iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 4 **Issue:** VIII **Month of publication:** August 2016

DOI:

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Aerodynamic Characteristic Analysis of UAV (Unmanned Aerial Vehicle) By Using CFD

Naresh S[1], Dr.V. Ravi[2]
[1]PG student, [2]Professor, Department of Mechanical engineering, BITM, Ballari, Karnataka, India

Abstract—the paper discusses the aerodynamics behavior of a baseline design and analysis of a UAV (Unmanned Aerial Vehicle). Analysis is presented, i.e. Steady-state, three- dimensional Computational Fluid Dynamics (CFD) at Mach 0.1 and 0.19 on a Ranger TUAV. In this method of analysis, Lift Coefficient (CL) and Drag Coefficient (CD) are measured and compared at respective Mach numbers. Pressure contours and velocity contours are plotted and the Turbulence area is predicted from streamline view, both extracted from ANSYS ICEM CFD analysis. UAV has its own advantages such as, computerizing and autonomous without an on-board pilot by using digital and electronic system which manned aircraft don't have. Therefore, there is no risk of loss of life and it is easier to be maintained than manned aircraft.
Key Words: Unmanned Aerial Vehicle, Computational Fluid Dynamics, Ranger TUAV, CL, CD, ANSYS ICEM CFD.

I. INTRODUCTION

UAV is an acronym for Unmanned Aerial Vehicle, which is an aircraft with no pilot on board. UAVs can be remote controlled aircraft (e.g. flown by a pilot at a ground control station) or can fly autonomously based on pre – programmed flight plans or more complex dynamic automation systems. UAVs are currently used for a number of missions, including reconnaissance and attack roles. For the purposes of this article, and to distinguish UAVs from missiles, a UAV is defined as being capable of controlled, sustained level flight and powered by a jet or reciprocating engine. In addition, a cruise missile can be considered to be a UAV, but is treated separately on the basis that the vehicle is the weapon. The acronym UAV has been expanded in some cases to UAVS (Unmanned Aircraft Vehicle System).The acronym UAS (Unmanned Aircraft System) to reflect the fact that these complex systems include ground stations and other elements besides the actual air vehicles. The typical launch and recovery method of an unmanned aircraft is by the function of an automatic system or an external operator on the ground and also referred to as an un-piloted aerial vehicle and a remotely piloted aircraft (RPA) by the International Civil Aviation Organization (ICAO), is an aircraft without a human pilot aboard.

II. RANGER TATICAL UAV

RANGER is a tactical UAV system (TUAV) built as a Swiss-Israeli joint venture between Swiss aerospace enterprise Aviation and Israeli aerospace company Israel Aerospace Industries. Its design and some of its technology is based on the Scout UAV system by Israel Aerospace Industries

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 the UAV to land nearly anywhere, on grass or on concrete runways, on snow or ice. According to the manufacturer, RANGER is the only tactical UAV system worldwide certified to fly in civilian airspace as well as over populated areas. The RANGER system is in service with the Swiss Air Force under the designation ADS-95. From 1988 to 1999, The Air Force's RANGERs are also used by Swiss police for reconnaissance, search and rescue missions.

General and Performance characteristics of UAV

TYPE: TACTICAL UAV	
Dimension	
Length	4,61 m (15.13 ft)
Height	1,13 (3.71 ft)
Wingspan	5,71 m (18.73 ft)
Weight	
Maximum take-off weight	285 kg (628 lb)

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Payload	45 kg (99 lb)
Propulsion	
Engine type	2 cylinder 2 stroke
Performance	31,5 kW
Flight Performance	
Speed	240 km/h max.
Service ceiling	up to 18'000 ft
Range	up to 180 km (110 mi)
Endurance	up to 9 hours

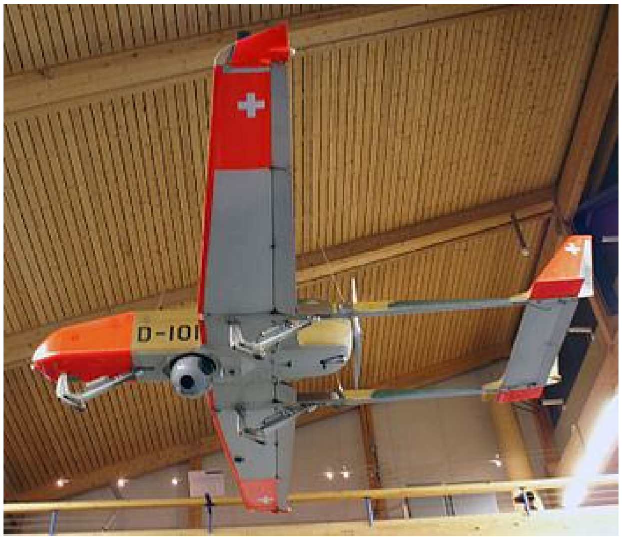


Fig-2.1Ranger TUAV

III. MATHEMATICAL MODEL

A. *Calculation of Flight Loads*

Analysis is performed with the ANSYS CFX solver, according to the turbulence and Navier stroke equation methods 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 wall. 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.

1) *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.

$M=U/c$

Where,

M- Mach number.

U- Flow velocity over boundaries.

c- Speed of sound.

$c= (\gamma*R*T) = 340m/s$

Where,

γ is the density of air 1.4

R is the gas consistent 287J/kg K

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

Drag Force

The drag force acting on a body in fluid flow can be expressed as

$F_D = 1/2\ C_D\ \rho\ V^2\ A$………………….. (1)

Where, F_D = drag force (N), C_D = drag co-efficient, ρ = density of fluid (kg/m^3), V = flow velocity (m/s), A = body area (m^2)

Lifting Force

The lifting force acting on a body in a fluid flow can be expressed as

$F_L = 1/2\ C_L\ \rho\ v^2\ A$ ………………… (2)

Where, F_L = lifting force (N), C_L = lifting co-efficient, ρ = density of fluid (kg/m^3), V= flow velocity (m/s), A = body area (m^2)

Calculation for co-efficient of Drag & Lift

WKT From, equation-1 we have, for fuselage with velocity-34.3m/s at Mach no 0.1

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

Drag co-efficient (C_D) =0.0168

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

Lift co-efficient (C_L) =0.0195

Table for forces on different parts at a Mach no 0.1 with a velocity of 34.3m/s.

Location	Drag Force N(X)	Side Force N(Y)	Lift Force N(Z)	Area m²	Velocity m/s	Density kg/m³	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 Aeliron	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 Aeliron	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

TABLE 3.1- Forces on Different Parts

IV. CFD APPROACH & METHODOLOGY

Computational Fluid Dynamics (CFD) is one of the branches of fluid dynamics that uses numerical methods and algorithms to analyze the problems. It is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes.

A. Methodology Of Using CFD

1) The fluid continuum is discretized: i.e., field variables are approximated by their values at a finite number of nodes.
2) The equation of motions is discretized: i.e., approximated in terms of values at the nodes.

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Differential or Integral Equation → Algebraic Equation

3) The system of Algebraic equations is solved to give values at the nodes.

B. Steps Involved In CFD Process

The process of performing a single CFD simulation is split into four components:

1) Creating the Geometry/Mesh
2) Defining the Physics of the Model
3) Solving the CFD Problem
4) Visualizing the Results in the Post-processor

V. 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-processor for the flow analysis.

A. 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 modeling features are primarily meant to 'clean-up' an imported CAD model.

B. Creating the Geometry and Flow Domain

The ANSYS workbench platform provides superior bi-directional connections to all major CAD systems, powerful geometry modification and creation tools with ANSYS Design modeler, advanced meshing technologies in ANSYS meshing.

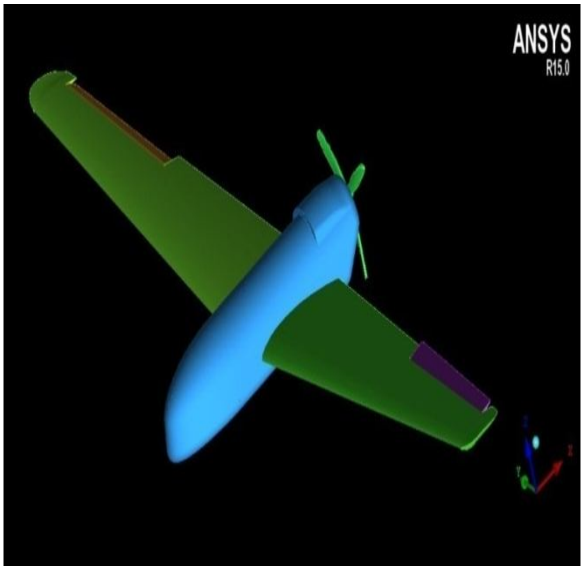


Fig 5.1-Geometry of model

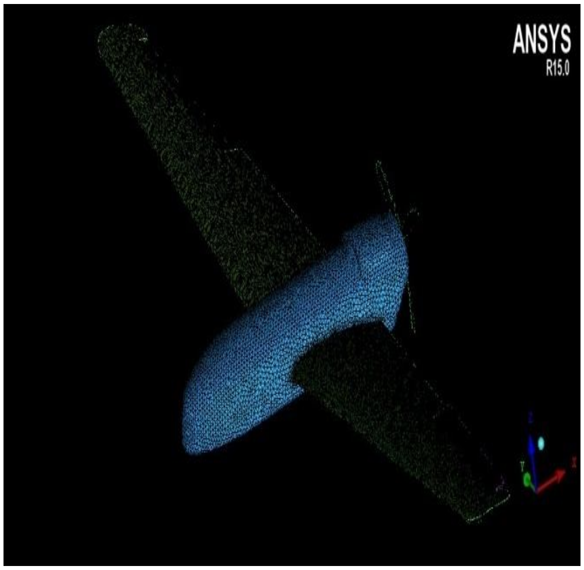


Fig 5.2-Meshed model

C. 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.A finite volume and density mesh is generated using unstructured tetrahedral cells in the area closely surrounding the aircraft, to allow for the complexities of the geometry, along with a prismatic boundary layer mesh 6 cells thick on the aircraft's wetted surface.

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

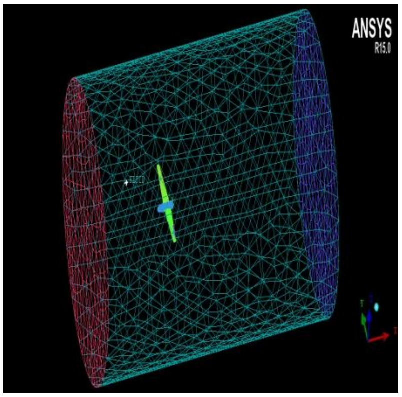


Fig 5.3-Geometry in a Domain

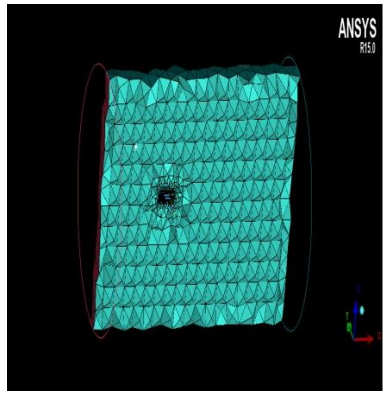


Fig5.4-Volume Mesh of Domain

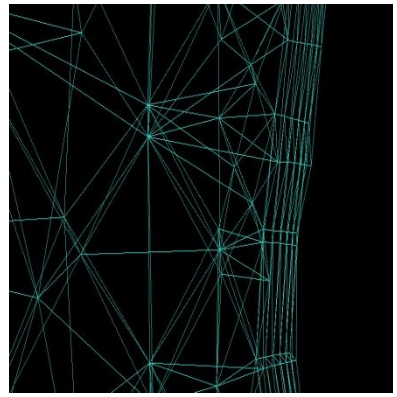
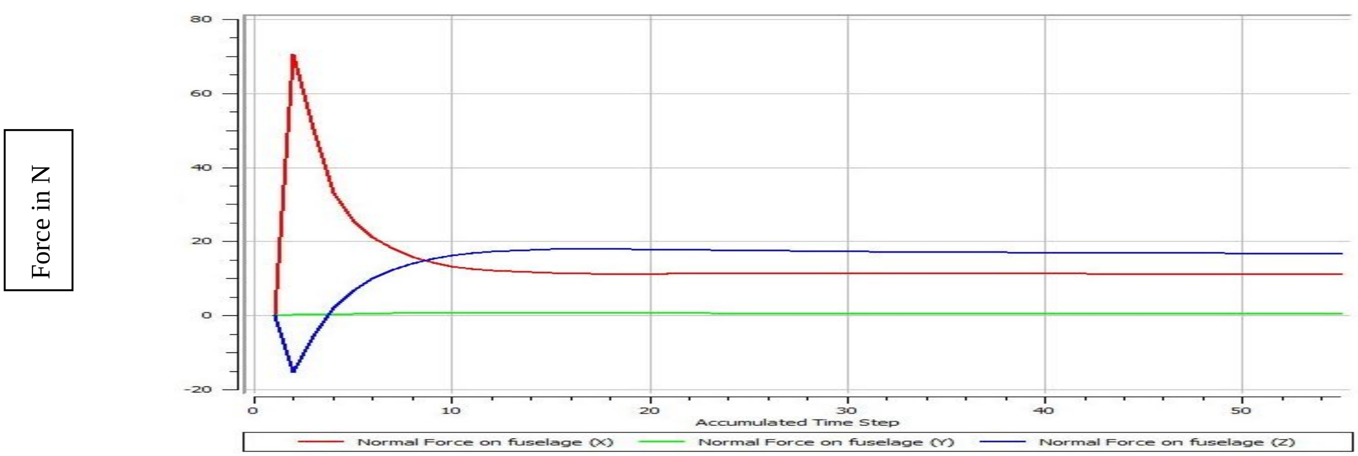


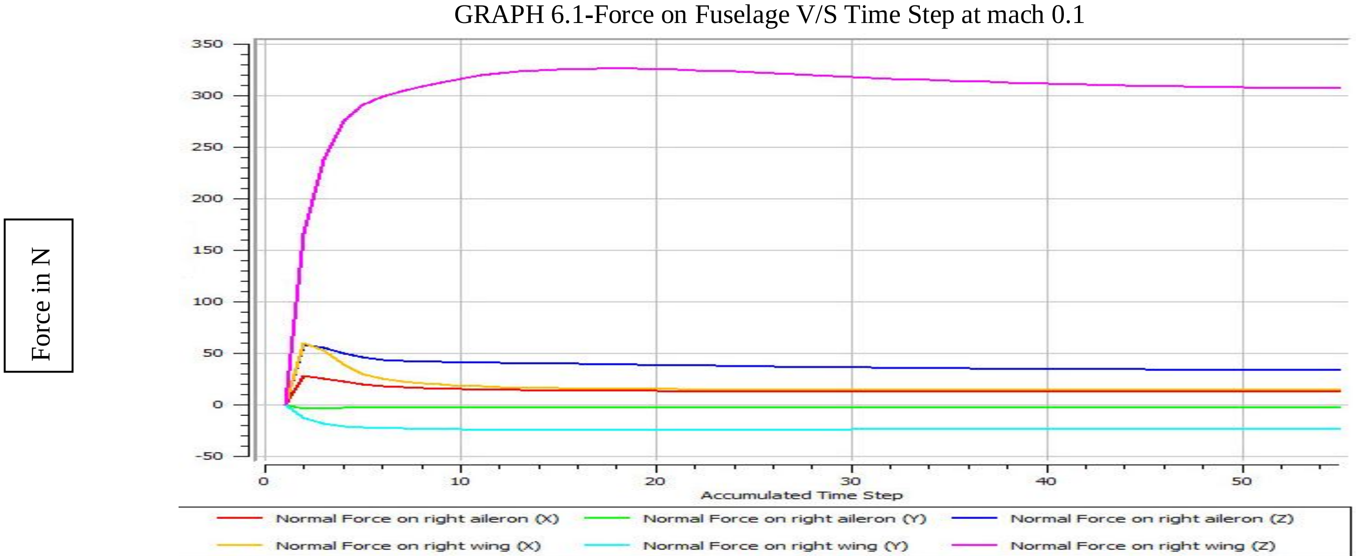
Fig 5.4- Prism Layers on Geometry

VI. RESULT AND DISCUSSION

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

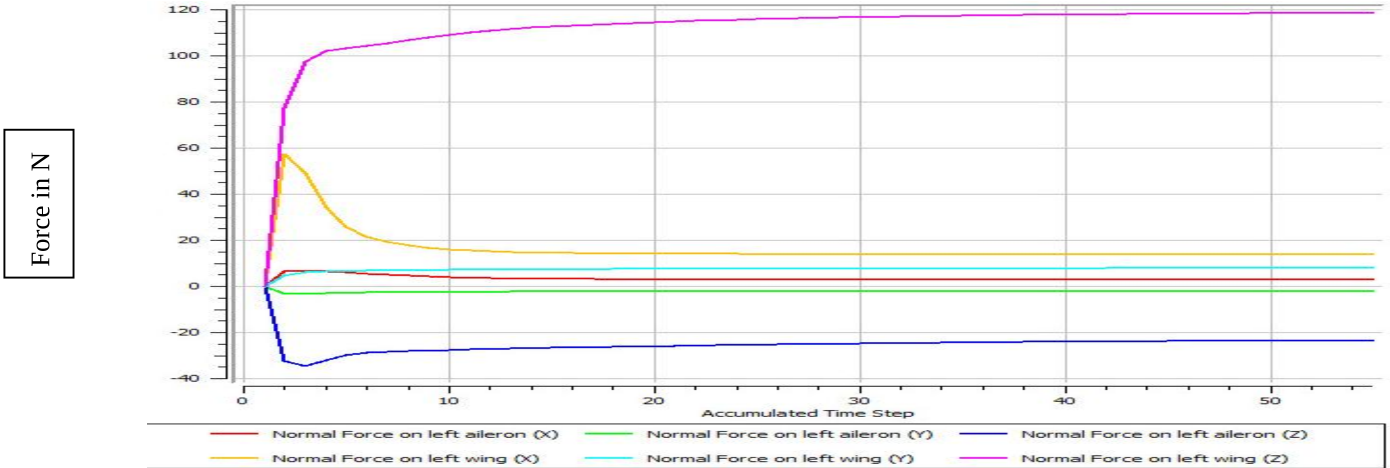


GRAPH 6.1-Force on Fuselage V/S Time Step at mach 0.1



GRAPH 6.2-Force on Right Wing & Aileron V/S Time Step at mach 0.1

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering
Technology (IJRASET)



GRAPH 6.3-Force on Left Wing & Aileron V/S Time Step at mach 0.1

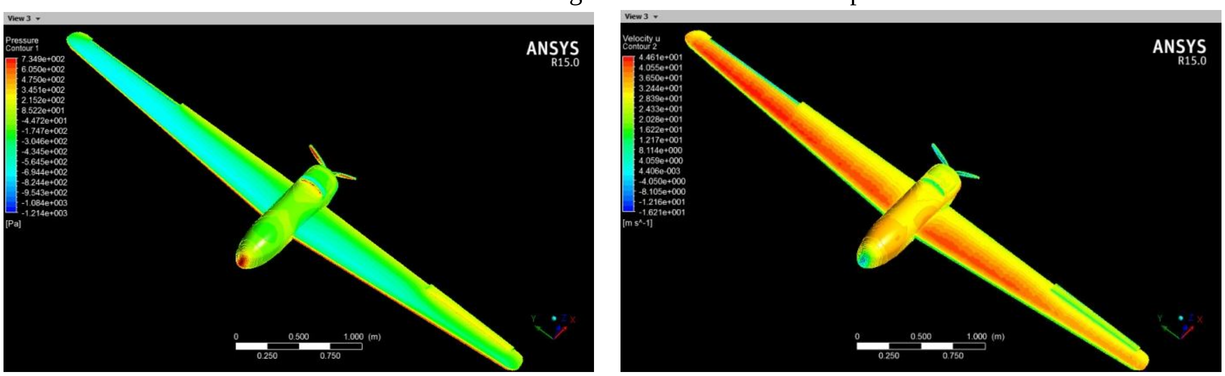


Fig 6.1-Pressure & Velocity Contour at Mach 0.1

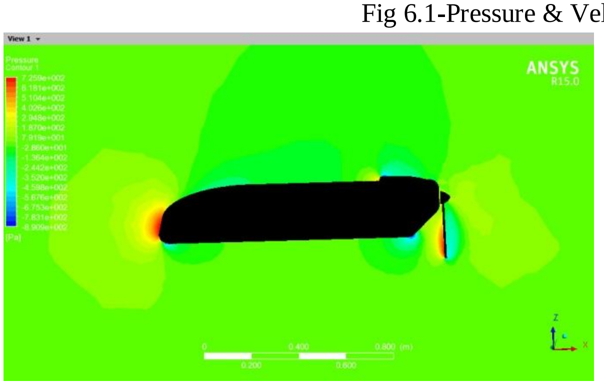


Fig 6.2- Sectional view of Pressure Contour at Mach 0.1

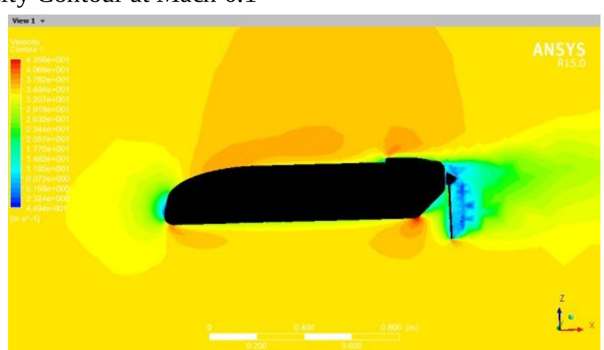


Fig 6.3- Sectional view of velocity Contour at Mach 0.1

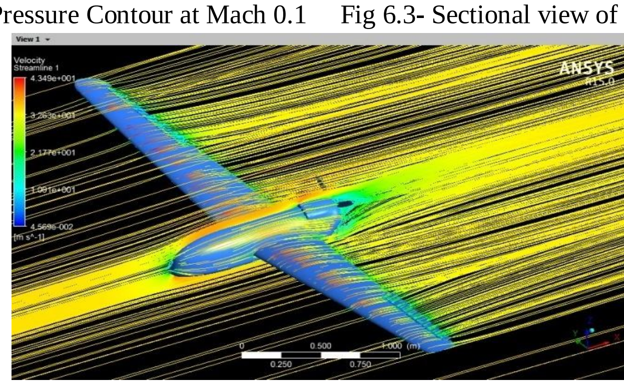


Fig 6.4-Streamline View at Mach 0.1

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

VII. CONCLUSION AND SCOPE FOR FUTURE WORK

A. Conclusion

In this study, the computational analyses of the aerodynamic characteristics of RANGER, TUAV were carried out using CFD. The main objective of this analysis was to evaluate the most appropriate design that will improve the aerodynamic performance of RANGER, TUAV.

The primary aim of this project has been to investigate the potential loads on different parts of a structure of an aircraft. A review of the literature in this field revealed that very few such studies have been published to date. For this a typical and challenging design was considered and preliminary flow analysis was carried out for the determination of the loads acting on the different parts.

From the results the following points can be inferred:

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

B. Scope For Future Work

There is a scope to extend this work with regard to modifications. Any kind of modification can be incorporated in to the model and can be analyzed in similar way as mentioned in the analysis. In current project the work carried out for UAV's.

Despite the advances resulting from the work contained in this work, several issues remain to be addressed.

1) Work can be extend with the different velocity in the sense of Mach no.
2) Using different UAV's may be a HALE or MALE.
3) This scope can also be carried out at different Angle of Attack which is crucial at fight take-off conditions to find the lift and drag forces.
4) Carrying out at steady i.e. in wind tunnel and also in transient or dynamic state.

REFERENCES

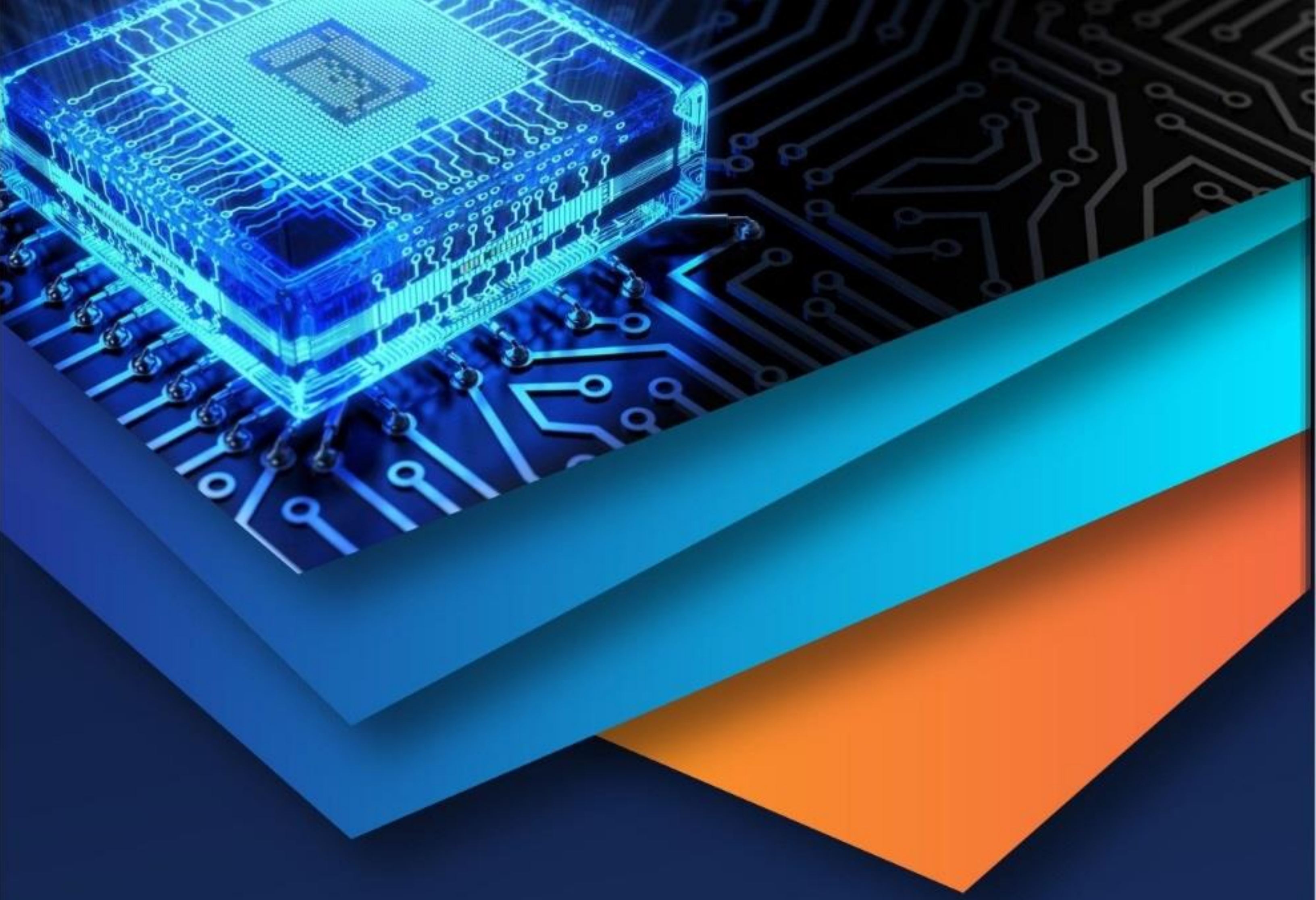
[1] International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 2, Issue-9, Sept.-2014 CFD Analysis of An RC Aircraft Wing CFD analysis of an rc aircraft wing shreyas Krishnamurthy, suraj jayashankar, sharath v rao, rochen krishna t s, shankargoud nyamannavar

[2] Design and analysis of a light cargo uav prototype Anastasios P. Kovanis, Vangelis Skaperdas, John A. Ekaterinaris.

[3] Reverse Engineering and Aerodynamic Analysis of a Flying Wing UAV Navabalachandran, Low Jun Horng, G. Leng. Aeronautical Engineering Group, Department of Mechanical Engineering, National University of Singapore.

[4] Design and Analysis of Morphing Wing for Unmanned Aerial Vehicles by Vlad Paul Galantai, Masters of Applied Science, Department of Mechanical and Industrial Engineering, University of Toronto 2010.

[5] International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Index Copernicus Value (2013): 6.14 | Impact Factor (2014): 5.611 Volume 4 Issue 11, November 2015 Design and Fluid Flow Analysis of Unmanned Aerial Vehicle (UAV) by B. Ravi Theja, Dr. M. Satyanarayana Gupta.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY