한국추진공학회 2010년도 춘계학술대회 논문집 pp.331~334 2010 KSPE Spring Conference

Transonic characteristics for AGARD Wing 445.6 by numerical simulation

Wenjuan Ye* · Youngshin Lee** · Jinhai Lan***

ABSTRACT

The supersonic speeds slowing down by shock waves is a common problem during the transonic region. So how to study the status of shock on the surface of airplane and wings is crucial adjective during transonic region. However, the theoretical and computational transonic flow problems were very hard. This paper introduced using Navier-Stokes Schemes to study characteristics of AGARD Wing 445.6 by ANSYS CFX in transonic region. From simulations results, as the Mach number increases, shock waves appear in the flowfield, getting stronger as the speed increases, these shock waves will lead to a rapid increase in drag.

Key Words: Transonic, AGARD Wing 445.6, Shock Wave, Reynolds-Averaged Navier-Stokes

1. Introduction

Transonic phenomenon occurs when there is transition from subsonic to supersonic local flow in the same flowfield (typically with freestream Mach numbers from M = 0.8 to 1.2). Usually the supersonic region of the flow is terminated by a shock wave, allowing the flow to slow down to subsonic speeds. This complicates both computations and wind tunnel testing. It also means that there is very little analytic theory available for guidance in designing for transonic flow conditions[1]. Note that hypersonic vehicles with bow shocks necessarily have a region of subsonic flow behind the shock, so there is an element of transonic flow on those vehicles.

As the Mach number increases, shock waves appear in the flowfield, getting stronger as the speed increases. The shock waves lead to a rapid increase in drag, both due to the emergence of wave drag, and also because the pressure rise through a shock wave thickens the boundary layer, leading to increased viscous drag. Thus studying the status of shock on the surface of airplane and wings is crucial adjective during transonic region. However, the theoretical and computational transonic flow problems were very hard. Initially, very few analytic or numerical methods were available. Earll Murman and Julian Cole made the major breakthrough. Using transonic small disturbance theory, they came up with a scheme that could be used to develop a practical computational method[2].

* 충남대학교 공대 기계설계실험실
** 충남대학교 공대 기계설계실험실
*** 충남대학교 공대 기계설계실험실
이영신, E-mail: leeys@cnu.ac.kr

The next logical development was to add viscous effects to the inviscid calculations, and to switch to the Euler equations for the outer inviscid flow. By now, many researchers were working on computational flow methodology, which had become an entire field known as CFD[3].

In 1950s, the swept wings was developed to delay the drag rise Mach number in German. AGARD Wing 445.6 which was standard swept wing studied by many researchers with lots of methods. So in this paper, the distribution of pressure on the surface of AGARD Wing 445.6 was studied by numerical method which utilizes the Reynolds-Averaged Navier-Stokes (RANS) in ANSYS CFX to study characteristics in transonic region.

2. Numerical Analysis

2.1 Flow Governing Equations

For transonic flow conditions with insignificant nonlinear effects, the time domain Euler and Reynolds-Averaged Navier-Stokes equations (RANS) simulations produce similar flutter boundaries. However, Euler solutions are unable to produce accurate flow boundaries when the nonlinear flow effects are significant, thus the RANS equations must be used for these cases [4]. So in this paper, the RANS equations were used for solving.

The Reynolds-Averaged Navier-Stokes equation for compressible flow given in CFX by [5]:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot \{\rho U \otimes U\} = \nabla \cdot \{\tau - \rho \overline{u \otimes u}\} + S_M \quad (1)$$

And the associated continuity equation given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (2)$$

where U is velocity, u is the time varying component, ρ is density, τ is the molecular stress tensor, S_M is the source term.

2.2 AGARD Wing 445.6

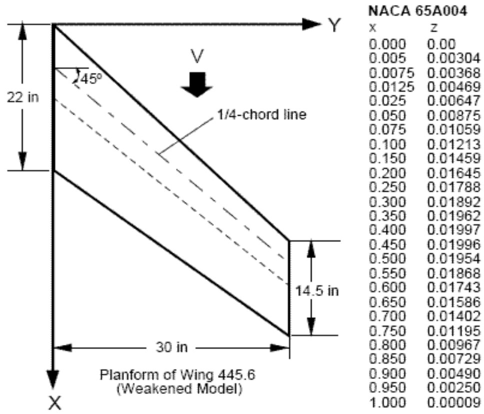


Fig. 1 Geometry of AGARD 445.6 test case [6]

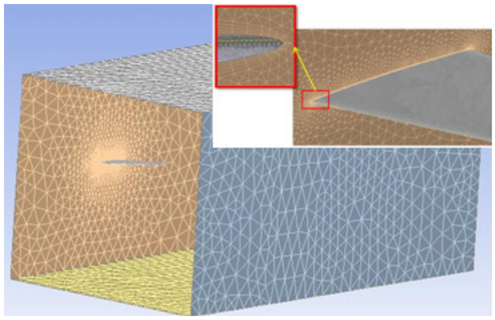


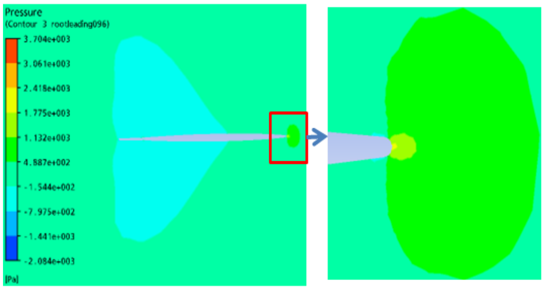
Fig. 2 Mesh for domain

The AGARD Wing 445.6 is one of the most frequently used benchmarks of study. This test case has a panel aspect ratio of 4.0, a quarter-chord sweep of 45º, a taper ratio of 0.6, and a NACA 65A004 aerofoil section. The semi-span of this model was 30inch and the root chord was 22inch, as shown in Fig.1.

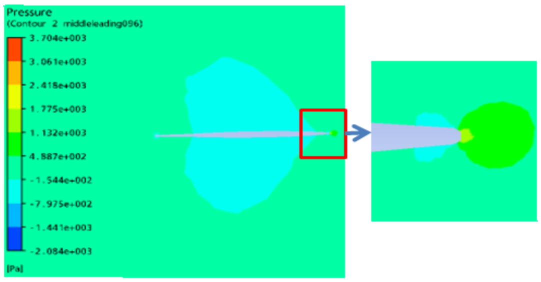
The computational fluid domain model was discrete with the prism elements layers around the wing and filled the rest space with tetrahedron elements by means of CFX Meshing, which is used to generate high quality grid, see Fig. 2.

2.3 Results and Discussions

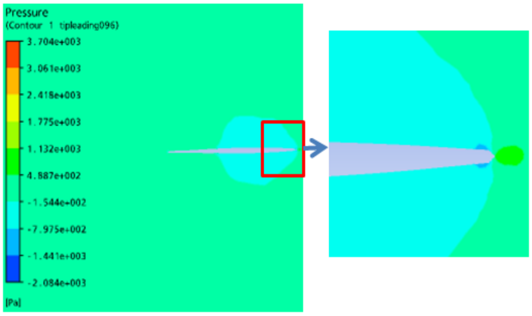
Figure 3(a~i), taken from CFX, shows the development of the flow with increasing Mach number from 0.960 to 1.141, and along with the wing chord from root to tip. From the Figures, the most obvious is the all the cases in transonic had symmetry shock waves which is due to the symmetry of wing configuration. At some Mach number the local flow becomes sonic at a single point on the upper surface where the flow reaches its highest speed locally. That is the critical Mach number. As the freestream Mach number increases further, a region of supersonic flow develops just like the Fig. (a)&(d)&(g). And the shock wave moved from the leading to trailing. At each Mach number, it's obvious that shake waves moved to the leading edge from root to tip, meanwhile, the shock waves were decreasing.



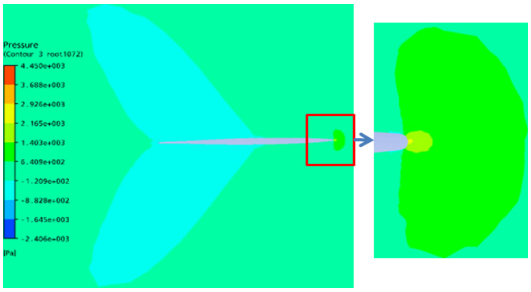
(a) Root of Wing at Mach Number of 0.960



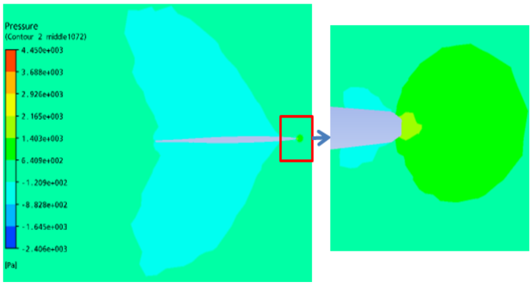
(b) Middle of Wing at Mach Number of 0.960



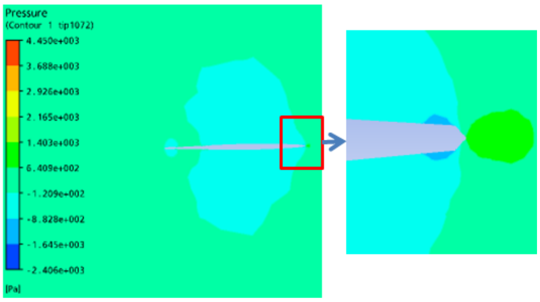
(c) Tip of Wing at Mach Number of 0.960



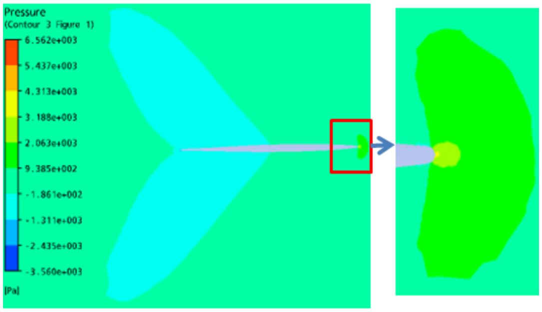
(d) Root of Wing at Mach Number of 1.072



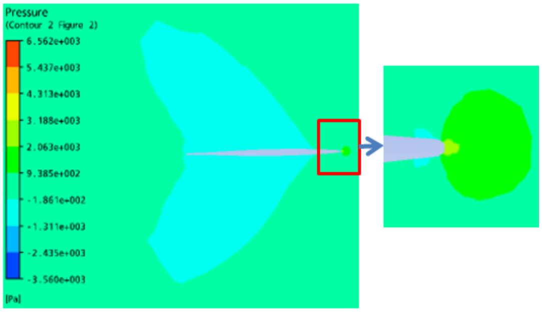
(e) Middle of Wing at Mach Number of 1.072



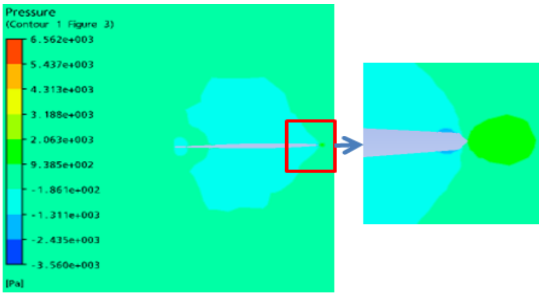
(f) Tip of Wing at Mach Number of 1.072



(g) Root of Wing at Mach Number of 1.141



(h) Middle of Wing at Mach Number of 1.141



(i) Tip of Wing at Mach Number of 1.141

3. Conclusions

In this paper, we used the numerical method based on Reynolds-Averaged Navier-Stokes (RANS) to discuss the aerodynamic properties of AGARD 445.6 wing.

Its characteristics are summarized as follow:

1. For transonic case, there are shock waves on the both of surfaces, tracing from the wing-tip leading edge to the wing root trailing edge.
2. As the Mach number increases, the shock moves after and becomes stronger. As the Mach number continues to increase, a supersonic region and shock wave also develops on the lower surface.
3. As the Mach number approaches one, the shocks move all the way to the trailing edge. Finally, when the Mach number becomes greater than one, a bow wave appears just ahead of the airfoil, and the shocks at the trailing edge become oblique. These shock waves are the basis for the supersonic wing.

References

1. Foss. R. L., "From Propellers to Jets in Fighter Aircraft Design," AIAA Paper 78-3005, The Evolution of Aircraft Design, pp.51-64, 1978.
2. Murman, E. M, and Cole, J. D., "Calculation of Plane Steady Transonic Flows," AIAA J., Vol. 9, No. 1, 1971.
3. Antony, J., "Full-Potential, Euler, and Navier-Stokes Schemes," Applied Computational Aerodynamics, Vol. 125, pp. 39-88, 1990.
4. Li, Z. Y., "Parallel computations of 3D unsteady compressible Euler equations with structural coupling", Master Thesis, Purdue University, Indianapolis, 2002.
5. Rausch, E. and Batina, J., "Calculation of AGARD wing 445.6 flutter using Navier–Stokes aerodynamics", pp.3493-3476, 1993.
6. Yates, E.C., "AGARD standard configuration for dynamic response, candidate configuration I.-Wing 445.6", NASA TM 100492, 1987.