International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 13, Issue 4 (April 2017), PP.27-31

Fluid-Structure Interaction Over an Aircraft Wing

G.Vigneshwaran[1], M.Vijayaraghavan[2], K.Sivamanikandan[3],
K.Keerthana[4], K.Balaji[5],
[1,2,3,4]Student, [5]Assistant Professor,
Department of Aeronautical Engineering,
Karpagam Institute of Technology, Coimbatore, Tamil Nadu, India.

ABSTRACT:- Aircraft is a brilliant man-made structure which helps us to fly over the world. At the same time, aircraft is a complex structure to be checked and maintained for the aero elasticity due to aerodynamic properties. In this paper, the fluid-structure interaction problem in super critical NASA SC(2)-0412 airfoil is discussed. The main aim of this project is to find the best performance and deformation limit of the wing on different Mach numbers. This project is completely done by numerical methods of designing the wing using CATIA and flow properties in Computational Fluid Dynamics (CFD) method. Finally, the structural analysis for deformation is analysed in ANSYS. The analytical approach of fluid-structure interaction over an Aircraft wing is complex.

Keywords:- FSI, CATIA, ANSYS, Aero-elasticity, NASA.

I. INTRODUCTION

Fluid-structure interaction (FSI) is the interaction of some movable or some deformable structure with an internal or surrounding fluid-flow. Fluid-structure interaction can be stable or oscillatory. In fluid-structure interaction, it occurs when a fluid interacts with a solid structure, external pressure to it which may cause deformation in the structure. Fluid-structure interaction problem is often too complex to solve analytically, so they can be analysed by means of numerical simulation. In general, a fluid structure interaction system is classified into strongly and weakly coupled interaction. In the weakly coupled flow field, a structure in the flow field or containing flowing fluid deforms slightly or vibrates with small amplitude. In the strongly coupled fluid-structure system, the alteration of the flow field due to large deformation or high amplitude-vibration of the structure cannot be neglected. This paper is based on one way of FSI and strongly coupled flow field.

II. LITERATURE REVIEW

[1] T.Sai Kiran Goud explains the analysis of fluid-structure interaction on an aircraft wing. This strategy is known to be fraught with complications associated with the interaction between the two simulation modules. AGARD 445.6 wings will be created along with the fluid domain structure. The transonic flow in subsonic flow command (M= 0.9) over the wing will be simulated and the results will be certified by comparing the computational results with the previously published results.

[2] Jian Tang explained the computational Fluid-Structure Interaction of a Deformable Flapping Wing for Micro Air Vehicle Applications and in that he tells that the structural model is based on an asymptotic approximation to the equations of elasticity. Considering the slenderness as the small parameter, the equations are categorised into two independent variational problems, corresponding to the (i) cross-sectional, small-deformation and (ii) longitudinal and large deformation analyses.

[3] Chowla Sangeetha explains the Fluid-Structure Interaction on AGARD 445.6 wing at Transonic Speeds. Since extensive research has been done in the field of aero-elasticity using this model, this configuration was chosen. The main objective of this paper is to study the FSI over the wing of aircraft and determine the aero elastic properties through modelling as well as analysing the AGARD 445.6 wing structure using CATIA V5 to generate the solid model and the stress analysis is done using ANSYS-FLUENT.

[4] Yung-Gyo Lee explains the FSI and optimisation technique to optimise a wing shape on an unmanned aerial vehicle (UAV) for minimum cruise drag. The fluid solution is generated with Euler solver and structural analysis is performed with FEM solver. Sample points are selected by Kriging method and Design of Experiment (DOE) method is used for generation of an approximation model.

III.METHODOLOGY

A. Model description

The Tapered wing is widely used for much aero-elastic analysis. The experimental wing that has NASA SC(2)-0412 airfoil and an aspect ratio of 7.76, sweep angle 33.5°, taper ratio 0.26. The figure below shows the selected airfoil. This model is homogeneous and orthotropic in nature. Material properties of the wing are shown below. The material used here is an aluminium alloy.

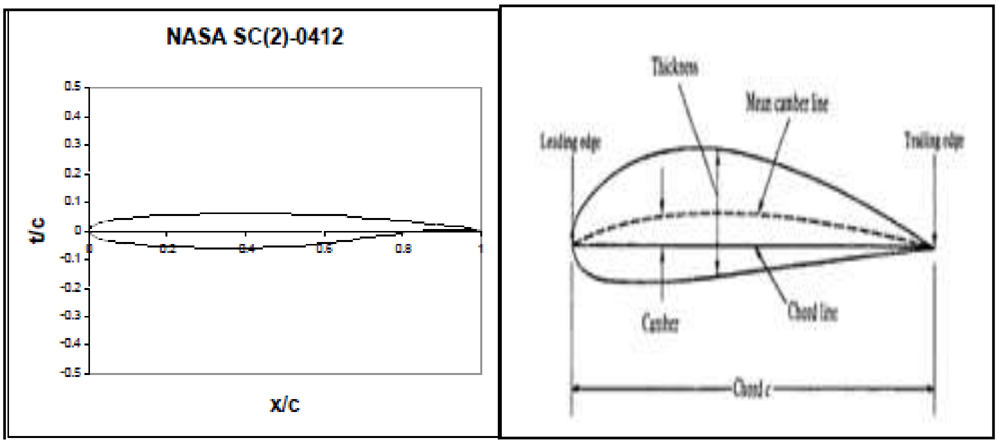

Fig.1:NASA SC(2)- 0412 Aerofoil and its description

Table I: Material Properties

S.No	Properties	Values
1.	Material	Aluminium alloy
2.	Density	2.77e-006 kg mm^-3
3.	Young’s modulus	71000 Mpa
4.	Poisson’s ratio	0.33
5.	Coefficient of thermal Expansion	2.3e-005 C^-1
6.	Compressive yield strength	280 Mpa
7.	Tensile yield strength	280 Mpa
8.	Tensile ultimate strength	310 Mpa

Table II: Wing Specifications

S.No	Specifications	Values
1.	Half wing span	40m
2.	Swept back angle	33.5^0
3.	Wing area	9,100ft
4.	Taper ratio	0.26

B. Modelling in CATIA

CATIA is the software used for designing the shapes and structural parts of the models. Using the above specifications the wing is designed using CATIA in the 3D view. The model with outer structure and inner parts are designed.

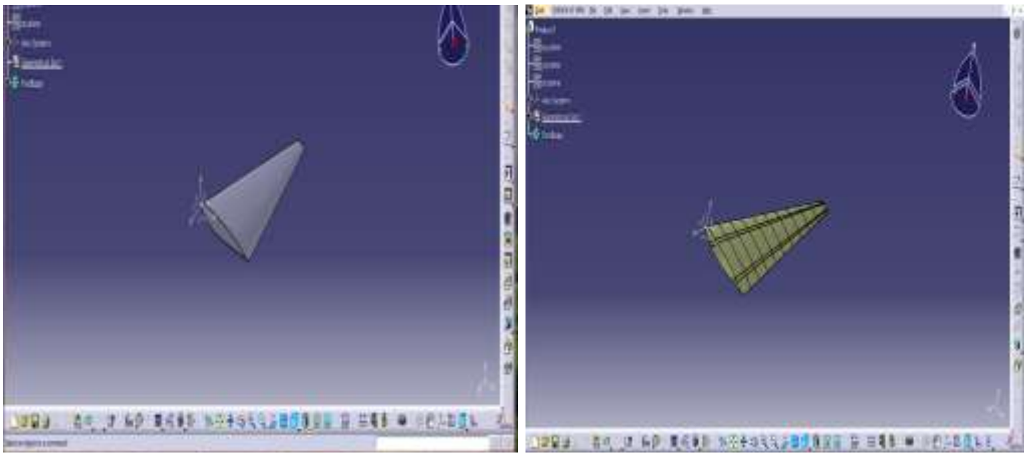

Fig.2: Design of NASA SC(2)-0412 wing using CATIA in the 3D view

IV. FLOW ANALYSIS

A. CFD Technique

The wing model designed in CATIA is taken for flow analysis. Here the technique used for flow analysis is Computational Fluid Dynamics (CFD). CFD is the numerical analysis used to analyse the fluid flows over the model. For flow analysis, here domains are applied to the wing model designed.

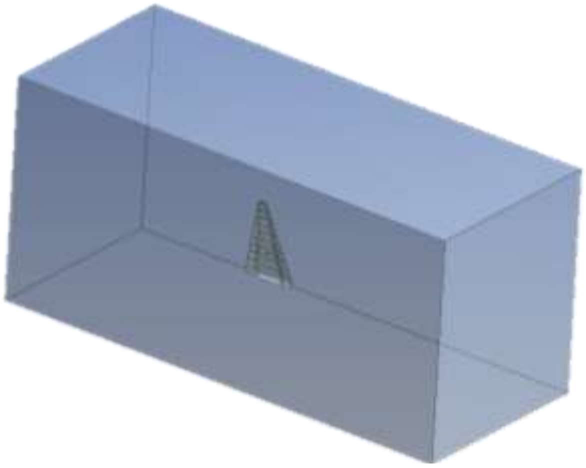

Fig.3: Wing model with domain

B. Meshing

Before doing flow analysis it is important to do meshing. Meshing is the important part of computer simulations because it shows drastic changes in results. Meshing means you create a mesh of some grid-points called 'nodes'. The results are calculated by solving governing equations (partial differential equation and Finite element method) numerically at each node of the mesh.

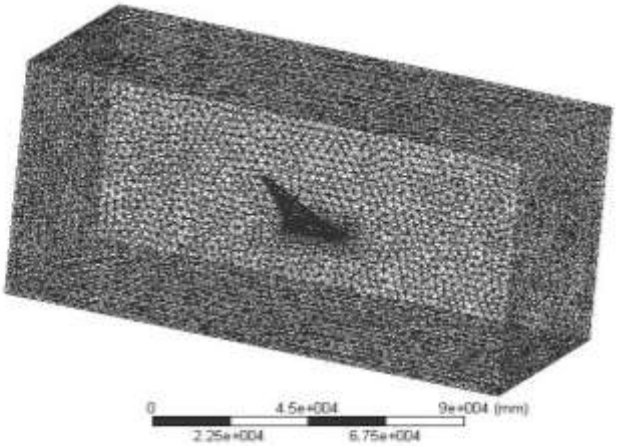

Fig.4: Wing model with meshing

C. Flow Analysis and Results

After meshing, the wing model is taken into flow analysis by giving input values of different Mach numbers at a cruise altitude of 35000 ft. Here the input Mach numbers are 0.82, 0.85 and 0.89 which is in the transonic region and the material for the flow analysis is air. As the wing taken for the analysis NASA SC(2)-0412 is mainly set to fly in the transonic region. By doing flow analysis, we get the pressure, lift and drag values so we get the best performance of the wing.

Table III: Performance of the wing

Mach Number	**Pressure Value**	**Lift Value**	**Drag Value**
0.82	21841.068	0.11319344	0.025093735
0.85	21579.08	0.11571787	0.027036934
0.89	21181.837	0.10306092	0.029905849

V. STRUCTURAL ANALYSIS

In structural analysis, the total deformation of the wing will be determined by analysing it in ANSYS software. The design and input values are same as given on fluid analysis and the material for structural analysis is Aluminium alloy. Then by solving structural analysis on the wing, the stress acting on the wing structure and the internal parts like spars, ribs are determined for 0.82, 0.85, 0.89 Mach numbers respectively. Also, the total deformation of the wing for corresponding Mach numbers is determined by the structural analysis made on the software.

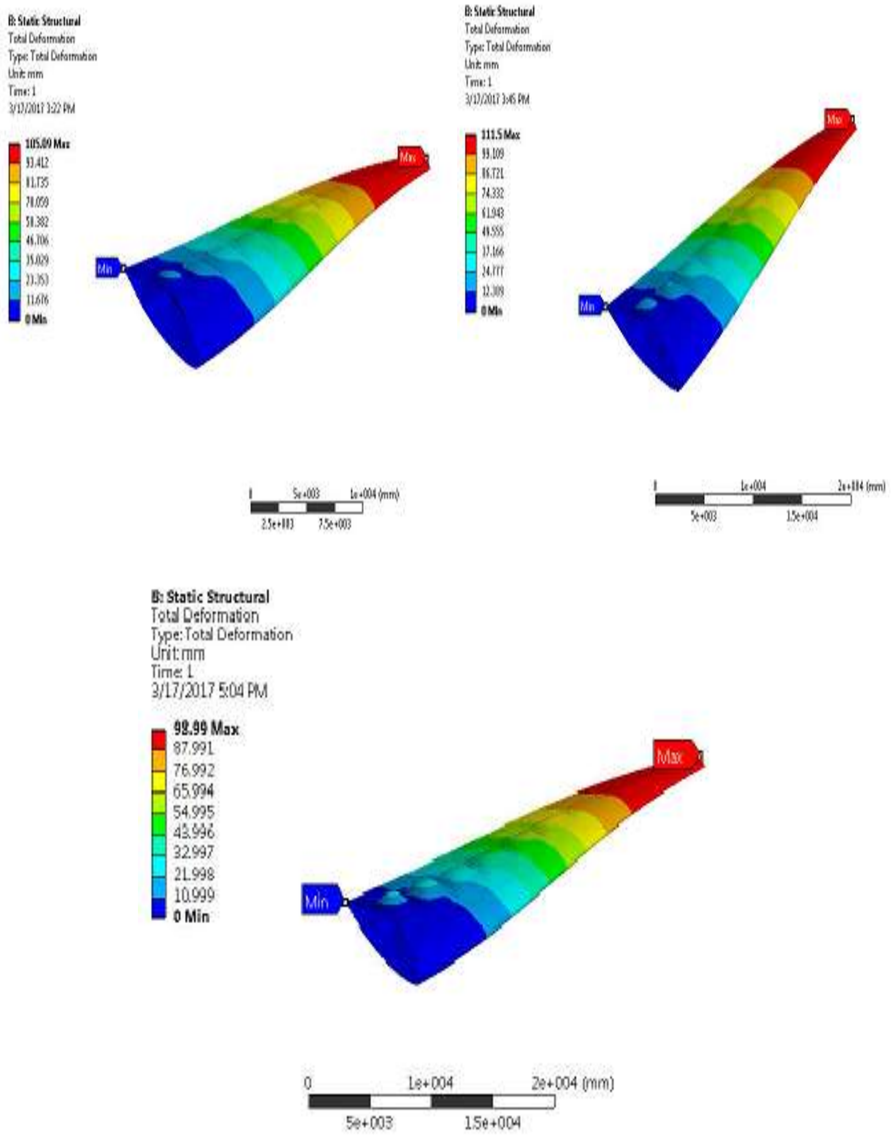

Fig.5: Total Deformation of the Wing at 0.82, 0.85, and 0.89 Mach numbers respectively

The results are displayed on the above Fig.5, where the total deformation of the wing is stated with minimum deformation at the leading edge and maximum deformation at the tailing edge. Here the leading edge is fixed with Aircraft's fuselage is the reason for minimum deformation on there. At the tailing edge, it is free and experiences more pressure from the air are the reason for maximum deformation on there. Thus for the three different Mach numbers, the images are displayed and the total deformation values are derived. In the transonic region, these three Mach numbers only will give the flight performance because the total deformation for 0.89 Mach number is dropped. So the Mach number increased further the wing will get damaged and the flight results in failure.

VI. CONCLUSION

It is concluded that the best performance of the wing chosen is at Mach number 0.85 because at this speed flights best lift value will be achieved. In that same Mach number, the total deformation value is quite good. By doing Fluid-structure Interaction this paper declares 0.85 Mach number will give the best performance of the Aircraft wing when comparing with other Mach numbers given. The graph plotted below shows the results of lift values from different Mach numbers. From the structural analysis, the total deformation for 0.85 Mach number is 111.5mm maximum deformation at the tail end.

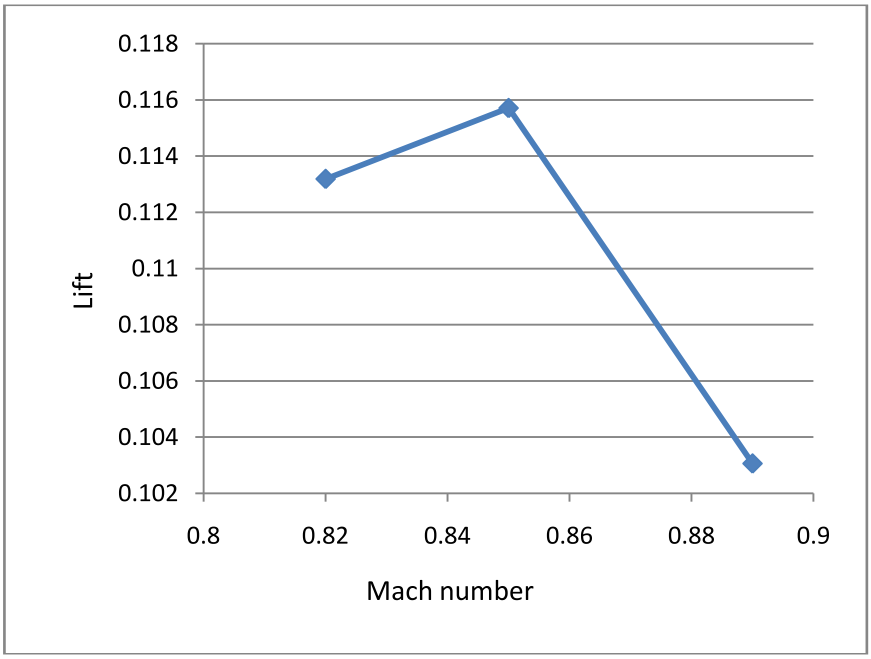

Fig.6: Graph plotted for Lift vs Mach number

ACKNOWLEDGMENT

With great pleasure and deep gratitude, authors wish and express their sincere gratitude to beloved Principal **Dr T. Ramachandran** for providing an opportunity and necessary amenities in carrying out this work and precise sincere thanks to all the staff members of Aeronautical Engineering whose assistance played a big role in this work and have been of immeasurable value.

REFERENCES

[1]. T. Sai Kiran Goud, Sai Kumar A, Dr S Srinivasa Prasad, “Analysis of Fluid-Structure Interaction on an Aircraft Wing”

[2]. Jian Tang, Satish Chimakurthi, Rafael Palacios, Carlos E.S. Cesnik, And Wei Shyy “Computational Fluid-Structure Interaction Of A Deformable Flapping Wing For Micro Air Vehicle Applications” by Aiaa 2008-615.

[3]. Chowla Sangeetha, Mr Veeranjaneyulu, Dr MSN Guptha, “Fluid-structure interaction on AGARD 445.6 wing at transonic speeds” Jawaharlal Nehru technological university, Hyderabad.

[4]. F. Liu, J. Cai and Y. Zhu, “Calculation of Wing Flutter by a Coupled Fluid- Structure Method”, University of California, Irvine, Irvine, California.

[5]. B. Nagaraj Goud, G. Sai Sathyanarayana, S. Shailesh Babu, T B S Rao “Dynamic aero elastic (flutter) instability characteristics of an aircraft wing” MRCET, Hyderabad.

[6]. Shiva.R, Manivel.M, Karthikeyan.J, “Flutter prediction based on the fluid-structure interaction of wing with winglet” Nehru institute of engineering and technology, Coimbatore published in IJRAME.

[7]. C.Bibin, Micheal Johnson Selvaraj and S.Sanju (2012), “Flutter analysing over an aircraft wing during cruise Speed”, *Elsevier. Procedia Engineering* 38 (2012) 1950 – 1961.