NUMERICAL ANALYSIS OF BLEND-WING-BODY AIRFRAME AND ENGINE AERODYNAMIC INTERFERENCE

Zhaoguang Tan, Yingchun Chen, Jiangtao Si, Yuan Zhong
Shanghai Aircraft Design and Research Institute of COMAC, Shanghai, China

Abstract

Blended–*Wing*–*Body (BWB) is a relatively new aircraft concept that has potential use as a commercial transport aircraft. BWB is basically a flying wing with the payload, i.e., passengers and cargo, enclosed in the thick, airfoil shaped, center section. Studies have shown remarkable performance improvements for BWB over a conventional subsonic transport configuration discussed above based on equivalent technology.BWB should not use the traditionalengine layout of wing mounted or fuselage mounted. Engineback installation becomes the preferred layout. However, back mounted engine is easy to produce the shock, separation, intake air distortion and other aerodynamic interference problems. The aerodynamic design of engine and airframe integration has become an important part of BWB development. And this paper focuses on this area.*

1 Calculation Methodology

1.1 Governing Equations and Discretization Scheme

The governing equation used in this paper was a 3-D integral form of RANS.

$$\frac{\partial}{\partial t}\iiint \boldsymbol{Q}\,dv + \iint_s \boldsymbol{f}\cdot\boldsymbol{n}\,ds = 0 \qquad (1)$$

Where v stood for the control volume, s stood for the surface of the control volume, Q was the conserved quantity, f was the sum of non-viscous and viscous fluxes passing through surface s, and n was the outward unit normal vector of a surface. The time and special discretization schemes were full implicit time marching scheme and high-order up wind scheme; and the turbulent model was k-ω SST[1].

1.2 Model of a Turbofan Engine

The flow characteristics of a turbofan engine is extremely complicated. However, the influences on aircrafts from engine are mainly due to intake and exhaust effects. Therefore, the internal flow of the engine could be negligible during this research, and once the specific boundary condition was selected to represent the intake and exhaust flow fields, the influences due to engine could be simulated. For a typical turbofan engine (or T.P.S), the simplified model was shown in fig. 1[1] [2].

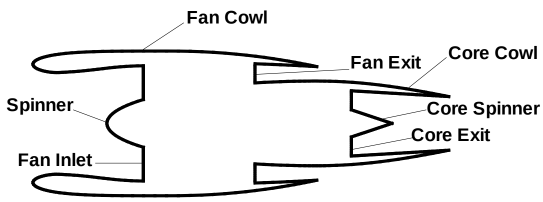


Fig. 1 Turbofan Engine Model

1.3 Definitions of Thrust and Drag

Precisely defining thrust and drag are very important when calculating the aerodynamics influences due to power planet. The methods to analysis thrust and drag were introduced in this section[3].

The shape of intake flow tube of an engine mainly depended on its working condition during subsonic flights. The net thrust of an engine could be obtained by taking engine cross-section as a control volume and capturing its internal flow. To analyze the thrust after installation, the loss of thrust due to installation should be considered. The typical working condition of a civil aircraft engine was shown in fig. 2. Flow bypassed the engine started to decelerate from far field (position 0) until the stagnation point (position i), then it accelerated until the maximum radius of the nacelle (position M). Similarly to flow around an airfoil, there was a suction due to flow from position i to position M. However, because of viscosity, separation and local shockwave during transonic flight, the suction could not totally balance the additive, and the difference was called spillage drag. Without considering pylon drag and interference drag, the drag due to nacelle could be written as:

$$D = \int_0^i (p - p_0)dA + \int_i^M (p - p_0)dA + \int_M^9 (p - p_0)dA + \int_i^9 \tau_w dA \quad (2)$$

Where D was the drag on nacelle, p was static pressure, A was projection area and w was the coefficient of friction of the nacelle wall.

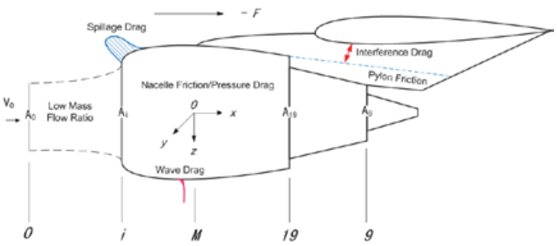


Fig. 2 Thrust Drag Definition

2 Validations

In order to validate the simplification of engine model, the generation of grids and the calculation of flow field, simulations of isolated powered engine model airflow were conducted and compared with experimental results.

A wind tunnel model from Institute of Aeronautics and Astronautics of Japan “NAL-AERO-02-01” T.P.S. (Turbine Powered Simulation) was used in this research. The 3-D model of this engine, shown in fig. 3, was obtained by rotating the 2-D boundary (given in the reference) about its axis.

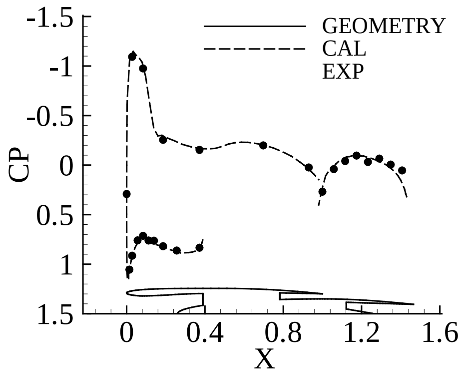
Fig.3 “NAL-AERO-02-01” Model Grid

The simulation results of intake and exhaust fields of two different working conditions were listed in table 1. (Re was 1E6, based on the maximum radius of the nacelle)

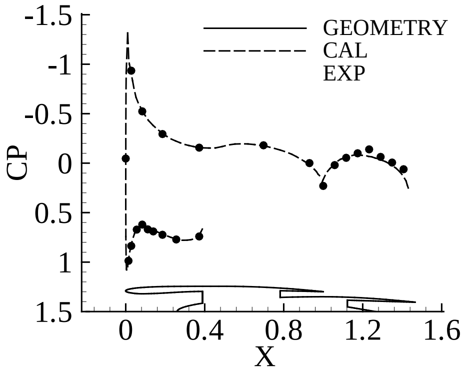
Table 1 Parameters of Two cases

Parameters	case1	case2
M_∞	0.8008	0.6024
α	0.0°	0.0°
M.F.R.	0.4973	0.4961
F.P.R.	1.3428	1.2052
F.T.R.	1.1086	1.0634
C.P.R.	0.0617	0.1003
C.T.R.	0.6907	0.7480

The pressure distributions on fan fairing and turbine fairing by simulations and experiments were shown in fig. 4. It showed the simulations and experiments matched and therefore validated the simulation methods.

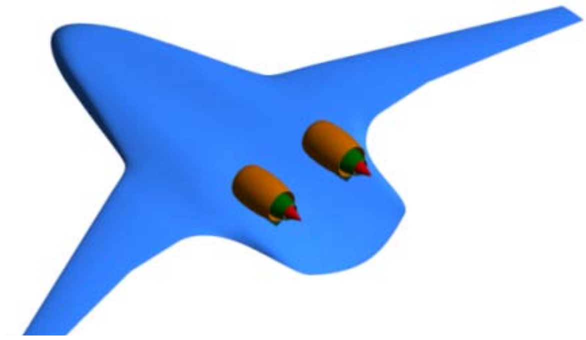


(a) Case 1

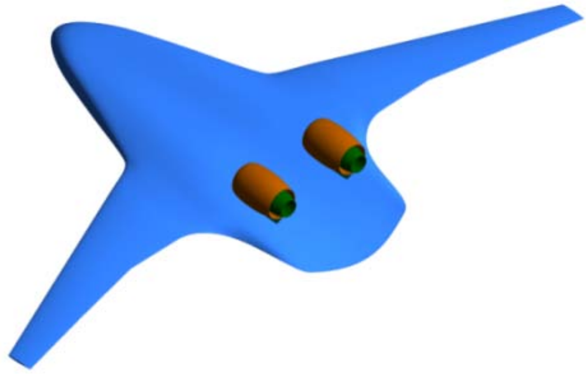


(b) Case 2

Fig.4 Cp Comparison



(a) Configuration with Power Nacelle



(b) Configuration with TFN Nacelle

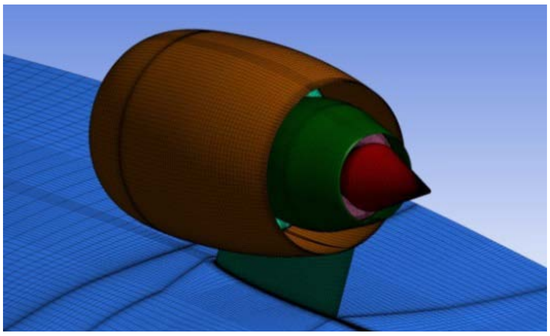
Fig.5 Calculated Two Configurations

3 Simulations of a BWB Full Aircraft Flowfield

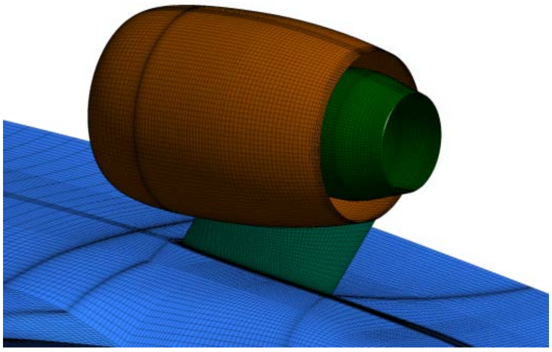
3.1 Simulation Models and Grids

Numerical simulations of a BWB layout aircraft were studied in this paper. Simulations of two configurations with power nacelle and through flow nacelle (TFN) were conducted (Fig. 5).

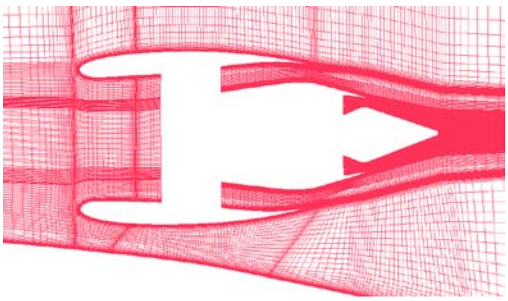
Multi-block structure grids were used in this research (fig. 6), which allowed the flexibilities of grid topologies and densities and the ability to capture complex geometries. The “O grid” was used in regions close to surfaces to maintain orthogonality and capture the boundary layers; while simple “H grid” was used in other regions.



(a) with Power Nacelle



(b) with TFN Nacelle



(c) Space Grid

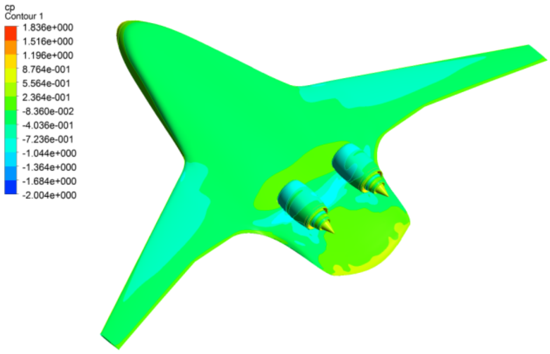
Fig .6 Grid of Two Configuration

3.2 Simulation Results

Simulations of both configurations were conducted with following working conditions: M = 0.80, Re = 4.8E7.

The surface pressure coefficients corresponding to M = 0.8, AOA = 2° were shown in fig. 7.

The polar corresponding to M = 0.8 was shown in fig. 8. Comparing the results, given the same lift, the drag of the power nacelle configuration was larger than the other. Near the cruise AOA (2 deg) the difference was about 0.0020. This difference was mainly due to the difference of the exhaust pressure between power nacelle and through flow nacelle; and it should be carefully noticed when predicting drag with CFD.



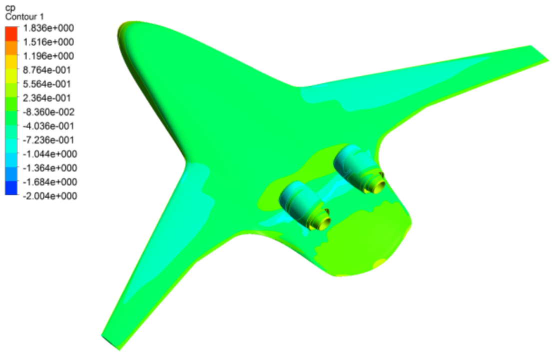


Fig. 7 Cp Contours Comparison

（M=0.8，α =2°）

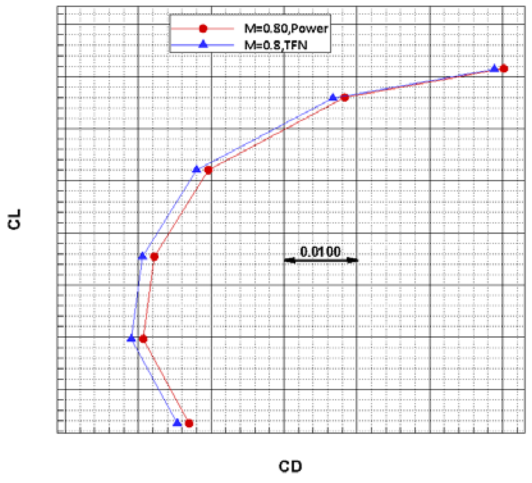


Fig. 8 Drag Polar Comparison（M=0.80）

3.3 Effects on Thrust due to Different Nacelle Height

A research about effects on thrust due to different nacelle height was also conducted. The purpose of this study is to find out the relevant regularities. Therefore, the realization of aircraft structure and layout is not considered. Some of the height changes will be very large.In order to purely study the effects from engine position, the pylons were removed. The simulation model was shown in fig. 9.

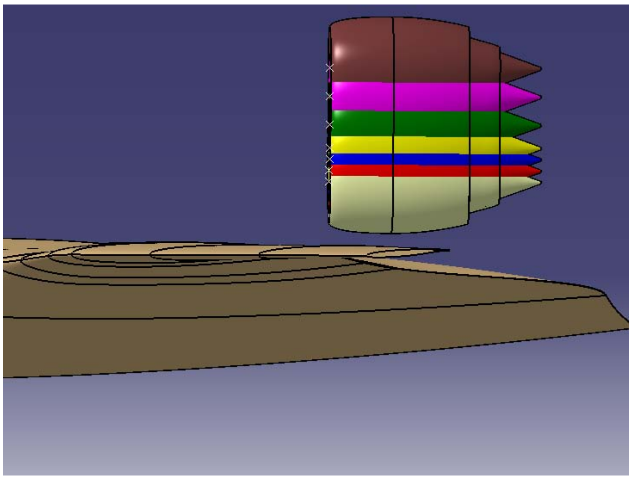


Fig. 9 Different Engine Height

The blue line in fig. 10 stood for isolated engine thrust. Engine thrust grew firstly then decayed when increased the height of nacelle. When the height is small (less than 0.5 m), engine thrust was less than the isolated value due to separation. However, when the height was large enough that the effects of separation could be ignored, fuselage conducted beneficial effects on engine which made the engine thrust larger than the isolated value. While the height kept increasing these effects became weaker so the engine thrust decayed. This phenomenon was different from aircraft with typical layout and gave a reference to further BWB designs.

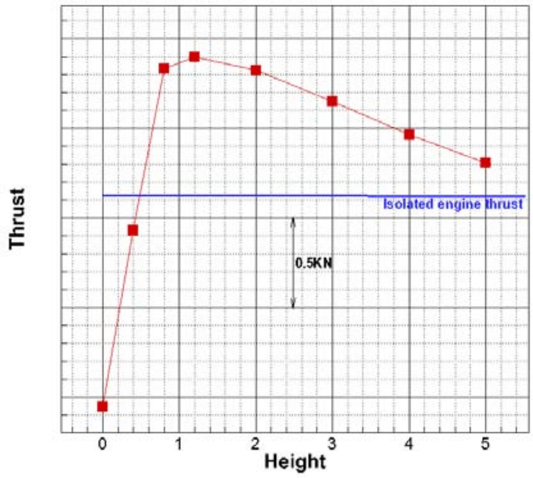


Fig. 10 Variation of Thrust with Engine Height

4 Conclusion

In this paper, the aerodynamic interference between the airframe and the engine of the aircraft is numerically studied. The main conclusions are as follows:

1) The simplified model of engine and the method of thrust drag definition can accurately simulate the power effect of the engine.

2) Aircraft drag with power nacelle will increase significantly than with TFN nacelle.

3) Thrust of the engine will also change with the height of the fuselage.

References

[1] Tan Zhaoguang, ChenYingchun, Si Jiangtao. Study of Power Influences to the Wing-Mounted Civil Aircraft Aerodynamic Characteristics [J]. Journal of Aircraft, 2014, 51(2):629-636.
[2] Tan Zhaoguang , Chen Yingchun, and LiJie .Numerical simulation method for the powered effects inairframe/propulsion integration analysis[J]. Journal of Aerospace Power. 2009,8:1766-1772
[3] Liu kaili, Ji Changrui, Tan Zhaoguang, Zhang Kunyuan, Zhang Huiliu, Si Jiangtao. Numerical Study on Low Speed Aerodynamic Performance of Large Bypass Ratio Engine TPS Nacelle [J]. Journal of Propulsion Technology. 2015,2 186-193
[4] Hirose N, Asai K, and Ikawa K. Transonic 3-D Euler analysis of flows around fan-jet engine and T.P.S. (Turbine Powered Simulator) [R], NAL-TR-1045, 1989.
[5] Li Jie, Li Fengwei,and E Qin. Numerical simulation of transonic flow over wing-mounted twin-Engine transport aircraft[J]. Journal of Aircraft,2000,37(3): 469-478.

Copyright Statement