Numerical Investigation of Aeroelastic Deformation Effect on the NASA Common Research Model

Lu Zhang¹, Zhongyun Xiao¹* Shuai Ma¹ and Fan Liu¹

¹ China Aerodynamics Research and Development Center, MianYang, SiChuan, 621000, China
*Corresponding author’s e-mail: xiaozy@cardc.cn

Abstract. The risk of the aeroelastic deformation of the wings to the flight safety has become increasingly prominent for commercial transport aircraft due to the requirements for lightweight structure. In this work, Reynolds averaged Navier-Stokes (RANS) method coupled with transfinite interpolation were employed to investigate the influence of static elastic deformation of wing on aerodynamic characteristics of NASA CRM. Both bend and twist distributions were scaled in numerical simulations based on dynamic pressure, which was shown to be reasonable by the comparisons between the calculations and measurements. The numerical results show premature flow separation in the middle and outboard of the wing for original configuration. Furthermore, the flow separation was suppressed by the upward bending of the wing and negative twist in the leading edge, and the aerodynamic characteristics of the aircraft were improved by the aeroelastic deformation.

1. Introduction

Safety is the most important in the design of civil aircraft. Under the influence of the coupling between the elastic structure and the steady aerodynamics, the wing of the aircraft would undergo elastic deformation during flight, which is the so-called static aeroelastic problem. Many problems unfavourable to flight safety arise due to the elastic deformation, including changes in aerodynamic load, lift efficiency and static stability margin, wing torsional divergence, decreases in control efficiency and aileron reversal. The elastic deformation effect becomes more significant for modern civil aircraft because the requirements for lightweight structure.

Many factors restrict the development of experiments to investigate the effects of aircraft static aeroelasticity in wind tunnel. On the one hand, it is difficult to design and manufacture the flexible model, and the cost is also considerable. On the other hand, the measurement technology of elastic deformation has a high threshold. These objective constraints have promoted numerical simulation to play an important role in the investigation of aircraft aeroelasticity. In recent years, the improvement of computing power of computers has led to the development of research on aircraft aeroelasticity based on computational fluid dynamics (CFD) and computational structural dynamics (CSD).

High-precision numerical simulation need to base on high-fidelity geometry. In the numerical simulation of aeroelasticity, the shape of the research object will change rigidly or elastically, which requires dynamic mesh technology. In previous numerical investigations [1], the aeroelasticity of wing was usually considered at transonic speed, and insufficient attention was paid at low speed. In current investigation, the NASA CRM was used to investigate the effect of elastic deformation on the aerodynamic characteristics of the aircraft based on RANS method, and the numerical results are compared with available measurements.
2. Test Case

2.1. Model description

NASA Common Research Model (CRM) [2-3] was used in present work, which is a typical configuration of modern commercial transport aircraft. There are several different configurations were tested in wind tunnel, and the configuration was employed in the current investigation which consists of a modern supercritical transonic wing, a fuselage and a tail with incidence angle of 0°, as shown in Fig. 1. The mean aerodynamic chord is equal to \( c = 0.189 \) m, and the reference area is \( S = 0.28 \) m\(^2\). The experiment has been performed in National Transonic Facility (NTF) [4], Ames Unitary Plan Wind Tunnel (UPWT) [5], European Transonic Wind tunnel (ETW) [6], and transonic wind tunnel of Japan Aerospace Exploration Agency (JATW)[7]. The ETW data was used in present work.

![Figure 1. NASA Common Research Model](image)

2.2. Aeroelastic deformation of CRM

The range of angle of attack tested in ETW is wider, so the flow separation phenomenon is more complicated than other wind tunnels. In addition to the conventional aerodynamic tests, the aeroelastic deformation of the main wing and tail was also measured in ETW. The low dynamic pressure at low speed in NTF leads to small elastic deformation, so the aerodynamic characteristics are not greatly affected by the elastic deformation of wing, and which is confirmed in the numerical results in 5th drag prediction workshop (DPW-V) [1]. On the contrary, the aeroelastic deformation needs to be considered in the numerical prediction based on measurements of ETW because the higher aerodynamic loads of wings caused by higher dynamic pressure.

![Figure 2. Wing bending (left) and twist (right) distribution](image)
However, the bend and twist distributions obtained in measurements cannot be directly applied to numerical simulations. There are differences in flow conditions between the test for elastic deformation of the wings and the test for aerodynamic force. Both bend and twist distributions were scaled proportionally based on the ratio of the dynamic pressure in different tests. The scaled bend and twist distributions was presented in Fig. 2 and Fig. 3, which shown the main wing with positive bending deflection and untwist deformation. The large bending deformation of main wing occurs at 10° and 12° with 7mm displacement of wing tip, and maximum twist appears at 12°. Nevertheless, there was a small deformation at 18° where large flow separation appeared.

3. Computational Description

3.1. The grid system
The structured multi-block grid was applied in numerical simulations, and the grid topology was H-type. The grid was densified in the main wing and tail area where the flow separation appears as the increasing angle of attack. The O-type topology was applied in the boundary layer, and the wall-normal distance of the first cell satisfies $y^+=1$. The total cells number is $1.2 \times 10^7$ in the whole flowfield. The grid topology and the details was shown in Fig. 4 and Fig. 5.

3.2. Numerical methods
In present work, the Navier-Stokes equations were discretized by a cell-central finite volume method. The spatial derivative of the inviscid flux was based the flux-difference splitting method of Roe, and the third-order MUSCL interpolation [8] was employed. Then the viscid flux was discretized with second-order central differences. The Flow equations were solved with Menter’s two-equation SST model [9] turbulence enclosure.
3.3. Grid deformation method

In order to avoid the uncertainty caused by artificial adjustment of the grid, and to maintain the consistency of the grid in boundary layer, transfinite interpolation (TFI) [10] was used to generate the grid after elastic deformation of the wing, which has high grid deformation efficiency because it is an algebraic method.

Figure 6. Deformed wing generated by TFI

For the three-dimensional structured grid, One-dimensional and two-dimensional TFI need to be used firstly to determine the position of the corner points and edges of the grid block, as well as the displacement of the surface nodes, then the information of the internal nodes of the grid block is obtained with three-dimensional TFI. Fig. 6 shown the deformed wing generated by TFI method.

3.4. Computing environments

The computations were performed with the high-performance computing cluster in present work, which has 153 nodes and each node contains two eight-core processors Intel Xeon E5-2670 with main frequency of 2.6Hz. Parallel computing with 2448 cores can be performed by the computing cluster based on message passing interface (MPI) technology. The strategy of parallel computing is to allocate computing resources according to the size of the grid block to balance the load of the processor.

4. Result and Discussion

In this section, the aerodynamic characteristics of the original CRM and deformed configuration, including deformation of main wing and tail, was predicted by Reynolds-averaged Navier-Stokes (RANS) method. According to available measurement, the numerical result was compared with each other and discussed.

4.1. Lift and drag coefficient

The comparisons of lift coefficient and drag coefficient between calculations and measurements are presented in Fig. 7 and Fig. 8. The lift and drag coefficient between the original and deformed configuration has little difference at small angle of attack, and the difference increases as the angle of attack increases, because increasing deformation of bend and twist, especially near the stall angle of attack where has the large deformation of wings. However, it was shown that the big difference between numerical and experimental result at an angle of attack greater than 12° where stall occurs, the main reason can be attributed to the RANS model has inaccurate prediction of flow separation.
4.2. Flow separations and pressure distribution

The comparison of the streamlines on the wing surface and the pressure coefficient contour are presented in Fig. 9, where the original configuration was shown in the upper column and the deformed one in the lower column. In order to better understand the difference in flow caused by the elastic deformation of the wing, chordwise static pressure distributions were extracted from numerical results in four different spanwise wing sections, 28.3%, 50.2%, 60.3% and 95%, corresponding to the inboard, middle and outboard of wing. The $C_p$ cuts at 10° and 12° angle of attack, which correspond to the maximum lift, were compared respectively in Fig 10 and Fig 11.

The flow separation appears in the middle and outer of main wing at $\alpha=10^\circ$ to the original configuration, while the flow is completely attached to the deformed configuration. From the $C_p$ distribution in fig3, on the inner wing, the numerical results of two configurations shows good agreement with the experiment on the inner wing ($\eta=28.3\%$) where the flow is attached. Furthermore, the $C_p$ of original configuration has significant difference at $\eta=50.2\%$, and the difference still exists in other two spanwise locations.

Figure 9. Surface streamlines and Cp contour (top: original configuration, bottom: deformed configuration)
The flow on the wing of the deformed configuration remains attached at $\alpha=12^\circ$, which is similar to $\alpha=10^\circ$. But the flow separation of the original configuration shows further expansion of the inner wing. The comparison of static pressure distribution between the original wing and measurement shows minor differences at the inboard section where wing deformations are very small, and original results deviates from the experiment in the middle and outer sections due to the large elastic deformation.

The numerical results of two configurations are in good agreement with each other and the experimental results because the very small deformation at low angle of attack. As the angle of attack increase, the aerodynamic load of the wings increases and consequently aero-elastic deformation of the wings increases. The deformation changes the original aerodynamic characteristics due to changes in the local angle of attack, the flow separation was suppressed by the upward bending of the wing and negative twist in the leading edge suppress, so the stall on the wing was delayed. The agreement between the calculations based on deformed configuration and measurements shows that it is reasonable to adjust the aero-elastic deformation of wing based on the dynamic pressure in wing tunnel. Furthermore, it is very necessary to consider aero-elastic deformation in the calculations, even at low speed, because the deformation in the middle and outboard of the wing cannot be ignored near the stall angle of attack.

5. Conclusion
A numerical investigation of the effect of elastic deformation on aerodynamic characteristics carried out by solving Reynolds averaged Navier-Stokes equations, and transfinite interpolation method was used to generate the grid after elastic deformation of the wing. The numerical results such as lift and drag coefficient, streamlines and pressure distributions at spanwise section were investigated and compared with wind tunnel results.
The deformation changes the original aerodynamic characteristics. The numerical results show false flow separation in the middle and outboard of the wing for original configuration. However, the good consistency was presented by the calculations considering aero-elastic deformation of the wing, which shows that it is reasonable to adjust the aero-elastic deformation of wing in numerical simulations based on the dynamic pressure in wing tunnel. Furthermore, the flow separation was supressed by the upward bending of the wing and negative twist in the leading edge, and the aerodynamic characteristics of the aircraft were improved by the aeroelastic deformation. As a result, it is necessary to consider aero-elastic deformation in the calculations, even at low speed, to the configuration of modern commercial transport aircraft.

References
[1] Keye, S. Togiti, V. Eisfeld, B. Brodersen, O. Rovers, M.B. (2013) Investigation of Fluid-Structure-Coupling and Turbulence Model Effects on the DLR Results of the Fifth AIAA CFD Drag Prediction Workshop. In: Fluid Dynamics and Co-located Conferences. San Diego. 2509.
[2] Vassberg, J.C. Dehaan, M. Rivers, M. Development of a common research model for applied CFD validation studies. In: 26th AIAA Applied Aerodynamics Conference. Hawaii. 6919
[3] Vassberg, J.C. Tinoco, E.N. Mani, M. (2014) Summary of the fourth AIAA computational fluid dynamics drag prediction workshop. Journal of Aircraft, 51: 1070-1089.
[4] Rivers M B., Dittberner A. (2014) Experimental investigations of the NASA common research model. Journal of Aircraft, 51: 1183-1193.
[5] Rivers M B, Dittberner A. (2011) Experimental investigations of the NASA Common Research Model in the NASA Langley National Transonic Facility and NASA Ames 11-ft transonic wind tunnel. In: 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. Orlando. 1126.
[6] Quix, H., Hensch, A.K., (2015) Dynamic Measurements on the NASA CRM Model tested in ETW. In: 53rd AIAA Aerospace Sciences Meeting, Kissimmee. 1097.
[7] Ueno, M. Kohzai, T. Koga, S. (2013) 80% scaled NASA Common Research Model wind tunnel test of JAXA at relatively low Reynolds number. In: 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. 493.
[8] Van Leer, B. (1979) Towards the ultimate conservative difference scheme V: A Second Order Sequel to Godunov's method. Journal of Computational Physics, 32:101-136.
[9] Menter, F. R. (1994) Two-equation eddy viscosity turbulence models for engineering applications. AIAA Journal, 32: 1598-1605.
[10] Dubuc, L. Cantariti, F. Woodgate, M. (2000) A grid deformation technique for unsteady flow computations. International Journal for Numerical Methods in Fluids, 32: 285-311.