NUMERICAL SIMULATION OF AXISYMMETRIC BASE FLOW IN THE PRESENCE OF PROPULSIVE JET

Debasis Chakraborty*, Pradeep Kumar*, R. Balu* and V. Adimurthy*

Abstract

Accurate estimation of flow parameters in the base region in the presence of propulsive jet is very important for the design of satellite launch vehicles. Simpler semi-empirical procedures can give only average values of the parameters in the base region, whereas it is expected that because of the presence of two different streams these parameters may vary. Numerical exploration of axisymmetric base flow in the presence of propulsive jet is performed by solving Navier Stokes equations with k-ε turbulence model. The computed base pressures are compared with the experimental values. Values of base pressure near the edges are much lower than the values in the central region.

Introduction

One of the major aerothermodynamic problems encountered by satellite launch vehicles during the ascent phase in the atmosphere is the problem of base heating, which is caused due to energy transfer from the rocket exhaust to the vehicle base. Understanding of the mechanisms involved in this energy transfer and having quantitative estimation of the problem are very important for the structural and thermal design of any launch vehicle.

Numerical computation of the base flow can be very helpful to understand this complex flow phenomena. In fact, most decisive progress in the base flow calculation certainly comes from the solution of the time averaged Navier Stokes equations with the turbulence closure [1,2,3,4,5,6]. Although, still extremely costly in computer time and not always quantitatively satisfactory, this approach allows a truly realistic prediction of the flow field structure. It is also probably the most straightforward way to extend the prediction capability to three-dimensional configuration, whereas the extension of simple semiempirical correlation methods to three dimensional flows is extremely hazardous and leads to nearly inextricable difficulty. Simpler semiempirical procedure [8] can predict only one average value of base pressure in the whole region. In the experimental work of Herrin and Dutton [9], it has been observed that even for flow past backward facing step without propulsive jet, significant lateral variation of base pressure exists. Sahu [10] simulated numerically the experimental condition of Herrin and Dutton [9] and predicted the radial variation of base pressure. This study reveals that $k-\varepsilon$ turbulence model can predict the radial variation of base pressure, where as the other algebraic turbulence models fail to predict the base pressure accurately. However, with a propulsive jet, very limited experimental or numerical studies are reported in the literature regarding the lateral variation of pressure in the base region. Petrie and Walker [11] compared numerical calculation of several workers [6, 12, 13] with experimental results of base flow in the presence of propulsive jet. The comparisons between the computation and experiment show a poor agreement and there were significant variations in the results from nominally similar prediction methods. Grid independence of the solutions were also not demonstrated. The accurate prediction of base pressure is very important for example, in determining the base drag characteristics with propulsive jets. This paper addresses to the numerical simulation of axisymmetric base flow in the presence of propulsive jet and in particular the problem of lateral base pressure variations.

In this work, numerical simulation of axisymmetric base flow in the presence of propulsive jet is performed to determine base pressure accurately. Navier Stokes equations are solved along with k-ε turbulence model using the software Aeroshape3D (AS3D) [7]. The results of the numerical computations are compared with the experimental value and the radial variation of base pressure along the base is presented.

---

*Aero and Flight Dynamics Group, Vikram Sarabhai Space Centre, Thiruvananthapuram-695 022, India

Manuscript received on 02 Dec 2000; Paper reviewed, revised and accepted on 08 Mar 2001

The Methodology

The details of the software Aeroshape3D (AS3D) are given in Ref.[7]. AS3D uses rectangular adaptive cartesian mesh to solve Navier Stokes equations on any arbitrary three dimensional body using time marching methodology. Turbulence is modeled by k-ε turbulence closure. The fluxes across the cell interface are computed by using a Riemann solver with mini-mod limiter to limit the fluxes across the interface. The scheme is of the TVD type with grid adaptation based on the flow gradients. The software has been validated extensively for various aerodynamics and propulsion problems and is being used routinely for solving design problems of aerospace vehicles. Important results of validation are summarized in References [14,15].

Results and Discussion

The axisymmetric base flow experiment of Reid and Husting [16] has been simulated numerically in the present work. In the experiment, the pressure in the annular base region in the presence of propulsive jet is measured. The model geometry used in the study is shown in Fig.1 along with the related dimensions. The point A at the nozzle throat at the axis of symmetry is considered as origin. Although the experimental study deals with various base configurations, in the present analysis only the configuration with nozzle divergence angle (δ) of 5 deg is considered. Further geometrical details are available in Ref.[16].

The inflow parameters of the simulations are same as those for experiment of Reid and Husting [16]. The free stream Mach number, static temperature and static pressure are 2.0, 162.77 K and 0.0107 Mpa respectively. The Mach number and static temperature at the throat of central propulsive jet 1.0 and 162.77. The static pressure of the jet at throat is varied (in the range of 0.0178 Mpa - 0.089 Mpa) to obtain various total pressure ratios (in the range of 0.4 to 2.0) of the two streams.

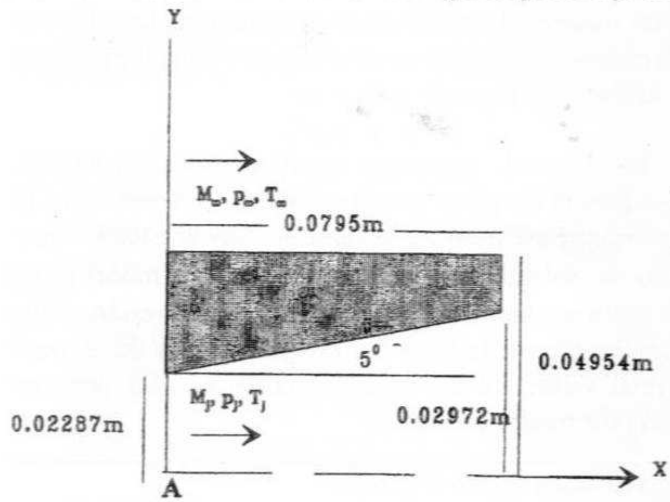

Fig. 1 Geometrical details of the model used in the study

The initial grid structure is shown in Fig.2 which contains 2514 cells. Although a uniform 50 x 50 initial grid is chosen, a few more cells have been added by the software to capture the body. The outflow boundary is taken at a distance of 12 body radius whereas the far field boundary is placed at a distance of 3 body radius. As the solution methodology is grid adaptive based on flow gradients, the number of cells keeps on increasing as the solution proceeds and as the flow develops. The grid structure at the end of 7000 iterations are shown in Fig.3. The grids are seen to be very fine in the region where there are significant flow gradients. The number of cells increased to 39572 at 7000 iteration from the initial value of 2514 cells.

The convergence of the solution is demonstrated by comparing the computed base pressure profiles for the case of $P_j/P_\infty = 0.4$ at different levels of iteration in Fig.4. Base pressure has been nondimensionalised with p_∞ (10700 Pa), whereas y_{base} (=0.04954 m) is used for nondimensionalising the radial distance. It can be seen from the figure that the maximum deviation of the base pressure between 5000 and 7000 iterations at any radial location is within 2%. Based on this observation it has been concluded that the solution has converged. Also it has been noticed that the base pressure is maximum near the center of the base and its value is much lower near the edge.

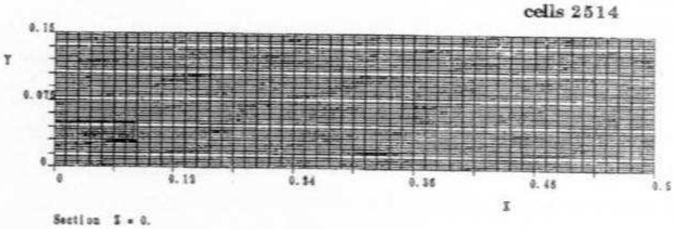

Fig. 2 Initial grid structure

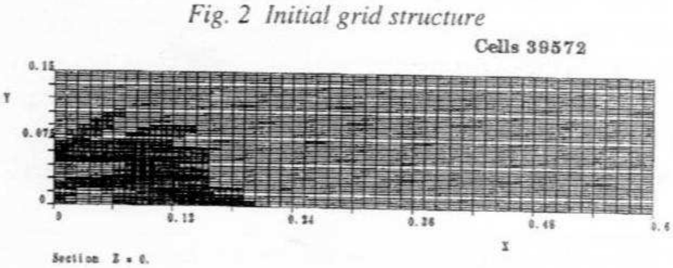

Fig. 3 Grid structure after 7000 iteration

The recirculating flow and the dead air zone in the base region are presented through the velocity vector plot in Fig.5. Only a portion of the flow field highlighting the feature in the base region is presented. The dead air zone and the recirculating flow is clearly visible in the figure. The length of the recirculating zone is about 10 mm from the base.

Radial variation of nondimensionalised base pressure obtained from the computation (marked AS3D in the figure) is compared with the experimental value [16] and the two dimensional semiempirical procedure BASE2D [8] in the Fig.6 for two pressure ratios (P_j/P_∞) of 0.4 and 1.8. The AS3D results are showing higher values compared to the experimental results (measurements of base pressure have been made at a distance of 0.045 m from the axis of symmetry). Whether the modeling of turbulence is playing any role to explain this mismatch is yet to be studied. As stated earlier, the simpler semiempirical procedure can give only one average value of base pressure throughout the base region, and radial variation of base pressure cannot be predicted through this approach. Moreover, with the increase of total pressure ratios the flow is departing more and more from two dimensionality, BASE2D is predicting much higher base pressure to AS3D for $P_j/P_\infty$of 1.8 compared to AS3D and experimental data.

Conclusions and Recommendations

Axisymmetric base flow in the presence of propulsive jet is explored numerically by solving Navier Stokes equations with $k-\varepsilon$ turbulence model by using the existing software Aeroshape3D. The computed base pressure is observed to be higher than the experimental value. The base pressure is found to be not uniform along the radial distance. The values near the edges of the base are much lower than away from the edges.

Detailed measurements of pressure in the base region in presence of propulsive jet are recommended to determine the radial variation of base pressure. This can help to understand the complex base flow phenomena and further validate the computational tools a great deal. The modeling of turbulence can be studied further to determine whether the modeling has any effect in the mismatch between the experimental and numerical values.

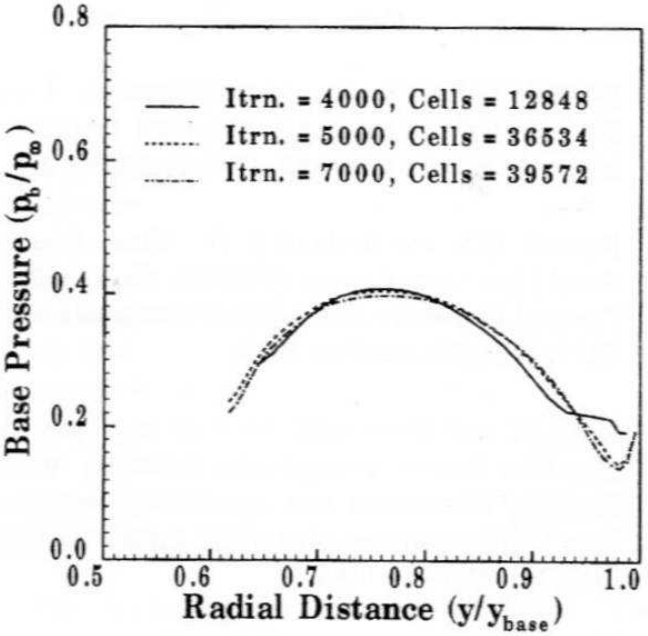

Fig. 4 Base pressure profile variation with iterations

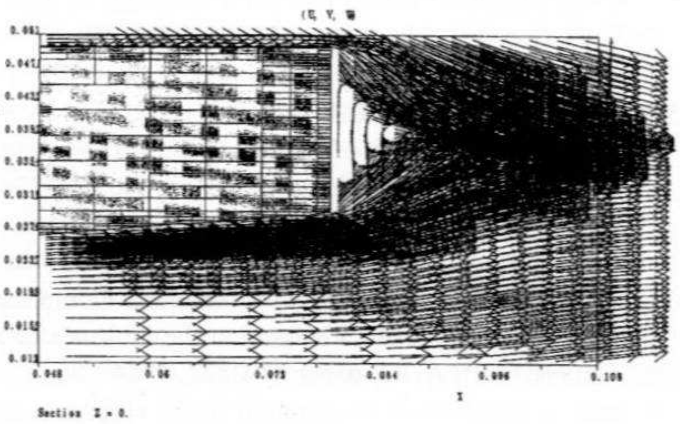

Fig. 5 Velocity vector plot in the base region

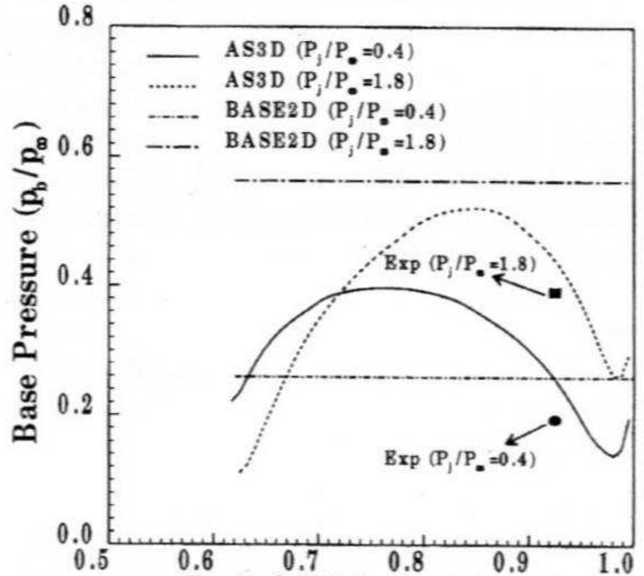

Fig. 6 Comparison for nondimensionalised base pressure profile

References

1. Deiwert, G.S., "Supersonic Axisymmetric Flow Over Boattails Containing a Centered Propulsive Jets", AIAA Journal, Vol.22, 1984, pp.1358-1365

2. Deiwert, G.S. and Rothmund, H., "Three Dimensional Flow Over Conical Afterbody Containing a Centered Propulsive Jet - a Numerical Simulation", AIAA Paper No.83-1709, 1983.

3. Sinha, N. and Dash, S.M., Overlap Approach for Base Flow Analysis of Supersonic Velocities : Basic Modeling Description and Application Including Base Combustion and Multiphase Effects", AIAA Paper No. 85-1673, 1985.

4. Sahu, J., Nietubitz, C.J. and Steger, J.L., "Numerical Computation of Base Flow for a Projectile at Transonic Speeds", AIAA Paper No.82-1358, 1982.

5. Sahu, J., Nietubitz, C.J. and Steger, J.L., "Navier Stokes Computation of Projectile Base Flow with and without Injection", AIAA Paper No.83-0224, 1982.

6. Sahu, J. and Nietubitz, C.J., "Numerical Computation of Base Flow for a Missile in the Presence of a Centered Jet", AIAA paper 84-0527, 1984.

7. Aeroshape3D - Technique and General Description, Computational Aerodynamics System Co., Moscow, 1994.

8. Debasis Chakraborty., "BASE2D - A Two dimensional Procedure to Calculate Base Flow"" Report No. VSSC/ARD/TR-004/Aug'98, Aerodynamics Division, Vikram Sarabhai Space Centre, Thiruvananthapuram, 1998.

9. Herrin, J.L. and Dutton, J.C., "An Experimental Investigation of the Supersonic Axisymmetric Base Flow Behind a Cylindrical Afterbody", Report No. UILU 91-4004, 1991.

10. Sahu, J., "Numerical Computation of Supersonic Base Flow with Special Emphasis on Turbulence Modeling", AIAA Journal, Vol.32, No.7, 1994, pp.1547-1549.

11. Petrie, H.L. and Walker, B.J., "Comparison of Experiment and Computation for a Missile Base Region Flowfield with a Centered Propulsive Jets", AIAA Paper No.85-1618, 1985.

12. Deiwert, G.S., Andrews, A.E. and Nakahashi, K., "Theoretical Analysis of Aircraft Afterbody Flow", AIAA Paper No.84-1524, 1984.

13. Thomas, P.D., Reklis, R.P., Roloff, R.R. and Conti, R.J., "Numerical Simulation of Axisymmetric Base Flow on Tactical Missile with Propulsive Jet", AIAA Paper No. 84-1658, 1984.

14. Pradeep Kumar., Swaminathan, R., Unnikrishnan, C., Ashok, V. and Adimurthy, V., "Modelling of jet Impingement Forces during Hot Stage Separation of a Launch Vehicle", AIAA Paper No. 98-3156, 1998.

15. Ashok, V., Devasia, K.J. and Jeyajothiraj, P., "Computational Studies for Flow Over Second Launch pad Jet Deflector Configurations", Report No. VSSC/ARD/TM/SLP-002/99, Aerodynamics Division, Vikram Sarabhai Space Centre, Thiruvananthapuram, 1999.

16. Reid, J. and Hastings, R.C., "The Effect of a Central Jet on the Base Pressure of a Cylindrical Afterbody in a Supersonic Stream", Report No. ARC R & M 3224, 1961.