Vibrational Characteristics of AGARD 445.6 Wing in Transonic Flow

J S Chaitanya¹, Arun Prasad¹, B Pradeep¹, P L N Sri Harsha¹, S Shali² and S R Nagaraja¹

¹Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, Amrita University, India
²Department of Civil Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, Amrita University, India
Email: shali_135@yahoo.co.in

Abstract. This paper presents the application of Computational Fluid Dynamics (CFD) and Fluid Structure Interaction in ANSYS to do vibrational analysis on an aircraft wing in transonic region. A simulation study is conducted on a wing by modelling it in a solid modelling software. Further CFD analysis is performed at different Mach numbers to identify pressure variations at different locations on the wing. Transient structural analysis is carried out to study the variations in displacement of the wing with time. The post processing is done for determining the structural frequency and thereby to establish the flutter boundary in the transonic range.

1. Introduction

An aircraft is a very complex engineering structure which experiences vibrational effects both internally and externally. Aerodynamic problems in general are often difficult to solve analytically. Experimental or numerical simulation can be used to analyse these models to simulate the interaction of fluids with surfaces defined by boundary conditions. Flutter of an aircraft wing is a major concern in the field of aeroelasticity. Vasanth Dhanagopal et al. [2] coupled the CFD solver with the governing structural equations of motion and with the help dual time stepping approach to establish the flutter boundary. T Sai Kiran Goud et al. [3] in his article emphasises on the complications arising with the interaction between the fluid module and the wing module. The paper discusses the difficulties of fluid structure interaction problems when done analytically and thereby use ANSYS workbench to determine the stresses induced corresponding to the flow. Chowla Sangeetha et al. [4] discussed the fluid structure interaction problem by determining the initial boundary conditions of AGARD 445.6 wing and also studies the variation in stress across the wing. The fundamental structural behaviour of the AGARD 445.6 wing is studied under some practical load conditions and the effects of Fluid structure interaction are validated with available experimental results. Liu et al. [6] has used a coupled Computational fluid dynamics (CFD) and Computational Structural dynamics (CSD) solver to simulate the aeroelastic system on the time domain in order to determine its stability.
boundary is then established by solving the flutter equation on the frequency domain with indicial responses as input.

In the present work, AGARD 445.6 wing has been taken into consideration to study the vibrational behavior of an aircraft wing in aerodynamic flow. The analysis is done in ANSYS-FLUENT and the stability boundary of the wing is determined as the condition where the structure oscillates with a constant magnitude. The flutter speed index is also defined for the identification of the transonic characteristics.

2. Wing Modelling
The wing cross section is modelled by the NACA airfoil NACA65a004. The airfoil NACA 65a004 is a symmetric airfoil which comes under 6 digit series. This has an area of minimum pressure 50% of chord with a lift coefficient of zero and has maximum thickness of 4% of the chord. The standard dimensions of the AGARD 445.6 Wing in shown in Figure 1.

![Figure 1. AGARD 445.6 Wing](image1)

The root chord ($C_r$) and tip chord ($C_t$) dimensions are 0.558m and 0.368m respectively. The half span length is 0.762m. The aspect ratio is 1.65 and taper ratio is 0.66. The wing is swept back at an angle of 45 degrees. The angle between quarter chord line and the perpendicular plane to the wing is 45°.

The material used for the wing structure is laminated mahogany. The orthotropic properties of this material are density $\rho = 381.98$ kg/m$^3$, Parallel young’s modulus= 3151 MPa, Orthogonal young’s modulus= 416.2 MPa, Rigidity Modulus $G = 439.2$ MPa and Poisson’s ratio= 0.31.

The coordinates are imported from MS Excel to CATIA V5 using Macros. The solid model obtained from the CATIA V5 is saved with .igs extension to import it to the ANSYS workbench. The imported solid model in ANSYS is as shown in Figure 2.

![Figure 2. Wing Model in ANSYS](image2)

3. Modal analysis
The modal analysis is performed on the wing in order to determine the natural frequencies and mode shapes with a fixed support at the root chord. The first 4 modes are taken into consideration to validate the wing model as shown in Table 1. Figure 3 shows the first bending frequency mode and Figure 4 shows the first torsion frequency mode. The obtained natural frequencies are compared with the previous study on the same model as in Ref [1].
Table 1. Natural Frequencies of the model

| Modes | Frequency | Ref. [1] |
|-------|-----------|----------|
| 1     | 7.2       | 9.46     |
| 2     | 46.02     | 39.44    |
| 3     | 70.74     | 49.71    |
| 4     | 88.4      | 94.39    |

4. Aerodynamic analysis

Aerodynamic analysis is performed to determine the pressure variations over the wing surface. The solid model is imported to ANSYS-Design Modeler and a fluid domain is created around the wing structure where the flow of fluid is constrained.

The fluid domain is meshed as shown in Figure 5. The region of wing which first comes in contact with the air flow is the leading edge of the wing. In the leading edge, the air flow is separated and at the trailing edge they merge. Hence the mesh on the fluid domain is a fine mesh close to the leading and trailing edge as shown in Figure 6. Transient analysis is preferred as it is a time domain solution and includes higher order terms dealing with time and hence we can get better accuracy than steady state solutions. In ANSYS-FLUENT transient analysis on the fluid domain is done with pressure based solver. The analysis is carried out with a time step of 0.001 for 10s to get the solution convergence. The pressure coefficients of the wing structure are plotted for different locations across the span. The Figure 7 and Figure 8 show the pressure coefficient at zero span and full span respectively for a Mach number 0.98.
The upper line indicates the pressure coefficient values on the upper camber line and lower line indicates the pressure coefficient values on lower camber line. These two lines almost coincide as it a symmetric airfoil. The area between these lines helps to determine the lift factor. As CFD analysis is done on this wing with 0 angle of attack there is no difference in values of these two lines and thus no lift generated.

5. Transient Structural Analysis

One way FSI and a transient structural analysis is done in ANSYS-FLUENT to determine the deformations on the wing structure due to air flow over the wing. Aerodynamic analysis is done for the fluid domain with the established boundary conditions.

In transient structural the wing will be under consideration. The meshing of wing in structural analysis is shown in Figure 9. The solution is shared from FLUENT to Transient Structural so that the pressure loads are imported for every element in the generated mesh. The imported pressure loads are shown in the Figure 10. The transient analysis is carried for 10s with time step 0.05s. The same procedure is carried out for different Mach numbers to obtain the displacement vs time plot.

6. Flutter Speed Index

Fast Fourier Transform (FFT) is done to find the frequency of structure, and the bending structural frequencies are determined for each Mach number. This bending structural frequency is used to determine the speed index value for the corresponding Mach number.

The flutter speed index is calculated by the formula given below

\[ V_f^* = \frac{U_\infty}{\left(\frac{b^2}{2\pi \mu \omega}\right)} \]  \hspace{1cm} (1)

Where \( U_\infty \) is the free stream velocity, \( b \) is the half-chord length, \( \omega \) is the structural frequency of the wing, and \( \mu \) is the mass ratio.

\[ \mu = \frac{m}{(\pi b^2 \rho l)} \]  \hspace{1cm} (2)
Where \( m \) is the mass of wing, \( l \) is the length of the wing and \( \rho \) is the density of free stream air. The speed index diagram has been plotted by considering the structural frequencies in bending mode of the wing. In the present simulation study, it is the bending displacement data that is analysed for the frequency. The torsional frequency data were not extracted as the present analysis was incapable of extracting the same.

7. Results

The displacement pattern of the structure with the time series is determined. The time-displacement plots at 0.57 Mach, 0.86 Mach and 1.14 Mach are shown in Figure 11, Figure 12 and Figure 13 respectively. Flutter boundary is considered as the conditions corresponding to the oscillations with constant amplitude, the figures show that the structure is at the flutter boundary.

Figure 11. Displacement vs Time at 0.57 Mach

Figure 12. Displacement vs Time at 0.86 Mach

Figure 13. Displacement vs Time at 1.14 Mach

Figure 14 shows the variation of speed index with the Mach number. For the Mach regime between 0.8 to 1.2 which is the transonic region, it is seen that there is a dip in the speed index graph. This dip is attributed to the nonlinearities and unsteadiness which are characteristics to the transonic region. It has been observed that a maximum dip is occurring at Mach 0.86. As we further increase the value of Mach number the speed index rises. The speed index graph in Figure 15 shows the variation of flutter index value in Ref [6]. It is seen that the trend of the graph is in good agreement whereas there is a difference in the range of values of flutter speed index. This is due to the fact that in the present work, the analysis was not able to extract the torsional frequency. The flutter speed index is computed with respect to the bending mode.
8. Conclusion
The analysis has been performed in ANSYS Workbench to obtain the vibrational characteristics of the AGARD 445.6 wing. The wing was studied under transonic flow conditions. Computational fluid dynamics analysis is performed on the wing to have a basic understanding on the pressure variations on the surface of the wing. The transient analysis gives the time domain solution for the wing which is utilized to extract the structural frequency. The plot of Mach number and its corresponding flutter index is important to observe the behavior of the wing when it is in the transition zone between subsonic and supersonic speeds. From the flutter index diagram, dip is observed in transonic regime and at a certain Mach number the wing is most unstable.

References

[1] Yates E C 1988 AGARD standard aeroelastic configurations for dynamic response I wing 445.6. AGARD Report 765, North Atlantic Treaty Organization, Group for Aerospace Research and Development. Vol 10.
[2] Dhanagopal Vasanth, Varghese Mathew T and Sharanappa V Sajjan 2006 Time Domain Flutter Analysis of the AGARD 445.6 Wing. 18th Annual CFD Symposium, CFD division of Aeronautical Society of India.
[3] Goud Sai Kiran T, Sai Kumar A and Srinivasa Prasad S 2014 Analysis of Fluid-Structure Interaction on an Aircraft Wing. International Journal of Engineering and Innovative Technology (IJEIT). Vol. 3.
[4] Sangeetha Chowla, Veeranjaneyulu and Guptha M S N 2015 Fluid Structure Interaction on AGARD 445.6 wing at Transonic Speeds. International Journal of Engineering Trends and Applications. Vol. 2
[5] Dowell Earl H, Robert Clark and David Cox 2004 A Modern Course in Aeroelasticity. Kluwer academic publishers. Vol 116
[6] Liu F, Cai J and Zhu Y 2001 Calculation of wing flutter by a coupled Fluid Structure Method. Journal of aircraft. Vol. 38
[7] Isogai K 1979 On the Transonic-Dip Mechanism of Flutter of a Sweptback Wing. AIAA Journal Vol. 17
[8] Anderson and John D 1991 Fundamentals of Aerodynamics. McGraw-Hill, Inc. Vol 2.