Numerical Simulation Method with Engine Power and Analysis of Jet Interaction

Jialei Yu*, Tiejun Liu, Tuliang Ma and Chen Zhai
Shanghai Aircraft Design and Research Institute, Shanghai 201210, China

*Corresponding author: yujialei@comac.cc
liutiejun@comac.cc, matuliang@comac.cc, zhaichen@comac.cc

Abstract. To explore the effects of engine jet on overall aircraft aerodynamic performance, an aerodynamic numerical simulation method and the interference effects of aircraft with wing-mounted nacelle were studied. In aspect of numerical simulation, based on isentropic flow relationship and automatic mass-flow matching method, a high-precision / high-efficiency numerical simulation of inlet and exhaust boundary conditions is achieved. The calculation result is in good agreement with the experimental values, which validates the calculation method and boundary condition treatment. Through numerical simulation, it is found that the ejection of engine jet can accelerate the air flow, reduce the lower wing pressure, enhance the upper wing shock wave and increase the pitch down moment. At the same time, the effect of jet brings blowing drag and loses the cruise efficiency. The airflow acceleration caused by the narrow channel composed of engine, wing and pylon may cause strong aerodynamic interference. It can be effectively weakened by adjusting the pylon contraction shape based on the near-field pressure distribution. In this way, the shock in the inner side of the pylon can also be eliminated.

1. Introduction
In the design of modern civil airliner, it is very important to evaluate the interaction between wing and engine for improving the aerodynamic performance of the aircraft. Jet interference may destroy the ideal pressure distribution of the wing, resulting in deterioration of the whole aircraft performance. In jet calculation, the drag and thrust are coupled, which brings great difficulties to aerodynamic analysis. NASA Langley Research Center has conducted a lot of research on the interference drag by using turbine power simulator with experimental method[1]. In terms of numerical simulation, based on the far-field method, Destarac D[2] proposed a Thrust-Drag Bookkeeping method based on the CFD results with power nacelle. In order to improve the comprehensive performance of aircraft and engine, this paper carries out jet interference evaluation and optimization design method, so as to reduce the loss of aerodynamic performance as much as possible and obtain the best cruise efficiency.

2. Numerical Simulation Method
Due to the complexity of engine combustion and working process, appropriate boundary conditions are usually set at the engine intake/exhaust port to make the engine intake and exhaust effect consistent with the actual situation, instead of simulating the inner flow of the engine. In order to study the aerodynamic effect of turbofan engine of Civil Aircraft, based on the principle of isentropic flow, the total temperature ratio/total pressure ratio is set at the exhaust port and the MFR(Mass Flow Rate)
is set at the engine intake. The static pressure condition is adopted for the engine intake, and the target MFR of the intake boundary is given. In the calculation process, the static pressure is adjusted according to the mass flow condition of the interface, so that the actual MFR converges to the defined target MFR as soon as possible. The Thrust-Drag Bookkeeping method is used to clearly distinguish the aerodynamic drag acting in the system from the thrust of the engine[3].

3. Validation Of Numerical Calculation

3.1. Turbine Powered Simulator
T.P.S.(Turbine Powered Simulator) is the wind tunnel model of "NAL-AERO-02-01" from Japan Aerospace Technology Research Institute[4].Its calculation grid is shown in Figure 1. The calculation states:Mach Number 0.801, Reynolds number 1.8E7 and the angle of attack 0 degree. See Table 1 for engine intake and exhaust boundary conditions, the MFR is 0.497, the TR(total temperature ratio) of the external duct is 1.109, the PR(total pressure ratio) of the external duct is 1.343, the TR of inner duct is 0.612, and PR is 0.92126.

![Figure 1. Calculation grid of T.P.S.](image)

|                      | MFR  | PR  | TR   |
|----------------------|------|-----|------|
| Intake               | 0.497| -   | -    |
| External duct        | -    | 1.343| 1.109|
| Inner duct           | -    | 0.92126| 0.612|

Table 1. Intake and exhaust boundary conditions of T.P.S.

Figure 2 shows the comparison between the calculated value of surface pressure and the experimental value. It can be seen that the calculated value is in good agreement with the experimental value.

![Figure 2. Pressure distribution of T.P.S.](image)
3.2. Numerical Simulation of DLR-F6

The DLR-F6 is a classical numerical simulation example with TFN (through-flow nacelle). Firstly, the no-power numerical simulation is carried out by using the TFN and compared with the experimental results [5] to verify the accuracy of the numerical method. Then, the intake and exhaust boundary conditions are added in the nacelle for jet simulation.

3.2.1. Numerical Simulation of DLR-F6 with TFN. The calculation grid is shown in Figure 3. The calculated states: Mach number 0.75, angle of attack 1 degree, Reynolds number 3 million.

![Figure 3. Calculation grid of DLR-F6.](image)

Figure 4 shows the comparison between the CFD value of surface pressure and the experimental value. It can be seen that the CFD value is in good agreement with the experimental value, indicating that the numerical calculation method used in this paper is reliable.

![Figure 4. Comparison between CFD results and test results of DLR-F6.](image)
3.2.2. Numerical Simulation of DLR-F6 with Power Nacelle. Based on DLR-F6 nacelle, a cross section is artificially selected and the engine intake and exhaust boundary conditions are given. The calculated states: Mach number 0.75, angle of attack 1 degree, Reynolds number 3 million. In the no-power state, the MFR is equal to the power state, and the total temperature and total pressure of the exhaust port is set as the total temperature and total pressure of the incoming flow. See Table 2 for engine intake and exhaust boundary conditions.

**Table 2.** Intake and exhaust boundary conditions of DLR-F6.

|                | MFR | PR  | TR  |
|----------------|-----|-----|-----|
| No-Power State | 0.4 | 1.452 | 1.112 |
| With-Power State | 0.4 | 2.134 | 1.254 |

Figure 5 shows the Mach number distribution at the meridional plane. It can be seen that the Mach number distribution is reasonable. Near the exhaust exit, a strong shear flow is formed between the external air flow and the exhaust air flow. The Mach number of the exhaust air flow gradually approaches the incoming Mach number in the far field with the diffusion of the flow. The calculation results in Figure 6 show that the jet effect also brings blowing drag and loses cruise efficiency.

Figure 5. Mach number contour comparison of DLR-F6 between no-power and with-power state.

Figure 6. Drag polar comparison of DLR-F6 between no-power and with-power state.
3.3. Analysis and Optimization of Jet Interference

A wide-body civil aircraft is used to study jet interference. The calculation state: Mach number 0.85, CL 0.5, Reynolds number 5 million. Figure 7 is the calculation grid, and the intake and exhaust boundaries are shown in Table 3.

![Figure 7. Calculation grid of wide-body civil aircraft.](image)

### Table 3. Intake and exhaust boundary conditions.

|       | MFR | PR  | TR  | PR  | TR  |
|-------|-----|-----|-----|-----|-----|
| No Power | 0.606 | 1.505 | 1.145 | 1.505 | 1.145 |
| With Power | 0.606 | 2.127 | 1.219 | 1.725 | 2.906 |

The jet effect causes the shock position of the upper wing surface to move forward, and the shock intensity increases. The impact on the lower wing surface is mainly concentrated in the area near the nacelle, which decreases with the increase of the distance from the section position to the engine. Under the influence of the engine jet, Figure 8 shows the velocity at the inner side and outer side of the pylon. The engine jet causes airflow interference on the inner side of the pylon, but this phenomenon does not occur on the outer side of the pylon. The main reason for this difference is the existence of the fuselage, the sweep/dihedral angle of the wing and the installation mode of the nacelle, the flow channel between the pylon/inner wing/engine nacelle is narrower than that on the outside.

![Figure 8. Velocity comparison between the inner side and outer side of the pylon.](image)

Based on the near-field velocity distribution, this paper proposes an engineering design method to adjust the contraction shape on the inner side of the pylon. Optimized modification of pylon inner side is shown in Figure 9(Solid line: Original shape; Dotted line: Optimize shape). At the shock wave position on the lower surface of the wing, expand the local flow channel and increase the local pressure. The result in Figure 10 shows this method effectively weakens the jet interference and the pressure distribution on the lower wing surface is obviously improved.
4. Conclusions
In this paper, the practicability and efficiency of the jet boundary treatment method is illustrated by the numerical simulation of the T.P.S and DLR-F6. The jet influence of a wide body aircraft is studied, and it is clear that the pylon aerodynamic interference comes from the narrow channel composed of engine, wing and pylon. An engineering design method for adjusting the pylon contraction shape based on near-field velocity distribution is proposed. This method can effectively weaken the aerodynamic interference of jet effect, which can provide certain reference data and theoretical basis for the aerodynamic optimization design of large passenger aircraft.

Acknowledgments
Sponsored by Natural Science Foundation of Shanghai, No.20ZR1470700.

References
[1] Henderson W P, Patterson. Propulsion installation characteristics for turbofan transports[J]. aiaa journal, 1983.
[2] Destarac D, Vooren J. Drag/thrust analysis of jet-propelled transonic transport aircraft; Definition of physical drag components[J]. Aerospace Science & Technology, 2004, 8(6):545-556.
[3] Kimzey W F, Covert E E, Rooney E C, et al. Thrust and Drag: Its Prediction and Verification || Summary and Conclusions[J]. 1985.
[4] Hirose N, Asai K, Kawamura R. 3D-Euler flow analysis of fanjet engine and turbine powered simulator with experimental comparison in transonic speed[C]/ 20th Fluid Dynamics, Plasma Dynamics and Lasers Conference. 2013.
[5] Laflin K R, Klausmeyer S M, Zickuhr T, et al. 42 nd aiaa aerospace sciences meeting and exhibit summary of data from the second aiaa cfd drag prediction workshop (invited).