Aerodynamic Studies of Flying Wing Configuration

Mahantesh Katagi
M.Tech in (Thermal Power Engineering)
Siddaganga Institute of technology
Tumkur, India

Mr. Manish Kumar Singh
Sr. Scientist, CTFD Division,
National Aerospace Laboratories
Bangalore, India

Mr. M. Shivashankar
Associate Professor
Dept. of Mechanical Engg.
SIT, Tumkur.

***Abstract* - This paper summarizes the study of Aerodynamics Characteristics of flying wing configuration using CFD techniques. The various aerodynamic parameters like coefficient of lift, coefficient of drag, pressure coefficient and moment coefficient are calculated using computational techniques. These parameters are calculated for different Mach number and different angles of attack so that variation of important aerodynamic parameters can be observed and analyzed. In the present work, external flow analysis over flying wing configuration is carried out to obtain the solution for all required parameters. In this research, parameters such as stall speed, critical Mach number, stall angle of attack, and drag divergence Mach number are obtained for each test case and the results are compared with the available experimental results.**

Keywords—Stall speed, Critical Mach no, Stall angle of attack, and Drag divergence Mach no

1.INTRODUCTION:

A flying wing is a tailless fixed wing aircraft there is no definite fuselage, most of the crews, payload and equipment is housed inside the main wing structure. The concept of flying wing has come into existence because of its excellent payload and range capabilities and it produces less drag compared to conventional aircraft. As we know that tail and fuselage can contribute more to drag but in this configuration there is no definite fuselage and tail section, so large amount of drag is reduced therefore performance is improved and also less amount of fuel is required.

The unmanned aerial vehicles (UAVs) are also have the flying wing configuration and they are remotely piloted or self-piloted aircrafts. They have the capacity to carry cameras, sensors, and other, communications equipment or other payloads. The mini UAVs are not just small version of larger aircrafts but they are fully functional, militarily capableand where conventional aerodynamic theories doesn't holds good. The UAVs have significant importance in some fields like in military services and rural search-rescue purposes [1].

The external flow analysis is carried outthrough CFD techniques over the flying wing configuration model and studied the variations of aerodynamic parameters. The working fluid used in our analysis is air at sea level conditions and simulation has been carried out for various Mach numbers and angles of attack. In this work two solvers have been used which are pressure based and density based solvers with suitable turbulence models, which solves RANS (Reynolds Averaged Neivier Stokes) equations. In this work Spalart Alamaras turbulence model is selected. The important graphs of aerodynamic parameters, particularly variations of coefficient of lift and coefficient of drag with the change in Mach number shown in preceding section.

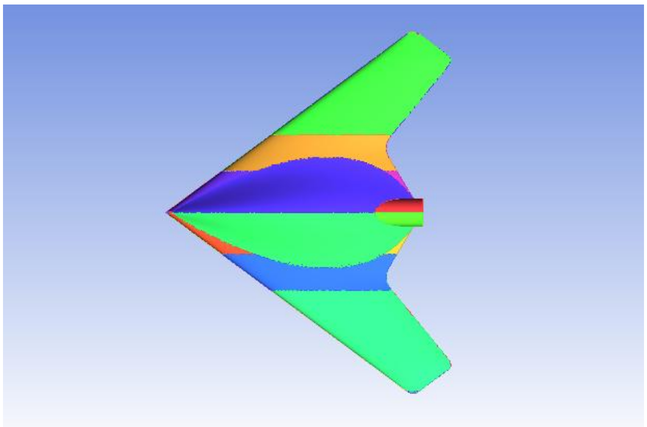

Fig. 1: Flying wing geometry

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

2. MODELLING AND GRID GENERATION:

The model close to the given geometric details (given in Table 1) is created using well known commercial software CATIA V5. It also includes building of computational fluid domain around the flying wing configuration by giving appropriate sizes at upstream and downstream of the wing configuration. This domain is sub divided into subdomains and cells or elements during grid generation process. Grid generation process has been carried using meshing tools like ICEM CFD 14, Pointwise and Workbench,Basically there are threetype's of grids are possible, one is structured grids, another one is unstructured grids and hybrid grids, but in the present work organizational grids are used for CFD analysis.

ISSN: 2278-0181
Vol. 4 Issue 05, May-2015

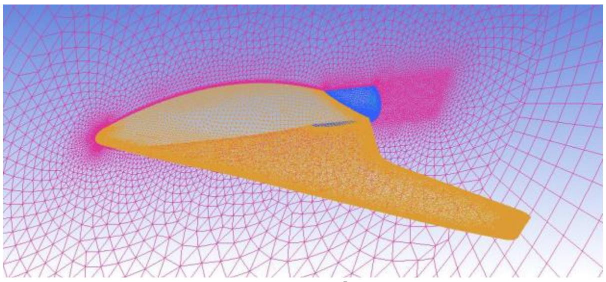

Fig 2: Grid

Reference Area	$0.087m^2$
Moment reference point (X,Y,Z)	(0,0,0)nose of the model
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 conducted for two grids, one with 2.5million cells and another with 7.5million cells at M=0.13 and different angles of attack from -4^0 to 24^0. The results of this study is concluded that the results for both the grids is almost similar which can be seen from thefigures 3 and 4, then one grid is considered which is having lesser elements for further simulation.

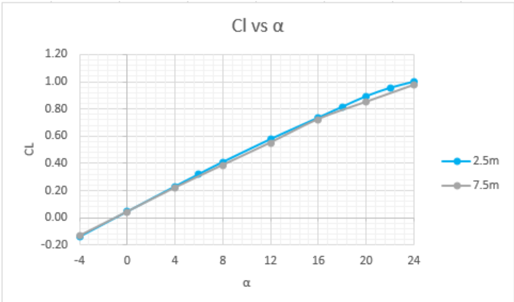

Fig: 3

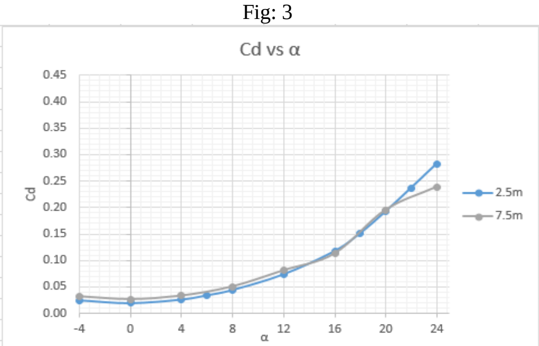

Fig: 4

3. CFD ANALYSIS:

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

Lift coefficient:

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

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

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

Where L is the lift force. ρis the density of air, V_∞ is the relative velocity of the air flow, S is the planform area, C_l is the lift coefficient.

Drag coefficient:

Aero dynamics drag is the natural resistance of air to the movingaero plane through air and it is partially controlled by pilot. It is acting parallel to the flight path and opposite to the thrust. Drag coefficient is obtained by rearranging drag equation, [2]

$$D = 0.5 * \rho * v_{\infty}^2 * s * C_d \quad (3)$$

$$C_d = \frac{D}{0.5 * \rho * v_{\infty}^2 * s} \quad (4)$$

Where D is the drag force, ρ is the density of air.$v_\infty$is velocity of air, s is the planform area and C_d is the drag coefficient.

Pressure coefficient:

It can be defined by,

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

Where P is the pressure at the point at which pressure coefficient is evaluated, P_∞ is the pressure in the free stream(i.e., remote from any disturbance), ρ is the density of air at sea level condition, V_∞ is the free stream velocity of the fluid or the velocity of body through the fluid.Cp of zero value indicates the pressure is the same as the free stream pressure, Cp of one indicates the pressure is the stagnation pressure and the point is a stagnation point. [2]

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

Temperature (T)	288.15k
Density (ρ)	1.225kg/m^3
Pressure (P)	101325Pa
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 there are three boundary locations were identified, which are inlet, wing wall, symmetry. At the inlet pressure far-field boundary conditionsare applied, for wing configuration wall conditions are applied and for symmetry surface symmetry conditions are applied. At each boundary location proper data has to be supplied because the information supplied at boundary will have greater impact on the final solution.

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

4. RESULTS AND DISCUSSION:

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

The very first fig. 5 shows graph of C_L vs α, here coefficient of lift is calculated for different angles of attack from -4^0 to 24^0. As the angle of attack increases pressure at the lower surface of the wing increases and on upper side it decreases because of the fact that air speed on the upper side of the wing increases as the air has to travel for longer distance on the surface so 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 C_d vs α, in this graph as the angle of attack increases the area opposed to the flow of air increases therefore coefficient of drag increases. The next fig 7 shows the graph of C_m vs α, here it is observed that as the angle of attack increases pithing moment coefficient increases due to instability during flight takeoff and landing.

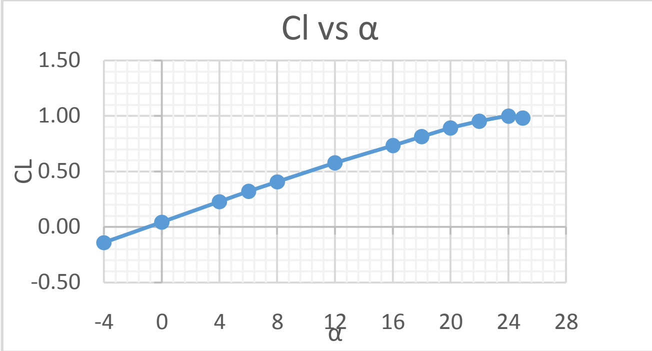

Fig: 5

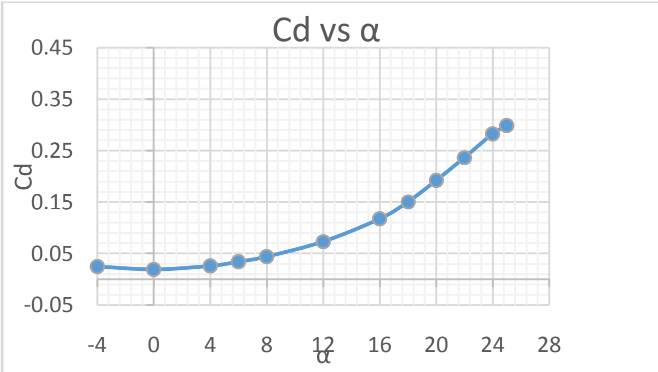

Fig: 6

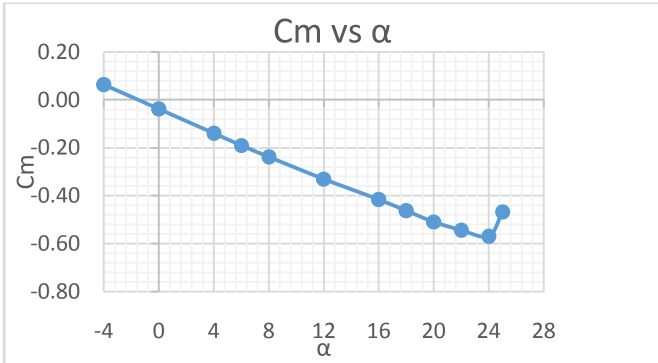

Fig: 7

Further the simulation is done for different Mach numbers and different angles of attack. In the fig 8 it clearly shows that as the Mach number increases coefficient of lift increases 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 available for lift. That's why C_l decreases in the transonic region.

The fig 9 shows C_d vs Mach number, coefficient of drag increases slightly and it is almost constant up to M=0.8 , after that it suddenly increases up to M=1. This phenomenon is called drag divergence and the corresponding Mach number at which drag divergence takes place is called drag divergence Mach number.

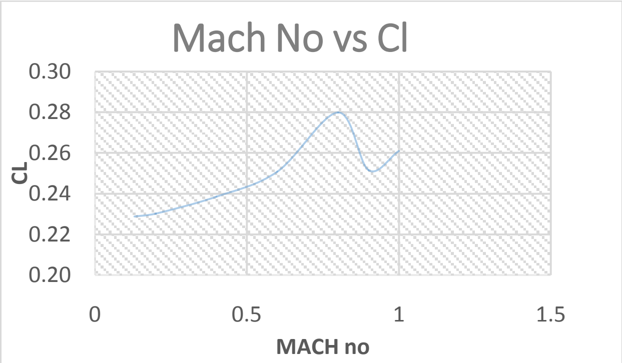

Fig: 8

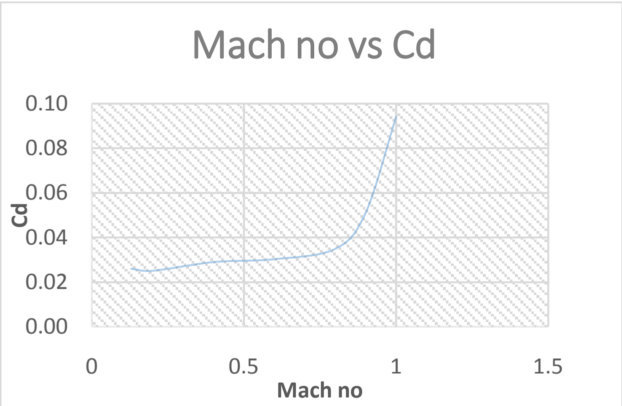

Fig: 9

At Mach=0.13 and 6^0 angle of attack Coefficient of pressure plot is shown in figure 10. The pressure above the wing surface reduces as the angle of attack increases when compared to the pressure below the wing. Pressure difference between the upper and lower surface increases as the angle attack increases that's why wing displaces in upward direction at higher angles of attack leading to the increase in lift coefficient. The wing is placed in XZ plane and section plane is taken in X direction at x=150mm from nose of the model. At this section coefficient of pressure values have been taken.

At higher Mach number i.e. in transonic region from 0.8 to 1.2 the shock waves have been observed on the upper surface of the wing. In this region flow variables change abruptly because of flow separation, here sudden decrease in the pressure can be observed. In the transonic region pressure drag increases that's why coefficient of drag increases which is shown in the fig 9.

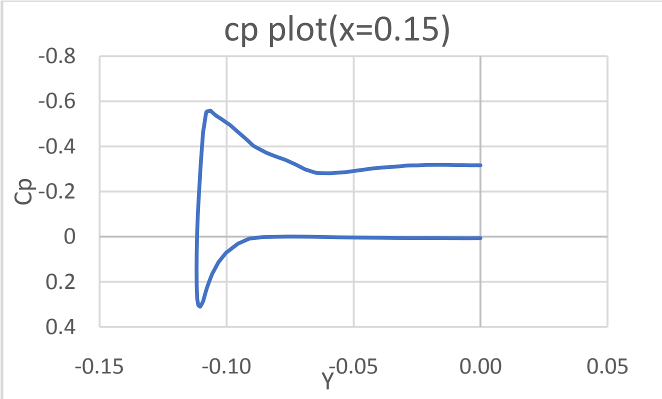

Fig 10:Plot of Coefficient of Pressure

5. 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 25^0.
2. C_L increases as angle of attack increases up to stall.
3. C_d 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 C_L reduces.

6. ACKNOWLEDGEMENT:

I would like to express my sincere gratitude to Mr. Manish Kumar Singh for his guidance and assistance in this study work. I also sincerely thanks to Mr. M. Shivashankar for his support and guidance to carry out this study work successfully.

7. REFERENCE:

(1) Thesis by Navabalachandran Department of Mechanical Engineering, National University of Singapore.
(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).