

# **CFD Simulations of Aircraft Body with Different Angle of Attack and Velocity**

Shivasharanayya Hiremath<sup>1</sup>, Anandkumar.S.Malipatil<sup>2</sup>

<sup>1,2</sup>Department of Thermal Power Engineering, VTU PG center, Gulbarga, India

**ABSTRACT:** In this project we describe the complete process of modeling and simulation of computational fluid dynamics (CFD) problems that occur in engineering practice. We focus mainly on the simulation of the airflow around the aircraft. The fluid flow simulations are obtained with the CFX software package ANSYS. We use the solver based on the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE). The important part is the preparation of the model with the software ANSYS. We will describe the preprocessing including the creation and modification of the surface mesh in ANSYS and the three-dimensional volume grid generation. We discuss the generation of the three-dimensional grid by the snappy Hex Mesh tool, which is included in the ANSYS package. Further, we present a way of analyzing the results and some of the outputs of the simulations and following analysis. The CFD simulations were performed on the computational model of the commercial aircraft. The computations were performed for different model settings and computational grids. It means that we considered laminar and turbulent flow and several combinations of the angle of attack and inlet velocity.

This case study aims to perform a CFD analysis on an aircraft model using CFX solver. While performing the simulations, meshing techniques, pre-processing and post processing sections and evaluation of a simulation is being learnt. Coefficient of lift and drag were also recorded as a user input data. These values were also compared by running two different simulations with one change of input parameter i.e. angle of attack and inlet velocity.

**KEYWORDS:** Aircraft body, Coefficient of Lift, Coefficient of Drag, CFD, Angle of attack.

## **I. INTRODUCTION**

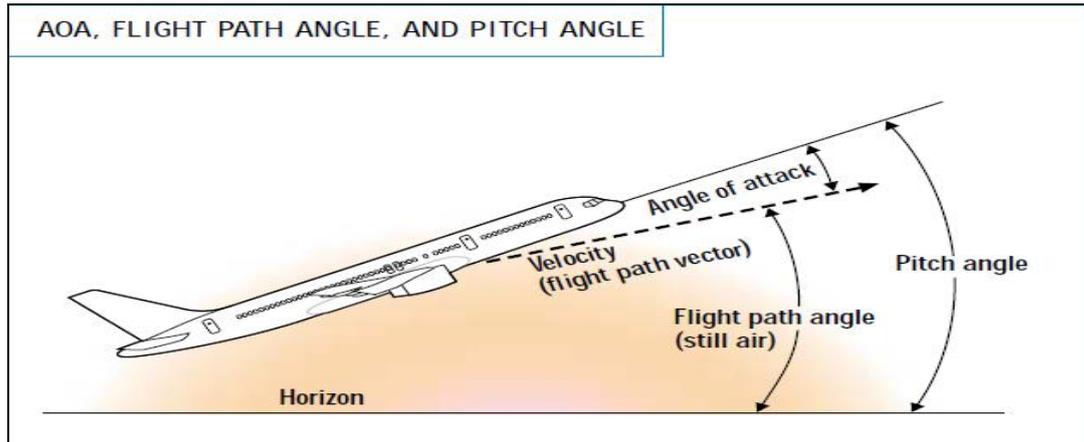
An **aircraft** is a machine that is able to fly by gaining support from the air, or, in general, the atmosphere of a planet. It counters the force of gravity by using either static lift or by using the dynamic lift of an airfoil, or in a few cases the downward thrust from jet engines. In an aircraft, multidisciplinary design environments are involved for achieving the critical mission requirements like payload-capacity, endurance, maneuverability, fuel-consumption, noise-emission etc. These in-flight performance parameters depend on the aerodynamic characteristics like lift, drag, vortex etc.

One of the foremost important aerodynamic force is induced drag, which is a drag caused by lift. It takes up approximately 33% of the total drag of the aircraft when in cruise and it is even more significant at low speed, accounting for 80%-90% of the aircraft drag, especially during landing and taking off situations. Streamline over an airfoil causes pressure difference between the top and bottom surface. However on a finite wing, there is a leakage of air molecules at the wing tip which causes downwash, thus generating vortices at the trailing edge of the wing. Wing tip sails are attached to the wings in such a way they use local airflows about the wing tips induced by the generation of lift on the wing to produce thrust. The environmental factors are like noise, air pollution around airports and impacts of climate change and other factor also play important role for future growth. Air travel now days impacts on the environment will gradually becoming power factor on aircraft design. It is important that to reduce emission **CO<sub>2</sub>** to achieve goals of 2020 launched by Europe commission.

Drag reduction is a great challenge but there is certainly room for improvements. The drag Breakdown of a civil transport aircraft shows that the skin friction drag and the lift-induced drag constitute the two main sources of drag, approximately one half and one third of the total drag for a typical long range aircraft at cruise conditions (Reneaux, 2004). This is why specific research on these topics has been initiated researchers and it seems that Hybrid Laminar Flow technology and innovative wing tip devices offer the greatest potential. Aircraft performance improvement can also be obtained through trailing edge optimization, control of the shock boundary layer interaction and of boundary layer separation. There are two key considerations in discussing drag. First, drag cannot yet be predicted accurately with high confidence levels (especially for unusual configuration concepts) without extensive testing (Sloof, 1988), and

secondly, no one is exactly sure what the ultimate possible drag level really is that can be achieved for a practical configuration. To this extent, aerodynamic designers are the dreamers of the engineering profession

### ANGLE OF ATTACK



**Figure 1: Angle of attack on plane**

Angle of attack is the angle between a reference line on a body (often the chord line of an airfoil) and the vector representing the relative motion between the body and the fluid through which it is moving. Angle of attack is the angle between the body's reference line and the oncoming flow. This article focuses on the most common application, the angle of attack of a wing or airfoil moving through air.

The **lift coefficient** ( $C_L$ ,  $C_a$  or  $C_z$ ) is a dimensionless coefficient that relates the lift generated by a lifting body due to the density of the fluid around the body, its velocity and an associated reference area. A lifting body is a foil or a complete foil-bearing body such as a fixed-wing aircraft.  $C_L$  is a function of the angle of the body to the flow, its Reynolds number and its Mach number. The lift coefficient  $c_l$  refers to the dynamic lift characteristics of a two-dimensional foil section, with the reference area replaced by the foil chord.

In fluid dynamics, the **drag coefficient** (commonly denoted as:  $c_d$ ,  $c_x$  or  $c_w$ ) is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air or water. It is used in the drag equation, where a lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag. The drag coefficient is always associated with a particular surface area.

## II. BACKGROUND

In the coming century, the impact of air travel on the environment will become an increasingly powerful influence on aircraft design.. Nevertheless, the design of aircraft to reduce fuel burn and hence CO2 Emission remains a key long-term objective; the paper considers the potential offered by new technology and new design concepts in this arena.

## III. METHODOLOGY AND BOUNDARY CONDITIONS

### 1. Creation of geometry

The following section shows the 2D diagram used to create the model in CATIA V5 R20.

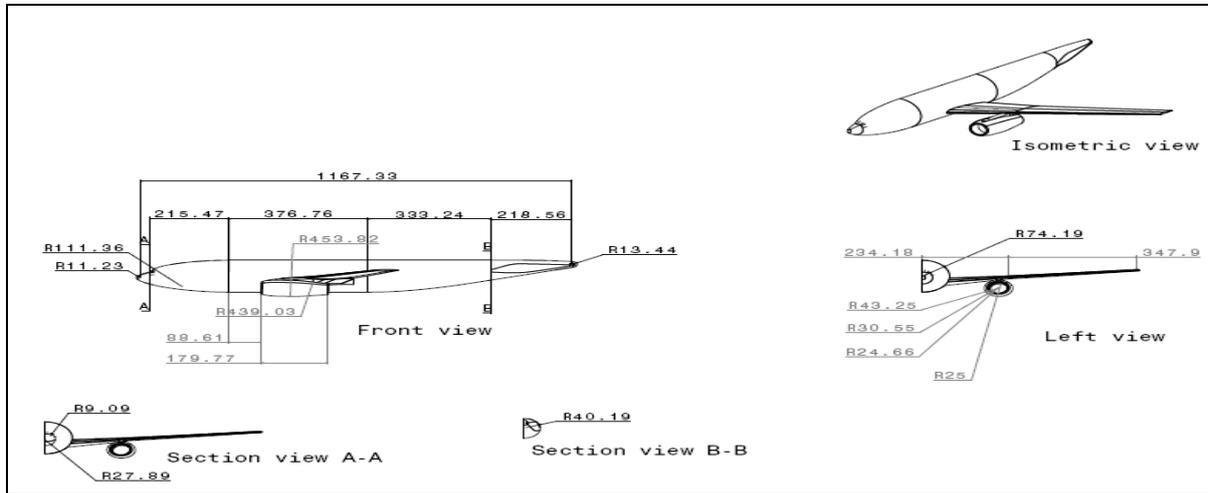


Figure 3.1: geometry creation of aircraft

## 2. Mesh generation

A model of an aircraft was given and the aircraft was symmetrical laterally. The model was then brought into ANSYS WB and was meshed appropriately. Meshing tools like Inflation and mapping couldn't be employed while meshing as its giving very fine mesh. Consequently this brings an error in CFX-Pre. A picture below shows the mesh which was carried out in this analysis.

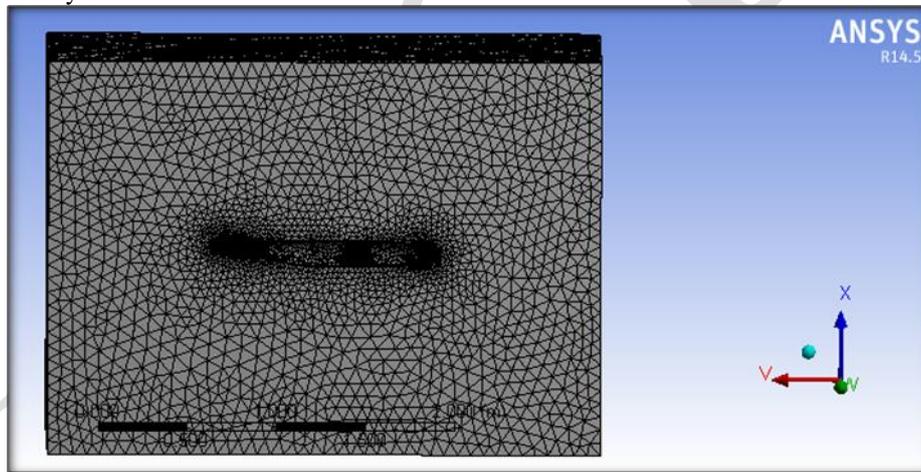


Figure 3.2: Grid generated around the fuselage for CFD analysis

## 3. ANALYSIS

CFD Analysis and study of results are carried out in 3 steps: Pre-processing, Solving and Post-processing by using CFX solver in ANSYS work bench.

**Velocity Inlet:** The inlet boundary conditions involve velocity components for varying angle of attack, turbulence intensity and turbulent viscosity ratio.

**Pressure Outlet:** Ambient atmospheric condition is imposed at outlet.

The velocity components are calculated for each angle attack case as follows. The x-component of velocity is calculated by  $x = u \cos \alpha$  and the y component of velocity is calculated by  $y = u \sin \alpha$ , where  $\alpha$  is the angle of attack in degrees.

**Symmetry:** This is the boundary wall to see the analysis.

**Operating condition:** The inlet velocity taken for fuselage 250 m/sec, for different angle of attack of air at 1.49deg, 5deg, 10deg, 15deg, 20deg, 25deg and at STP (Temperature=300k, Pressure=101325pa) as the fluid medium.

**Properties of Air:**

Density:  $\rho = 1.225 \text{ kg/m}^3$

Viscosity:  $\mu = 1.7894 \times 10^{-5} \text{ kg/m-s}$

**Solvers selection:** Pressure based solvers.

**Mathematical models:** K-  $\epsilon$  standard wall function

**Solution controls:** Gauss-Seidel flow turbulence energy.

**Momentum:** Second Order Upwind Scheme

**Initialization:** Inlet Values

**Force Monitors:** Lift and Drag

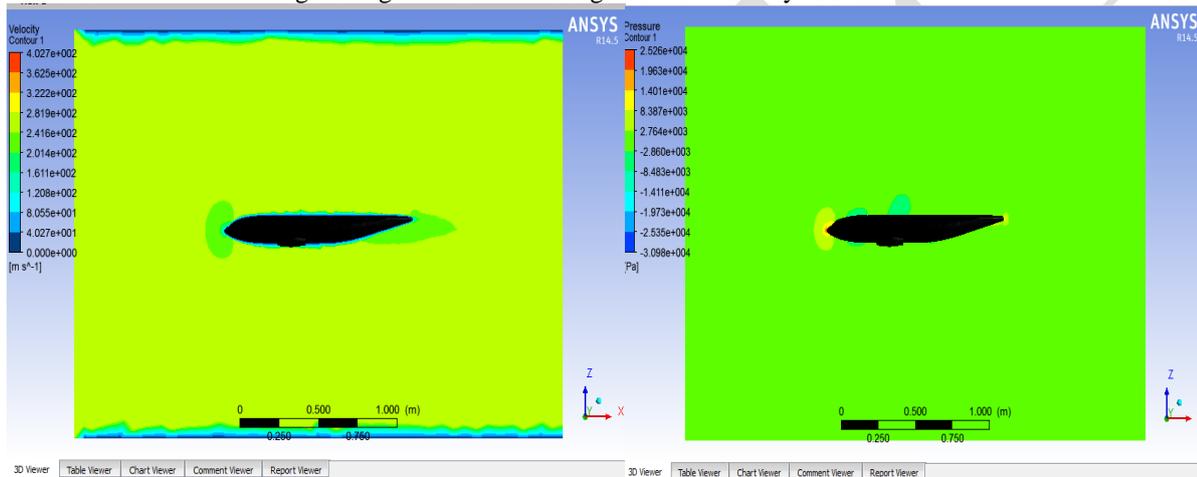
**Reference values:** Inlet Values

**Convergence Limit:**  $1 \times 10^{-6}$

**IV. RESULTS AND DISCUSSION**

Analysis set up of all cases is carried out in ANSYS CFX Solver. Result analysis is done using ANSYS Post processor. K-epsilon turbulence model is used for Analysis. Velocity and pressure plots are plotted for all the cases of study.

**Test case I:** For this fuselage at angle of attack 1.49deg and with velocity 250m/s

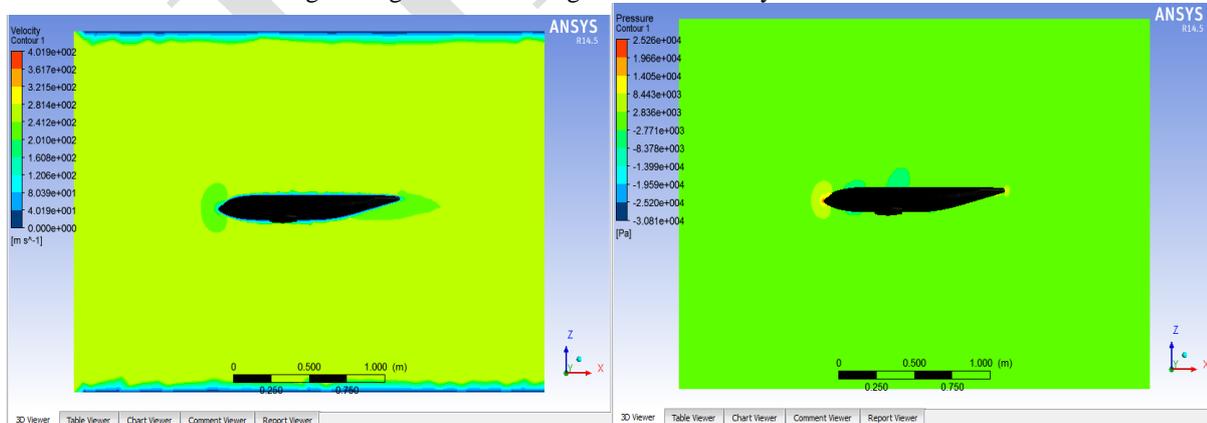


Velocity contour

Pressure contour

**Figure 4.1: Velocity contour and pressure contour at angle of attack 1.49deg**

**Test case II:** For this fuselage at angle of attack 2deg and with velocity 250m/s

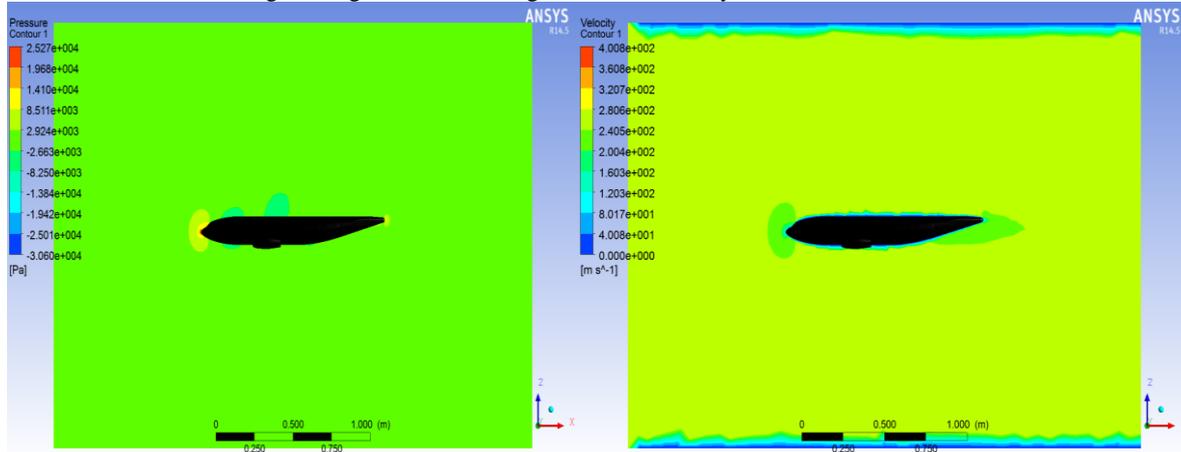


Velocity contour

Pressure contour

**Figure 4.2: Velocity contour and pressure contour at angle of attack 2deg**

**Test case III:** For this fuselage at angle of attack 5deg and with velocity 250m/s

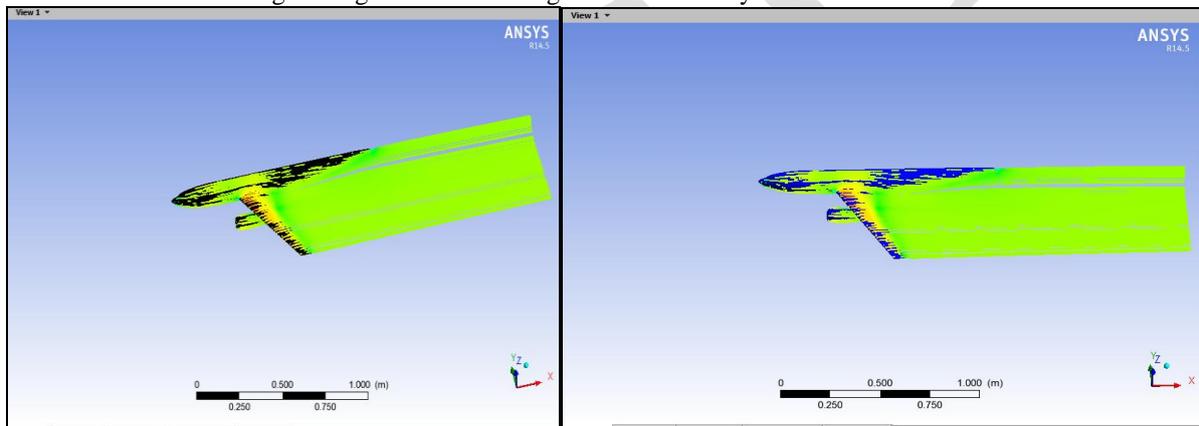


Pressure contour

Velocity contour

**Figure 4.3: Velocity contour and pressure contour at an angle of 5deg**

**Test case IV:** For this fuselage at angle of attack 10deg and with velocity 250m/s

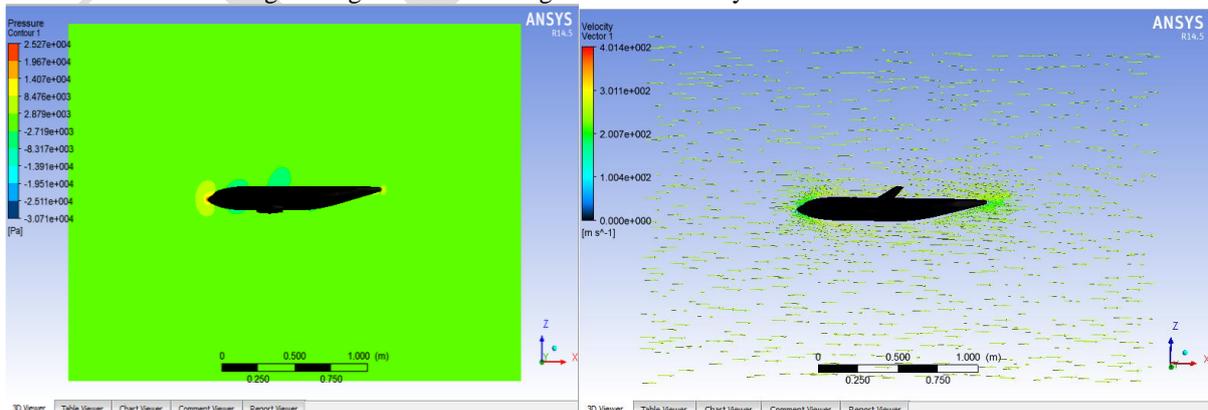


Pressure contour

Velocity contour

**Figure 4.4: Pressure contour and Velocity contour at angle of attack 10deg**

**Test case V:** For this fuselage at angle of attack 15deg and with velocity 250m/s

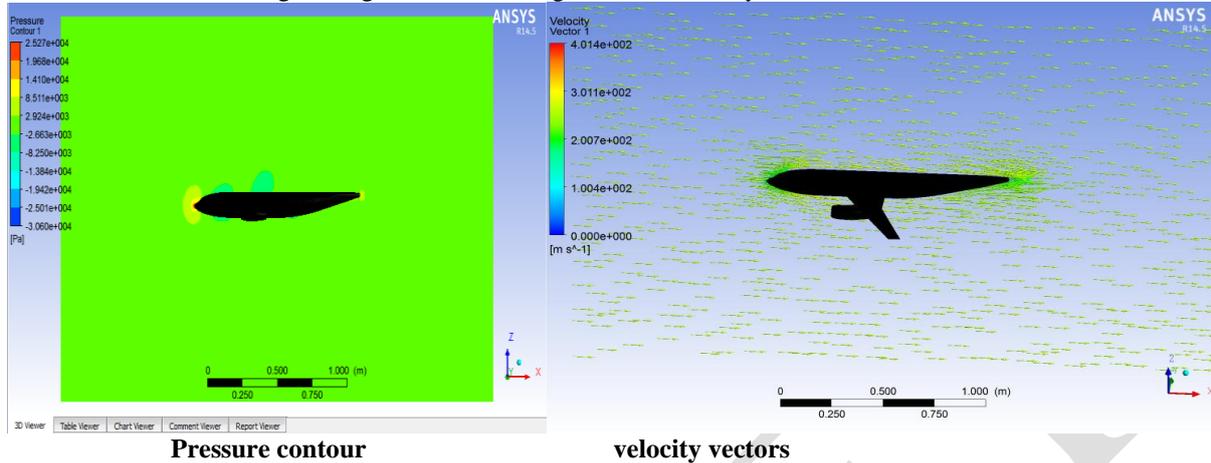


Pressure contour

Velocity vectors

**Figure 4.5: Pressure contour and Velocity vectors at an angle of attack 15deg**

**Test case VI:** For this fuselage at angle of attack 18deg and with velocity 250m/s



**Figure 4.6: Pressure contour and Velocity vectors at an angle of attack 18deg**

Above Figures shows the pressure distribution and velocity of air effects on aircraft. It is assumed that a stream air against the aircraft, which is fixed in air. It is equivalent to aircraft is moving in space. When the aircraft located in the stream of air velocity the flow apart near the leading edge and flow along the upper and lower surface of aircraft. Further downstream the velocity gradually decreases and reaches the trailing edge approximately 85% of free stream velocity. The velocity at the lower surface looks similar, but its level is considerably lower. Velocity and pressure are dependent each other – Bernoulli’s equation says that increasing the velocity decreases the pressure and vice versa. Thus higher velocities on upper side of airfoil side result in lower than ambient pressure whereas the pressure lower side is higher than the ambient pressure.

Increasing angle of attack also changes the velocity distribution as shown in above figures at angle of attack 2deg, 5deg, 10deg, 15deg and 18deg respectively. The velocity on upper surfaces increases whereas the velocity on lower surface decreases depending on angle of attack. Thus increase in pressure difference increases the lift.

**Changing angle of attack:** For each angle of attack, the flow around the changes and pressure distribution varies on the aircraft. The relative velocity (air velocity) on upper and lower side of aircraft at 0° angle of attack is relatively small. Moreover, relative pressure difference results in small lifting force. Flow separation begins to occur at small angles of attack while flow attached flow over the wing still dominant.

One of the most important target is to control the flow separation with respect to drag reduction. The separation of boundary layer is associated with the large energy losses and in most applications adversely affects aerodynamic loads in the form of lift and drag increase. Reduction of drag on wing of an airplane can reduce the fuel consumption and saves more energy. At this condition increase in angle of attack, drag also increases. Figures shows positive pressures more compared to negative pressures. As the angle of attack increases the coefficient of lift also increase with respect that pressure on airplane effects the aerodynamic characteristics. At this state flow separation takes place at trailing edge and pressure leading edge is more than trailing edge resultant pressure is sum of upper surface positive pressure and negative pressure at lower surface of airplane. The center of pressure distribution changes as angle of attack increases.

The lower surface sometimes carries a positive pressure, but at many design conditions is actually pulling the wing downward. The pressure at the trailing edge is related to the airfoil thickness and shape near the trailing edge. Large positive values of at the trailing edge imply more severe adverse pressure gradients. **Pressure recovery region:** Pressure increases from its minimum value to the value at the trailing edge. This adverse pressure gradient is associated with boundary layer transition and **possibly separation**, if the gradient is too severe... which occurs at higher angles of attack!!!

**International Journal of Innovative Research in Science,  
Engineering and Technology**

(An ISO 3297: 2007 Certified Organization)

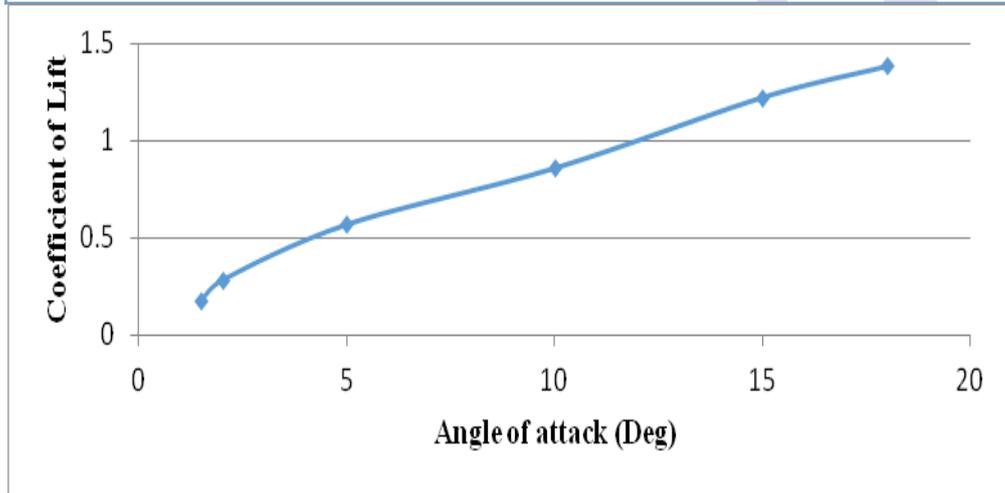
Vol. 3, Issue 10, October 2014

**V.GRAPHS AND TABLE**

After performing simulation method, the values of Coefficient of Lift and Drag with a different angle of attack and velocity were observed.

**Table 5: Angle of attack vs Coefficient of lift**

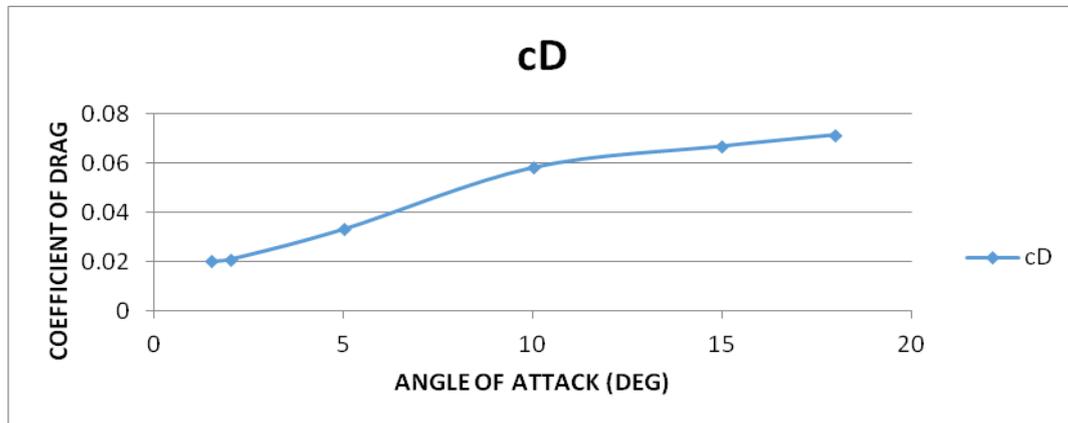
Angle of attack	Coefficient of lift
1.49	0.1801
2	0.2823
5	0.5688
10	0.8584
15	1.223
18	1.384



**Figure 5.1: Angle of attack vs coefficient of lift**

Angle of attack	Coefficient of drag
1.49	0.0201
2	0.0209
5	0.0331
10	0.0581
15	0.0668
18	0.0714

**Table 5.2: Coefficient of drag values at different angle of attack**



**Figure 5.3: Angle of attack vs Coefficient of drag**

Above performances, shows that though with increase in angle of attack, lift increase but this tend not go long. After few degrees of increase in angle of attack lift start reducing drastically, this angle of attack called critical angle of attack. It is also seen that increase in angle of attack flow ascends towards the middle of the aircraft wings. This induces more turbulence to the flow and increase the sound. In figure 5.3 shows that the coefficient of drag initially decreases as the angle of attack increases after that increases slightly and after few degrees drag increase when speed increases because drag is function of airspeed.

## V. CONCLUSION

- This work presents the simulated flow over an aircraft and it was observed that the lift increases as angle of attack increases and if the angle of attack is increased, center of pressure moves forward and if it is decreased, it moves rearward or towards trailing edges and center of gravity is fixed at one point.
- For an angle of attack 1.49deg with a speed 250m/s it was observed that the maximum velocity is 308.3m/s and total pressure is 22408Pa, and related coefficient of lift is 0.1801
- At 10deg angle of attack with speed 250m/s observed maximum velocity is 286m/s, total pressure is about 23056Pa, coefficient of lift and coefficient of drag are 0.8584 and 0.0501 respectively.
- The lift and drag depend on the airfoil shape and it is depending upon the velocity distribution, but also on the wing planform and on the wing area. It is possible to calculate the aerodynamic properties of differently sized airfoils or wings if all forces and moments are normalized.

## REFERENCES

- [1] Ryan Babigian and Shigeo Hayashibara (2009): "Computational study of the Vortex Wake Generated by a Three-Dimensional Wing with Dihedral, Taper and Sweep", 27<sup>th</sup> AIAA Applied Aerodynamics, Conference 22-25 June 2009.
- [2] K.P. Singh, J. S. Mathur, V. Ashok, and Debasis Chakraborty, "Computational Fluid Dynamics in Aerospace Industry in India", Volume 60, Number: 6, Defense Science Journal, 2010.
- [3] Mueller, T. J., (editor), "Aerodynamic characteristics of low aspect ratio wings at low Reynolds numbers" presented at the conference on fixed, flapping and rotary wing vehicles at very low Reynolds numbers, Notre Dame, In, June 5-7, 2000. Proceeding to be published as AIAA book in 2000.
- [4] Nathan Logsdon, Dr. Gary Solbrekken, "Procedure For Numerically Analyzing Airfoils And Wing Sections" Volume 2, Issue2 doctoral diss., University of Missouri – Columbia, December 2006.
- [5] F. X. Wortmann, "The Quest for High Lift at Low Reynolds Number", AIAA Paper 74-1018, MIT- Cambridge, Sept. 1974.
- [6] F. Le Chuiton, A. D Alascio, G. Barakos et al, "Computation of the Helicopter Fuselage Wake with the SST, SAS, DES and XLES Models", Volume97, pp 117-124, Springer eBook, January 24, 2008.
- [7] Takashi Misaka, Frank Holzapfel and Thomas Gerz, "Large Eddy Simulation of Wake Vortex Evolution from Roll-Up to Vortex Decay" on 49<sup>th</sup> AIAA Aerospace Science meeting including the New Horizons forum and Aerospace Exposition(AIAA 2011 1003), 4-7 January 2011, Orlando, Florida.
- [8] Christopher L. Rumsey, Susan X. Ying et al, "A CFD Prediction of High Lift : review of present CFD capability", Elsevier, Progress in Aerospace Sciences, volume 38, issue 2, February 2002.