International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 - 5161

Available at http://inpressco.com/category/ijcet

Research Article

Flow Analysis of Wing under Critical Mach Numbers using CFD

Srinivas G[Ȧ*] and P.Abhiram[Ȧ]

[Ȧ] Aeronautical Engineering Department, MIT-Manipal University, Manipal Karnataka-India.

Accepted 10 January 2014, Available online 01 February 2014, **Special Issue-2, (February 2014)**

Abstract

Wings are the main lift generating sources for any aerospace vehicle. The performance of an airborne vehicle largely depends on its wing design. Wing design is a complex process involving selection of many parameters called the wing design parameters. These parameters differ for different wing configurations. As per the mission requirement, these parameters are selected optimally. The main focus in this study is to understand wing design methodologies. It is necessary to understand how much lift has to be generated, what the required wing loading is and what will be the flight conditions where this wing will be used. A standard 3D wing validation has been completed before the designed wing aero-analysis is performed. With this validation the wing design methodology has been verified and standardized to be applicable for similar type of wing design in future. at initial stage the standard wing validation case has been studied. wing geometry has been modeled and full viscous flow has been calculated for the range of Mach no 0.7 and 0.84 at angles of attack 0 to 6.5 at the same Reynolds number (at which Wind Tunnel test has been carried out), to validate the CFD methodologies.

Keywords: *Lift, aerospace vehicle, design, wing loading.*

1. Introduction

The wing of an aircraft is the most important part of the aircraft as it lifts the whole weight of the aircraft. All the maneuvering of the aircraft is done by using the control surfaces (places to control the air flow and thereby producing the desired changes in the aircraft course) in the wings. Also the importance of wings is critical to an airplane; they produce lift that can sustain the airplane in the air. In flight, a wing has lower pressure on top and higher pressure on bottom due to Bernoulli's principle which in turn sucks the airplane into the air. The air on top must travel a longer distance than the air on bottom in order to meet up again because of the shape of the wing therefore causing the effects of Bernoulli's principle. The wings of an airplane are made from very strong aluminum, and are designed specifically to bend and move up and down (Anderson). Reason being, is because when the plane is under high results of turbulence the wings are supposed to absorb the wind instead of the cabin moving all over the place. Without wings the airplane would just be a really fancy car that cannot fly because it won't be sucked into the air.

2. Literature review

The design methodology for selecting the wing design parameters was based on the thesis submitted Milan Kumar Pal . In this technique Genetic algorithm was used by the author to design an optimized technique because of its simplicity and the capability to arrive at a global optimum solution. This technique was designed to mainly minimize the total drag in the transonic conditions. So for this purpose a Navier-Stokes solver (RANS) was used to evaluate the wing performance at all conditions.

Various design methods can be found in the literature. Among them, perhaps the most explored and widely used design is that of a glider. If we look at a glider next to a conventional powered plane, we'll notice a significant difference in the wings. While the wings of both are similar in general shape and function, those on gliders are longer and narrower than those on conventional aircraft. The wing design of a glider differs from that of a normal conventional aircraft. It has high aspect ratio compared to that of normal wings. These glider wings are capable of producing high aerodynamic efficiency with an advent of no propulsive unit in both subsonic and transonic regimes.

3. Computational Fluid Dynamics

CFD methods calculate the flow properties within each cell, using various convergence schemes to equate the flow properties along the boundaries connecting the cells. Gridding is especially important because the CFD results are highly sensitive to the shaping of the cells (Jack Moran).

The current so-called Navier-Stokes Codes actually use simplification in the handling of turbulence, which are the most difficult phenomena to analyze mathematically. Turbulence is handled with some type of separate

*Corresponding author: **Srinivas G**

statistically-calibrated model apart from the NS solution. The most sophisticated codes to date, the Large Eddy Simulation codes, use a statistically-based turbulence model for small-scale turbulence effects. Large eddy codes are capable of directly analyzing the larger turbulent eddies.

Comparisons of computed results with experimental data reveal that while semi-empirical codes offer predictions sufficient for preliminary design, the higher level CFD solutions gives better accuracy in aerodynamic coefficients predictions. The objectives of the CFD study are to demonstrate the capability of this project to successful prediction and validation of aerodynamic coefficients. The CFD simulations provide visualizations of the flow field that aid the understanding of the flow physics.

3.1 Boundary Conditions

Pressure far field conditions are used here to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries. This boundary condition is applicable only when the density is calculated using the ideal-gas law. Following information for a pressure far-field boundary is given; 1. Static pressure; 2. Mach number; 3. Temperature; 4 .Flow direction- flow direction is taking care of angle of attack, say for example at angle of attack 5 is given in X direction V*cos5 and Y direction V*sin5, where V is the velocity (Schmittt *et al* 1978).
Symmetry boundary conditions are used when the physical geometry of interest and the expected pattern of the flow solution have mirror symmetry. They can also be used to model zero-shear slip wall in viscous flows. Wall Boundary Conditions-On wing body wall boundary conditions are used. As the nature of flow is viscous the no-slip boundary condition is assigned to it. The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

3.2 Turbulence Model- The k-ω SST Model

In turbulence models that employ the Boussinesq approach, the central issue is how the eddy viscosity is computed. The model proposed by Wilcox solves two transport equations for two quantities k (turbulent kinetic energy), and ω (turbulent frequency). The *k*-ω based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients(Schmittt *et al* 1978). The *k*-ω models assume that the turbulence viscosity is linked to the turbulence kinetic energy and turbulent frequency via the relation equ 1-2. (ANSYS 12.0 Manual):

$$\mu_t = \rho \frac{k}{\omega} \tag{1}$$

The transport equations for k and ω are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta^t \rho k \omega \tag{2}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \tag{3}$$

k is the turbulent kinetic energy and is defined as the variance of the fluctuations in velocity, ρ is the density, U_j is the velocity vector, μ_t is the turbulent viscosity, P_k is the production rate of turbulence, ω is the turbulent frequency and σ_k, σ_w, β^t, β and α are constants with values given as $\sigma_k = 2$, $\sigma_w = 2$, $\beta^t = 0.09$, $\beta = 0.075$ and $\alpha = 5/9$.
Two-equation turbulence models are very widely used, as they offer a good compromise between numerical effort and computational accuracy. Two-equation models are much more sophisticated than the zero equation models. Both the velocity and length scale are solved using separate equations velocity.

4. Methodology

In 1972, the Aerodynamics Department designed a swept back wing very well instrumented to be used as an experimental support for basic studies of three-dimensional flows at high Reynolds number from low to transonic speeds (Sunada *et al* 2002). Wind tunnel data from this model called M6-wing have constituted a good base both for computer program assessment and for understanding various flow phenomena like shock wave-boundary layer interaction or flow separation (Schmitt, V *et al* 1978).

4.1 Geometry and Simulation Parameters

The investigation used CFD to determine the flow field and aerodynamic coefficients on a wing configuration. The analyses is performed at around M = 0.7 and 0.84, and at different angles of attack varying from α = 0 and 6.1 deg. After geometry creation flow domain is created. For validating the Cp distribution over the wing, the flow domain dimensions are taken exactly similar to the experimental test section dimensions

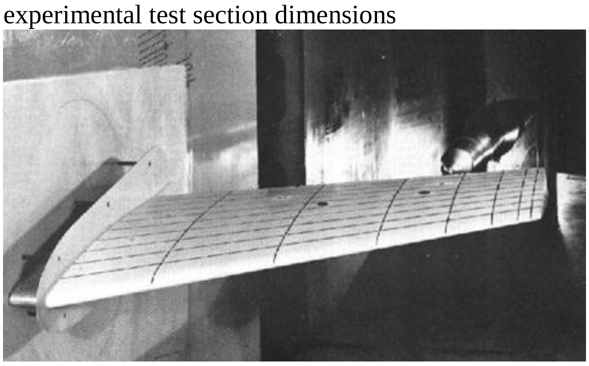

Fig.1 Wing geometry With Sections [Daniel P Raymer, Schmitt, V *et al* 1978]

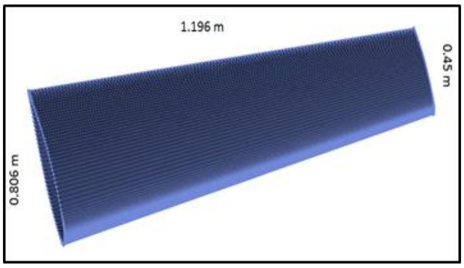

Fig.2 Wing Geometry created in CFD

4.2 Computational Mesh and Flow Simulation

The experimental test section used for Wing analysis was a square section of dimensions 5.4 m * 1.750 m *1.770 m (Milan Kumar Pal 2008). So the same dimensions are being considered here. Structured rectangular surface mesh has been generated for whole wing surface which yields almost 2.7 million cells. An implicit scheme was used in the ©Fluent to run this mesh. Following fig. shows the cut view of the meshing, which shows a very finer clustering has been done near the wing body to capture flow physics accurately.

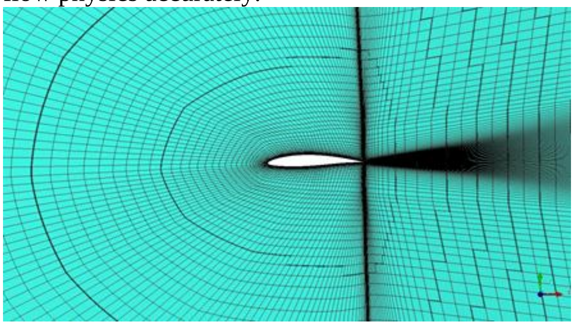

Fig 3. Structured Mesh around the wing

A nonslip wall boundary condition was used for the wing surfaces. A symmetry boundary condition was used to the surface which attaches to the Root-Chord, and a far-field pressure (non-reflecting) boundary condition was used for the outer boundary.

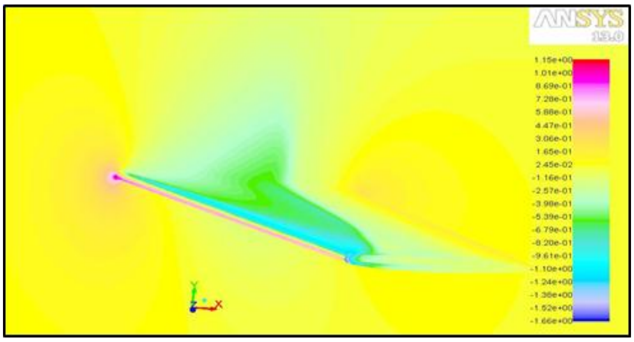

Fig. 4. Mach No.0.8 AOA= 3^0

Several choices for mesh generation around the configurations of interest were considered. Since it was mandatory to remesh only small areas of geometrical changes maintaining a grid independent flow solution in the remaining domain, single block unstructured methods were considered to be unsuitable.

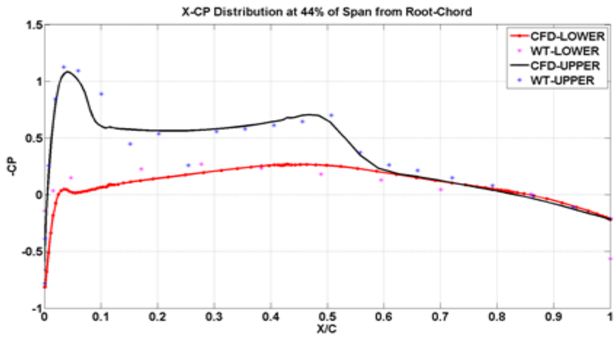

Fig. 5. Coefficient of Pressure Distribution of span from root chord

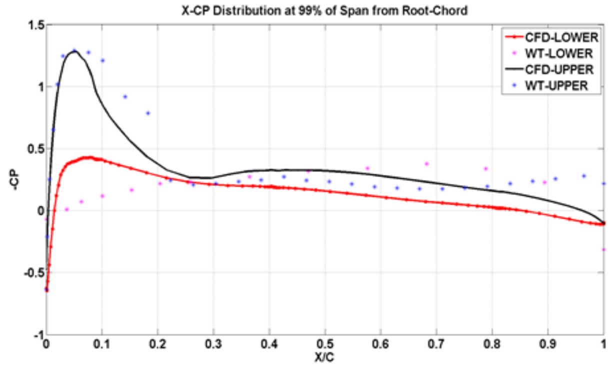

Fig. 6. Coefficient of Pressure Distribution of span from root chord

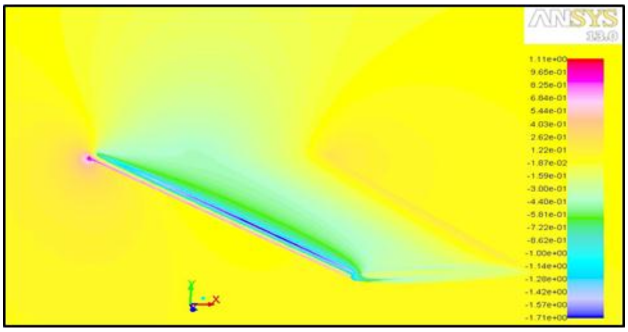

Fig. 7. Mach No.0.7 AOA= 3^0

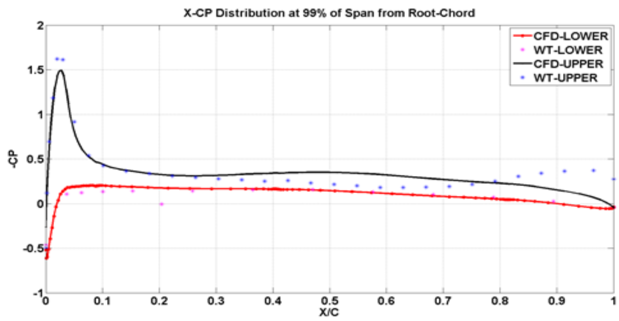

Fig.8 Coefficient of Pressure Distribution of span from root chord

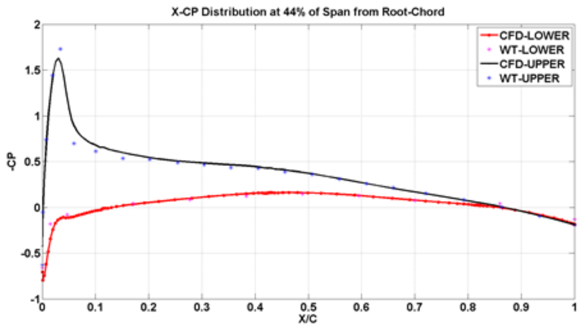

Fig 9. Coefficient of Pressure Distribution of span from root chord

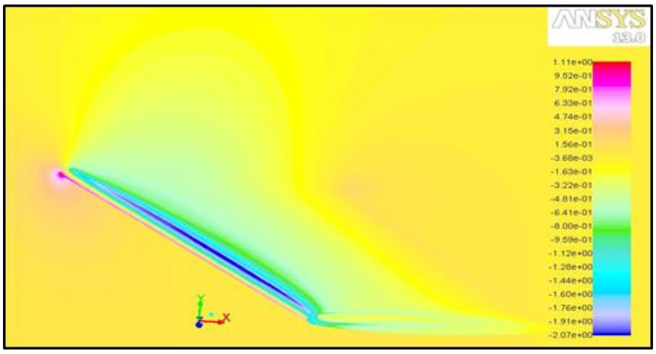

Fig. 10. Mach No.0.6 AOA= 3^0

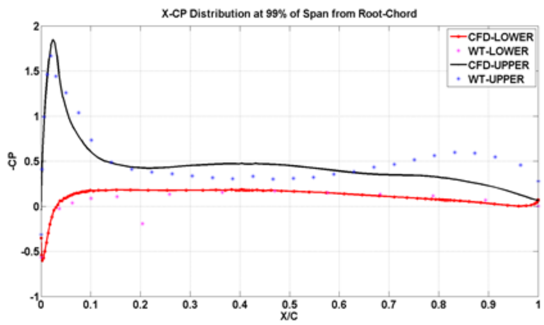

Fig. 11 Coefficient of Pressure Distribution of span from root chord

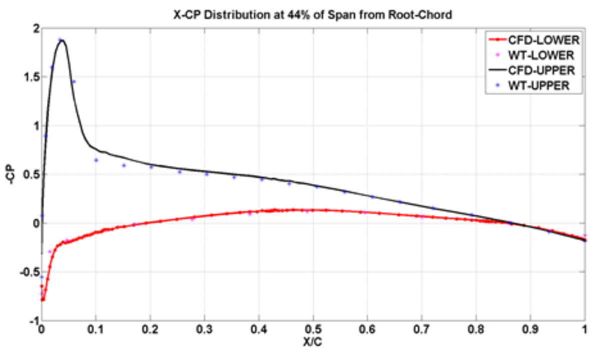

Fig. 12 Coefficient of Pressure Distribution of span from root chord

An structured method unfortunately was unavailable at the time of this study and a true multi-block structured grid generation was too time consuming. Therefore, a structured Chimera technique has been applied that combines a fast local remeshing with a preserved grid in the remaining domain. The flow analysis of wing and Coefficient of pressure distribution is shown in figure figures from Fig 4-12.

Conclusions

From the above Numerical Simulation through the CFD the following conclusion can be drawn.

1) The wing configuration designed here meets the initial mission requirement of a guided weapon by providing with the required normal force sufficient to compensate with the weight. The conditions are obtained to maximize the value of critical Mach number.
2) It has been observed that at angle of attack 3 deg the aerodynamic data is optimum when compared to Experimental work.
3) The wing performance is robust with a maximum aerodynamic efficiency of about 22 when it flies at a free-stream velocity of M=0.7 and angle of attack $\alpha=3°$.
4) Also from fig 4-9 at coefficient of pressure about the span is optimum for all the flight conditions. Hence by using the wing at of 3° aerodynamic forces required for shock free, streamlined flow can be achieved.
5) Analysis can be further carried to calculate the wing stalling speed which is generally observed at $\alpha > 18°$. Also the value of the critical mach number for different flow conditions can be calculated by suitably running the simulations.

References

Sunada, S., Yasuda, T., Yasuda, K., Kawachi, K.,Comparison of wing characteristics at an ultralow Reynolds number, *Journal of Aircraft*, Vol.39, pp.331-338, 2002.

Milan Kumar Pal Wing Design Optimization for a Subsonic Unmanned Aerial Vehicle, M.Tech Thesis, IIT Kharagpur, May 2008

Anderson, J.D. Jr. Computational Fluid Dynamics, *McGraw Hill International Editions*.

Jan Roksam Airplane Design Part-1: Preliminary Sizing of Airplanes

Anderson, J.D. Fundamentals of Aerodynamics, McGraw Hill Publications.

Schmitt, V. and Charpin. F. (1978) Pressure Distribution on the ONERA-M6-WING at Transonic Mach Numbers, *AGARD-AR-138*.

Daniel P Raymer Aircraft Design-A Conceptual Approach, *AIAA Publications*.

ANSYS ICEM CFD 11 Tutorial Manual January 2012

Fluent 13 User's Guide Fluent Incorporated 2010.