Effects of Nose-Bluntness Ratio on Aerodynamic Performance for Re-entry Vehicle

Sheikh Arslan Ali, Dr. Mukkarum Husain, Dr. M. Nauman Qureshi
mrmukkarum@yahoo.com

Abstract— **The objective of the present work is twofold, first is to validate the methodology of calculating aerodynamics performances of high speed vehicle using CFD software Fluent, and secondly to study the aerodynamic characteristics of different symmetric re-entry vehicles. Aerodynamic performances of symmetric re-entry vehicles with different configurations and orientations are computed. Focus is made on effects of nose-bluntness ratio on aerodynamic performances for re-entry vehicle. Static aerodynamic coefficients (angles of attack of 0^0 to 20^0) for spherically blunted cones having different cone half-angle and nose bluntness ratio are calculated using CFD. Comparisons between present numerical and available experimental results were made and found in good agreement. Measured numerical results validate the methodology to calculate aerodynamic performance of re-entry vehicle and confirm that the effects of nose-bluntness ratio are small for the large half cone angle configurations as compared to small half cone angle configurations.**

I. Introduction

Allen and Eggers [1] showed that the heat load experienced by a re-entry vehicle is inversely proportional to the drag coefficient, i.e., the greater the drag; lesser be the heat load. Since then re-entry techniques rely on the use of very blunt, high-drag configurations that avoid high heating rates at the expense of poor controllability. The U.S. Mercury, Gemini and Apollo and the Soviet Vostok, Voskhod and Soyuz were ballistic re-entry vehicles [2-6]. These vehicles had ablative heat shields, and used parachutes to further slow down at the last period of flight, examples are shown in Figure 1. Lift is generated when the vehicle flies at a nonzero angle of attack and the (L/D) of vehicle is relatively small. Drag on the vehicle depends on the density of the air, the shape, mass, nose diameter and roughness. The atmospheric maneuvering capability is attained by trim at a nonzero angle of attack with the center of gravity off the centerline and relatively near the blunt spherical heat shield.

Computational Fluid Dynamics [7-9] (CFD) as a computational technology is extremely appropriate to build up the concept of numerical test rig. It is also becoming a mature discipline for high speed applications. Re-entry vehicle requires a precise understanding of all physical phenomena that happened in the flow field to evaluate its aerodynamics and aerothermodynamics performance. This requires a number of wind tunnel and flight tests which are costly and time consuming. CFD can be used as a numerical test rig to significantly reduce the number of wind tunnel and flight tests.

The aim of the current work is to study the effect of aerodynamic characteristics of different symmetric re-entry vehicles [10]. Four configurations of different half cone angles of 30^0 and 60^0; and nose bluntness ratio of 0.25 and 0.50 are generated and numerically simulated. Focus is made on effects of nose-bluntness ratio on aerodynamic performances for re-entry vehicle. Static aerodynamic coefficients (angles of attack of 0^0 to 20^0) for spherically blunted cones having different cone half-angle and nose bluntness ratio are calculated using CFD. Comparisons between present numerical and available experimental [11] results are made and found in good agreement. Measured numerical results confirm that the effects of nose-bluntness ratio are small for the large half cone angle configurations as compared to small half cone angle configurations.

Figure 1: Ballistic Re-entry Vehicles, taken from NASA websites

II. Model Geometries

Four different cone models used for present study are listed in Table 1. Models are shown in Figure 2.

Table 1: Geometric Details of Cone Models

θ (degree)	Rn/Rb	Rb (cm)
30	0.25	2.529
30	0.50	2.530
60	0.25	2.531
60	0.50	2.543

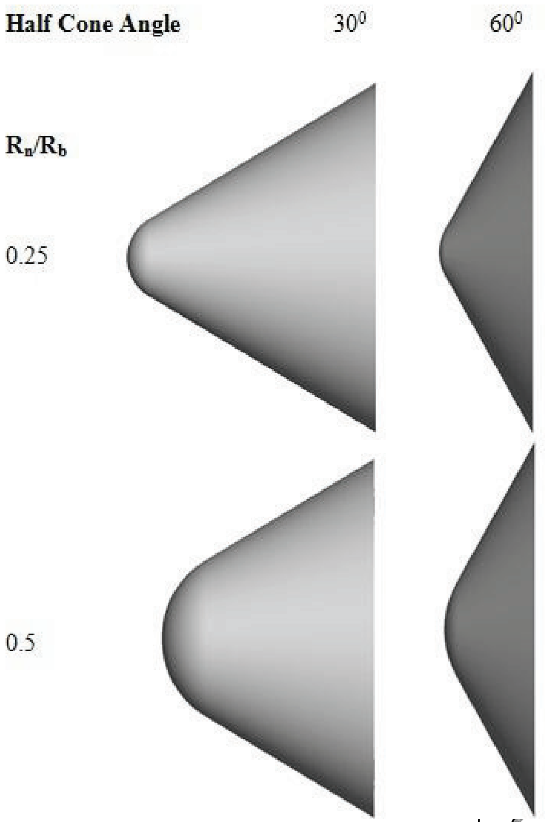


Figure 2: Different Configuration used in present study

III. MESH GENERATION AND SOLVER SETUP

Gridgen software is used to generate mesh around selected configurations as shown in Figure 3. Approximately 0.4 million hexahedral cells are generated on each configuration. Density-based Roe-FDS for solver and Sutherland formula for viscosity is used. In all simulations, the flows are assumed to be laminar, and the free stream conditions listed in Table 2 assured that the perfect gas assumption is still valid. Simulations are carried out on workstation of 2 x quad core 2.4 GHz and 16 GB RAM. Good convergence is achieved specially in case of smaller half cone angle.

TABLE 2: FREESTREAM CONDITIONS

M_∞	P_∞	T_∞
5.9	193.79 pa	54.13 k

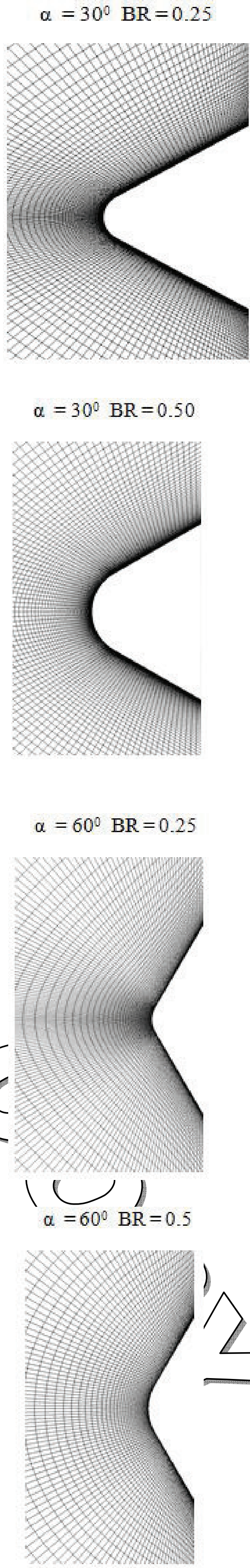
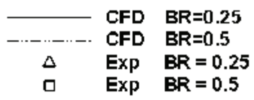


Figure 3: Mesh around the wall of cone

IV. Results and Discussion

Different blunt cone configurations are simulated using CFD and compared with available experimental data. Present computations are in good agreement with experimental results, which insure the capability of CFD to use as a numerical wind tunnel for calculating aerodynamics performances of high speed vehicle configurations.

Figures 4-6 show comparison of CFD and experimental results for 30^0 half cone angle configurations. In case of axial force, CFD results are slightly over predicted but showing same behavior as depicted by experimental results. CFD results for normal force and pitching moment are in good agreement with experimental results. CFD results confirm that the effect of nose-bluntness ratio on aerodynamic performance is significant in the case of small half cone angle configurations. Increase in axial force while decrease in normal force and pitching moment is noticed with increase in bluntness ratio. Figures 7-9 show comparison of CFD and experimental results for 60^0 half cone angle configurations. CFD results for axial force, normal force, and pitching moment are in good agreement with experimental results. CFD results confirm that the effect of nose-bluntness ratio on aerodynamic performance is small in the case of large half cone angle configurations.



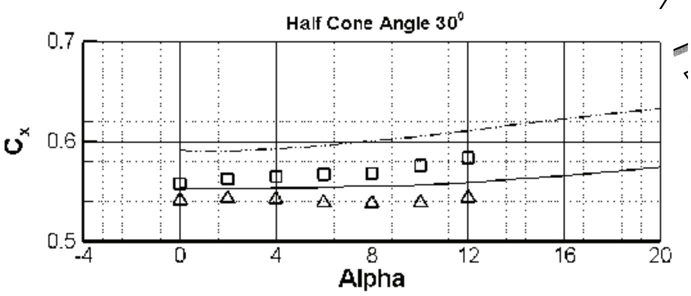


Figure 4: Comparison of Axial Force Coefficient ($\theta = 30^0$)

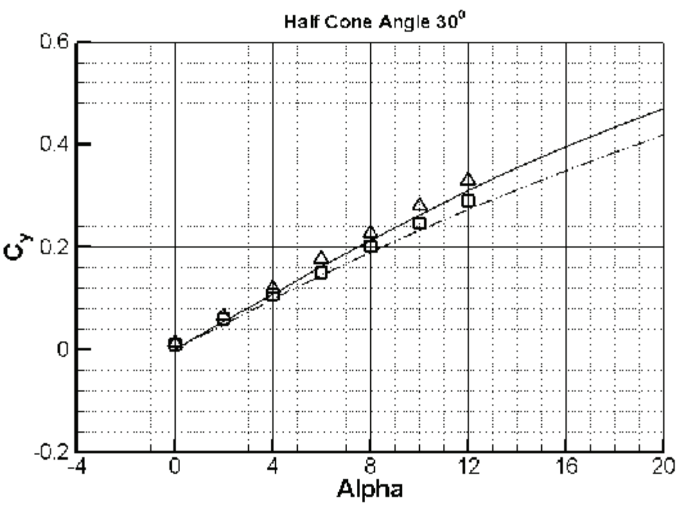


Figure 5: Comparison of Normal Force Coefficient ($\theta = 30^0$)

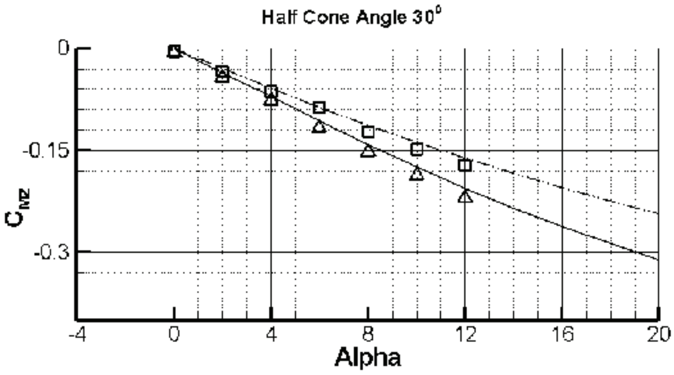


Fig 6: Comparison of Pitching Moment Coefficient ($\theta = 30^0$)

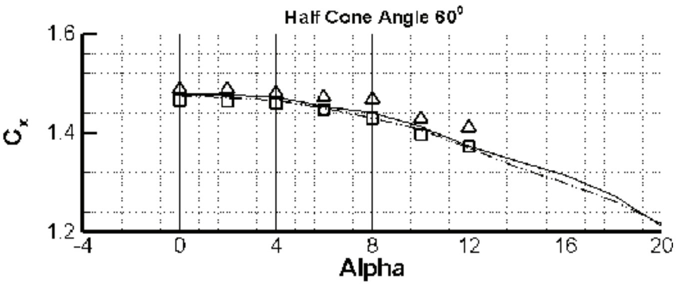


Figure 7: Comparison of Axial Force Coefficient ($\theta = 60^0$)

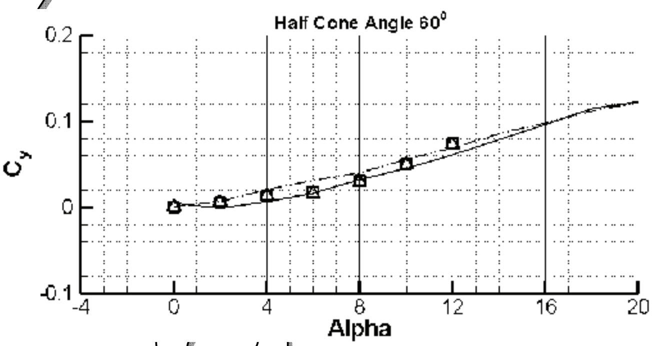


Figure 8: Comparison of Normal Force Coefficient ($\theta = 60^0$)

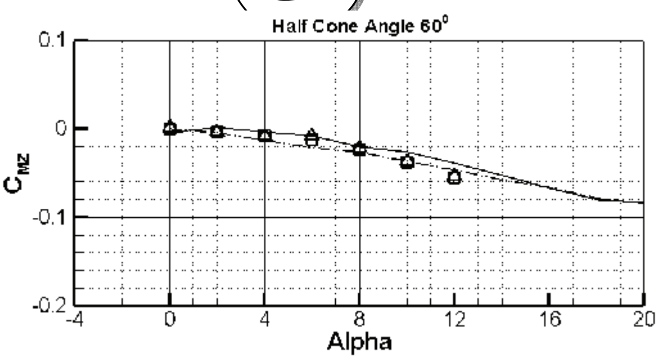


Figure 9: Comparison of Pitching Moment Coefficient ($\theta = 60^0$)

V. CONCLUSION

In the present study, the methodology of calculating the aerodynamics performances of high speed vehicle using CFD software Fluent is validated and aerodynamic performance of different symmetric re-entry vehicles are studied. Numerical results are in good agreement with experimental data and depict that CFD as a computational technology is appropriate to develop the concept of numerical test rig. CFD results also confirm that the effects of nose-bluntness ratio are small for large half cone angle configurations as compared to small half cone angle configurations..

REFERENCES

[1] Allen, H. J., Eggers, A. J., Jr., "A Study of the Motion and Aerodynamic Heating of Missiles Entering the Earth's Atmosphere at High Supersonic Speeds," NACA TN-4047, Oct. 1957.

[2] R. C. Mehta, "Three-Dimensional Inviscid Flow field Computation over Various Reentry Configurations," 45th AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2007-133.

[3] Robert B. Hruschka, Gisu Park, Sean O'Byrne, Harald Kleine, "PLIF-Velocity Measurements and Computations of a Hypersonic Wake Flow Field," 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference, AIAA 2009-7261.

[4] Kamal M. Shweyk, B.F. Tamrat, Abdi Khodadoust, "Parametric Shape Study of Capsule-Type Vehicles during Atmospheric Re-entry," AIAA Atmospheric Flight Mechanics Conference and Exhibit 21 - 24 August 2006, Keystone, Colorado, AIAA 2006-6140.

[5] Antonio Viviani, Giuseppe Pezzella, Davide Cinquegrana, "Aerothermodynamic Analysis of an Apollo-like Reentry Vehicle," 14th AIAA/AHI Space Planes and Hypersonic Systems and Technologies Conference, AIAA 2006-8082.

[6] James L. Brown, Joseph A. Garcia, David J. Kinney, "An Asymmetric Capsule Vehicle Geometry Study for CEV," 45th AIAA Aerospace Sciences Meeting and Exhibit, 8 - 11 January 2007, Reno, Nevada, AIAA 2007-604.

[7] J. C. Tannehill, D. A. Anderson, R. H. Pletcher, "Computational Fluid Mechanics and Heat Transfer," Second Edition, January 1997.

[8] John D. Anderson, Jr., "Computational Fluid Dynamics, the Basics with Application," McGraw-Hill Series in Mechanical Engineering, 1995.

[9] Toro EF., "Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction," Springer, 1997.

[10] Mukkarum Husain, Shamoon Jamsheed, "Parametric Studies of Hypersonic Aerodynamic Performances of Asymmetric Re-entry Vehicles," IBCAST 2011, Islamabad, Pakistan, January 11 – 14, 2011.

[11] Robert L. Calloway and Nancy H. White, "Measured and Predicted Shock Shapes and Aerodynamic Coefficients for Blunted Cones at Incidence in Air at Mach 5.9," NASA Technical Paper 1652, May 1980.