ISSN 2250-3153

Examination of Aerodynamic Flying Wing Configuration

Bharat Shahapur [*], K Sriveni [**], D Meenakshi [***]

[*] Department of Mechanical Engineering, Avanthi Institute of engineering and technology, Hyderabad.
[***] M.tech Student, AKRG College of Engg and Tech, Nallajerla.

Abstract - This paper presents the examination of Aerodynamics flying wing configuration using Computation fluid dynamics (CFD) techniques. The parameters of aerodynamic such as coefficient of drag, coefficient of lift, moment coefficient and pressure coefficient are calculated by CFD. The aerodynamic parameters are determined for different angles of attack and Mach number so that deviation of salient aerodynamic parameters can be observed and analyzed. In this work, Solution is obtained for all aerodynamic parameters by considering the effect of external flow over flying wing configuration. In this research work parameter are stall speed, stall angle of attack, drag divergence Mach number and critical Mach number. These parameters are obtained for every test case and the comparison is made between tested results and available experimental results.

Index Terms- CFD, Stall speed, Stall angle of attack, Critical Mach no and Drag divergence Mach no

I.INTRODUCTION

An aircraft flying wing has a tailless fixed wing and there is no fixed fuselage, payload, crews and all these equipment is housed inside the prime wing structure. The aerodynamic flying wing concept has come into existence because of its superior payload and range capabilities and it produces less drag than a conventional aircraft. As we know the fuselage and tail can contribute more to drag but in this configuration there is no fixed fuselage and tail section, so large amount of drag is minimized therefore performance is improved and also less amount of fuel is used.

The flying wing configuration is used in unmanned aerial vehicles which is self-piloted or remotely piloted aircrafts. They have the capability to carry cameras, payloads, sensors and other communications equipment. The mini or micro UAVs is not just small version of larger aircrafts but they are highly functional, militarily capable and where a traditional aerodynamic theory does not obey. The UAVs have significant importance in particular fields like in military services and rural search-rescue purposes.

The Computational fluid dynamics technique is used to carry out the analysis of external flow over the flying wing configuration model and the variations of aerodynamic parameters are studied. In this analysis used working fluid is air at sea level conditions and simulation part has been carried out for different Mach numbers and angles of attack. In this research work two solvers are used such as density based and pressure based solvers with suitable turbulence models, which solves Reynolds Averaged Neivier Stokes (RANS) equations. In this work the selected turbulence model is Spalart Alamaras turbulence model. The relevant graphs of aerodynamic parameters are plotted particularly variations of coefficient of drag and coefficient of lift with the change in Mach number shown in preceding section.

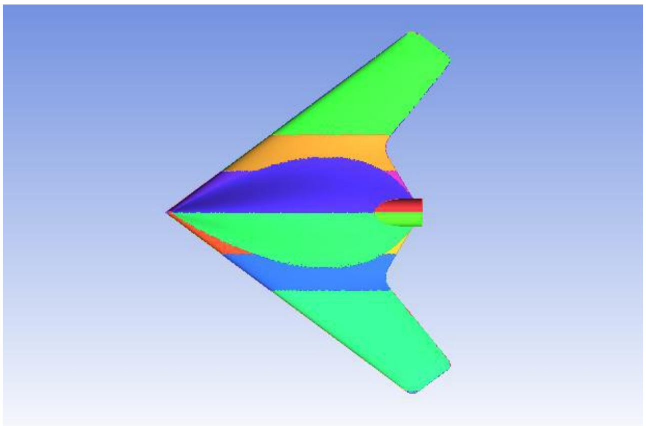

Fig. 1: Flying wing geometry

Fig. 1 shows the geometry of the flying wing configuration which is considered in the present work. The analysis is made half and symmetry part which is attached inside the fluid domain.

II. MODELLING AND GRID GENERATION:

The geometric model details are given in Table 1, and are created using familiar commercial software CATIA V5. It also includes building of computational fluid domain around the flying wing configuration by considering appropriate sizes at upstream and downstream of the wing configuration. During grid generation process the domain is divided into cells and sub domains. Grid generation process has been carried out by meshing tools like ICEM CFD 14, Point wise and Workbench, Basically there are three different types of grids are possible, structured grids, unstructured grids and hybrid grids. In the present work combined grids are used for CFD analysis.

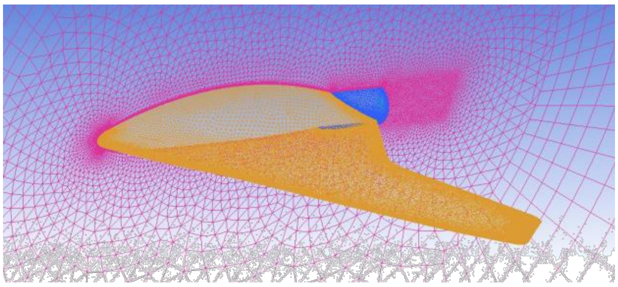

Fig 2: Grid

Moment reference point [X,Y,Z]	[0,0,0] nose of the model
Reference Area	0.087m2
Moment reference arm length	0.499m
Mean Aerodynamic Chord(MAC)	0.278m
Root Chord Length	0.499m

Table 1: Geometrical Parameters

Grid independence study is carried out for two grids, one with 2.5 million cells and another with 7.5 million cells at M=0.13 and variation of angles of attack from -40 to 240. This study represents the results for both the grids are almost similar and it can be seen from the figures 3 and 4, which is having lesser elements that grid is considered for further simulation.

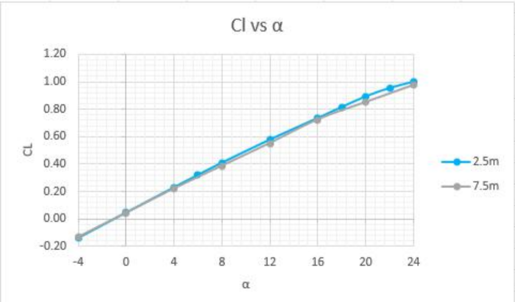

Fig: 3

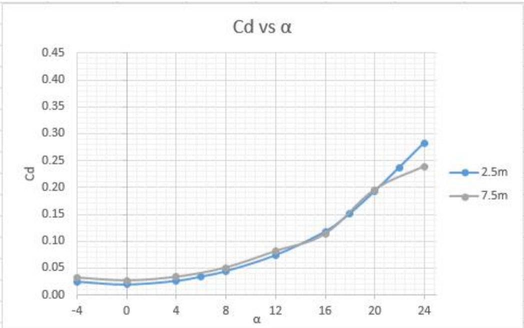

Fig: 4

III. CFD ANALYSIS:

The aerodynamics parameters are coefficient of lift, pressure coefficient, lift to drag ratio and drag coefficient. These are important parameters for any aircraft. In the analysis these parameters are analyzed by varying and Mach number and angles of attack. Aerodynamic drag and lift are based on normal stress and shear stress. The shear stress is restrained by surface wall roughness.

Lift coefficient:

Lift force is an artificial force and it is controlled by Pilot. Lift force is influenced through wings and acts perpendicular to relative wind and wing span. The concept of Center of pressure is the base to the direction and lift force. The lift force on the wings depends on velocity of fluid, planform area density of fluid and also on the lift coefficient value, the equation is given by, [2]

$$L = 0.5 * \rho * v_{\infty}^{2} * s * C_{l} \qquad (1)$$

$$C_{l} = \frac{L}{0.5 * \rho * v_{\infty}^{2} * s} \qquad (2)$$

Where, L indicates the lift force, ρ indicates density of air, V∞ indicates relative velocity of the air flow, s indicates planform area and Cl indicates coefficient of lift.

Drag coefficient:

Aerodynamics drag is the natural resistance to the moving aeroplane through air and it is partly controlled by pilot. It acts parallel to flight path and opposite to thrust. Drag coefficient is obtained by arranging drag equation [2].

$$D = 0.5 * \rho * v_{\infty}^{2} * C_{d} * s \qquad (3)$$

$$C_{d} = \frac{D}{0.5 * \rho * v_{\infty}^{2} * s} \qquad (4)$$

Where D indicates drag force, ρ indicates density of air, v_{∞} indicates velocity of air, s indicates planform area and Cd indicates drag coefficient.

Pressure coefficient:

It can be defined by,

$$C_{p} = \frac{p - p_{\infty}}{0.5 * \rho * v_{\infty}^{2}} \qquad (5)$$

Where P indicates pressure point at which pressure coefficient is evaluated, p_{∞} indicates pressure in the freestream (i.e., remote from any disturbance), ρ indicates density of air at sea level, V∞ indicates velocity of freestream of the fluid. Cp of zero value indicates freestream pressure and Cp of one indicates the stagnation pressure at stagnation point [2].

In ANSYS Fluent 14.0 there are 2 solvers used, one is pressure based solvers for low speed incompressible flows, M<0.3 and other one is density based solvers for high speed compressible flows, M>0.3. In this present work, above solvers are used as per the given conditions and the viscosity of fluid is considered to CFD simulation in the flow domain. The external flow analysis on working fluid is at sea level condition. The following table shows the properties of air at sea level condition.

Density (ρ)	1.225 kg/m^3
Pressure (P)	101325 Pa
Temperature (T)	288.15 k
Gas constant of air (R)	287.057
Dynamic viscosity	$1.789*10^{-5}$Pa.sec

Table 2: Properties of air at sea level [3].

Boundary conditions:

In this problem three boundary locations were identified which are inlet, symmetry and wing wall. The applied boundary conditions for inlet, symmetry and wing wall are pressure far-field boundary conditions, surface symmetry conditions and wing configuration wall conditions respectively. The proper data has to be supplied at each boundary location to get higher impact on the final solution

The mesh file is imported to ANSYS Fluent 14, depending upon Mach number and selected solvers are density based or pressure based solvers. Ideal gas is air material which is selected as pressure far-field conditions are given at inlet. If density based solver is selected then it is helpful to choose implicit formulation method for faster convergence of the final solution. In the initial running stage the oscillations in solutions are more and after some iteration the solution will be become stable then that time courant number is increased for faster convergence. Many numbers of turbulence models are available in the solver for RANS calculations, but in this work the applied model is Spalart-Alamaras. This model solves one equation at each cell of the fluid domain and these equations are Reynolds Averaged Navier Stokes (RANS) Equations. Further the suitable solution methods are applied to solve RANS equations.

IV. RESULTS AND DISCUSSION:

CFD analysis has shown that all the parameters of aerodynamics are predicted adequately. The following graphs are obtained after the simulation. All the graphs are plotted for M=0.13 and different angles of attack.

The very first fig. 5 shows graph of CL vs α, here lift coefficient is determined for different angles of attack from -40 to 240. As the angle of attack increases pressure at the lower side of the wing increases and on upper side it decreases because of the fact that air speed on the upper surface of the wing increases as the air has to travel for longer distance, according to Bernoulli's theorem as the air speed increases pressure at that location decreases gradually. That's why because of this pressure difference lift increases. The fig.6 shows the graph of Cd vs α, in this graph as the angle of attack increases the area resists to the flow of air increases therefore drag coefficient increases. The next fig.7 shows the graph of Cm vs α, here it's observed that as the angle of attack increases the pitching moment coefficient also increases due to instability during flight takeoff and landing.

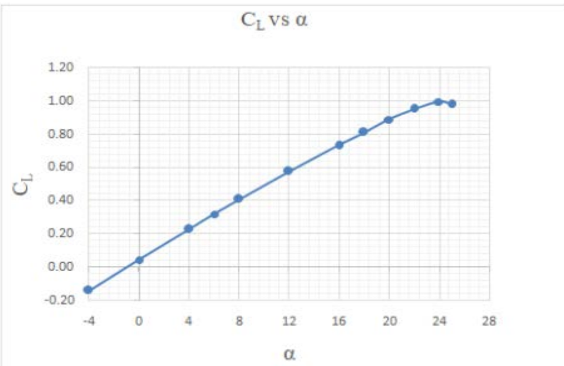

Fig.5

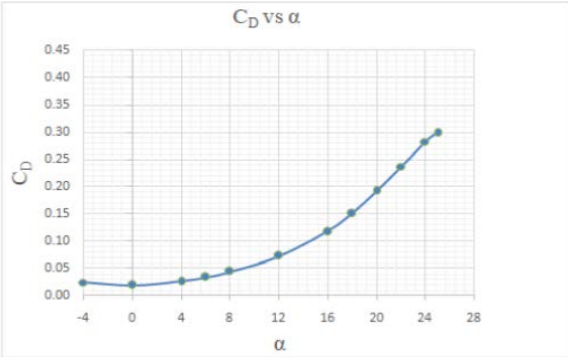

Fig.6

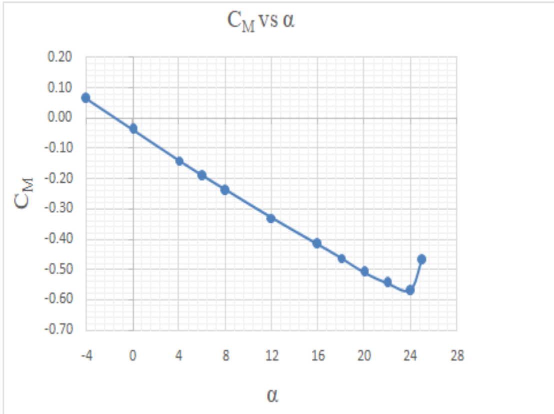

Fig.7

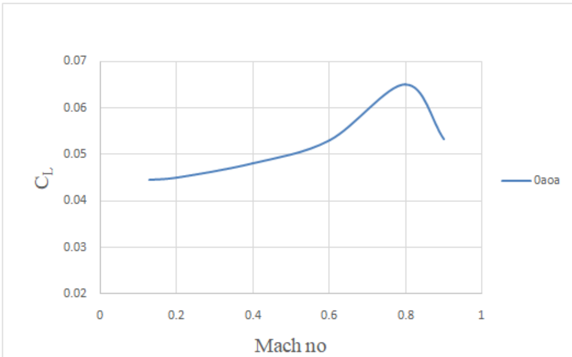

Fig.8

The fig.8 shows Cd vs Mach number, coefficient of drag increases slightly and it is almost constant up to M=0.8, after that it suddenly rising up to M=1. This phenomenon is known as drag divergence and the corresponding Mach number at which drag divergence arise is called drag divergence Mach number.

Further the simulation carried out for different Mach numbers and different angles of attack. In the fig.8 shows that as the Mach number increases the lift coefficient also increases but only up to M=0.8 after that it decreases in the transonic region, because of presence of shock wave interaction with the boundary layer. So the pressure on the upper side of the wing is more and hence less pressure difference is there for lift. That's why Cl decreases in the transonic region.

ISSN 2250-3153

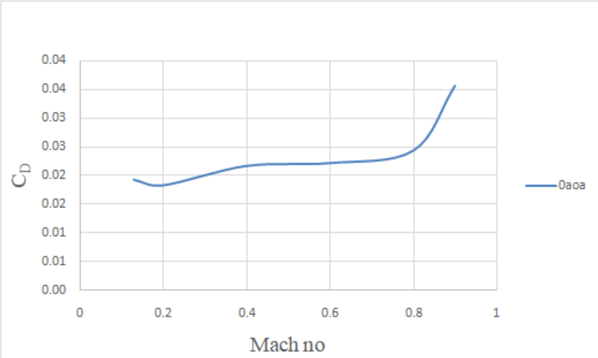

Fig.9

At higher Mach number i.e. in transonic region from 0.8 to 1.2 the shock waves have been observed on the upper side of the wing. In this region flow variables change without prior indication because of flow separation, here sudden decrease in the pressure can be noticed. In the transonic region pressure drag increases and that increases drag coefficient which is shown in the fig 9.

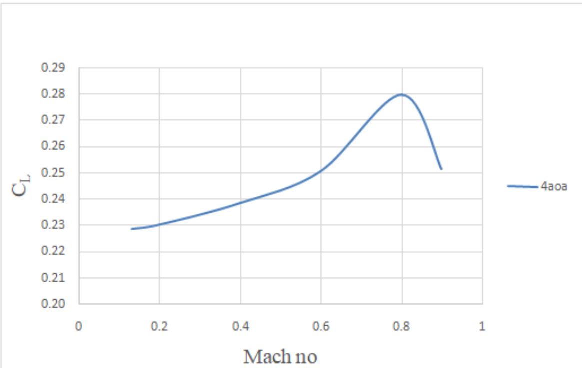

Fig.10

The fig.10 shows that Coefficient of pressure plot At Mach=0.13 and 60 angle of attack. As the angle of attack increase the pressure above the wing surface reduces when compared to the pressure below the wing surface. Pressure difference between the upper side and lower side of wing increases as the angle of attack increases that's why wing displaces in upward direction at larger angles of attack leading to the increase in the coefficient of lift. The wing is placed in XZ plane and section plane is taken in X direction at x =150mm from nose of the aeroplane. At this section the Pressure coefficient values have been noted.

V. CONCLUSION:

Based on the CFD analysis of flow over flying wing configuration the following conclusions can be drawn,

1. Stall angle of attack is observed at 250.
2. CL increases as angle of attack increases up to stall.
3. Cd increases as angle of attack increases up to M=1.
4. Drag divergence Mach number is observed at M=0.8
5. Due to presence of shock wave in the transonic region CL reduces.

. REFERENCE:

(1) Ho, S Nassef, H., Pomsinsi Thesis N., Tai, Y-C. and Ho, C.-M (2003) Unsteady Aerodynamics and Flow Control for Flapping Wing Flyers. Progress in Aerospace Sciences , 39, 635-681. https://doi.org/10.1016/j.paerosci.2003.04.001

(2) www.langleyflyingschool.com

(3) www.researchgate.net/publictopics

(4) A computational study of the low-speed flow over the 1303 UCAV Configuration (M.T.Arthur and K.Petterson).

(5) Aerodynamic Studies over a Maneuvering UCAV 1303 Configuration. (M.S. Chandrasekhara1 & LT. Brian K. McLain2)

(6) Unstructured CFD Aerodynamic Analysis of a Generic UCAV Configuration. (Neal T. Frink, Magnus Tormalm, Stefan Schmidt).