the stream area depends on prior studies in which the space size was progressively expanded until there was no more a discernible impact on the airplane's lift and drag. A limited volume and thickness cross section is produced utilizing unstructured tetrahedral cells as a part of the zone nearly encompassing the flying machine, to take into account the complexities of the geometry, alongside a kaleidoscopic limit layer network 6 cells thick on the airplane's wetted surface. Organized hexahedral cells are then used to characterize the remaining stream Space.

**Figure4.** Geometry in a Domain**Figure5.** Volume Mesh of Domain**Figure6.** Prism Layers

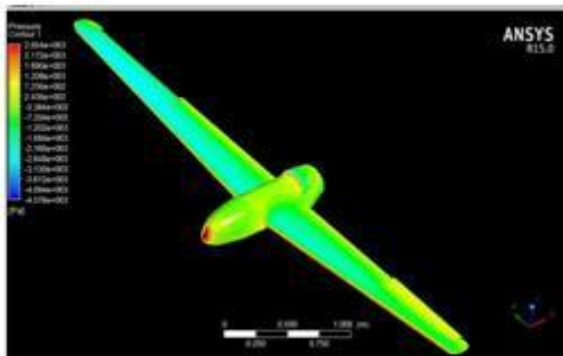
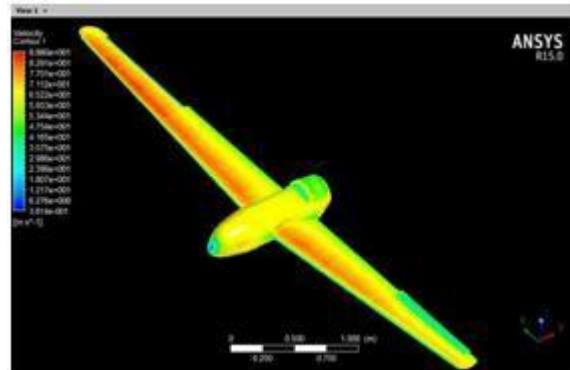
5. Results and Discussion

With initial conditions 34.3 m/s and 65.17 m/s free stream velocity in the x-direction, a temperature of 298K, and a pressure of 1 atm. At different Mach numbers of 0.1 & 0.19 the results of plots and stream lines are shown below.

5.1 At Mach 0.1

**Figure7.** Pressure Contour at Mach 0.1**Figure8.** Velocity Contour at Mach 0.1**Figure9.** Sectional view of pressure counter**Figure10.** Sectional view Velocity counter

5.2 At Mach 0.19

**Figure11:** Pressure Contour at Mach 0.19**Figure12:** Velocity Contour at Mach 0.19

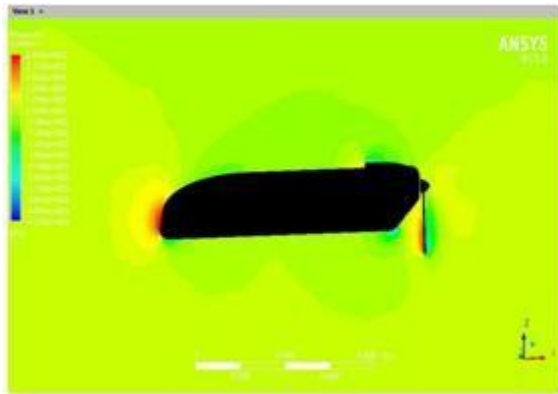
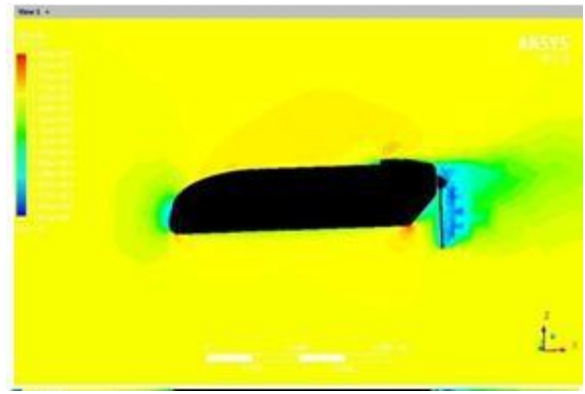
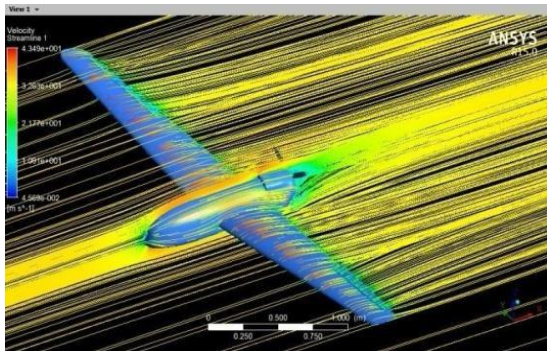
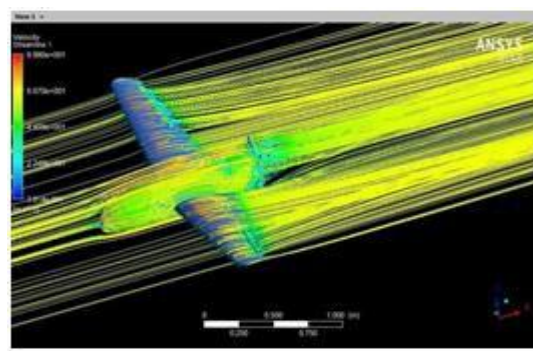
**Figure13:** Sectional view of pressure counter**Figure14:** Sectional view Velocity counter**Figure15:** velocity streamlines view at mach 0.1**Figure16:** velocity streamlines view at mach 0.19

Figure 7 to 16 shows the pressure and velocity counters at different Mach numbers

6. Conclusion and Scope for future work

6.1 Conclusion

In this study, the computational investigations of the streamlined qualities of RANGER, TUAV were completed utilizing CFD. The fundamental goal of this examination was to assess the most proper outline that will enhance the streamlined execution of RANGER, TUAV.

The essential points of this anticipate has been to research the potential burdens on various parts of a structure of an air ship. A survey of the writing in this field uncovered that not very many such studies have been distributed to date. For this a normal and testing configuration was considered and preparatory stream examination was done for the determination of the heaps following up on the distinctive parts.

From results the accompanying focuses can be derived:

- Material distribution is more where stress is high and material is made void where stress is considerably low.
- All design aspects can be met with minimal material.

6.2 Scope for future work

There is a degree to extend this work as to changes. Any sort of alteration can be fused into the model and can be investigated comparative path as specified in the examination. In current venture the work completed for UAV's.

In spite of the advances coming about because of the work contained in this work, a few issues stay to be tended to.

- Work can be extend with the different velocity in the sense of Mach no.
- Using different UAV's may be a HALE or MALE.

References

- [1] "Wind tunnel experiments and CFD analysis of blended wing body unmanned aerial vehicle at mach 0.1 and mach 0.3" by wirachman wisnoe, rizal effendi mohd nasir, wahyu kuntjoro and aman mohd ihsan mama.
- [2] *Int'l J of aeronautical & space sci.* 15(4), 374-382 (2014) DOI: 10.5139/IJASS.2014.15.4.374 "computational analysis of the aerodynamic performance of a long endurance UAV" by wonjin jin and yung gyo lee.
- [3] *International journal of mechanical and production engineering* , ISS: 2320-2092, volume -2, issue – 9, sept 2014 "CFD analysis of an RC aircraft wing CFD analysis of an rc aircraft".
- [4] "Design and analysis of a light cargo UAV prototype" anastasios p. kovanis, Vangelis skarperdas, john A. Ekaterinaris.
- [5] Hess, J.L. and Friedman, D.M., "Analysis of Complex Inlet Configurations Using a Higher-Order Panel Method," AIAA paper 83-1828, presented at the *AIAA Applied Aerodynamics Conference*, Danvers, Massachusetts, July 1983.
- [6] Bristow, D.R., "Development of Panel Methods for Subsonic Analysis and Design," *NASA CR-3234*, 1980.
- [7] Ashby, Dale L.; Dudley, Michael R.; Iguchi, Steve K.; Browne, Lindsey and Katz, Joseph, "Potential Flow Theory and Operation Guide for the Panel Code PMARC", *NASA NASA-TM-102851* 1991.
- [8] K. Sreelakshmi, B. Niharika, "Design and Analysis of mini-UAV", *International Journal of Mechanical and Production Engineering Research and Development*.
- [9] Woodward, F.A., Dvorak, F.A. and Geller, E.W., "A Computer Program for Three-Dimensional Lifting Bodies in Subsonic Inviscid Flow," *USAAMRDL Technical Report, TR 74-18, Ft. Eustis, Virginia*, April 1974
- [10] Katz, J. and Maskew, B., "Unsteady Low-Speed Aerodynamic Model for Complete Aircraft Configurations," *AIAA paper 86-2180*, presented at the *AIAA Atmospheric Flight Mechanics Conference, Williamsburg Virginia, August 1986*.
- [11] B Niharika, V Saibasha N Madhavi, "Design and Analysis of Humpback Whale Wing with Winglets", *International Journal of Civil Engineering & Technology*
- [12] K. Shiva Shankar, M. Satya narayana Gupta, G. Parthasarathy, "Comparative Study of CFD Solvers for Turbulent Fuel Flow Analysis to Identify Flow Nature", *International Journal of Civil Engineering & Technology*
- [13] MD Khaleel, Marampalli Shilpa, L. Farooq,"Modeling and CFD Analysis on One Stage of Turbine of Gas Turbine Engine", *International Journal of Civil Engineering & Technology*.