Chinese Journal of Aeronautics, (2015), **28**(4): 1023–1033

Chinese Society of Aeronautics and Astronautics
& Beihang University
Chinese Journal of Aeronautics
cja@buaa.edu.cn
www.sciencedirect.com

Drag prediction method of powered-on civil aircraft based on thrust drag bookkeeping

Zhang Yufei [a], Chen Haixin [a,*], Fu Song [a], Zhang Miao [a,b], Zhang Meihong [b]

[a] *School of Aerospace Engineering, Tsinghua University, Beijing 100084, China*
[b] *Shanghai Aircraft Design and Research Institute, Shanghai 201210, China*

Received 30 September 2014; revised 4 December 2014; accepted 29 January 2015
Available online 24 June 2015

KEYWORDS

Discharge coefficient;
Drag prediction;
Powered-on configuration;
Thrust drag bookkeeping;
Velocity coefficient

Abstract A drag prediction method based on thrust drag bookkeeping (TDB) is introduced for civil jet propulsion/airframe integration performance analysis. The method is derived from the control volume theory of a powered-on nacelle. Key problem of the TDB is identified to be accurate prediction of velocity coefficient of the powered-on nacelle. Accuracy of CFD solver is validated by test cases of the first AIAA Propulsion Aerodynamics Workshop. Then the TDB method is applied to thrust and drag decomposing of a realistic aircraft. A linear relation between the computations assumed free stream Mach number and the velocity coefficient result is revealed. The thrust losses caused by nozzle internal drag and pylon scrubbing are obtained by the isolated nacelle and mapped on to the in-flight whole configuration analysis. Effects of the powered-on condition are investigated by comparing through-flow configuration with powered-on configuration. The variance on aerodynamic coefficients and pressure distribution is numerically studied.

1. Introduction

The powered-on nacelle has a significant effect on aerodynamic characteristics of civil aircraft. The engine jet could reduce the pressure on the wing's lower surface[1] and induce interference drag. The shock location of the wing on its upper surface might be changed by the powered-on nacelle[2] and increase wave drag. The existence of the airframe could also alter the performance of the engine, introducing additional thrust loss. In the civil aircraft design practice, the thrust drag bookkeeping (also called thrust drag accounting, TDB) procedure is necessary to decompose the thrust of the exhaust system and the drag of the airframe and to point out the source of interference.[3] Usually, the airframe and engine are designed and manufactured by different companies in the modern aircraft industry. When integrated, the interference between exhaust system and airframe could induce significant drag.[4] Accurately and rationally predicting and decomposing the performances of the airframe and engine, as well as their interference effect, is important to improve the propulsion/airframe

---

* Corresponding author. Tel.: +86 10 62789269.
E-mail addresses: zhangyufei@tsinghua.edu.cn (Y. Zhang), chenhaixin@tsinghua.edu.cn (H. Chen).

Peer review under responsibility of Editorial Committee of CJA.

ELSEVIER Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.cja.2015.06.015
1000-9361

integration design, and on the other hand, split the responsibilities and contributions of the two sides.

Flight test[5] and turbofan powered simulator (TPS)[6–8] are reliable methods for evaluating the interference drag and do TDB for realistic civil aircraft. The flight test can only be used after the aircraft is produced and the cost is very expensive. The TPS test for full aircraft configuration is also expensive. And the TPS test conditions, such as the fan pressure ratio and the Reynolds number, sometimes may not be completely consistent with the real fight condition.

The nozzle internal drag is a main source of thrust loss and is a primary issue of TDB. The quantity of the nozzle internal drag is about 1.5%–2.0% of the engine thrust.[9] It is a big value as it is equivalent to about 15–20 drag counts of drag coefficient (1 drag count = 0.0001). Interference among the engine jet, pylon and the wing also induces about 0.3%–0.9% thrust loss.

The fundamental theory of TDB is the control volume analysis method of fluid mechanics.[10] By integrating the momentum variations of the nacelle fan nozzle and core nozzle, the internal drag of the nozzles, which can cause the thrust losses, will be obtained.[11] Two dimensionless parameters, discharge coefficient and velocity coefficient, C_d and C_v, are often used to indicate the performance of nacelle nozzle.

The accuracy of velocity coefficient is important for the nacelle thrust loss calculation. Wright's error estimation[12] shows that a 0.1% uncertainty of velocity coefficient could cause a 5% uncertainty of internal drag for a typical nacelle with a velocity coefficient around 0.98. If the velocity coefficient of the nacelle is 0.99, 0.1% uncertainty could cause a 10% uncertainty on internal drag.

In industrial applications, the nozzle internal drag is usually measured by flight simulation chamber on the ground and in static air.[9,13] The measured C_d and C_v results are assumed to be only varying with nozzle pressure ratio and then used directly in the flight condition, where the thrust of the exhaust system is calculated by the ideal thrust of isentropic expansion subtracting the thrust losses of the nozzles.[14]

The computational fluid dynamics (CFD) method is an appropriate method for TDB and propulsion/airframe integration in the aircraft design process. In the Boeing company, the propulsion effect had been considered even in their panel method code in the 1980s.[15] In recent years, the TDB based on CFD method is going through a rapid development in the aircraft design process, such as computing the thrust loss of nacelle chevron[16,17] and validating the engine efficiency.[18]

The key issue of CFD-based TDB is the prediction of the nozzle performance coefficients. Earlier results of CFD showed that[14,19] for three-dimensional exhaust nozzle configurations, velocity coefficient had a typical accuracy in the range of 0.5% to 1.0%. This accuracy is not quite adequate for modern aircraft design. The CFD method has received its rapid development in recent years, as the computational resources go through an explosive growth. Numerical schemes and turbulence models are also greatly improved. The American Institute of Aeronautics and Astronautics (AIAA) held the first Propulsion Aerodynamic Workshop (PAW 1) in 2012.[20] The basic objective is to evaluate and improve the state-of-art of nozzle performance prediction.[21,22] A series of nozzle test cases with experimental data was used for CFD verification and validation in the PAW 1.

In this paper, a drag prediction process fully based on Reynolds averaged Navier–Stokes (RANS) computation is introduced. The process is first derived by applying the control volume theory on a powered-on nacelle. CFD code's accuracy is then validated by the nozzle test cases of the PAW 1. Finally the whole method is applied to the thrust and drag decomposition of a realistic configuration of a civil aircraft.[23] Variation of the nacelle velocity coefficient and the influence of the engine jet on the wing characteristics are both investigated.

2. Control volume analysis of a powered-on nacelle

2.1. Flow regime definition

Fig. 1 shows the control volume usually used for a powered-on nacelle.[11] The planes E0, E9 and E19 represent the far field boundaries. Plane E1 is the nacelle inlet lip surface. Plane E12 is the fan face station. Planes E7 and E17 are the core nozzle entrance (or core turban exit station) and fan nozzle entrance (or fan exit station), respectively. Planes E8 and E18 are the nozzle exit planes of the core and the fan nozzle. Such a station number designation follows the conventional way in the engine industry.[3] The V_{Pre} and V_{Post} are the pre-entry control volume and post-exit control volume.

The whole computation domain can be divided into drag domain and thrust domain. The inflow stream tube from plane E0 to E1 and the fan jet stream tube from E18 to E19, as well as the engine's external surface, are used as the borders separating the two domains. If the inflow stream tube is not straight, there will be a pressure force Φ_{Pre} acting on this pre-entry tube and such a force will contribute to the control volume V_{Pre}. Similarly post-exit stream tubes are coaxially formed by the core jet and fan jet, which are from E8 to E9 and E18 to E19, respectively. If the jet goes through an expansion or a contraction, the tube is not straight and a pressure force Φ_{Post} will exert effects on V_{Post}.

On the external surface of the nacelle, the summation of pressure force and viscous force is named as Φ_{Ext}. Φ_{Aft} is the resultant of the pressure and friction forces which exert on the exposed engine cowl and plug surfaces by the fan and core jets after E8 and E18 planes.

2.2. Force analysis of control volume

Define F_8 and F_{18} as the overall gross thrusts of the E8 and E18 control planes. The "thrusts" are formed by momentum forces and pressure forces. Similarly, define F_1 as the gross force on the inlet lip plane. The expressions of the forces are shown in:

$$\begin{cases} F_8 = \dot{m}_8 u_8 + (P_8 - P_{\text{amb}})A_8 \\ F_{18} = \dot{m}_{18} u_{18} + (P_{18} - P_{\text{amb}})A_{18} \\ F_1 = \dot{m}_1 u_1 + (P_1 - P_{\text{amb}})A_1 \end{cases} \tag{1}$$

where the subscripts represent the respective stations on the control volume in Fig. 1, the subscript "amb" represents the ambient air condition, $\dot{m}$ is the mass flow rate, u the flow velocity, P the pressure and A the area.

On the far field planes E0 and E9 + E19, the static pressures are equal to the ambient pressure. By the momentum balance and mass flow conservation on the control volumes V_{Pre} and V_{Post}, we can get

$$F_8 + F_{18} + \Phi_{\text{Post}} + \Phi_{\text{Aft}} = \dot{m}_8 u_9 + \dot{m}_{18} u_{19} \tag{2}$$