Computational Analysis of the Flow Field Near the Boat-tail Region of Annular Plug Nozzles*

Kozo FUJII**, Kazuhiro IMAI***
and Tetsuya SATO**

Flow fields over annular plug nozzles are computationally simulated and the boat-tail drag characteristics are clarified and discussed based on the simulation results. The plug nozzle configuration is taken from the ATREX (Air Turbo Ramjet) engine under development at the ISAS. The simulations are carried out for the axi-symmetric nozzle configuration using compressible Navier-Stokes equations. The computed result shows that there exists a low pressure region downstream of the Prandtl-Meyer expansion on the boat-tail region and the separation shock wave appears due to the interaction of the main flow and the nozzle plume downstream. The result also shows that the primary cause of the boat-tail drag turns out to be the low pressure region downstream of the Prandtl-Meyer expansion. The flow field with the secondary flow injection is then simulated and the computed result shows that the pressure in the region in front of the separation shock wave increases and the secondary flow injection successfully reduces the boat-tail drag. The effect of the turbulence model used in the computation is discussed to ensure the reliability of the solutions.

Key Words: Spacecraft, Plug Nozzle, Compressible Flows, Computational Fluid Dynamics

1. Introduction

One of the current interests of future space transportation systems is a reusable TSTO, and the key technology of successful TSTO's may be highly efficient air-breathing engines creating large thrust for wide range of flow conditions. Therefore, analysis of the nozzle region is very important for a good design of such engines and eventual success of the next-generation space transportation systems.

Under such considerations, the Air Turbo Ramjet engine (so called ATREX engine) is under development at the Institute of Space and Astronautical Science (ISAS) for the possible future use as an

engine of the flying-back booster of TSTO (Two-stage to Orbit) transportation systems⁽¹⁾. One of the current problems of this Ramjet engine is the amount of drag induced by the boat-tail region. At transonic or low supersonic speeds, the boat-tail drag may go up to more than 20% of the total thrust. Figure 1 shows the schematic picture of the boat-tail region of the ATREX engine. Plug nozzle is used to achieve high thrust for the entire flight path, and there occurs a low-pressure region at the boat tail between strong expansion of the free-stream and the plume boundary from the cowl lip. This low-pressure region causes, so called "boat-tail drag". CFD analysis of the flow fields has been conducted for the last two years to clarify the flow mechanism and develop a better design of the boat-tail configuration.

In the present paper, some of the CFD results obtained by the boat-tail flow simulations are presented and are discussed for the clear understanding of the flow mechanism of the boat-tail drag. The secondary flow injection is tried to reduce the boat-tail drag, and the effect is discussed based on the computed results for the plug nozzle with and without the secondary

* Received 20th February, 2002 (No. 02-4031)

** The Institute of Space and Astronautical Science, Sagami-hara, Kanagawa 229-8510, Japan. E-mail: fujii@flab.eng.isas.ac.jp

*** Department of Aeronautics and Astronautics, School of Engineering, University of Tokyo, 7-3-1 Hongo, Bunkyo-Ku, Tokyo 113-8656, Japan. (Currently) Aerodynamics Research Section, Nagoya Aerospace Systems, Mitsubishi Heavy Industries Ltd.

Fig. 1 Boat-tail region of the ATREX plug nozzle

flow injection. The results are applicable to any types of boat-tail configurations, and therefore give important information for engine designs.

2. Computational Approach

The basic equations are three-dimensional thin-layer Navier-Stokes equations. The three-dimensional equations are used although the computations in the present paper are restricted to be axis-symmetric flows as future applications include the cases at angles of attack and the cases of cluster-type secondary flow injections. Thin-layer approximation is used to avoid the increase of the computer time as our experience for many applications showed that the computed results with the full viscous terms gave no different solution from the solution with thin-layer viscous terms under the grid resolution currently available. The computation for the full viscous terms would increase the total computer time about 10% more in our computer program, which is not negligible. The convective terms are computed by the SHUS scheme (one of the monotone AUSM-type schemes) developed by Shima and Jonouchi⁽²⁾ and the extension to high-order space accuracy is achieved by the MUSCL interpolation⁽³⁾. The viscous terms are evaluated by the central differencing and Baldwin-Lomax algebraic turbulence model⁽⁴⁾ is mainly used. The effect of the turbulence model is discussed later. Only the steady-state solutions are considered and the LU-ADI factorization implicit time integration algorithm⁽⁵⁾ is used. The computer code used in this research has been used for a wide variety of CFD applications and was validated by the comparison with experiments^{(6),(7)}.

Body geometry and the computational grids are shown in Figs. 2 and 3. Two grid systems; one for the internal region of the nozzle ($101 \times 17 \times 81$) and the other for the outer flow region ($347 \times 17 \times 151$), are used as the overset grids. The flow is assumed to be axis-symmetric and only the region of 6.5 degrees in

Fig. 2 Computational grids (outer region)

Fig. 3 Computational grids (inner nozzle region)

the circumferential direction is considered with the periodic boundary condition at the circumferential boundary. The arrow in Fig. 3 is the direction and the location of the secondary flow injection. The basic Navier-Stokes equations are modified for the information exchange between each grid zone. The interface method used in the present study is called "Fortified Solution Algorithm (FSA)" where source terms are added to the basic equations as forcing terms to enforce the solutions for the other zones. The details of the algorithm can be found in Ref. (8).

The geometry of the nozzle and the body is taken from the experiment conducted at the ISAS for the ATREX engine development. The design pressure ratio is 9.86 and the nozzle area ratio is 1.92. The free-stream Mach number is 1.5 and the Reynolds number is 1.37×10^7 based on the body length for the results presented in this paper. The ratios of the chamber total pressure for the internal nozzle flows and the free-stream static pressure are 4.5, 5.5 and 6.5. These pressure ratios are chosen as the pressure ratios would be 4 - 7 when it flies at subsonic, transonic to low supersonic speeds and the problem of the boat-tail drag occurs in this flow regime. Note that the experiment was also conducted under these conditions. The flow field details are mainly discussed for the pressure ratio 6.5 as the length of the manuscript is restricted.

The non-slip boundary condition is given at the body boundary. The free-stream flow with the turbu-

lent boundary layer given by the velocity distribution law of the $1/7$ th power is given near the body surface at the inflow boundary. The supersonic flow is assumed at the outflow boundary and all the physical variables are extrapolated there. In the wind-tunnel experiment, air induced from the air tank of the wind tunnel was used as the internal gas inside the nozzle. The total condition of the internal nozzle flow is given by specifying the total pressure and the density to be same as the experiment. The densities are 2.298, 3.379 and 3.807 for the total pressure (same as the pressure ratio) 4.5, 5.5 and 6.5, respectively. A part of the nozzle flow is used as the flow from the sonic nozzle of the secondary flow injection. Therefore, the total flow condition of the secondary flow injection is given as that of the internal flows. The perfect gas assumption is adopted with the ratio of specific heats, $\gamma=1.4$. All the physical variables are normalized by the their free-stream values. The viscosity coefficients are computed by the sum of the laminar and turbulent eddy viscosity coefficients. Similar expression is used for the thermal conductivity with the Prandtl number $Pr=0.72$. The formulation is same as what is used in many technical papers on the Navier-Stokes simulation of compressible flows and can be, for instance, found in Ref.(9).

3. Results and Discussions

The basic flow structure without the secondary flow injection is discussed first, and then the flow structure with the secondary flow injection is discussed next.

The computed Mach number contour plots are shown in Fig. 4 for the pressure ratio of $\xi=6.5$ with no injection. Strong expansion is observed along the boat-tail region and, as a result, a low pressure region appears. Separation shock wave appears in front of the nozzle plume and there occurs a recirculating region between the separation shock wave and the plume as shown in the close-up view in Fig. 4. Figure 5 shows the longitudinal pressure distributions over the boat-tail region and the plug. The C_p is the pressure coefficient defined by the free-stream density and velocity; $C_p = \frac{P - P_\infty}{1/2 \rho U_\infty^2}$. The pressure at the recirculating region behind the separation shock wave is lower than the free-stream pressure and it becomes a cause of the drag. However, much wider low pressure region (wide in the longitudinal direction) by the expansion appears in front of the separation shock wave, as observed in Fig. 5. Note that the body is axis-symmetric. As this wide low-pressure region is located in the outer region of the boat tail where the projected area of the body in the streamwise direction

Fig. 4 Mach number contour plots ($\xi=6.5$) with the close-up view of the velocity vector plots

Fig. 5 Pressure distributions over the boat-tail region and the plug surface

is large, this low pressure region contributes to the drag increase very much. On the other hand, the contribution by the narrow low pressure region behind the separation shock wave is small as the projected area is small. The pressure recovery of the wide low pressure region located forward turns out to be the key for the reduction of the boat-tail drag.

The close-up view of the C_p distributions near the separation shock wave region are plotted in Fig. 6 for the three pressure ratios. Separation shock wave moves slightly forward and the boat-tail drag decreases when the pressure ratio is increased. However, the C_p level of the wide low pressure region

Fig. 6 Pressure distributions over the boat-tail region and the plug surface: close-up view of the region near the injection for three pressure ratios

in front of the separation shock wave is same for all the three pressure ratios. As the body is axis-symmetric, the drag in the boat-tail region can be roughly estimated by the axial C_p distributions. By such considerations, the result seems to indicate that the boat-tail drag would become small if the separation shock wave be moved forward and the low-pressure region be made small for any pressure ratio.

The secondary flow injection was tried for the purpose of reducing the low pressure region. The total conditions of the secondary flow were set to be the same as the internal flows, and sonic condition is given at the secondary jet exit. The pressure and density at the exit can be written as follows by the isentropic expansion from the total conditions.

$$P_j = P_{0,inner} \left(\frac{2}{\gamma+1} \right)^{\frac{\gamma}{\gamma-1}}, \quad \rho_j = \rho_{0,inner} \left(\frac{2}{\gamma+1} \right)^{\frac{1}{\gamma-1}}$$

where P_j and ρ_j are the pressure and the density at the exit, and $P_{0,inner}$ and $\rho_{0,inner}$ are the pressure and the density at the total flow condition of the inner nozzle flow, respectively. The direction of the injection is set to be normal to the free-stream to avoid the direct thrust increase by the injection.

Figure 7 shows the Mach number contour plots for the case with the secondary flow injection. Compared to Fig. 4, separation shock wave moves forward and is located in front of the secondary flow injection. The boundary layer separates in front of the injection, and a new shock wave is observed. Recompression shock wave appears as the injected flow interacts with the nozzle plume, and the flow field becomes complex. The expansion of the nozzle plume is stronger due to the change of the ambient pressure. The secondary flow injection alters the whole flow field in this area. In Fig. 8, the pressure distributions

Fig. 7 Mach number contour plots ($\xi=6.5$) — with secondary injection

Fig. 8 Pressure distributions over the boat-tail region and the plug surface: with the secondary flow injection

for this case are plotted and compared to the case without injection shown in Fig. 5. The low-pressure region moves forward towards the flat part of the body when the secondary flow injection is used. The pressure upstream of the secondary flow injection increases due to the shock wave that appears upstream of the injection. The pressure in the re-circulating region downstream also increases. The boat-tail drag decreases due to the change of the wall

Fig. 9 Boat-tail drag for three different pressure ratios—comparison of with and without injection

pressure.

Figure 9 shows the comparison of the boat-tail drag with and without the injection at three pressure ratios. The definition of the boat-tail drag computation is,

$$D = \frac{\iint_A (P_\infty - P_{\text{wall}}) dA}{\frac{1}{2} \rho_\infty U_\infty^2 A}$$

Here, P_{wall} is the axial component of the wall pressure and A is the projected area of the boat-tail region to the axial direction. P_∞ , ρ_∞ and U_∞ are the free-stream pressure, density and velocity, respectively. The result shows the secondary flow injection effectively reduces the boat-tail drag at any pressure ratio. The effect of drag reduction becomes larger as the pressure ratio increases.

There would be the thrust increase when the whole mass used for the secondary injection directly flows into the plume. It should be noted that the resultant thrust increase is much smaller than the thrust increase due to the secondary flow injection for any pressure ratio. Figure 10 shows the thrust component comparison of the plug nozzle with and without the secondary flow injection. In this figure, boat-tail drag is plotted as the negative thrust and the net thrust is the total thrust minus the boat-tail drag. The result indicates that each component of the thrust does not change much and the secondary flow injection is shown to be effective for achieving higher thrust. There is pressure reduction on the plug surface due to the secondary flow injection as shown in Fig. 8, but the total negative contribution is small as the area of projection is small near the axis. In general, the secondary flow injection changes the flow mechanism, which is the main reason for the net thrust increase.

For the problem under consideration, only the

Fig. 10 Comparison of the thrust component with and without the secondary flow injection

limited experimental data is available. The experiment was conducted by Sato, et al. in 1997 at the ISAS^{(10),(11)} and the detail can be found in Ref.(12). The experiment showed that there was total pressure loss inside the test section due to the oblique shock wave at the nose and other obstacles such as air supplier. Because of those, the Mach number measured near the flat region in front of the boat-tail was 1.02 although the free-stream Mach number was set to be 1.5. For the comparison with the experiment, the computation for the free-stream Mach number 1.02 was carried out. The computed density gradient was compared with the schlieren picture taken in the experiment in Fig. 11. The qualitative features of the flow field; plume boundary and shock systems, such separation shock wave, recompression shock wave and trailing shock wave are captured in the computation although the locations of the shock waves show some disagreement. Note that the free-stream flow coming to the boat-tail region is not uniform in the experiment due to the obstacles such as the support system and the air supplier above inside the test section. The thrust efficiencies η are compared with the experiment for three pressure ratios in Fig. 12. The thrust efficiency η is defined as the ratio of the net thrust and the ideal thrust. The net thrust is the thrust computed by the simulation and the ideal thrust is the thrust that would be obtained when the nozzle flow expands isentropically. The agreement with the experiment in this figure is good for this global data although the computation estimated the thrust efficiency slightly lower than the experiment. It may be due to the disturbance in the free-stream flows mentioned above.

Fig. 11 Comparison of the schlieren picture (upper half) and the computed density gradient (lower half) (computation at Mach number 1.02)

Fig. 12 Comparison of the thrust coefficients for three pressure ratios (computation at Mach number 1.02)

4. Effect of Turbulence Models

There is always a discussion how accurate the solutions are when some turbulence model is used. In this study, Baldwin-Lomax model, the well-known and efficient algebraic turbulence model is used. The model tends to have large eddy viscosity in the recirculating region, and the Degani-Schiff modification⁽¹³⁾ is introduced. The Goldberg's one equation turbulence model⁽¹⁴⁾ is also tried for the comparison. The computed Mach number contours for the result with (a) original Baldwin-Lomax model, (b) Baldwin-Lomax model with Degani-Schiff modification, (c) Goldberg's 1 Eq. Model are plotted in Figs. 13(a) – (c), respectively. The case with the secondary flow injection is chosen as the flow field is relatively complex. The flow condition is same as Fig. 7. There is little difference in the global structure of the flow field except in front of the secondary flow injection. The

(a) Original Baldwin-Lomax model

(b) B-L model with Degani-Schiff modification

(c) Goldberg's 1 Eq. model

Fig. 13 Effect of the turbulence model: Mach number contour plots for the case with the secondary flow injection ($\xi=6.5$)

result for Degani-Schiff modification shows λ -type shock structure, and others do not. The C_p distributions shows difference in this region, and the resultant boat-tail drag shows about 10% difference although not shown here. However, the effect of the secondary flow injection is almost the same for the result with any turbulence model. The result indicates that the detailed structure of the flow field depends on the turbulence model used in the computation, but the qualitative discussion based the solutions with any turbulence model (at least tried) is acceptable.

5. Conclusions

Numerical simulations of the flows over a annular

plug nozzles were conducted and the boat-tail drag characteristics were clarified. The simulations were carried out for the axi-symmetric configuration using Navier-Stokes equations. The computed result showed that the primary cause of the boat-tail drag is the low pressure region downstream of the Prandtl-Meyer expansion. The flow field with the secondary flow injection was next simulated and the result showed that the secondary flow injection effectively increases the pressure in that region and the boat-tail drag is reduced. The computation of the boat-tail drag showed that the secondary flow injection is useful at all the three pressure ratios used in the present research. The effect of the turbulence model used in the computation was discussed and the results indicated that the global structure of the flow field does not depend on the turbulence model.

References

- (1) Tanatsugu, N., Development Study on ATREX Engine, 7th International Spaceplane and Hypersonic Systems & Technology Conference, Norfolk, VA, (1996).
- (2) Shima, E. and Jounouchi, T., Role of CFD in Aeronautical Engineering/No. 14—AUSM Type Upwind Scheme, Proc. 14th NAL Symposium on Aircraft Computational Aerodynamics, (1997), pp. 7-12.
- (3) Thomas, J.L., van Leer, B. and Walters, B.W., Implicit Flux-Split Schemes for the Euler Equations, AIAA Paper 85-1680, (1985).
- (4) Baldwin, B.S. and Lomax, H., This Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, (1978).
- (5) Obayashi, S., Matsushima, K., Fujii, K. and Kuwahara, K., Improvement in Efficiency and Reliability for Navier-Stokes Computations Using the LU-ADI Factorization Algorithm, AIAA Paper 86-0338, (1986).
- (6) Fujii, K. and Obayashi, S., Navier-Stokes Simulations of Transonic Flows over a Wing Fuselage Combination, AIAA Journal, Vol. 25, No. 12 (1987), pp. 1587-1596.
- (7) Fujii, K. and Tamura, Y., Effect of Engine Integration on the Aerodynamic Characteristics of a Spaceplane, International Journal of Computational Fluid Dynamics, Vol. 8, No. 4 (1997), pp. 235-246, Gordon and Breach Science Publishers.
- (8) Fujii, K., Unified Zonal Method Based on the Fortified Solution Algorithm, Journal of Computational Physics, Vol. 118 (1995), pp. 92-108.
- (9) Anderson, D.A., Tannehill, J.C. and Pletcher, R.H., Computational Fluid Mechanics and Heat Transfer, (1984), McGraw Hill.
- (10) Sato, T., Tanatsugu, N., Naruo, Y., Hatta, H., Omi, J., Tomike, J. and Minami, R., Development Study on ATREX Engine System, IAF-97-S. 5.01, 48th International Astronautical Congress, Turin, (1997).
- (11) Sato, T., Tanatsugu, N., Naruo, Y., Omi, J., Tomiike J. and Minami, R., Development Study for ATREX Engine Flight Test, 21st International Symposium on Space Technology and Science, Omiya, Japan, (1998).
- (12) Murooka, T., Experimental Investigation of the Plug Nozzle Boat-tail Drag Reduction, Master's Thesis, Dept. Aeronautics and Astronautics, School of Engineering, University of Tokyo, (in Japanese), (1997).
- (13) Degani, D. and Schiff, L.B., Computation of Turbulent Supersonic Flows around Pointed Bodies Having Crossflow Separation, J. Computational Physics, Vol. 66, No. 1 (1986).
- (14) Goldberg, U., Pointwise Turbulence Modeling for Engineering Applications, International CFD Workshop on Supersonic Transport Design, Tokyo, (1998).