Numerical and Experimental Analysis of Subsonic and Transonic Base Pressure

Wang Xiaobing¹, He Binhua¹*, Zhang Hong², Zhao Zhongliang¹, Li Yuping¹, Chen Jianzhong¹
1. China Aerodynamics Research and Development Center, Mianyang, China
2. Chengdu Aircraft Design & Research Institute, Chengdu, China

wangxbice@163.com

Abstract. Base drag of space vehicles with large, truncated base usually forms a considerable component of overall drag. The base drag is usually difficult to be accurately predicted due to the intrinsic complexity of base flow. The subsonic and transonic base pressure characteristics were studied by large scale wind tunnel test and Reynolds-Averaged Navier-Stokes/Large Eddy Simulation (RANS/LES) hybrid numerical simulation. The test results show that interferences of both tail sting and ventral blade varies evidently with Mach numbers. The RANS/LES hybrid method was proved to be effective as the calculated time-average base pressure and base drag agree well with wind tunnel test, and detailed flow structures were successfully captured. The base flow pattern is complicated in sub- and transonic regime that either single support scheme is not sufficient to obtain an accurate measurement of base drag. Various support schemes and high precision numerical method are recommended in similar studies.

1. Introduction
Accurate overall drag of space vehicles is needed in flight trajectory simulation and design, especially for the long-time, unpowered glide-back process. The large and truncated base usually produce a base drag which may be even larger than forebody drag, but more difficult to be predicted as the base flow includes complicated flow structures like unsteady detached-eddy shedding, vortices interaction of various scales, and existence of shock waves and expansion waves. Mostly, the wind tunnel based base pressure measurement is subjected to interferences of support schemes, the accuracy of computational fluid dynamics (CFD) result, unfortunately, is also restricted by grid quality, discretization scheme and turbulence closure.

As one of the most typical problem in aerodynamics, base flow has been studied over half a century. Among all the methods, the semi-empirical analysis should be mentioned at first, which is probably the oldest and most classic. Back in 1950, Chapman⁴ developed an approximate semi-empirical theory for base pressure in a viscous fluid for some simple shapes, and studied the effects of Mach number, Reynolds number, profile shape and the type of boundary-layer flow. The AeroPrediction Code (APC), after several decades of development, has already included the base drag computation methodology for axisymmetric bodies and blunt trailing edge fins, with corrections for effect of fins and for sub- and transonic speeds⁵. Most methods based on semi-empirical theory are restricted by aerodynamic configuration and the estimation error will increase near transonic speed.

Ground simulations including wind tunnel experiment and CFD are common methods nowadays for base pressure analysis and base drag prediction. Paciorri et al⁶ investigated the base flow of a space launcher during the early ascent phases in a subsonic wind tunnel. McArthur et al⁷ investigated the...
effect of rotating cylinder combination at the lee-ward four edges of the three-dimensional bluff body in ground proximity using both wind tunnel experiment and CFD. Guo et al.\textsuperscript{[5]} studied the effects of the wings on the revolution body base flow-field and base drag using pressure measurement and schlieren visualization in a wind tunnel. Khankhasaeva et al.\textsuperscript{[6]} numerically studied the supersonic base flow of an aircraft model, by unsteady Reynolds averaged Navier-Stokes equations (URANS) with the Spalart-Allmaras (S-A) and Menter’s SST turbulence model. However, most studies were focusing on simple shapes (e.g., slender bodies of revolution and blunt landing capsules) under low speed or supersonic and hypersonic speed, but few on complicated space vehicles in sub- and transonic regime. In present research, a typical winged re-entry space vehicle was chosen to study the subsonic and transonic base pressure characteristics using wind tunnel experiment and CFD. The support interferences of wind tunnel experiment were investigated and detailed flow patterns by CFD were analysed.

2. Research Methodology

2.1. Experiment setup

The base pressure measurement was accomplished in the 2.4m Transonic Wind Tunnel of CARDC, an intermittent, injection driven wind tunnel with the test section of 2.4m×2.4m. The Mach numbers of the experiment ranged from 0.3 to 1.2, and the model was scaled properly to suit the test section. Tail and ventral support schemes were employed, as well as ventral support with dummy tail sting, which are given in Figure 1. Over 100 taps were distributed on the base, with small area elements divided. Figure 2 gives different pressure taps distribution for different support schemes. The mean base pressure level was therefore calculated by area weighted averaging. As for cases the tail sting exists, six extra taps were distributed in the afterbody cavity to calculate the pressure around the sting. Equation 1 shows how to obtain the base drag coefficient from the measured pressure.

$$C_{D_b} = \frac{\cos \alpha \cos \beta}{qS_{ref}} \cdot \left\{ 2 \cdot \sum \left( p_\infty - p_i \right) \cdot A_i + \left( p_\infty - p_{sting} \right) \cdot A_{sting} \right\}$$  

(1)

Where $C_{D_b}$ is the base drag, $\alpha$ is the angle of attack, $\beta$ is the sideslip angle, $q$ is dynamic pressure, $S_{ref}$ is the reference area, $p_\infty$ is the static pressure, $p_i$ and $A_i$ are the measured pressure and area of element $i$ of the base, $p_{sting}$ is the measured pressure around the tail sting, and $A_{sting}$ is the cross section area of the sting.

(a) Tail support  
(b) Ventral support  
(c) Ventral support with dummy sting

Figure 1. Support schemes for wind tunnel experiment
2.2. Computational method

The CFD grid includes several type of cells: Anisotropic hexahedron cells for boundary layer flow simulation, Cartesian cells for domains away from the wall to decrease errors caused by grid direction inconsistency, tetrahedron cells for linking up hexahedron and Cartesian cells to ensure flux conservation. The base region was locally refined to capture detailed flow structures (Figure 3). The limited-numerical-scales (LNS) method by Batten and Goldberg\cite{[7]} was adopted, which uses RANS model with nonlinear $k$-$\varepsilon$ turbulence closure in boundary layers, and blends to an anisotropic form of the Smagorinsky model in regions of uniformly-refined mesh to achieve LES calculation, so that accuracy can be promoted without evident increase in calculating load.

Figure 3. CFD grid of base flow region (Upperside: no support; Lowerside: tail support)
3. Results and Discussion

3.1. Experiment results
Provided that the secondary interferences are negligible, influence quantities caused by tail support and ventral support can be obtained via Equation 2 and 3 respectively:

\[ \Delta C_{Db,\text{tail}} = C_{Db,\text{ventral+tail}} - C_{Db,\text{ventral}} \]  
\[ \Delta C_{Db,\text{ventral}} = C_{Db,\text{ventral+tail}} - C_{Db,\text{tail}} \]  

Where \( \Delta C_{Db} \) is the support interference of base drag coefficient, and the subscripts denote different support schemes.

The angles of attack for following results are all 0°. Figure 4 shows the calculated support interferences. The x-axis is Mach number and the y-axis is the change in base drag caused by different support schemes. The solid squares represent the ventral support, and the hollow triangles represent the tail support. Generally, the ventral support makes base drag larger, while the tail support makes base drag smaller. Furthermore, some other conclusions can be draw: the magnitudes of both types of support interferences are approximately equal for subsonic speed, but for Mach numbers close to 1.0, the effect of ventral support reaches to the minimum value of approximately zero, while that of tail support reaches to its maximum. When Mach numbers exceeds 1.0, however, the impact of ventral support increases abruptly, as the shock wave induced by the support blade diminishes the overall base pressure level evidently. The tail support interference decays to zero when Mach number reaches 1.2 as the supersonic speed can remarkably reduce the backfield interference. Overall, it can be considered that the adoption of ventral support increases the base drag mainly at \( M \geq 1.0 \), and the tail support, however, makes base drag smaller by changing the base shape and producing extra high pressure region around the tail sting when \( M = 0.3~1.05 \).

![Figure 4. Support interference by wind tunnel experiment](image-url)

Figure 5 gives the base pressure distributions of different types of support schemes and different Mach numbers, from the wind tunnel experiment. Firstly, we can obviously see that when the base surface is changed by the tail sting, the base pressure distribution is different as the pressure level around the tail sting is increased, so that the tail sting decreases the base drag. For Mach number equals 1.05, the base pressure level is decreased evidently by the ventral blade. For Mach number equals 1.2, although the pressure distribution is changed by tail sting, the averaged base pressure remains unchanged, so the tail support interference is quite small.
3.2. **Computational results**

Cases without support and with tail support at different Mach numbers were calculated among which the physical time step was set to $1 \times 10^{-4}$ s.

Figure 6 gives the base drag converge history over a certain time period, which indicates that the base drag fluctuates with time by RANS/LES hybrid method and remains stable by unsteady RANS method. Moreover, from Figure 7 we can concluded that the time-averaged base drag calculated by RANS/LES hybrid method agree better with experimental results than results by unsteady RANS method.

**Figure 5.** Base pressure distribution

**Figure 6.** Base drag history over time step

**Figure 7.** Time-averaged results of different CFD methods comparing with experimental data
Figure 8. Base flow pattern by different methods (Y vorticity)

Figure 8 gives the base flow pattern by unsteady RANS method and RANS/LES hybrid method, respectively. Comparing to the traditional unsteady RANS method, the RANS/LES hybrid method can obtain more detailed flow structures, including lots of unsteady, small-scale eddies. Using the time averaged results of RANS/LES simulation, a comparison was made by the numerical simulation and wind tunnel experiment. Figure 9(a) shows the base drag over Mach numbers, and Figure 9(b) shows the base pressure distribution. Both pictures indicate that the numerical results agree well with experimental results within Mach numbers between 0.3–1.05 despite some slight differences.

Figure 9. Validation of CFD with wind tunnel based base pressure measurements

More transient base flow patterns are given in Figure 10, with several Mach numbers, that contribute to further analysis. The geometric corner of the model induces flow separation and forms a recirculation zone, including unsteady, small-scale eddies. A free shear layer separates the recirculation zone and the outer flow region. As the flow develops downstream, the free shear layer becomes unstable and gradually grows into unsteady wake flow. Expansion waves can be seen at high Mach numbers. As Mach number increases, the free shear layer shrinks towards the middle and the recirculation zone becomes smaller, making expansion waves stronger, and base pressure lower. The body flap alleviates the streamline curvature, so that expansion waves of the lower part are relatively weaker.
**Figure 10.** Mach number contours of base region, by CFD
4. Conclusion

Following conclusions can be drawn from present study:

a. The wind tunnel support interferences show significant changes with Mach numbers. Generally, ventral support increases base drag mainly at Mach numbers above 1.0, while tail support decreases base drag dramatically until Mach number reaches 1.2.

b. The numerical simulation by RANS/LES hybrid method is proved to be effective when it obtained results that agree well with experimental results, meanwhile captured unsteady flow structures like small-scale eddies, free shear layer destabilization and expansion waves.

c. The base flow pattern is complicated in sub- and transonic regime that either single support scheme is not sufficient to obtain an accurate measurement of base drag. It is recommended to use various support schemes (including dummy tail sting) and high precision numerical method in similar research.

5. References

[1] Chapman D R. An analysis of base pressure at supersonic velocities and comparison with experiment. NACA TN 2137, 1950.

[2] Moore F G, Moore L Y. Improvements to Power-On Base Pressure Prediction for the Aeroprediction Code. Journal of Spacecraft and Rockets

[3] Paciorri R, Sabetta F, Valenza F, Faucci R, Passaro A, Baccarella D. Base-Pressure Experimental Investigation on a Space Launcher in Subsonic Regime. Journal of Spacecraft and Rockets, 2013: 50(3).

[4] McArthur D, Burton D, Bell J R, Sheridan J. Numerical and Experimental Investigation of the Effect of Multiple Rotating Cylinders on Base Pressure of a Three Dimensional Bluff Body in Ground Proximity. 30th AIAA Applied Aerodynamic Conference, AIAA 2012-2656.

[5] Guo H, Li S. Effect of Wings on the Base Flow of Bodies of Revolution. Acta Aerodynamica Sinica, 1994: 12(3)

[6] Khankhasaeva Y V, Borisov V E, Lutsky A E. Numerical Study of the Supersonic Base Flow around Aircraft Model. IOP Conf. Series: Journal of Physics: Conf. Series 1009, 2018.

[7] Batten P, Goldberg U, Chakravarthy S. Interfacing statistical turbulence closures with large-eddy simulation. AIAA J, 2004,42(3): 485-492.

[8] Franck S, Sébastien D, Philippe G, Roxan C. Numerical simulations of projectile base flow. AIAA-Aerospace-Sciences-Meeting-and-Exhibit-44th-Reno-NV-Jan, 2006.

[9] Makoto U, Yukimitsu Y, Masaaki Y, Michel L, Jean O. Assessment of experimental and computational transonic base pressure using flight data. Journal of Spacecraft and Rockets, 2007, 44(6): 1241-1249.

[10] Jean-François D, Klaus A H. Computations of Turbulent Flow over a Backstep. AIAA-Fluid-Dynamics-Conference-and-Exhibit-37th-Miami-FL-June, 2007.

[11] Tao Y, Li Y, Zhang Z, Zhao Z, Liu Z. Transonic wing stall of a blended flying wing common research model based on DDES method, Chinese Journal of Aeronautics, 2016, 29(6): 1506–1516

[12] Tao Y, Liu Z, Xiong N, Sun Y, Lin J. Optimization of positional parameters of close formation flight for blended wing-body configuration. Heliyon 4 (2018) e01019.doi: 10.1016/j.heliyon.2018. e01019